Works 2013

Water Networks

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

Water Networks

TechnoLogismiki

Water Networks

Publisher

TechnoLogismiki

Editors

Fotis Fotopoulos
Aristotelis Charalampakis

Technical Assistance

Antigoni Egglezou

Printed: September 2012 in Athens, Greece.

Table of Contents

Chapter V View

Chapter VI Data

Chapter

1 About the program

1.1 What does the program do?

Water Networks is a computer program that performs extended period simulation of hydraulic and water quality behavior within pressurized pipe networks. A network consists of pipes, nodes (pipe junctions), pumps, valves and storage tanks or reservoirs. The program tracks the flow of water in each pipe, the pressure at each node, the height of water in each tank, and the concentration of a chemical species throughout the network during a simulation period comprised of multiple time steps.

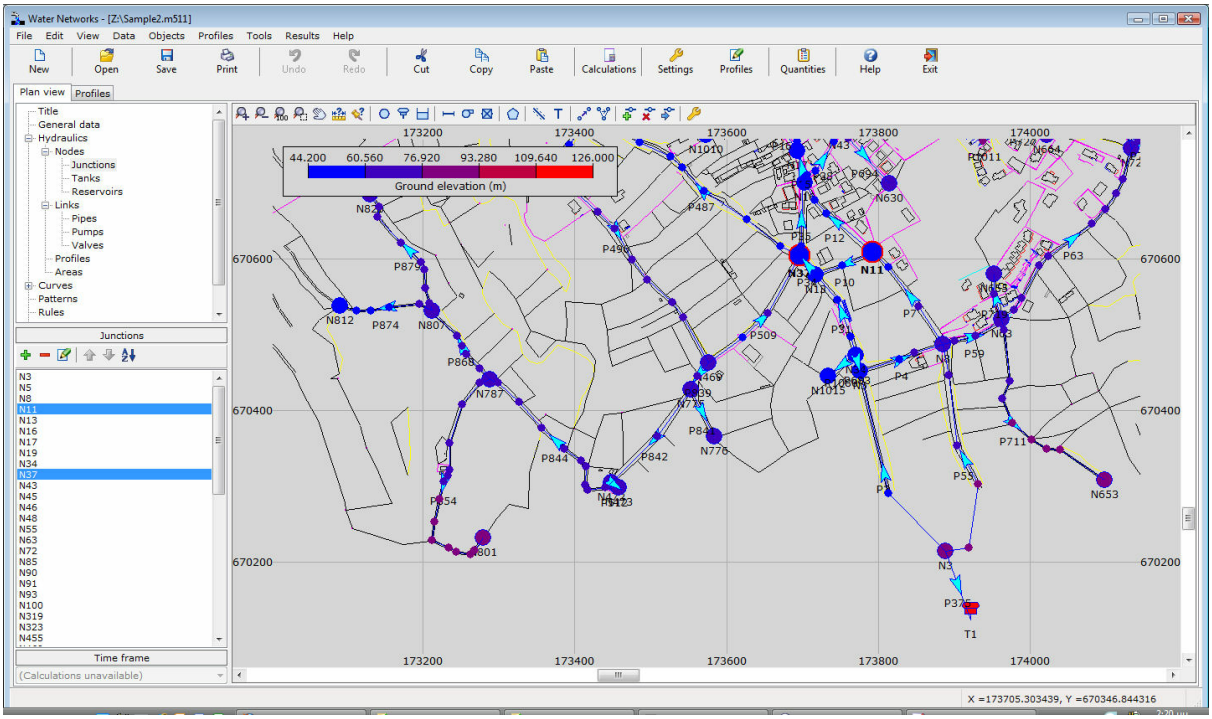
In addition to chemical species, water age and source tracing can also be simulated. The program is designed to be a research tool for improving our understanding of the movement and fate of drinking water constituents within distribution systems. It can be used for many different kinds of applications in distribution systems analysis.

Sampling program design, hydraulic model calibration, chlorine residual analysis, and consumer exposure assessment are some examples. The program can help assess alternative management strategies for improving water quality throughout a system.

These can include:

- altering source utilization within multiple source systems
- altering pumping and tank filling/emptying schedules
- use of satellite treatment, such as re-chlorination at storage tanks
- targeted pipe cleaning and replacement.

The program is based on and fully compatible with EPA EPANET 2. It offers unique design tools and extensive import/export capabilities through DXF/GTM/GIS files.



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

- Handles Water / Irrigation networks
- Easy data input in plan view and/or profile view with embedded spreadsheets.
- Fully compatible with American regulations (ASCE & WPCF) as well as Greek Regulations
- Easy data input from drawings and / or spreadsheets
- Automatic data input based on logical rules
- Active profile drawings with CAD capabilities
- Conduit / Manhole / Trench specifications
- One-click generation of professional reports
- One-click generation of plan view and profile drawings
- One-click generation of quantities report, including pipe lengths, excavations, backfill etc.

1.2 Minimum requirements

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

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

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

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

1.3 Technical support

Support through the Internet

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

Support by e-mail

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

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

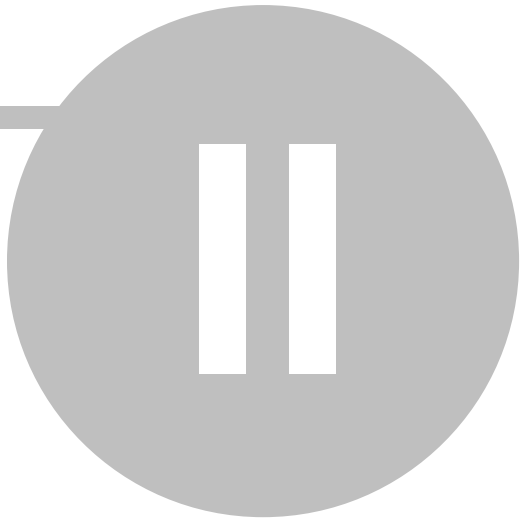
The normal response time is within two business days. If your inquiry cannot be answered by e-mail, a customer service representative will contact you by telephone.

Interactive Support

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

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

Chapter



2 User Interface

2.1 Methodology

2.1.1 Computational Methods

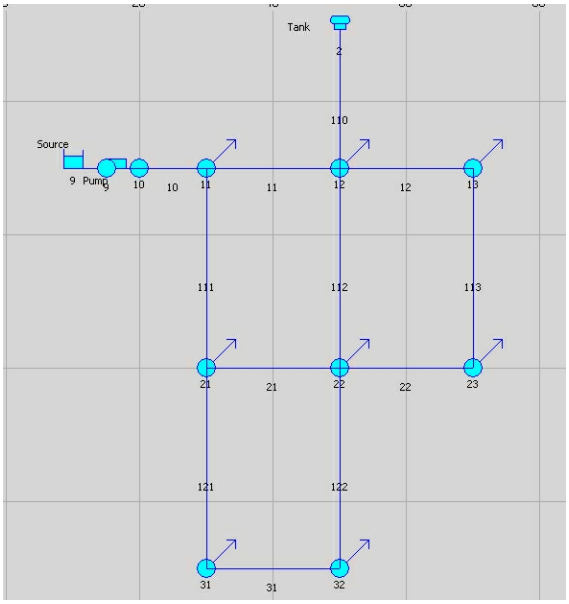
The program and this manual are based on EPA's EPANET 2, which is a computer program that performs extended period simulation of hydraulic and water quality behavior within pressurized pipe networks.

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

- Physical components
- Non-physical components
- Hydraulic simulation model
- Water quality simulation model

2.1.2 Physical components

Water Networks models a water distribution system as a collection of links connected to nodes. The links represent pipes, pumps, and control valves. The nodes represent junctions, tanks, and reservoirs. The figure below illustrates how these objects can be connected to one another to form a network.



Physical components may include:

- Junctions
- Reservoirs
- Tanks
- Emitters

- Pipes
- Pumps
- Valves

2.1.2.1 Junctions

Junctions are points in the network where links join together and where water enters or leaves the network. The basic input data required for junctions are:

- elevation above some reference (usually mean sea level)
- water demand (rate of withdrawal from the network)
- initial water quality

The output results computed for junctions at all time periods of a simulation are:

- hydraulic head (internal energy per unit weight of fluid)
- pressure
- water quality

Junctions can also:

- have their demand vary with time
- have multiple categories of demands assigned to them
- have negative demands indicating that water is entering the network
- be water quality sources where constituents enter the network
- contain emitters (or sprinklers) which make the outflow rate depend on the pressure.

2.1.2.2 Reservoirs

Reservoirs are nodes that represent an infinite external source or sink of water to the network. They are used to model such things as lakes, rivers, groundwater aquifers, and tie-ins to other systems. Reservoirs can also serve as water quality source points.

The primary input properties for a reservoir are its hydraulic head (equal to the water surface elevation if the reservoir is not under pressure) and its initial quality for water quality analysis.

Because a reservoir is a boundary point to a network, its head and water quality cannot be affected by what happens within the network. Therefore it has no computed output properties. However its head can be made to vary with time by assigning a time pattern to it.

2.1.2.3 Tanks

Tanks are nodes with storage capacity, where the volume of stored water can vary with time during a simulation. The primary input properties for tanks are:

- bottom elevation, where water level is zero
- diameter or shape if non-cylindrical
- initial, minimum and maximum water levels
- initial water quality

The principal outputs computed over time are:

- hydraulic head which is the water surface elevation
- water quality

Tanks are required to operate within their minimum and maximum levels. The program stops outflow if a tank is at its minimum level and stops inflow if it is at its maximum level. Tanks can also serve as water quality source points.

2.1.2.4 Emitters

Emitters are devices associated with junctions that model the flow through a nozzle or orifice that discharges to the atmosphere. The flow rate through the emitter varies as a function of the pressure available at the node:

$$q = C \cdot p^{\gamma}$$

where: **q**, the flow rate
p, pressure
C, discharge coefficient and
γ, pressure exponent.

For nozzles and sprinkler heads γ equals 0.5 and the manufacturer usually provides the value of the discharge coefficient in units of (L/s)/atm$^{0.5}$ or gpm/psi$^{0.5}$, stated as the flow through the device at a 1 atm (1 psi) pressure drop.

Emitters are used to model flow through sprinkler systems and irrigation networks. They can also be used to simulate leakage in a pipe connected to the junction (if a discharge coefficient and pressure exponent for the leaking crack or joint can be estimated) or compute a fire flow at the junction (the flow available at some minimum residual pressure). In the latter case one would use a very high value of the discharge coefficient (e.g., 100 times the maximum flow expected) and modify the junction's elevation to include the equivalent head of the pressure target. The program treats emitters as a property of a junction and not as a separate network component.

2.1.2.5 Pipes

Pipes are links that convey water from one point in the network to another. The program assumes that all pipes are full at all times. Flow direction is from the end at higher hydraulic head (internal energy per weight of water) to that at lower head. The principal hydraulic input parameters for pipes are:

- start and end nodes
- diameter
- length
- roughness coefficient for determining headloss
- status which can be either open, closed, or contains a check valve

The status parameter allows pipes to implicitly contain shutoff (gate) valves and check (non-return) valves (which allow flow in only one direction).

The water quality inputs for pipes consist of:

- bulk reaction coefficient
- wall reaction coefficient

These coefficients are explained more thoroughly in the water quality reactions section.

Computed outputs for pipes include:

- flow rate
- velocity
- headloss
- Darcy-Weisbach friction factor
- average reaction rate (over the pipe length)
- average water quality (over the pipe length).

The hydraulic head lost by water flowing in a pipe due to friction with the pipe walls can be computed using one of three different formulas:

- Hazen-Williams formula
- Darcy-Weisbach formula
- Chezy-Manning formula

The Hazen-Williams formula is the most commonly used headloss formula in the US while the Darcy-Weisbach formula is the most commonly used formula in Europe. The Hazen-Williams formula cannot be used for liquids other than water and was originally developed for turbulent flow only. The Darcy-Weisbach formula is the most theoretically correct. It applies over all flow regimes and to all liquids. The Chezy-Manning formula is more commonly used for open channel flow.

Each formula uses the following equation to compute headloss between the start and end node of the pipe:

$$h_L = A \cdot q^B$$

where: **h_L**, headloss (Length)
q, flow rate (Volume/Time)
A, resistance coefficient and
B, flow exponent

Each formula uses a different pipe roughness coefficient that must be determined empirically. Be aware that a pipe's roughness coefficient can change considerably with age.

With the Darcy-Weisbach formula the program uses different methods to compute the friction factor f depending on the flow regime:

- The Hagen–Poiseuille formula is used for laminar flow (Re < 2,000).
- The Swamee and Jain approximation to the Colebrook-White equation is used for fully turbulent flow (Re > 4,000).
- A cubic interpolation from the Moody Diagram is used for transitional flow (2,000 < Re < 4,000)

Pipes can be set open or closed at preset times or when specific conditions exist, such as when tank levels fall below or above certain set points, or when nodal pressures fall below or above certain values. This is implemented by using simple or rule based controls.

Minor Losses

Minor head losses (also called local losses) are caused by the added turbulence that occurs at bends and fittings. The importance of including such losses depends on the layout of the network and the degree of accuracy required. They can be accounted for by assigning the pipe a minor loss coefficient. The minor headloss becomes the product of this coefficient and the velocity head of the pipe, i.e.,

$$h_L = K\left(\frac{V^2}{2g}\right)$$

where: **K**, minor loss coefficient
V, flow velocity (Length/Time) and
g, acceleration of gravity (Length/Time^2)

For minor loss coefficients values for several types of fittings please see appendix.

2.1.2.6 Pumps

Pumps are links that impart energy to a fluid thereby raising its hydraulic head. The principal input parameters for a pump are its start and end nodes and its pump curve (the combination of heads and flows that the pump can produce). In lieu of a pump curve, the pump could be represented as a constant energy device, one that supplies a constant amount of energy (horsepower or kilowatts) to the fluid for all combinations of flow and head.

The principal output parameters are flow and head gain. Flow through a pump is unidirectional and the program will not allow a pump to operate outside the range of its pump curve.

Variable speed pumps can also be considered by specifying that their speed setting be changed under these same types of conditions. By definition, the original pump curve supplied to the program has a relative speed setting of 1. If the pump speed doubles, then the relative setting would be 2; if run at half speed, the relative setting is 0.5 and so on. Changing the pump speed shifts the position and shape of the pump curve (see the section on pump curves below).

As with pipes, pumps can be turned on and off at preset times or when certain conditions exist in the network. A pump's operation can also be described by assigning it a time pattern of relative speed settings. The program can also compute the energy consumption and cost of a pump. Each pump can be assigned an efficiency curve and schedule of energy prices. If these are not supplied then a set of global energy options will be used.

Flow through a pump is unidirectional. If system conditions require more head than the pump can produce, the program shuts the pump off. If more than the maximum flow is required, the program extrapolates the pump curve to the required flow, even if

this produces a negative head. In both cases a warning message will be issued.

2.1.2.7 Valves

Valves are links that limit the pressure or flow at a specific point in the network. Their principal input parameters include:

- start and end nodes
- diameter
- setting
- status

The computed outputs for a valve are flow rate and headloss. The different types of valves included in the program are:

- Pressure Reducing Valve (PRV)
- Pressure Sustaining Valve (PSV)
- Pressure Breaker Valve (PBV)
- Flow Control Valve (FCV)
- Throttle Control Valve (TCV)
- General Purpose Valve (GPV)

PRVs limit the pressure at a point in the pipe network. The program computes in which of three different states a PRV can be in:

- partially opened (i.e., active) to achieve its pressure setting on its downstream side when the upstream pressure is above the setting
- fully open if the upstream pressure is below the setting
- closed if the pressure on the downstream side exceeds that on the upstream side (i. e., reverse flow is not allowed).

PSVs maintain a set pressure at a specific point in the pipe network. The program computes in which of three different states a PSV can be in:

- partially opened (i.e., active) to maintain its pressure setting on its upstream side when the downstream pressure is below this value
- fully open if the downstream pressure is above the setting
- closed if the pressure on the downstream side exceeds that on the upstream side (i. e., reverse flow is not allowed).

PBVs force a specified pressure loss to occur across the valve. Flow through the valve can be in either direction. PBV's are not true physical devices but can be used to model situations where a particular pressure drop is known to exist.

FCVs limit the flow to a specified amount. The program produces a warning message if this flow cannot be maintained without having to add additional head at the valve (i. e., the flow cannot be maintained even with the valve fully open).

TCVs simulate a partially closed valve by adjusting the minor head loss coefficient of the valve. A relationship between the degree to which a valve is closed and the resulting head loss coefficient is usually available from the valve manufacturer.

GPVs are used to represent a link where the user supplies a special flow - head loss relationship instead of following one of the standard hydraulic formulas. They can be used to model turbines, well draw-down or reduced-flow backflow prevention valves.

Shutoff (gate) valves and check (non-return) valves, which completely open or close pipes, are not considered as separate valve links but are instead included as a property of the pipe in which they are placed.

Each type of valve has a different type of setting parameter that describes its operating point (pressure for PRVs, PSVs, and PBVs; flow for FCVs; loss coefficient for TCVs, and head loss curve for GPVs). Valves can have their control status overridden by specifying they be either completely open or completely closed. A valve's status and its setting can be changed during the simulation by using control statements.

Because of the ways in which valves are modeled the following rules apply when adding valves to a network:

- a PRV, PSV or FCV cannot be directly connected to a reservoir or tank (use a length of pipe to separate the two)
- PRVs cannot share the same downstream node or be linked in series
- two PSVs cannot share the same upstream node or be linked in series
- a PSV cannot be connected to the downstream node of a PRV.

2.1.3 Non-physical components

In addition to physical components, the program employs three types of informational objects – curves, patterns, and controls - that describe the behavior and operational aspects of a distribution system.

Non-physical components are:

- Curves
- Time patterns
- Controls

2.1.3.1 Curves

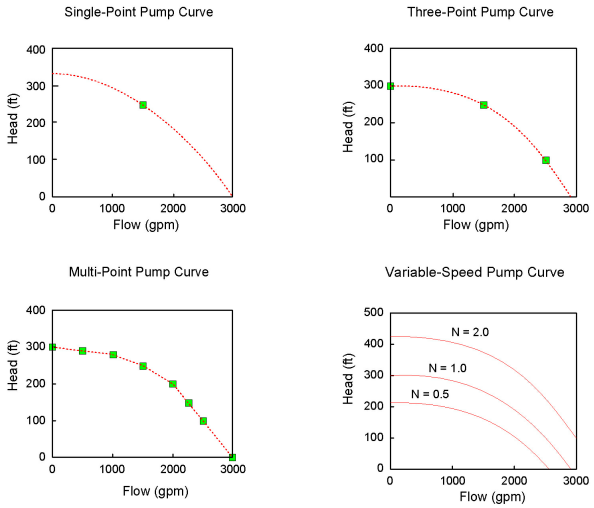
Curves are objects that contain data pairs representing a relationship between two quantities. Two or more objects can share the same curve. The program can utilize the following types of curves:

- Pump curves
- Efficiency curves
- Volume curves
- Head Loss curves

2.1.3.1.1 Pump curves

A pump curve represents the relationship between the head and flow rate that a pump can deliver at its nominal speed setting. Head is the head gain imparted to the water by the pump and is plotted on the vertical (Y) axis of the curve in feet (meters).

Flow rate is plotted on the horizontal (X) axis in flow units. A valid pump curve must have decreasing head with increasing flow. The program will use a different shape of pump curve depending on the number of points supplied (see Figure below):



Single-Point Curve
A single-point pump curve is defined by a single head-flow combination that represents a pump's desired operating point. The program adds two more points to the curve by assuming a shutoff head at zero flow equal to 133% of the design head and a maximum flow at zero head equal to twice the design flow. It then treats the curve as a three-point curve.

Three-Point Curve
A three-point pump curve is defined by three operating points: a Low Flow point (flow and head at low or zero flow condition), a Design Flow point (flow and head at desired operating point), and a Maximum Flow point (flow and head at maximum flow). The program tries to fit a continuous function of the form

$$h_G = A - B \cdot q^C$$

through the three points to define the entire pump curve. In this function:

hg, head gain
q, flow rate and
A, **B**, and **C** are constants

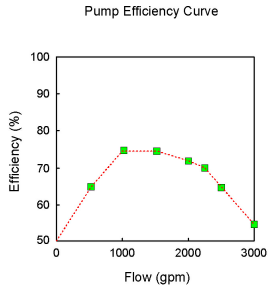
Multi-Point Curve
A multi-point pump curve is defined by providing either a pair of head-flow points or four or more such points. The program creates a complete curve by connecting the points with straight-line segments.

Variable Speed Pumps
For variable speed pumps, the pump curve shifts as the speed changes. The relationships between flow (Q) and head (H) at speeds N1 and N2 are:

$$\frac{Q_1}{Q_2} = \frac{N_1}{N_2}, \frac{H_1}{H_2} = \left(\frac{N_1}{N_2}\right)^2$$

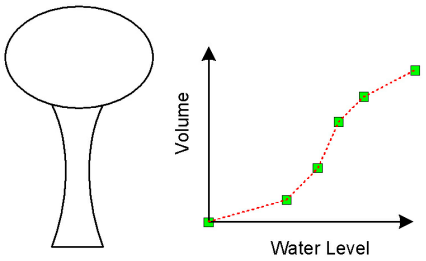
2.1.3.1.2 Efficiency curves

An efficiency curve determines pump efficiency (Y in percent) as a function of pump flow rate (X in flow units). An example efficiency curve is shown in the figure below. Efficiency should represent wire-to-water efficiency that takes into account mechanical losses in the pump itself as well as electrical losses in the pump's motor. The curve is used only for energy calculations. If not supplied for a specific pump then a fixed global pump efficiency will be used.



2.1.3.1.3 Volume curves

A volume curve determines how storage tank volume (Y in cubic feet or cubic meters) varies as a function of water level (X in feet or meters). It is used when it is necessary to accurately represent tanks whose cross-sectional area varies with height. The lower and upper water levels supplied for the curve must contain the lower and upper levels between which the tank operates. An example of a tank volume curve is given below.



2.1.3.1.4 Headloss curves

A headloss curve is used to described the headloss (Y in feet or meters) through a General Purpose Valve (GPV) as a function of flow rate (X in flow units). It provides the capability to model devices and situations with unique headloss-flow relationships, such as reduced flow - backflow prevention valves, turbines, and well draw-down behavior.

2.1.3.2 Time patterns

A Time Pattern is a collection of multipliers that can be applied to a quantity to allow it to vary over time. Nodal demands, reservoir heads, pump schedules, and water quality source inputs can all have time patterns associated with them. The time interval used in all patterns is a fixed value, set with the project's Time Options. Within this interval a quantity remains at a constant level, equal to the product of its nominal value and the pattern's multiplier for that time period.

Although all time patterns must utilize the same time interval, each can have a different number of periods. When the simulation clock exceeds the number of periods in a pattern, the pattern wraps around to its first period again. As an example of how time patterns work consider a junction node with an average demand of 10 GPM. Assume that the time pattern interval has been set to 4 hours and a pattern with the following multipliers has been specified for demand at this node:

Period	1	2	3	4	5	6
Multiplier	0.5	0.8	1.0	1.2	0.9	0.7

Then during the simulation the actual demand exerted at this node will be as follows:

Hours	0-4	4-8	8-12	12-16	16-20	20-24	24-28
Demand	5	8	10	12	9	7	5

2.1.3.3 Controls

Controls are statements that determine how the network is operated over time. They specify the status of selected links as a function of time, tank water levels, and pressures at select points within the network. There are two categories of controls that can be used:

- Simple Controls
- Rule-Based Controls

2.1.3.3.1 Simple based controls

Simple controls change the status or setting of a link based on:

- the water level in a tank
- the pressure at a junction
- the time into the simulation
- the time of day.

They are statements expressed in one of the following three formats:

```
LINK x status IF NODE y ABOVE/BELOW z
LINK x status AT TIME t
LINK x status AT CLOCKTIME c AM/PM
```

where:

x, a link ID label
status = OPEN or CLOSED, a pump speed setting, or a control valve setting
y, a node ID label

z, a pressure for a junction or a water level for a tank
t, a time since the start of the simulation in decimal hours or in hours:minutes notation
c, a 24-hour clock time.

Some examples of simple controls are:

Control Statement	Meaning
LINK 12 CLOSED IF NODE 23 ABOVE 20	Close Link 12 when the level in Tank 23 exceeds 20 ft.
LINK 12 OPEN IF NODE 130 BELOW 30	Open Link 12 if the pressure at Node 130 drops below 30 psi
LINK 12 1.5 AT TIME 16	Set the relative speed of pump 12 to 1.5 at 16 hours into the simulation
LINK 12 CLOSED AT CLOCKTIME 10 AM LINK 12 OPEN AT CLOCKTIME 8 PM	Link 12 is repeatedly closed at 10 AM and opened at 8 PM throughout the simulation

There is no limit on the number of simple control statements that can be used.

NOTE: Level controls are stated in terms of the height of water above the tank bottom, not the elevation (total head) of the water surface.

NOTE: Using a pair of pressure controls to open and close a link can cause the system to become unstable if the pressure settings are too close to one another. In this case using a pair of Rule-Based controls might provide more stability.

2.1.3.3.2 Rule based controls

Rule-Based Controls allow link status and settings to be based on a combination of conditions that might exist in the network after an initial hydraulic state of the system is computed. Here are several examples of Rule-Based Controls:

Example 1:

This set of rules shuts down a pump and opens a by-pass pipe when the level in a tank exceeds a certain value and does the opposite when the level is below another value.

```
RULE 1
IF TANK 1 LEVEL ABOVE 19.1
THEN PUMP 335 STATUS IS CLOSED
AND PIPE 330 STATUS IS OPEN
RULE 2
IF TANK 1 LEVEL BELOW 17.1
THEN PUMP 335 STATUS IS OPEN
AND PIPE 330 STATUS IS CLOSED
```

Example 2:

These rules change the tank level at which a pump turns on depending on the time of

day.

```
RULE 3
IF SYSTEM CLOCKTIME >= 8 AM
AND SYSTEM CLOCKTIME < 6 PM
AND TANK 1 LEVEL BELOW 12
THEN PUMP 335 STATUS IS OPEN
RULE 4
IF SYSTEM CLOCKTIME >= 6 PM
OR SYSTEM CLOCKTIME < 8 AM
AND TANK 1 LEVEL BELOW 14
THEN PUMP 335 STATUS IS OPEN
```

2.1.4 Hydraulic simulation model

The program's hydraulic simulation model computes junction heads and link flows for a fixed set of reservoir levels, tank levels, and water demands over a succession of points in time. From one time step to the next reservoir levels and junction demands are updated according to their prescribed time patterns while tank levels are updated using the current flow solution. The solution for heads and flows at a particular point in time involves solving simultaneously the conservation of flow equation for each junction and the headloss relationship across each link in the network. This process, known as "hydraulically balancing" the network, requires using an iterative technique to solve the nonlinear equations involved. The program employs the "Gradient Algorithm" for this purpose.

The hydraulic time step used for extended period simulation (EPS) can be set by the user. A typical value is 1 hour. Shorter time steps than normal will occur automatically whenever one of the following events occurs:

- the next output reporting time period occurs
- the next time pattern period occurs
- a tank becomes empty or full
- a simple control or rule-based control is activated.

2.1.5 Water quality simulation model

The water quality simulation model cosists of the following routines:

- Basic transport
- Mixing in storage tanks
- Water quality reactions
- Water age and source tracing

2.1.5.1 Basic transport

The program's water quality simulator uses a Lagrangian time-based approach to track the fate of discrete parcels of water as they move along pipes and mix together at junctions between fixed-length time steps. These water quality time steps are typically much shorter than the hydraulic time step (e.g., minutes rather than hours) to accommodate the short times of travel that can occur within pipes.

The method tracks the concentration and size of a series of non-overlapping segments of water that fills each link of the network. As time progresses, the size of the most upstream segment in a link increases as water enters the link while an equal loss in size of the most downstream segment occurs as water leaves the link. The size of the

segments in between these remains unchanged.

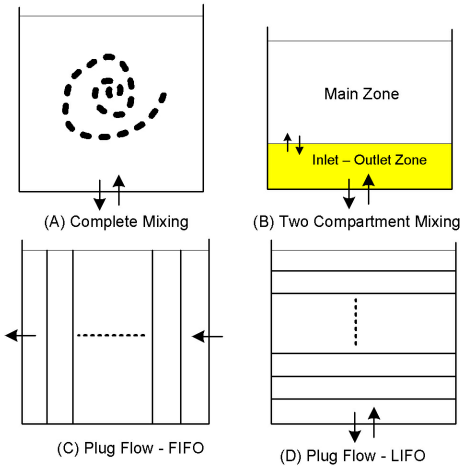
For each water quality time step, the contents of each segment are subjected to reaction, a cumulative account is kept of the total mass and flow volume entering each node, and the positions of the segments are updated. New node concentrations are then calculated, which include the contributions from any external sources.

Storage tank concentrations are updated depending on the type of mixing model that is used. Finally, a new segment will be created at the end of each link that receives inflow from a node if the new node quality differs by a user-specified tolerance from that of the link's last segment.

Initially each pipe in the network consists of a single segment whose quality equals the initial quality assigned to the upstream node. Whenever there is a flow reversal in a pipe, the pipe's parcels are re-ordered from front to back.

2.1.5.2 Mixing in storage tanks

The program can use four different types of models to characterize mixing within storage tanks as illustrated in the figure below:



- Complete Mixing
- Two-Compartment Mixing
- FIFO Plug Flow
- LIFO Plug Flow

Different models can be used with different tanks within a network.

The **Complete Mixing model** (Figure a) assumes that all water that enters a tank is instantaneously and completely mixed with the water already in the tank. It is the simplest form of mixing behavior to assume, requires no extra parameters to describe it, and seems to apply quite well to a large number of facilities that operate in fill-and-draw fashion.

The **Two-Compartment Mixing model** (Figure b) divides the available storage volume in a tank into two compartments, both of which are assumed completely

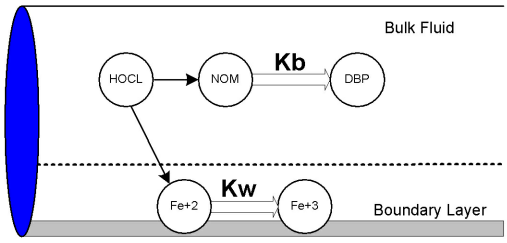
mixed. The inlet/outlet pipes of the tank are assumed to be located in the first compartment. New water that enters the tank mixes with the water in the first compartment. If this compartment is full, then it sends its overflow to the second compartment where it completely mixes with the water already stored there. When water leaves the tank, it exits from the first compartment, which if full, receives an equivalent amount of water from the second compartment to make up the difference. The first compartment is capable of simulating short-circuiting between inflow and outflow while the second compartment can represent dead zones. The user must supply a single parameter, which is the fraction of the total tank volume devoted to the first compartment.

The **FIFO Plug Flow model** (Figure c) assumes that there is no mixing of water at all during its residence time in a tank. Water parcels move through the tank in a segregated fashion where the first parcel to enter is also the first to leave. Physically speaking, this model is most appropriate for baffled tanks that operate with simultaneous inflow and outflow. There are no additional parameters needed to describe this mixing model.

The **LIFO Plug Flow model** (Figure d) also assumes that there is no mixing between parcels of water that enter a tank. However in contrast to FIFO Plug Flow, the water parcels stack up one on top of another, where water enters and leaves the tank on the bottom. This type of model might apply to a tall, narrow standpipe with an inlet/outlet pipe at the bottom and a low momentum inflow. It requires no additional parameters be provided.

2.1.5.3 Water quality reactions

The program can track the growth or decay of a substance by reaction as it travels through a distribution system. In order to do this it needs to know the rate at which the substance reacts and how this rate might depend on substance concentration. Reactions can occur both within the bulk flow and with material along the pipe wall. This is illustrated in the figure below. In this example free chlorine (HOCl) is shown reacting with natural organic matter (NOM) in the bulk phase and is also transported through a boundary layer at the pipe wall to oxidize iron (Fe) released from pipe wall corrosion. Bulk fluid reactions can also occur within tanks. The program allows a modeler to treat these two reaction zones separately.



Bulk Reactions

The program models reactions occurring in the bulk flow with n-th order kinetics, where the instantaneous rate of reaction (R in mass/volume/time) is assumed to be concentration-dependent according to:

$$R = K_b \cdot C^n$$

Where: **K_b**, a bulk reaction rate coefficient
C, reactant concentration (mass/volume) and
n, a reaction order.

K_b has units of concentration raised to the (1-n) power divided by time. It is positive for growth reactions and negative for decay reactions. The program can also consider reactions where a limiting concentration exists on the ultimate growth or loss of the substance. In this case the rate expression becomes:

$$R = K_b (C_L - C) C^{(n-1)}, n > 0, K_b > 0$$
$$R = K_b (C - C_L) C^{(n-1)}, n > 0, K_b < 0$$

Where: **C_L**, the limiting concentration.

Thus there are three parameters (K_b, C_L, and n) that are used to characterize bulk reaction rates. Some special cases of well-known kinetic models include the following:

Model	**Parameters**	**Examples**
First-Order Decay	C_L = 0, K_b < 0, n = 1	Chlorine
First-Order Saturation Growth	C_L > 0, K_b > 0, n = 1	Trihalomethanes
Zero-Order Kinetics	C_L = 0, K_b <> 0, n = 0	Water Age
No Reaction	C_L = 0, K_b = 0	Fluoride Tracer

The K_b for first-order reactions can be estimated by placing a sample of water in a series of non-reacting glass bottles and analyzing the contents of each bottle at different points in time. If the reaction is first-order, then plotting the natural log (C_t/ Co) against time should result in a straight line, where Ct is concentration at time t and Co is concentration at time zero. K_b would then be estimated as the slope of this line.

Bulk reaction coefficients usually increase with increasing temperature. Running multiple bottle tests at different temperatures will provide more accurate assessment of how the rate coefficient varies with temperature.

Wall Reactions

The rate of water quality reactions occurring at or near the pipe wall can be considered to be dependent on the concentration in the bulk flow by using an expression of the form:

$$R = \left(\frac{A}{V}\right) K_W C^n$$

where: **Kw**, a wall reaction rate coefficient and
A/V, the surface area per unit volume within a pipe (equal to 4 divided by the

pipe diameter).

The latter term converts the mass reacting per unit of wall area to a per unit volume basis. The program limits the choice of wall reaction order to either 0 or 1, so that the units of Kw are either mass/area/time or length/time, respectively. As with Kb, Kw must be supplied to the program by the modeler. First-order Kw values can range anywhere from 0 to as much as 5 ft/day.

Kw should be adjusted to account for any mass transfer limitations in moving reactants and products between the bulk flow and the wall. The program does this automatically, basing the adjustment on the molecular diffusivity of the substance being modeled and on the flow's Reynolds number. Setting the molecular diffusivity to zero will cause mass transfer effects to be ignored.

The wall reaction coefficient can depend on temperature and can also be correlated to pipe age and material. It is well known that as metal pipes age their roughness tends to increase due to encrustation and tuburculation of corrosion products on the pipe walls. This increase in roughness produces a lower Hazen-Williams C-factor or a higher Darcy-Weisbach roughness coefficient, resulting in greater frictional head loss in flow through the pipe.

There is some evidence to suggest that the same processes that increase a pipe's roughness with age also tend to increase the reactivity of its wall with some chemical species, particularly chlorine and other disinfectants. The program can make each pipe's Kw be a function of the coefficient used to describe its roughness. A different function applies depending on the formula used to compute headloss through the pipe:

Headloss Formula	**Wall Reaction Formula**
Hazen-Williams	$K_W = F / C$
Darcy-Weisbach	$K_W = -F / \log(e / d)$
Chezy-Manning	$K_W = F \cdot n$

where: **C**, Hazen-Williams C-factor
e, Darcy-Weisbach roughness
d, pipe diameter
n, Manning roughness coefficient and
F, wall reaction - pipe roughness coefficient

The coefficient F must be developed from site-specific field measurements and will have a different meaning depending on which head loss equation is used. The advantage of using this approach is that it requires only a single parameter, F, to allow wall reaction coefficients to vary throughout the network in a physically meaningful way.

2.1.5.4 Water age and source tracing

In addition to chemical transport, the program can also model the changes in the age of water throughout a distribution system. Water age is the time spent by a parcel of water in the network. New water entering the network from reservoirs or source nodes enters with age of zero. Water age provides a simple, non-specific measure of the overall quality of delivered drinking water. Internally, the program treats age as a

reactive constituent whose growth follows zero-order kinetics with a rate constant equal to 1 (i.e., each second the water becomes a second older).

The program can also perform source tracing. Source tracing tracks over time what percent of water reaching any node in the network had its origin at a particular node. The source node can be any node in the network, including tanks or reservoirs. Internally, the program treats this node as a constant source of a non-reacting constituent that enters the network with a concentration of 100. Source tracing is a useful tool for analyzing distribution systems drawing water from two or more different raw water supplies. It can show to what degree water from a given source blends with that from other sources, and how the spatial pattern of this blending changes over time.

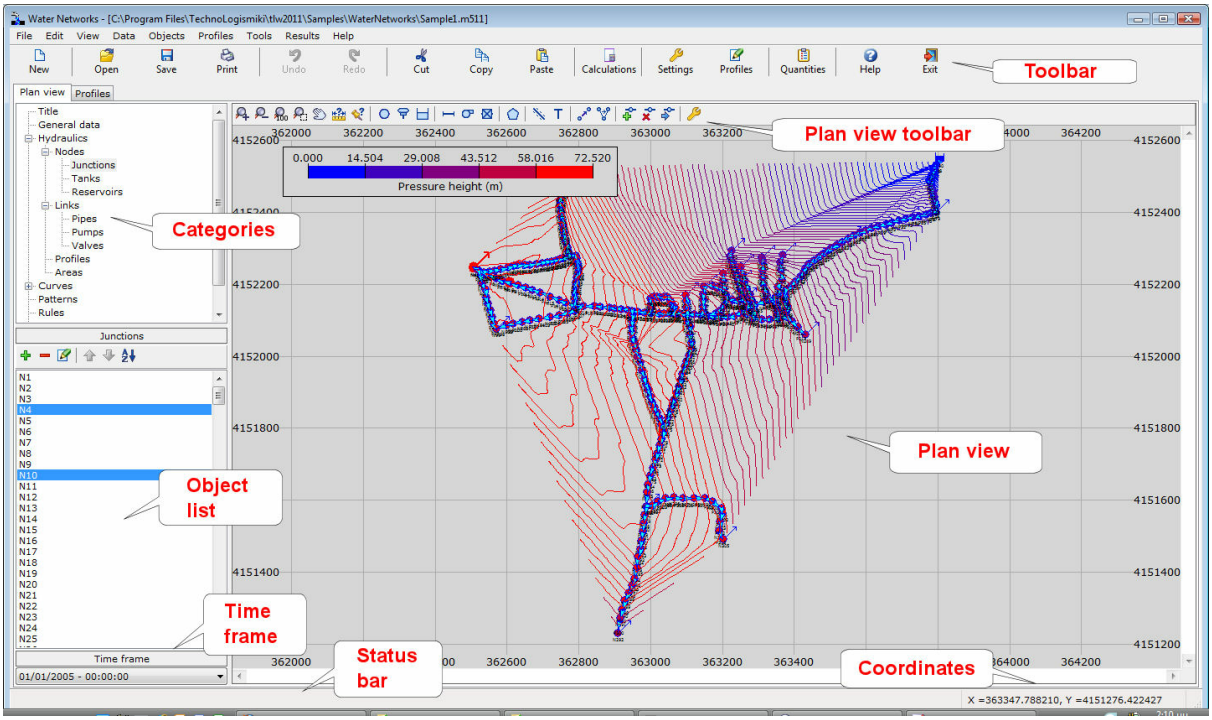
2.2 Main Window (Plan View tab)

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

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

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

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

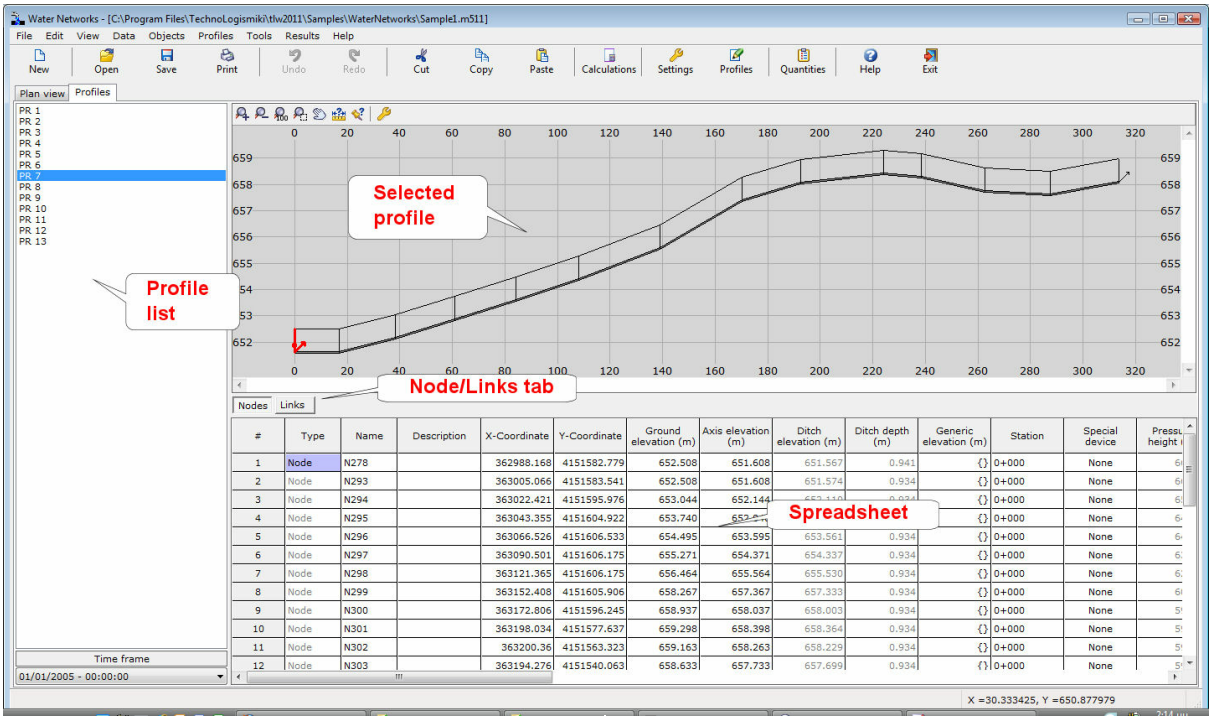


(Screenshot is from Sewer Networks program)

2.3 Main Window (Profiles tab)

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

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



(Screenshot is from Sewer Networks program)

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
  - Plan view
    - From DXF file
    - From ArcView Shapefile
    - From GTM
  - Network from ArcView Shapefile
  - Background from DXF
  - Satellite image
  - Network Data File
    - EPANET
    - TechnoLogismiki Cross v2.x
    - Encosoft Cross
  - Network from MAP file
  - Network Scenario file
  - Data from LandXML
- Export
  - Network to EPANET
  - MAP file
  - Network Scenario file
  - Plan view to BMP picture
  - Plan view to DXF
  - Plan view to ArcView Shapefile
  - Plan view to LandXML
- Print Setup
- Print
- Print to
  - Print to File
  - Print to Word
  - Print to Word (Formatted)
  - Print to Excel
- Exit

3.2 New project

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

To create a new project:

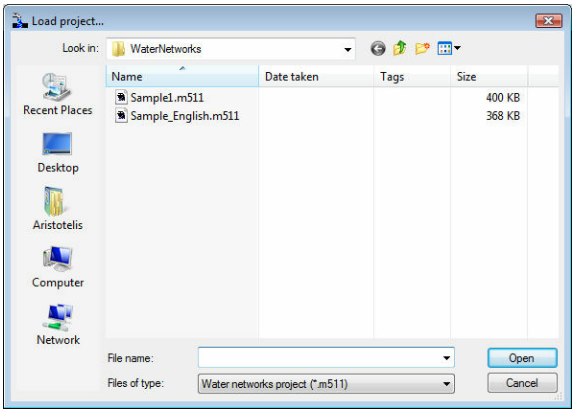
1. Select **New project** from the **File** menu.
2. If a project is already loaded and changes have been made, a warning message will appear that asks the user whether to save the changes or not.
3. The current project is erased and a new project is started.

3.3 Open project

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

To open an existing project:

1. Select **Open project** from the **File** menu.
2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Water Network project" with the extension .m511.
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\WaterNetworks

Supported file types

- **M11** (Water network project): Files created by versions 2012 and 2013 of Water Networks.
- **M511** (Water network project): Files created by versions 2011, 2010, 2009 and 2008, 2007, 5.0 and 4.0 of Water Networks.
- **BCK** (Backup files): If you have selected from program options the creation of backup copy when a file is loaded, then the file can be loaded by selecting Backup files (*.bck) from the Files of type drop-down list.
- ***.*** (All files): Displays all files in the current folder.

Backwards compatibility

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

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

3.4 Save project

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

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

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

To save the current project:

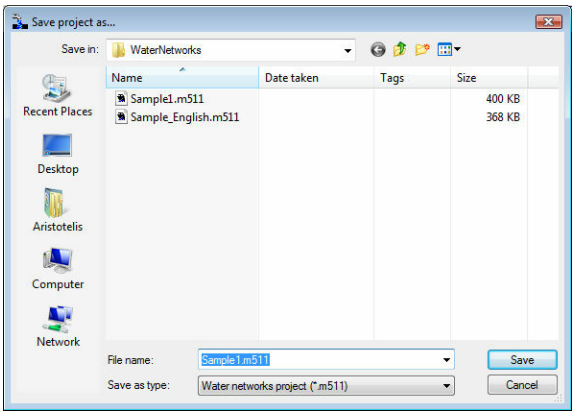
1. Select **Save project** from the **File** menu.
2. If the location of the file is already set, the project is saved to this file without any messages. If the filename is not set, a dialog box will appear that allows the selection of the filename and path.

3.5 Save project as

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

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

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



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

then the existing file is erased and the new file takes its place. If you answer No, the existing file remains intact but NO changes of the current project are saved.

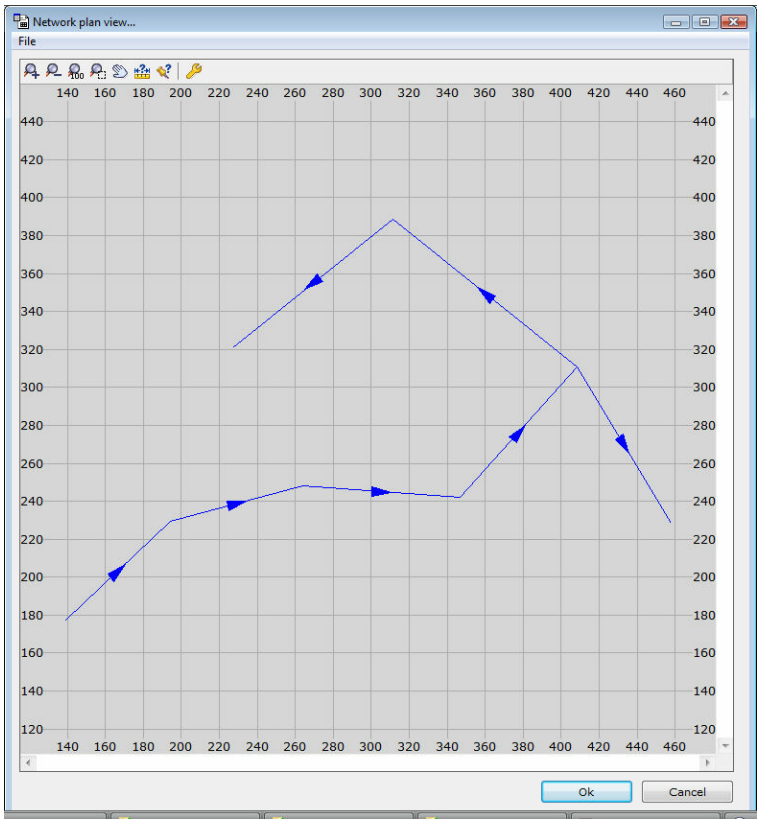
3.6 Import

3.6.1 Plan view

With this option, you can import plan view data from an external file.

To import plan view data from an external file:

1. Select **Import** from the **File** menu.
2. Select **Import plan view** from the **Import** menu. The following form appears:



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

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

To reverse the flow of the selected pipe:

1. Right-click on the selected branch.

2. Select **Reverse Flow** from the drop-down menu. The flow is reversed.

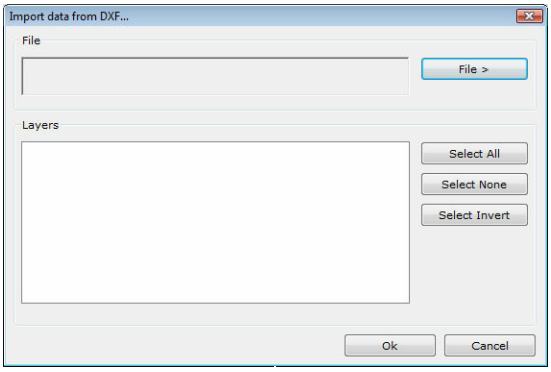
In order to manipulate the view of the drawing, use the toolbar. The button (from left to right) are as follows:

1. Zoom In: click anywhere on the drawing to zoom in.
2. Zoom Out: click anywhere on the drawing to zoom out.
3. Extends: view the whole drawing.
4. Window: zoom to a specified window.
5. Distance: calculate the distance between two points.
6. Coordinates: click anywhere on the drawing to show the coordinates.
7. Options: customize the appearance of the drawing.

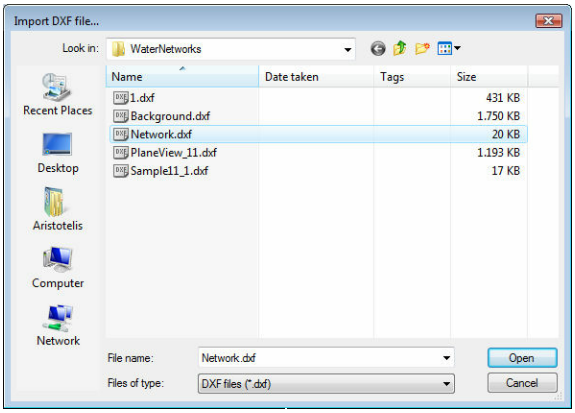
3.6.1.1 From DXF file

To import plan view data from a DXF file:

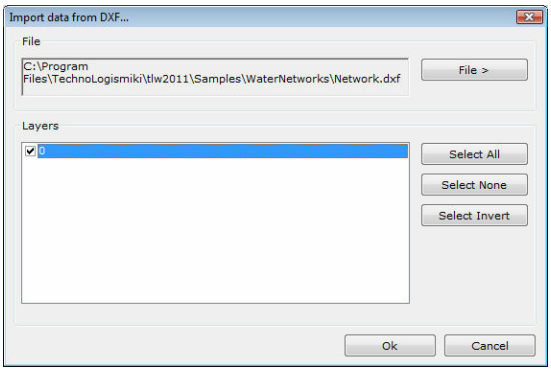
1. Select **Import From DXF file** from the **File** menu. The following form appears:



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



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



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

NOTE: The following tips may be useful:

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

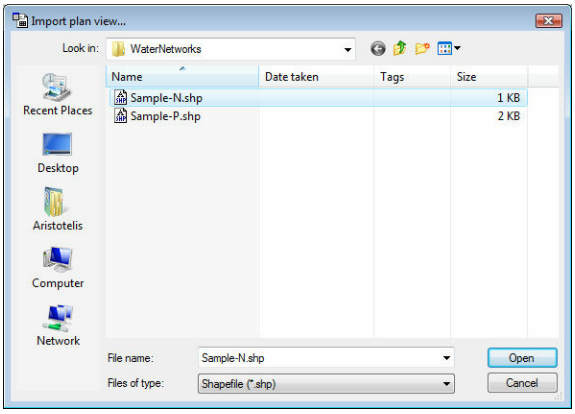
3.6.1.2 From ArcView Shapefile

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

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

To import plan view data from an ArcView Shapefile:

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

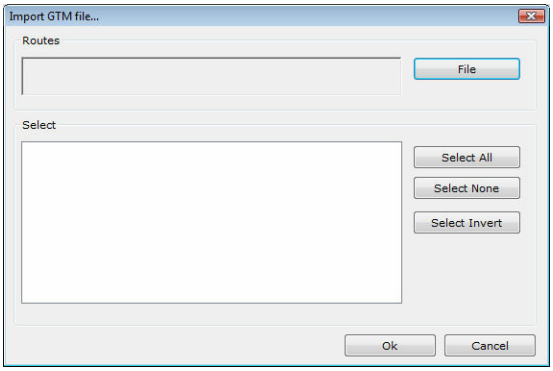


2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "ArcView shapefile" with the extension .shp.
4. Select the file by clicking on it.
5. Select **Open** to open and analyze the file. The current project data are erased. Select **Cancel**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

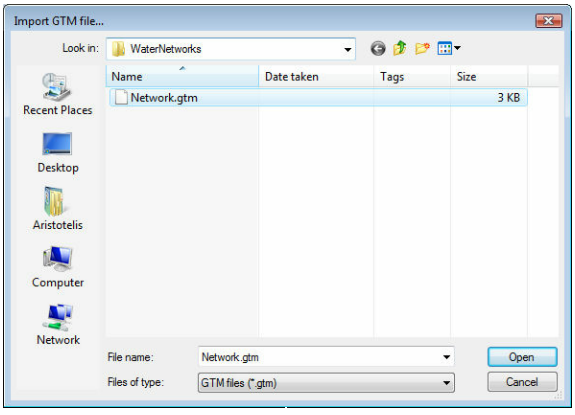
3.6.1.3 From GTM

To import plan view data from a GTM file:

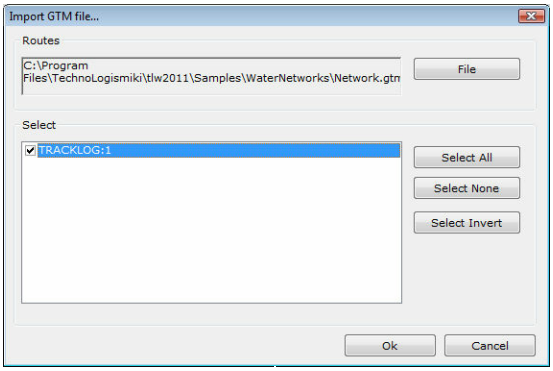
1. Select **Import from GTM file** from the **File** menu. The following form appears:



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



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



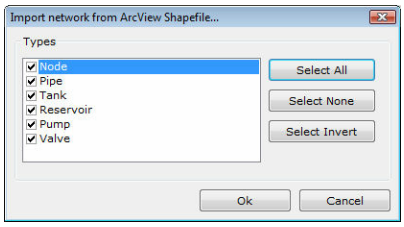
7. Select one or more routes or tracklog containing the data. Optionally, the coordinates of the stations can be used to calculate the distances between stations. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the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

3.6.2 Network from ArcView Shapefile

With this option, you can import node, pipe, tank, reservoir, pump and valve data from an ArcView Shapefile.

To import data from an ArcView Shapefile:

1. Select **Network from Arcview Shapefile** from the **File** menu. The following form appears:

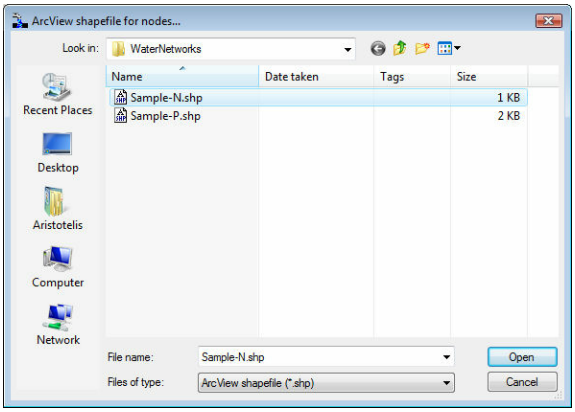


2. Select the type of objects that you wish to import. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
3. Select **Ok** to proceed. Select **Cancel** to cancel the operation and close the dialog box.
4. The supported shapefiles are the following:

Object Type	Supported Shapefiles
Nodes	point/pointM/pointZ

Pipes	polyline/polylineM/polylineZ
Tanks	point/pointM/pointZ
Reservoirs	point/pointM/pointZ
Pumps	polyline/polylineM/polylineZ
Valves	polyline/polylineM/polylineZ

6. The file selection dialog box appears:



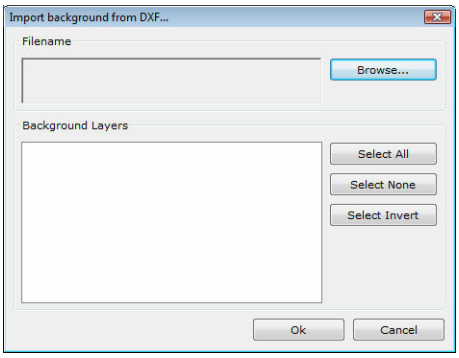
7. Select the path of the file.
8. Select the file type from the **Files of type** drop-down list. The default option is "DXF file" with the extension .dxf.
9. Select the file by clicking on it.
10. Select **Open** to open and analyze the file. Steps 6 to 10 are repeated for each selected object of step 1.

3.6.3 Background from DXF

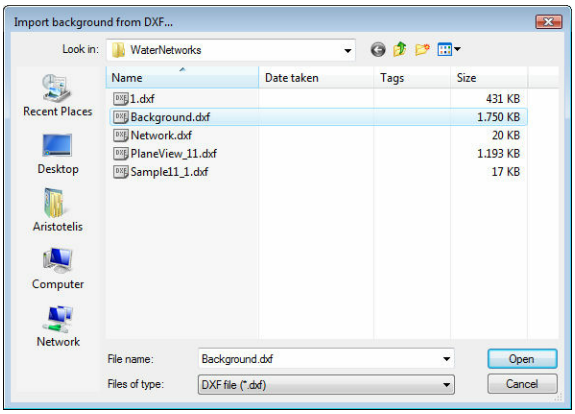
With this option, you can import background data from DXF files. The background is not active, but it is most helpful when designing a network. For performance reasons, only the most common entities of DXF files are imported, such as lines, points, polylines, arcs, circles, text etc.

To import background data from DXF:

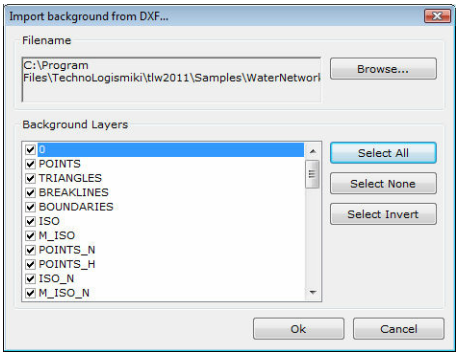
1. Select **Import** from the **File** menu.
2. Select **Background from DXF** from the **Import** menu. The following form appears:



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



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



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

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

NOTE: Since version 2010, it is possible to snap to background objects. For large or complex backgrounds, users may notice performance issues. It is recommended to disable snap capabilities once adding elements to the plane view is finished, in order to significantly accelerate the program.

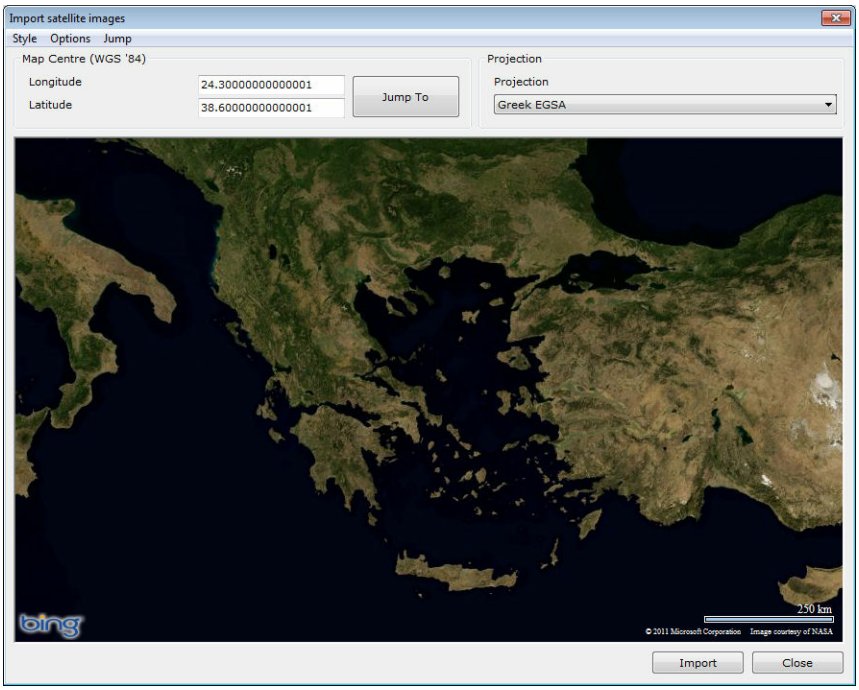
3.6.4 Satellite image

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

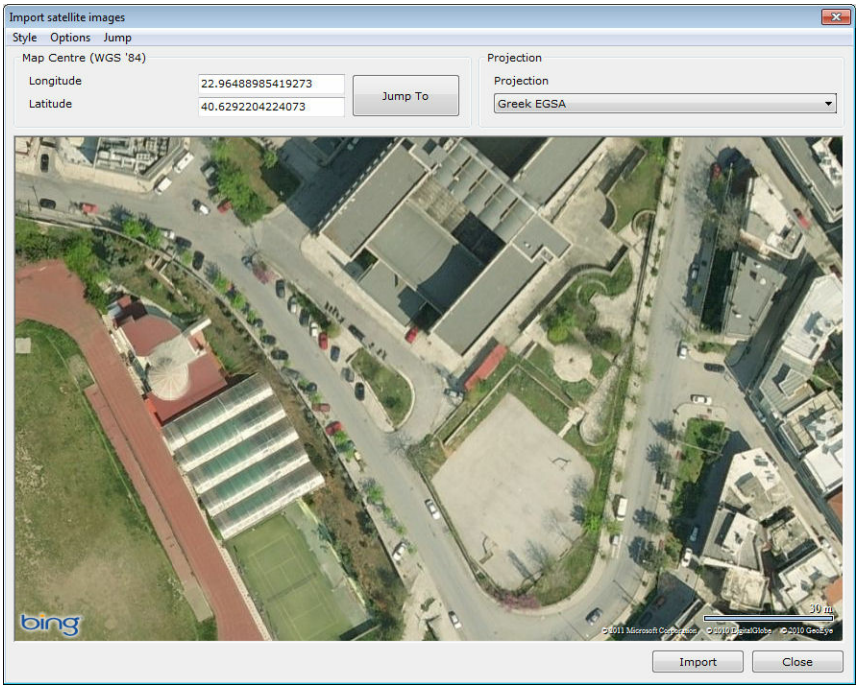
projected in the specified coordinate system.

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

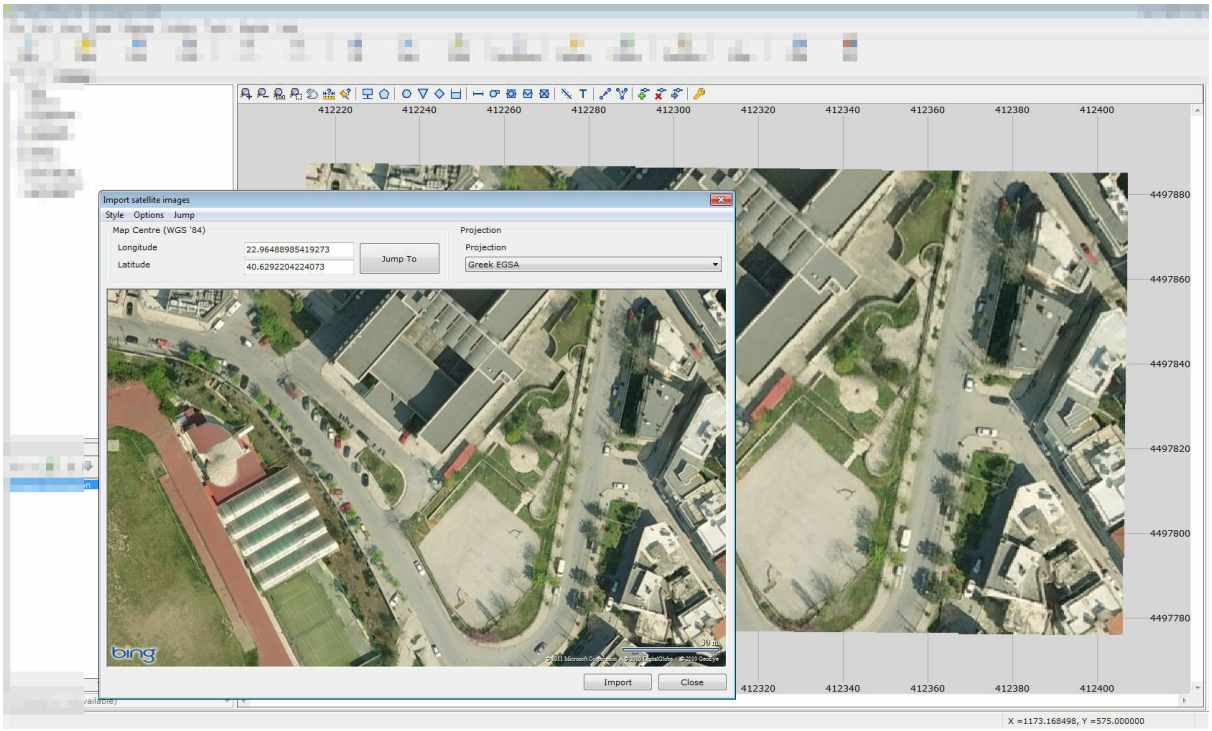
1. Select **Insert > Satellite image** from the **File** menu. The following form appears:



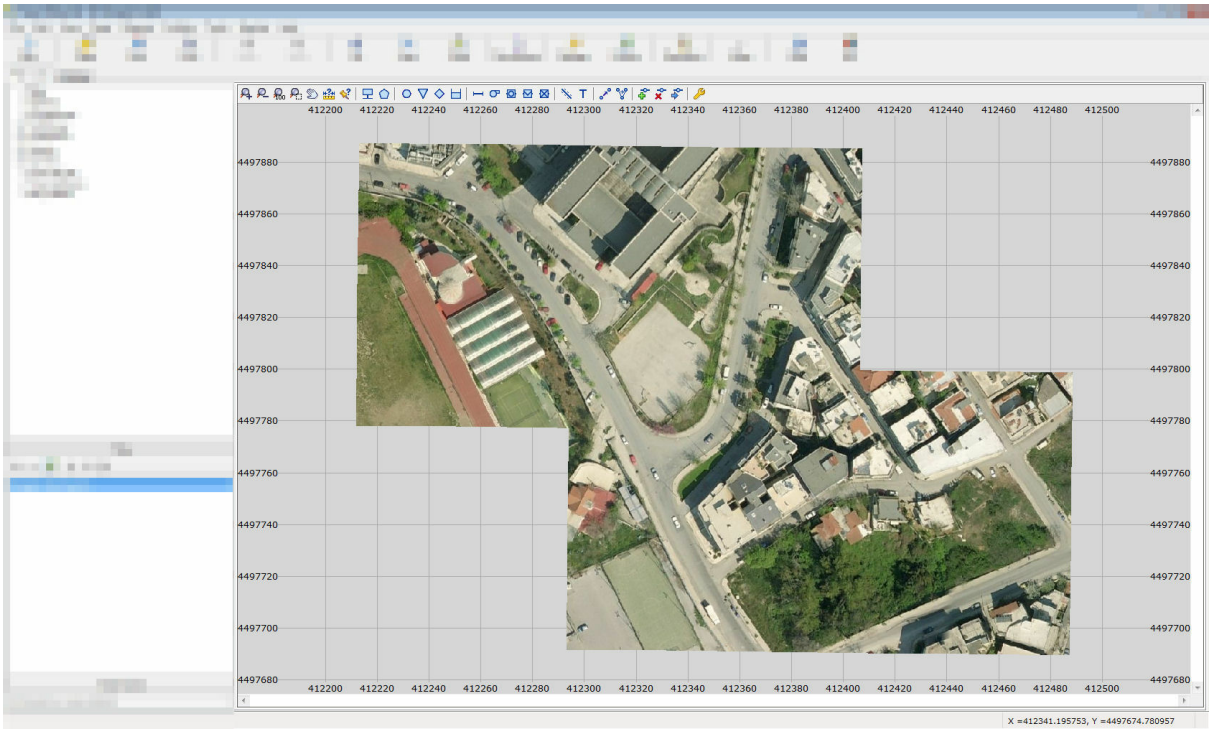
2. Navigate to the area of the project. You can pan the image by holding down the left mouse button. Using the roller you can change the resolution. Alternatively, enter the longitude and latitude in decimal degrees and press **Jump to:**



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



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



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

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

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

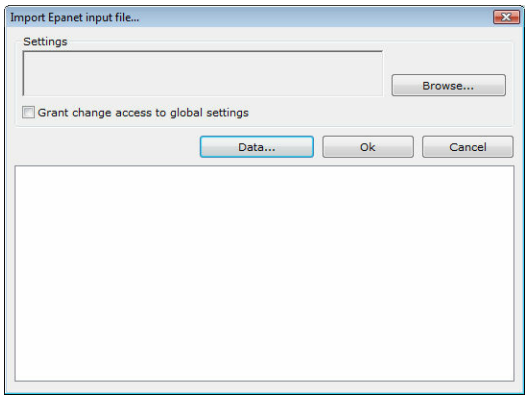
3.6.5 Network Data File

3.6.5.1 EPANET

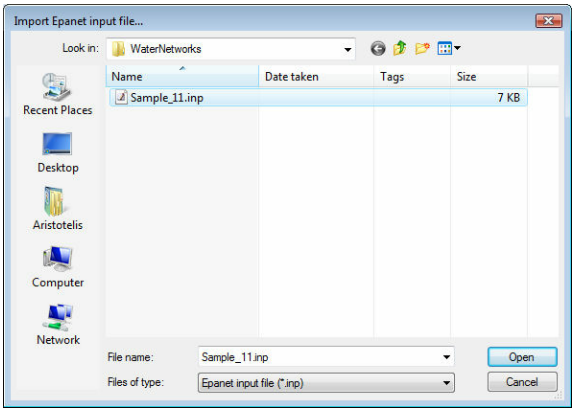
With this option, you can import network data from .inp files. These files can be created by many popular programs such as Water Networks by TechnoLogismiki, EPANET by EPA, WaterCAD by Heastad Methods etc.

To import network data from .inp files:

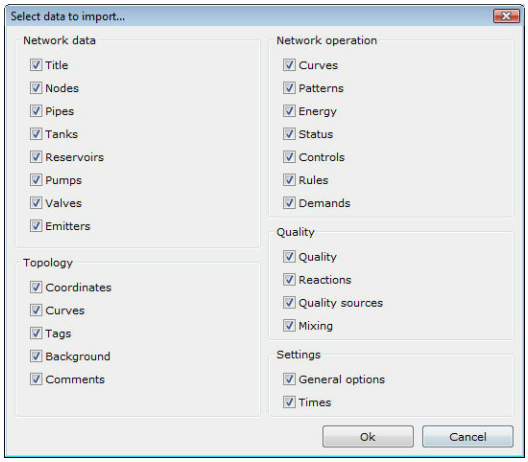
1. Select **Import** from the **File** menu.
2. Select **Network Data File** from the **Import** menu.
3. Select **EPANET** from the **Network Data File** menu. The following form appears:



4. Click **Browse** to select the file. The file selection dialog box appears:

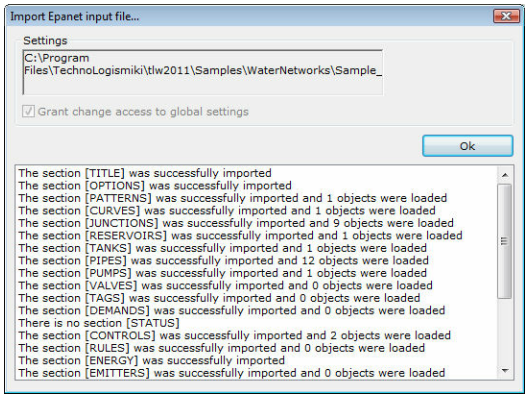


5. Select the path of the file.
6. Select the file type from the **Files of type** drop-down list. The default option is "Water network file" with the extension .inp.
7. Select the file by clicking on it.
8. Select **Open** to open and analyze the file.
9. Check **The file can change core program settings** if you wish to override program settings with data contained in the file.
10. By default, all data are imported. If you wish to import specific objects, select **Data...**. The following form appears:



- Select the objects that you wish to import.
- Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

11. Select **Ok** to proceed with the import. Select **Cancel** to cancel the operation and close the dialog box.
12. By selecting **Ok,** existing data are erased and the network is imported. Moreover, a list containing comments, errors etc is loaded:



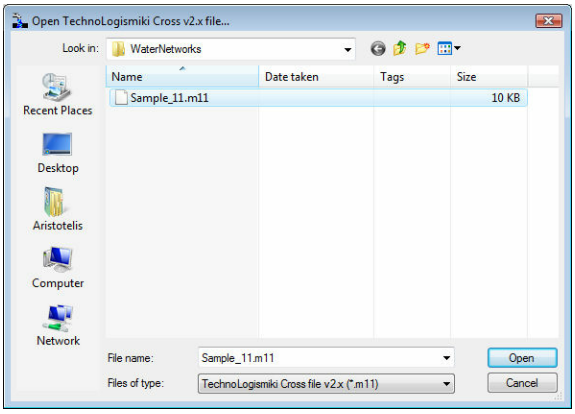
13. Select **Ok** to close the dialog box.

3.6.5.2 TechnoLogismiki Cross v2.x

With this option, you can import data from .m11 files created by Cross v2.x by TechnoLogismiki.

To import data from .m11 files created by Cross v2.x:

1. Select **Import** from the **File** menu.
2. Select **Network Data File** from the **Import** menu.
3. Select **TechnoLogismiki Cross v2.x**. from the **Network Data File** menu. The following form appears:



4. Select the path of the file.
5. Select the file type from the **Files of type** drop-down list. The default option is "TechnoLogismiki Cross file" with the extension .m11.
6. Select the file by clicking on it.
7. Select **Open** to import data from the file.

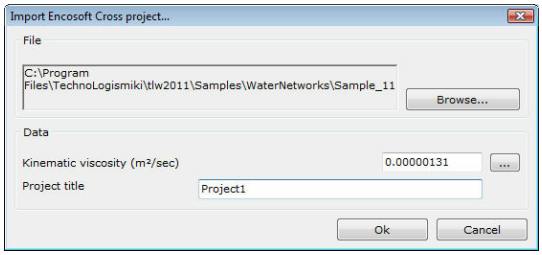
NOTE: After importing the data, you should always check the connectivity; due to the incompatibility of the data, some assumptions may be made during import.

3.6.5.3 Encosoft Cross

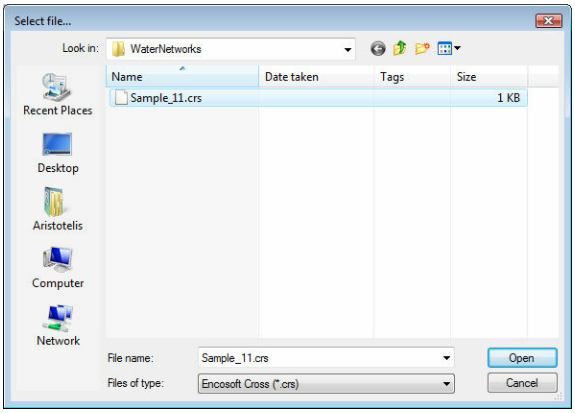
With this option, you can import data from .crs files that are created by Cross by Encosoft. All previous data are erased. No topographical data are included in this file; therefore you should manually enter the coordinates of the nodes.

To import data from .crs files:

1. Select **Import** from the **File** menu.
2. Select **Network Data File** from the **Import** menu.
3. Select **Encosoft Cross** from the **Network Data File** menu. The following form appears:



4. Click **Browse** to select the file. The file selection dialog box appears:



5. Select the path of the file.
6. Select the file type from the **Files of type** drop-down list. The default option is "Encosoft Cross" with the extension .crs.
7. Select the file by clicking on it.
8. Select **Open** to open and analyze the file.
9. Enter the **kinematic viscosity** of the fluid in m^2/s. Alternatively, select the button with the ellipses (...) to invoke the corresponding database.
10. Enter the **project title** by typing into the corresponding text box. Alternatively, you can enter the project title in project info.
11. Select **Ok** to proceed with the import. Select **Cancel** to cancel the operation and close the dialog box.

NOTE: Two other files, with the extensions .ko and .lo must exist in the same path for the import to be concluded successfully.

NOTE: After importing the data, you should always check the connectivity; due to the incompatibility of the data, some assumptions may be made during import.

3.6.6 Network from MAP file

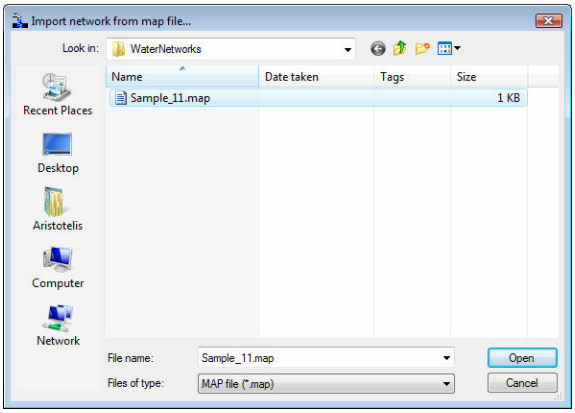
With this option, you can import network data from .map files. These files can be created by many popular programs such as Water Networks by TechnoLogismiki, EPANET by EPA, WaterCAD by Heastad Methods etc. These files contain the following information:

- Coordinates
- Labels
- Intermediate nodes
- Background

The above information is included in .inp files as well. The difference is that, in this case, only topological information is included.

To import network data from .map files:

1. Select **Import** from the **File** menu.
2. Select **Network from MAP file** from the **Import** menu. The following form appears:



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

3.6.7 Network Scenario file

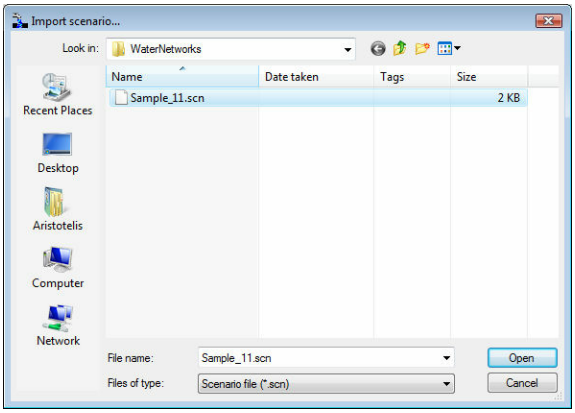
With this option, you can import scenario data from .scn files. These files can be created by many popular programs such as Water Networks by TechnoLogismiki, EPANET by EPA, WaterCAD by Heastad Methods etc. These files contain the following information:

- Demands
- Diameters
- Friction coefficients
- Initial qualities
- Reaction coefficients
- Rules

The above information is included in .inp files as well. With these files, you can easily create scenarios that correspond to normal operation, fires in various places etc.

To import scenario data from .scn files:

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



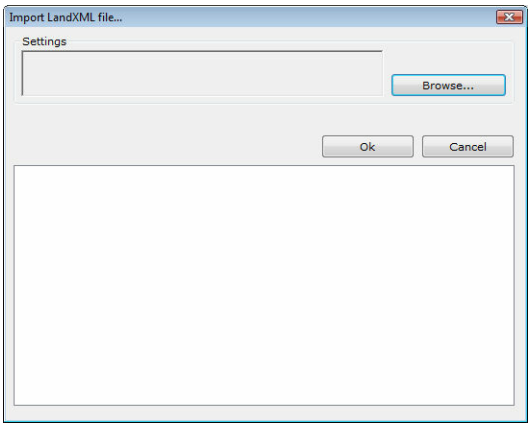
3. Select the path of the file.
4. Select the file type from the **Files of type** drop-down list. The default option is "Scenario file" with the extension .scn.
5. Select the file by clicking on it.
6. Select **Open** to open and analyze the file.

3.6.8 Data from LandXML

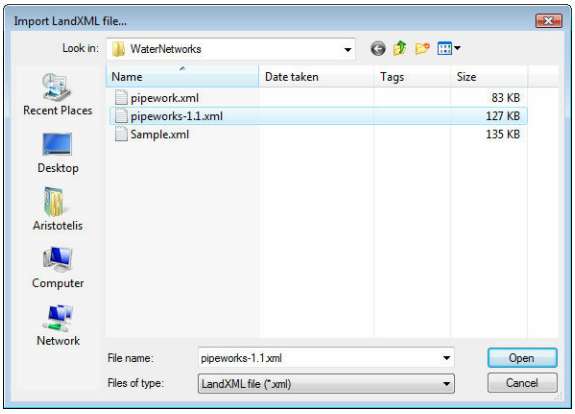
LandXML is an open source format specifically developed and tuned for exchanging design data used in the design / build process for land development and transportation projects. LandXML has evolved from previous efforts and is the cooperative work of representatives from all aspects of the engineering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

To import LandXML data:

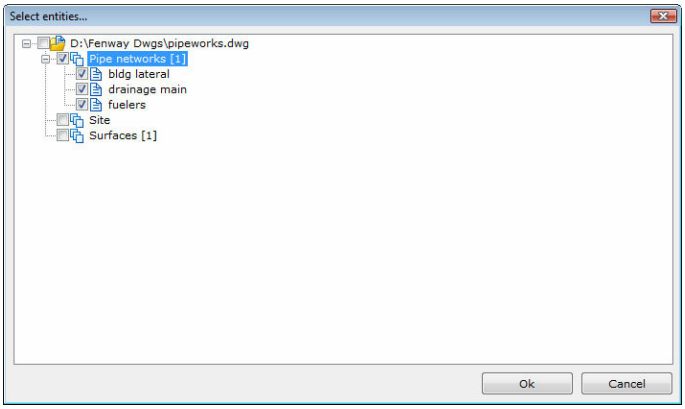
1. Select **Import** from the **File** menu.
2. Select **Data from LandXML** from the **Import** menu. The following form appears:



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



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



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

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

- LandXML
- Units*
- CoordinateSystem*
- Project*
- Application*
- Alignments
- CgPoints
- Amendment*
- GradeModel
- Monuments
- Parcels

- PlanFeatures
- PipeNetworks
- Roadways
- Surfaces
- Survey
- FeatureDictionary*

* No visual representation.

NOTE: In this version the following LandXML schemes are supported:
- v1.2 beta Build32 03.12.2007
- v1.1
- v1.0
- v0.88

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

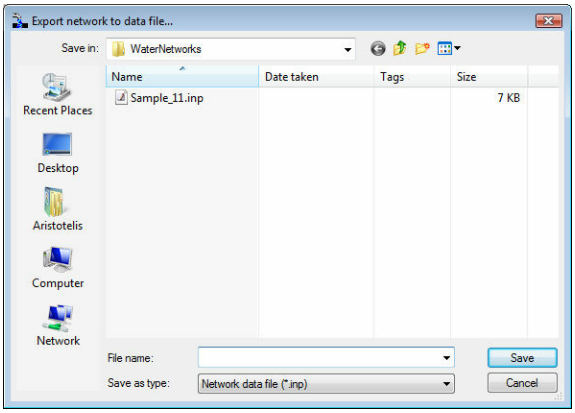
3.7 Export

3.7.1 Network to EPANET

With this option, you can create a .inp file containing network data. This file can be used by other programs such as EPANET by EPA or WaterCAD by Heastad Methods. Some data, such as hydrants and air release valves, are not supported by the file format and they are not exported.

To create a .inp file containing network data:

1. Select **Export** from the **File** menu.
2. Select **Network to Network Data File** from the **Export** menu. The following form will appear:



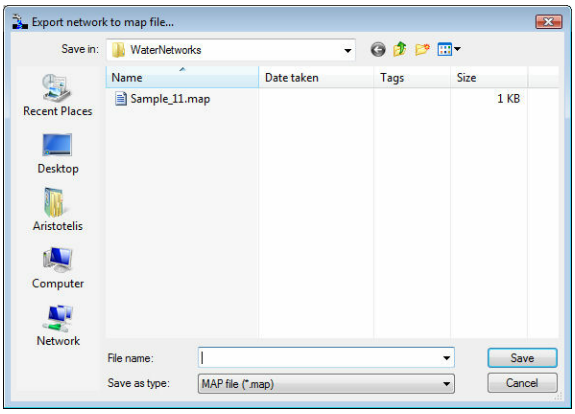
3. Select the path of the file.
4. Type the filename in the **File name** text box.
5. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

3.7.2 MAP file

With this option, you can create a .map file containing topological data. This file can be used by other programs such as EPANET by EPA or WaterCAD by Heastad Methods. Some data, such as hydrants and air release valves, are not supported by the file format and they are not exported.

To create a .map file containing topological data:

1. Select **Export** from the **File** menu.
2. Select **Network to MAP File** from the **Export** menu. The following form will appear:



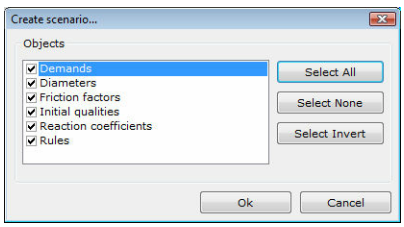
3. Select the path of the file.
4. Type the filename in the **File name** text box.
5. Select **Save** to save the file with the selected filename and path. Select **Cancel** to cancel the operation.

3.7.3 Network Scenario File

With this option, you can create a .scn file containing scenarios. This file can be used by the Water Networks or by other programs such as EPANET by EPA or WaterCAD by Heastad Methods. You can select one or more categories that contain the data you wish to export.

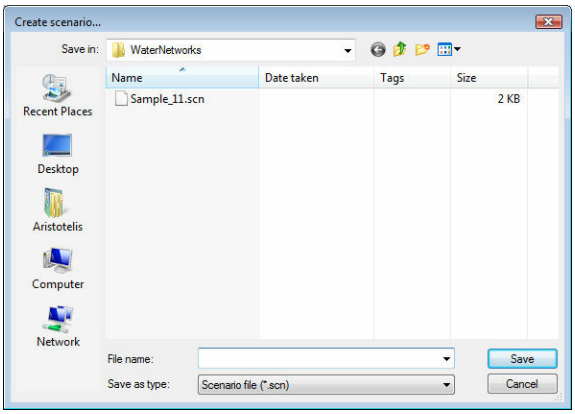
To create a .scn file containing scenarios:

1. Select **Export** from the **File** menu.
2. Select **Scenario** from the **Export** menu. The following form appears:



3. Select one or more objects that you wish to export. The quick keys (**Select all, Select None, Invert Selection**) can be used to quickly select all objects, deselect all objects and invert the current selection. At least one object must be selected. Objects

that do not vary can be omitted.
4. Select **Ok** to proceed with the export. Select **Cancel** to cancel the operation and close the dialog box.
5. The following form will appear:



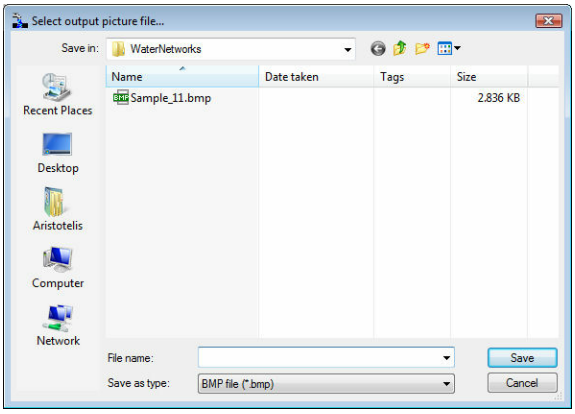
6. Select the path of the file.
7. Type the filename in the **File name** text box.
8. Select **Save** to save the project with the selected filename and path. Select **Cancel** to cancel the operation.

3.7.4 Plan view to BMP picture

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

To create a BMP file:

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



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

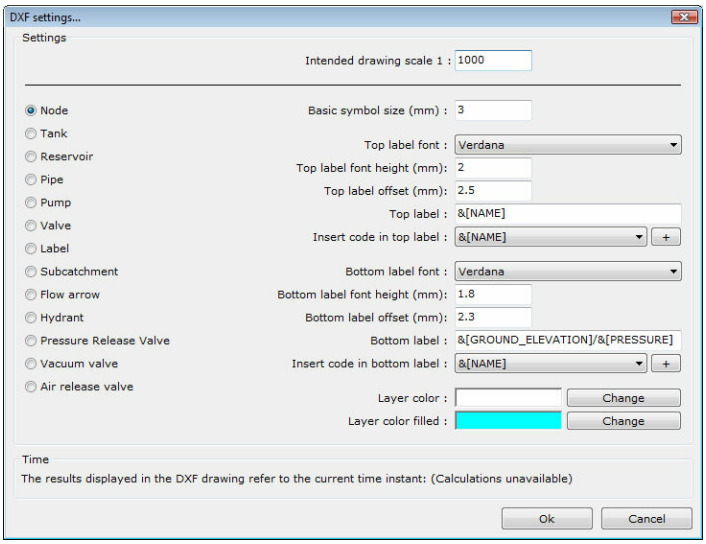
3.7.5 Plan view to DXF

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

- Station, station names
- Node data, pipe data
- Hydraulic calculations

To export the plan view to DXF file:

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



3. Select the appropriate settings:

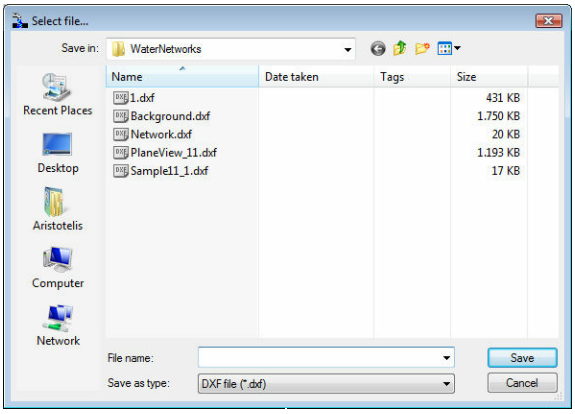
- **Intended drawing scale**: Select the intended drawing scale. The rest of the settings are measured in mm (millimeters) in the printed drawings. Based on the intended drawing scale, the sizes of all drawing elements are automatically derived.
- Select the object you wish to configure from the list on the left. Depending on the object, the following options may become available:
  - **Basic symbol size (mm)**: Enter the basic symbol size in mm (millimeters) in the printed drawing. For example, the basic symbol size for a junction refers to the diameter of the circle that represents it.
  - **Top label font**: select the font for the top label. All drawing elements are grouped into layers and font styles, so that they can be easily modified using CAD software.
  - **Top label font height (mm)**: Enter the top label font height in mm (millimeters) in the printed drawings. If the font height is selected to be 2mm and the scale is 1:1000, then the text height that will be used is equal to 2mm * 1000 = 2m. When printed, the desired font height will be obtained.

- **Top label offset (mm) :** Enter the top label offset from the center of the object, in mm in the printed drawing.
- **Top label** : Enter the text that will appear in the top label of the selected object. The text may contain **codes**. The codes have the prefix "&" and contain a special keyword within brackets. For example, if the top label is selected to be "&[NAME]", then the name of the object will be displayed. **Any combination of text and/or codes is allowed**. For example, if the model contains junctions named "J1", J2", etc and the manhole type is "T1", then the label "&[NAME] - &[MANHOLE_TYPE]" will create labels "J1 - T1", "J2 - T1" etc in the DXF drawing.
- **Insert code in top label**: Depending on the selected object, select the code from the list and click on the " + " button to enter it in the top label. The insertion point is the current cursor point in the previous field entitled "Top label".
- **Similar options may refer to the bottom label.**
- **Layer color:** Select the layer color of the selected object.
- **Layer color filled**: Select the layer color for the filled elements of the selected object.

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

- "&[NAME]"
- "&[STATION]"
- "&[DESCRIPTION]"
- "&[TAG]"
- "&[GROUND_ELEVATION]"
- "&[AXIS_ELEVATION]"
- "&[LENGTH]"
- "&[SHAPE]"
- "&[FLOW]"
- "&[VELOCITY]"
- "&[STATUS]"
- "&[LOSS]"
- "&[DEMAND]"
- "&[PRESSURE_ELEVATION]"
- "&[PRESSURE]"
- "&[REAL_DEMAND]"

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



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

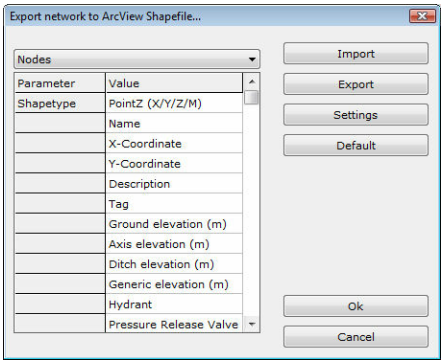
NOTE: If nothing is visible when viewing the newly created DXF file, select **Zoom Extents** to view the whole drawing.

3.7.6 Plan view to ArcView Shapefile

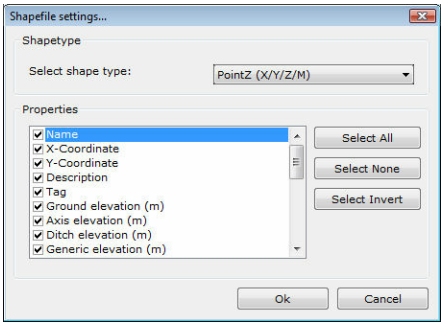
With this option, two ArcView Shapefiles containing the plan view data are created. The first file contains station data and the other contains pipe data. Note that apart from geometry, hydraulic calculation results are included in the Shapefiles.

To export the plan view to an ArcView Shapefile:

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



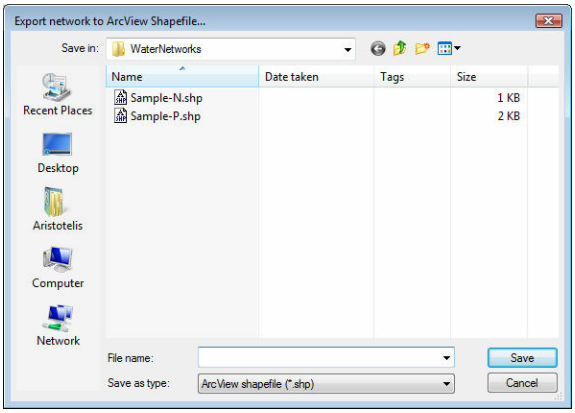
4. Select one of **Nodes**, **Pipes** from the drop-down list.
5. Select **Customization...** to customize the shape type and the properties that will be included:



6. Select the appropriate shape type.

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

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



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

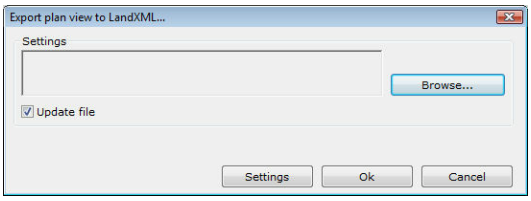
NOTE: If nothing is visible when viewing the newly created ArcView Shapefile, select **Zoom Extents** to view the whole drawing.

3.7.7 Plan view to LandXML

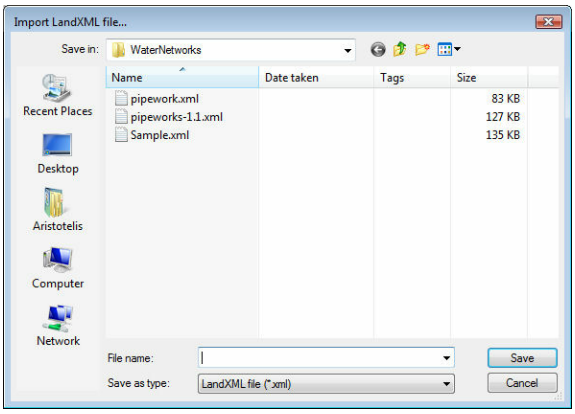
The plan view is exported (background, networks and calculations) to a LandXML file. For more information regarding programs supporting LandXML or for the LandXML scheme, please refer to LandXML forum.

To export plan view to LandXML:

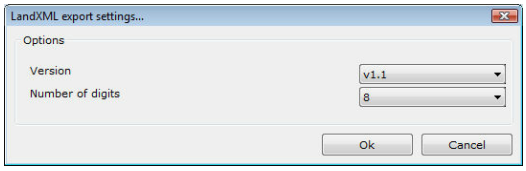
1. Select **Export** from the **File** menu..
2. Select **Plan view to LandXML** from the **Export** menu. The following form will appear:



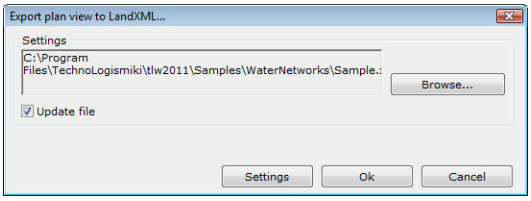
3. Click **Browse** to select an existing or a new LandXML file. The file selection dialog box appears.



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



8.1. Select the LandXML scheme that will be used to create / update the file. It is recommended to select 1.0 for maximum compatibility or the scheme version supported by the destination program (the program that will be used to read the exported file).
8.2. Select the number of decimal digits. It is recommended to select any number from 4 to 8 digits.
8.3. Click **Ok** to save changes. Click **Cancel** to ignore changes and close the form.



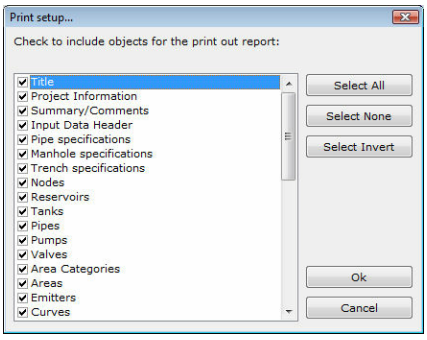
9. If an existing file has been selected, the **update file** check box is enabled. If this check box is selected then the target file will be updated with any changes made within the program.
10. Press **Ok** to close the form and finalize the export procedure or press **Cancel** to close the form and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

3.8 Print Setup

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

To modify the print setup:

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



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

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

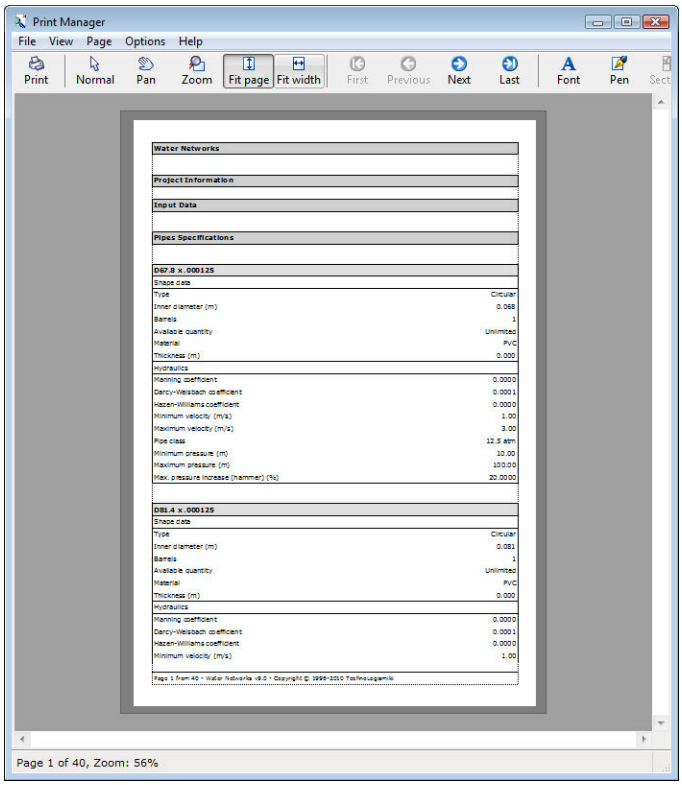
3.9 Print

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

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

To create a report:

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



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

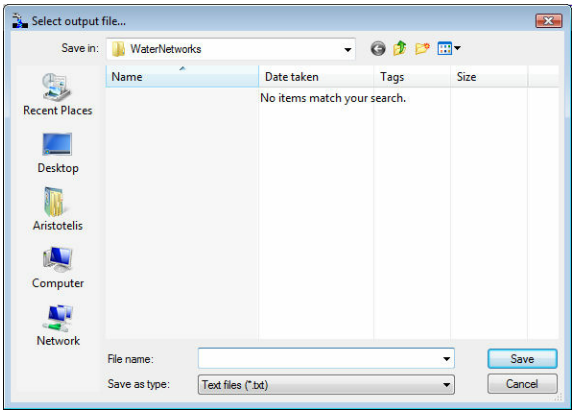
3.10 Print to

3.10.1 Print to File

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

To print to a text file:

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



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

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

3.10.2 Print to Word

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

To print the report to a Microsoft Word file:

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

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

3.10.3 Print to Word (Formatted)

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

To print the report to a formatted Microsoft Word file:

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

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

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

3.10.4 Print to Excel

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

To print the report to a Microsoft Excel file:

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

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

3.11 Exit

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

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

depending on what you have selected in General preferences.

To exit the program:

1. Select **Exit** from **File** menu.
2. If you are asked whether to save the changes or not, you can save changes or ignore them.
3. The program is terminated.

Chapter

4 Edit

4.1 Edit menu

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

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

4.2 Undo

Undo cancels the last committed change in the project.

To cancel the last committed change:

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

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

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

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

4.3 Redo

Redo cancels the latest undo command.

To redo the latest change that was undone:

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

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

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

4.4 Copy

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

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

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

To copy data to be used with Microsoft Excel:

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

4.5 Cut

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

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

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

To cut data to be used with Microsoft Excel:

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

4.6 Paste

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

To paste data from the clipboard to the data table:

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

To paste data from Microsoft Excel:

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

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

4.7 Select all

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

To select all:

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

4.8 Clipboard delimiter

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

To select the delimiter:

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

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

4.9 Clipboard decimal separator

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

To select the decimal separator:

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

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

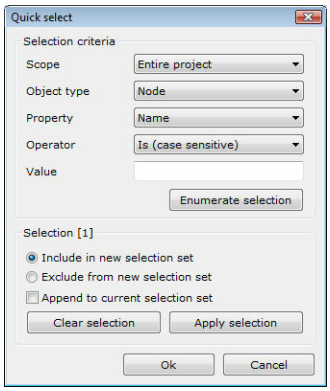
4.10 Select objects

4.10.1 Create selection

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

To modify the set of selected objects:

1. Perform calculations. This step is optional, but if you do not follow it, the results for some objects may not be available and the selection may not function properly.
2. Select **Select 0bjects** from the **Edit** menu.
3. Select **Create selection** from the **Select 0bjects** menu. The following form appears:



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

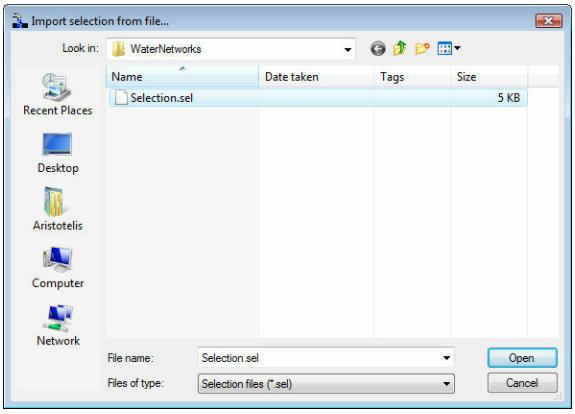
15. Select **Ok** to close the form and accept changes or **Cancel** to close the form and ignore any changed.

4.10.2 Load selection

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

To load a selection set from an external file:

1. Select **Select 0bjects** from the **Edit** menu.
2. Select **Load Selection** from the **Select 0bjects** menu. The following form appears:



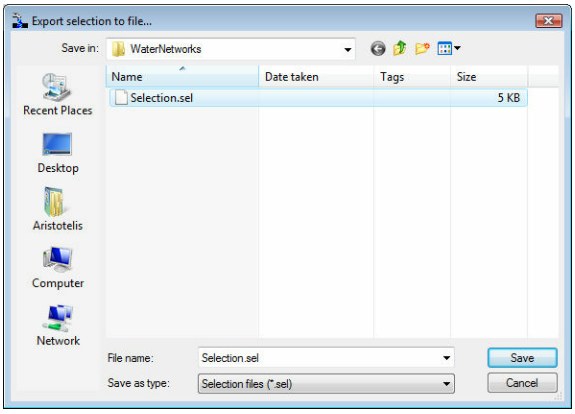
3. Select the path of the file.
4. Select the file type from the **Files of type** drop-down list. The default option is "Selection Files" with the extension .sel.
5. Select the file by clicking on it.
6. Select **Open** to open the selected file. Select **Cancel** to cancel the operation.

4.10.3 Save selection

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

To save the current selection set to an external file:

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



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

4.10.4 Clear selection

With this option, you can deselect all items.

To deselect all items:

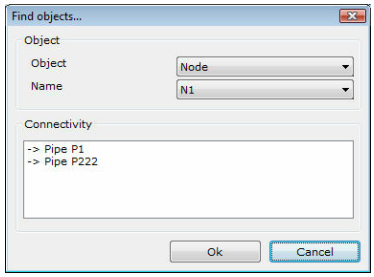
1. Select **Select objects** from the **Edit** menu.
2. Select **Clear Selection** from the **Select objects** menu. All objects are deselected.

4.11 Locate objects

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

To locate objects:

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



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

Chapter

5 View

5.1 View menu

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

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

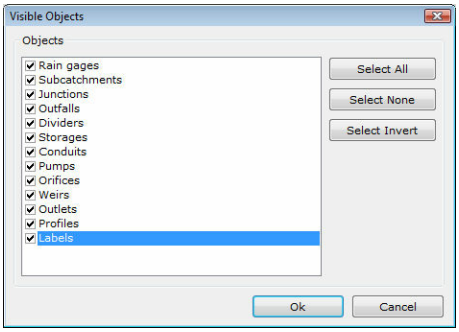
5.2 Plan view

5.2.1 Visible objects

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

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

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



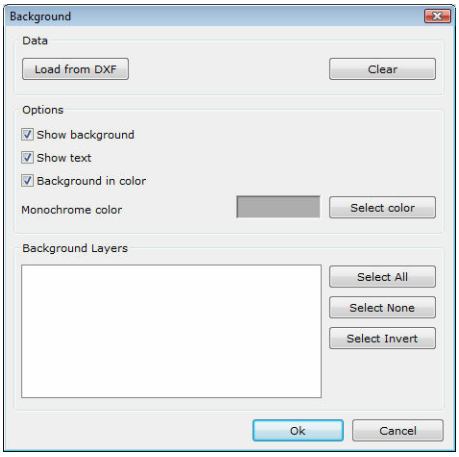
3. Select the object type(s) that you wish to be visible. The quick keys (**Select all, Select None, Select Invert**) can be used to quickly select all objects, deselect all objects and invert the current selection.
4. Select **Ok** to accept changes and close the form. Select **Cancel** to close the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

5.2.2 Background

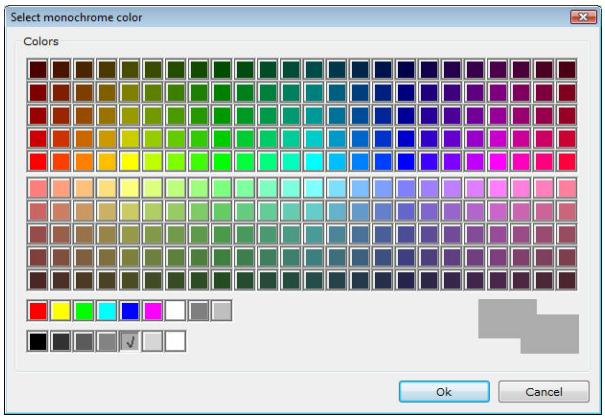
With this option, you have full access to the background data from DXF files. This includes the option to import data from a DXF file , under the **File > Import** menu.

To modify the background:

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



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



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

dialog box without saving the changes.

NOTE: The color palette follows standard CAD color palettes.

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

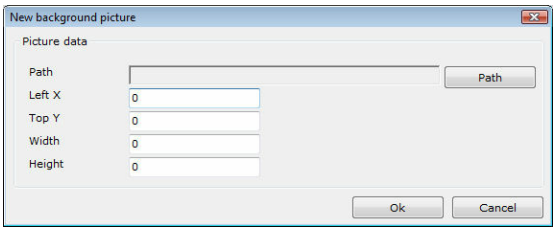
5.3 Background pictures

5.3.1 Add

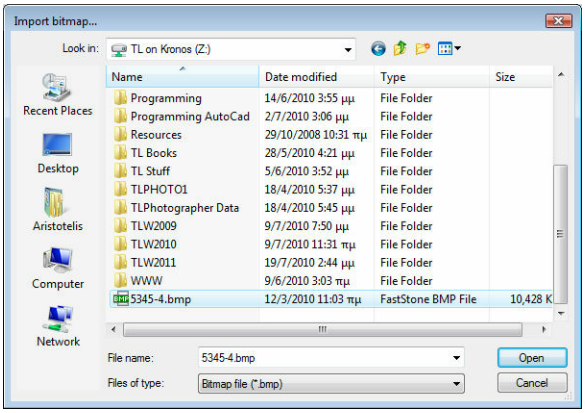
With this option, you can add a picture to the background.

To add a picture to the background:

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



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



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

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

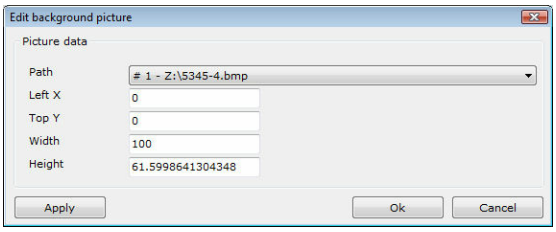
NOTE: The following image file types are supported: bitmaps (.bmp) and JPEG (.jpg).

5.3.2 Edit

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

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

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



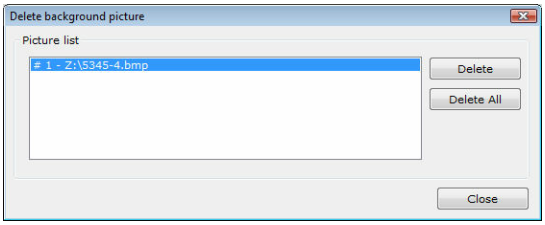
3. Select the picture from the drop-down list.
4. Make the appropriate changes. If you provide only the height or the width, the ratio of the source picture will be used to calculate the missing data.
5. Select **Apply** to apply the changes without closing the form. Select **Ok** to save the changes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

5.3.3 Delete

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

To delete one or more existing background pictures:

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



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

5.3.4 Show

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

To show or hide all pictures in the background:

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

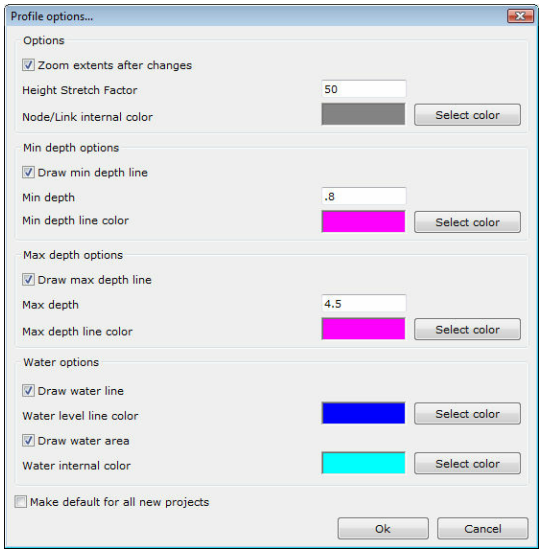
5.4 Profile

5.4.1 Options

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

To modify the appearance of the profile sketch:

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



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

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

Chapter

6 Data

6.1 Data menu

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

- Project info
- Project summary
- General data
  - General
  - Hydraulics
  - Times
  - Energy
  - Reactions
  - Quality
  - Checks
  - Excavation options
- Curves
  - Management
  - Add
  - Delete
  - Edit
  - Move
  - Sort
- Design period
- Consumptions
  - General
  - Population estimation
- Design consumptions
- Area categories
- Patterns
  - Management
  - Add
  - Delete
  - Edit
  - Move
  - Sort
- Rules
  - Management
  - Add
  - Delete
  - Edit
  - Move
  - Sort
- Rules
  - Management
  - Add
  - Delete

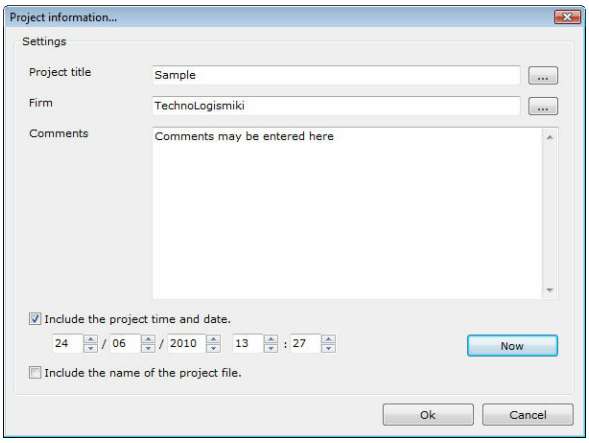
  - Edit
  - Move
  - Sort
- Conduit shapes
  - Management
  - Add
  - Delete
  - Edit
  - Import
  - Export
- Manhole specifications
  - Management
  - Add
  - Delete
  - Edit
  - Import
  - Export
- Trench specifications
  - Management
  - Add
  - Delete
  - Edit
  - Import
  - Export
- Network consistency
- Options
  - General preferences
  - Sketch
  - Grid editing
  - Customize toolbar
  - Default values
  - Algorithm

6.2 Project info

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

To add or modify the project information:

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



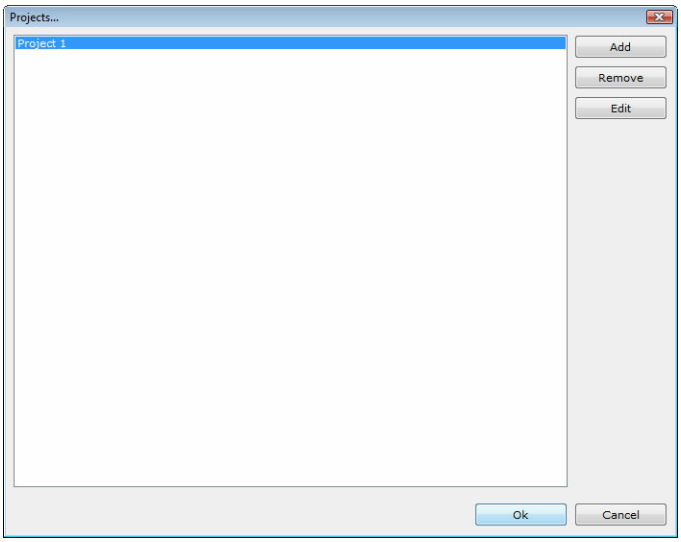
By selecting the buttons with the ellipses (...) next to the **project title** and **firm**, you can access the corresponding databases.

Project title database

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

To use the project title database:

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

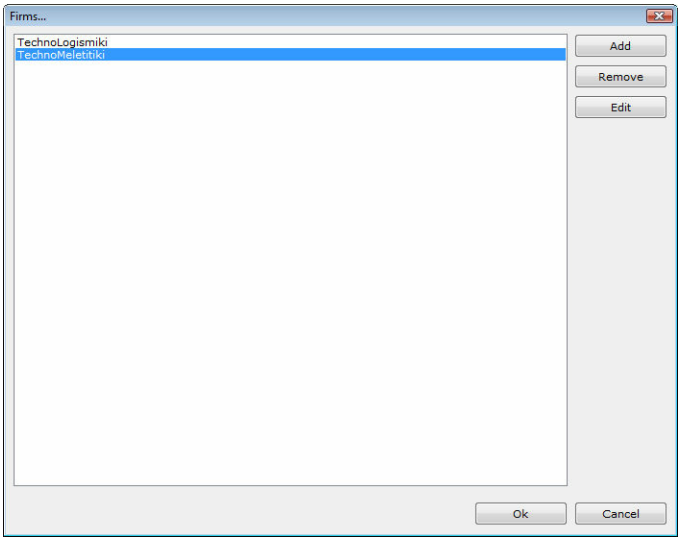


Firm database

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

To use the firm database:

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

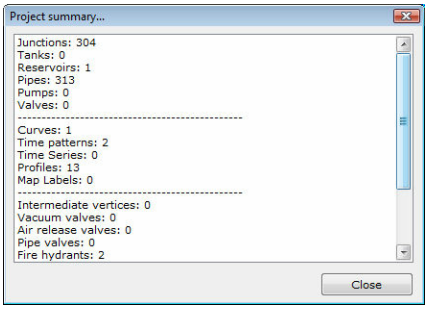


6.3 Project summary

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

To display the project summary:

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



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

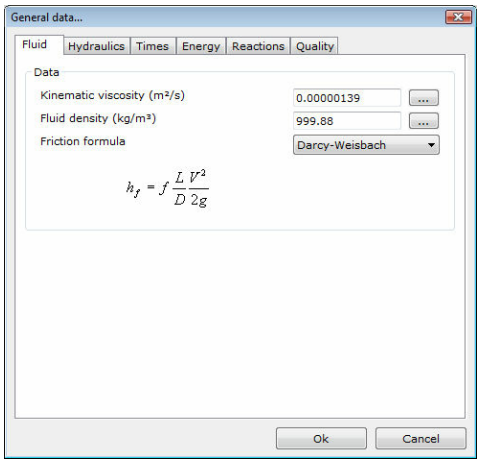
6.4 General data

6.4.1 General

With this option, you can enter friction data.

To enter friction data:

1. Select **General Data** from the **Data** menu.
2. Select the **Friction** tab:



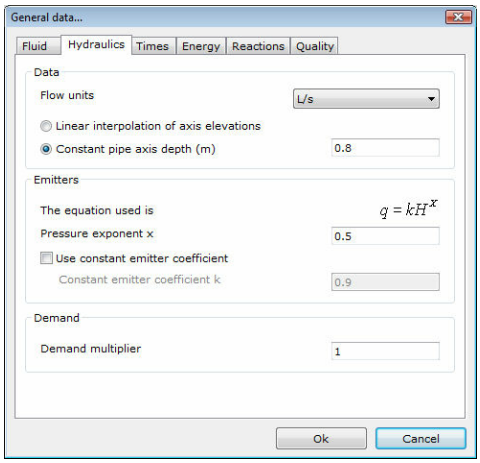
3. Enter the **kinematic viscosity** in m^2/s or ft^2/s. Alternatively, click the button with the ellipses (...) to invoke the corresponding database. This field is required in case of water quality analyses using the Darcy - Weisbach friction formula.
4. Enter the **fluid density** in kg/m^3 or lb/ft^3. Alternatively, click the button with the ellipses (...) to invoke the corresponding database. This field is required in case of water quality analyses using the Darcy - Weisbach friction formula.
5. Select the **friction formula.** This can be one of Manning, Hazen-Williams or Darcy-Weisbach. The selected formula is shown in a window.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.4.2 Hydraulics

With this option, you can enter additional hydraulic data. These are required in case of irrigation networks.

To enter additional hydraulic data:

1. Select **General Data** from the **Data** menu.
2. Select the **Hydraulics** tab:



3. Select the **flow units**. Depending on this setting, all values may revent to Metric or SI system. Values are not converted automatically from one unit system to another, it is the responsibility of the engineer to ensure correct values for all parameters and program variables.
4. Enter the **pipe axis depth** in m or ft. This will be subtracted from all ground elevations automatically upon creation of a new pipe.
5. Enter the **pressure exponent x.** For nozzles and sprinklers, the default value is 0.5.
6. Check **Use constant emitter coefficient** if the emitter coefficient is constant. In this case, enter its value in the corresponding text box. This value will override the values that may be entered in nodes. The manufacturer provides this value, in (L/s)/ $atm^{0.5}$ or gpm/$psi^{0.5}$, and it corresponds to the flow rate with pressure loss of 1 Atm (1 psi).
7. Enter the **demand multiplier** by typing into the corresponding text box. The default value is 1.0. All demands are multiplied by this number; for example, if the multiplier is equal to 2.0, all demands will be doubled.
8. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

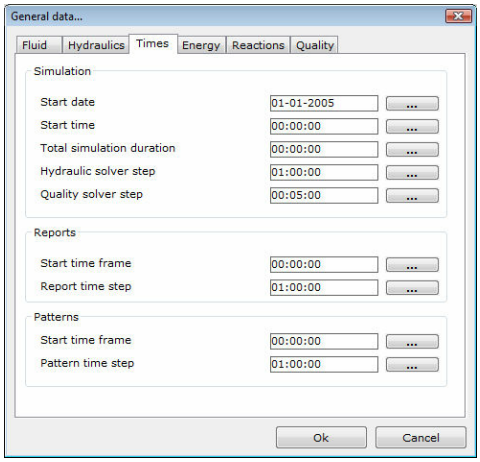
6.4.3 Times

With this option, you can enter time data. These are required in case of extended period analyses.

To enter time data:

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

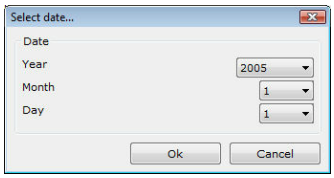
2. Select the **Times** tab:



3. Enter the appropriate data, as described below.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

To change the Start date:

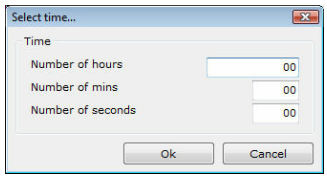
1. Select the corresponding button with the ellipses (...). The following form appears:



2. Select the appropriate **year, month,** and **day** from the corresponding drop-down lists.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

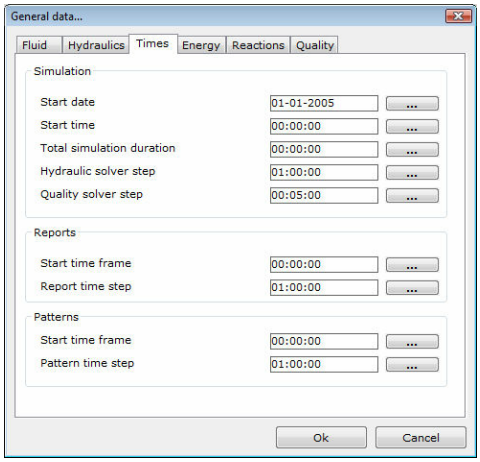
To change a time:

1. Select the corresponding button with the ellipses (...). The following form appears:



2. Enter the appropriate number of **hours, minutes** and **seconds**.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

The available fields are the following:



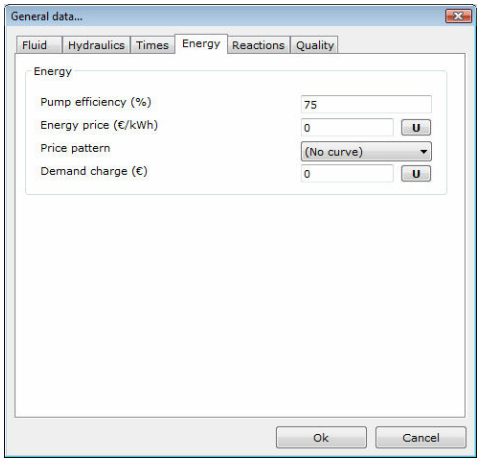
- **Start time**: it is used for compatibility reasons when linking to other models. It is ignored by the solver.
- **Total simulation duration**: if it is zero (00:00:00) then single period analysis is used. If it is non-zero, extended period analyses are performed.
- **Hydraulic solver step**: it is the time step that will be used in extended period analyses. The default value is 1 hour. It is ignored in case of single period analysis.
- **Quality solver step**: it is the time step that will be used in quality analyses. Usually it is much smaller than the hydraulic solver step. It is ignored in case of single period analysis or when no water quality analyses are performed.
- **Start time frame (for reports)**: in case of a simulation of great duration, you may wish to export results not from the beginning but from a later time instant.
- **Report time step**: you can use a large time step to limit the number of result sets. For a simulation of 120 hours and a hydraulic solver step of 1 hour, there will be 120 sets of results. If the report time step is 24 hours, the result sets are limited to 6.
- **Start time frame (for patterns)**: it refers to the starting time of patters.
- **Pattern time step**: it refers to the time step of patters. For example, if it is 2 hours then 12 values are sufficient for a day.

6.4.4 Energy

With this option, you can enter energy data regarding pumps. These are used when performing operational cost and energy analyses.

To enter energy data:

1. Select **General Data** from the **Data** menu.
2. Select the **Energy** tab:



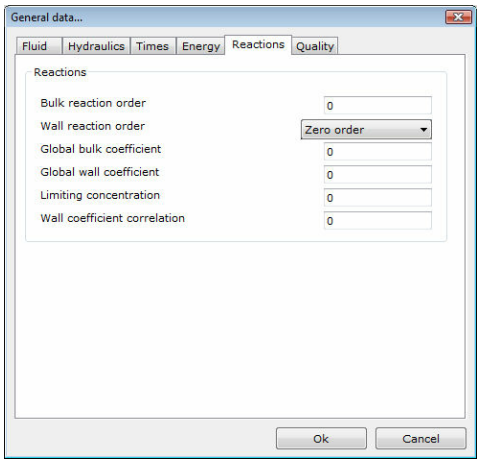
3. Enter the **pump efficiency** in %. This value will be used in case the efficiency of a specific pump is not defined and a pump efficiency curve is not used.
4. Enter the **energy price** in appropriate currency (€, $, etc) per kWh. If the operational cost will not be calculated, you can safely ignore this field.
5. Select the **price pattern** from the drop-down list. After defining a suitable pattern, you can use varying energy cost, as in the case of operation during the night as compared to operation in peak hours.
6. Optionally, enter the additional **demand charge** in appropriate currency (€, $, etc) per kWh.
7. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.4.5 Reactions

With this option, you can enter reaction data. These are used when performing water quality analyses.

To enter reaction data:

1. Select **General Data** from the **Data** menu.
2. Select the **Reactions** tab:



3. Enter the **bulk reaction order**. This is the exponent of the concentration when calculating the reaction rate. For first-order reaction enter 1.0, for second-order

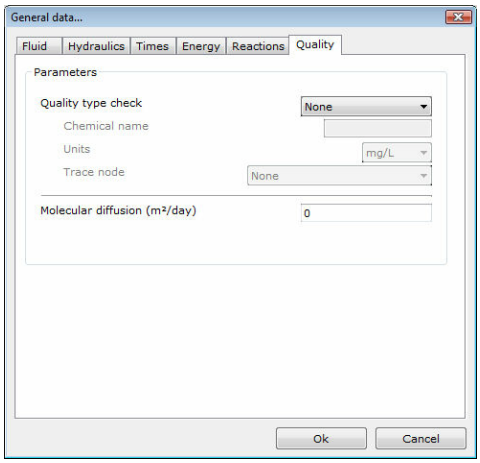
reaction enter 2.0 etc. In order to use kinetics according to Michaelis-Menton, enter any negative value.
4. Select the **wall reaction order** from the drop-down list. Zero-order reaction means constant reaction rate.
5. Enter the **global bulk coefficient.** This value will be used when no value is specified in pipe properties. A positive value means increase whereas a negative value means decrease (decay). Enter zero to ignore the phenomenon.
6. Enter the **global wall coefficient.** This value will be used when no value is specified in pipe properties. A positive value means increase whereas a negative value means decrease (decay). Enter zero to ignore the phenomenon. The unit system is m/day for first-order reaction and mass/m^2/day for zero-order reaction.
7. Enter the **limiting concentration.** This is the maximum (for increase) or minimum (for decrease) concentration. The reaction is analogous to the difference between the current concentration and the limiting concentration.
8. Enter the **wall coefficient correlation.** This is a coefficient that correlates the pipe asperity with the reaction rate.
9. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.4.6 Quality

With this option, you can enter water quality data. These are used when performing water quality analyses.

To enter water quality data:

1. Select **General Data** from the **Data** menu.
2. Select the **Quality** tab:



3. Select the **quality type check** from the drop-down list. This can be one of the following:
- **none**: no quality check will be performed.
- **chemical**: the concentration of a chemical will be calculated.
- **trace**: the percentage of water flow that origins from a specific node will be calculated.
- **age**: the time that water remains in the network will be calculated.

4. In case the quality check is of type **chemical**, enter the name of the **chemical** in the corresponding text box.

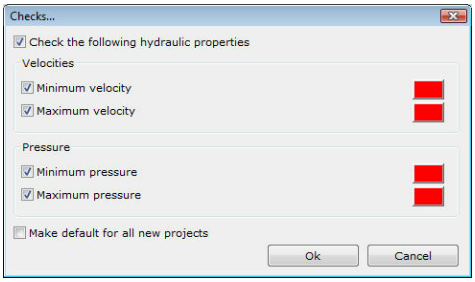
5. In case the quality check is of type **chemical**, select the **unit system** for the concentration. This can be one of mg/L or ug/L=10^{-6}g/L. Note that when the quality check is of type **trace** or **age**, the unit system is percentage and hours, respectively.
6. In case the quality check is of type **trace**, select the **trace node** from the drop-down list.
7. Enter the **molecular diffusion** in m^2/day. This value is used in the diffusion equation.
8. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.4.7 Checks

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

To set the maximum and minimum values of various parameters:

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



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

Velocities

- Check **Maximum velocity** to enable this check.
- Check **Minimum velocity** to enable this check.
- Press the corresponding colored button to change the font color of the results that fail these checks.

Pressure

- Check **Maximum pressure** to enable this check.
- Check **Minimum pressure** to enable this check.
- Press the corresponding colored button to change the font color of the results that fail these checks.

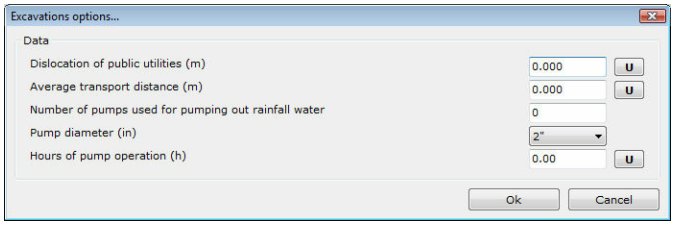
Press **Ok** to accept the changes. Press **Cancel** to close the form without any changes.

6.4.8 Excavation options

With this option, you can set several excavation options.

To set excavation options:

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



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

Data

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

6.5 Curves

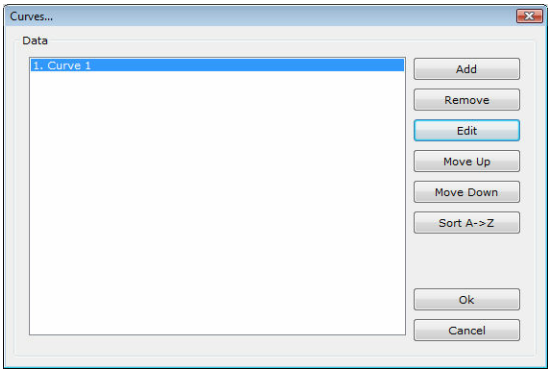
6.5.1 Management

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

- Pump curves
- Pump efficiency curves
- Volume - height curves
- Flow rate - headloss curves

To manage curves:

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

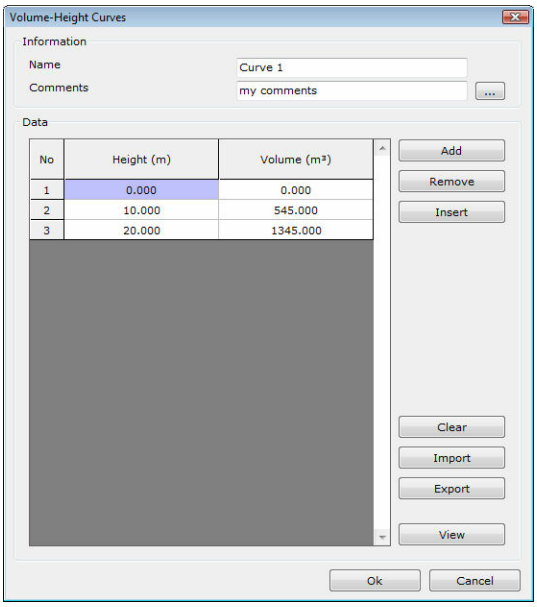


3. Make the necessary modifications.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.5.2 Add

To add a new curve:

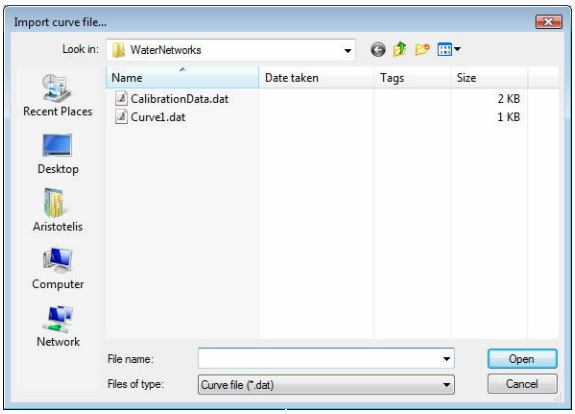
1. Press **Add**. The data form that corresponds to the specified curve type appears.



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

To import a curve from an external file:

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

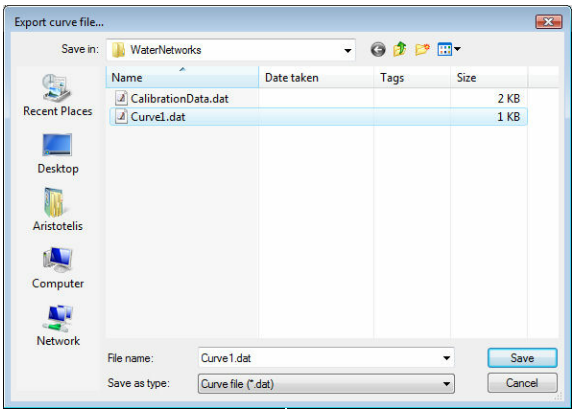


2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Curve file" with the extension .dat.
4. Select the file by clicking on it.
5. Select **Open** to open and analyze the file.

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

To export a curve to an external file:

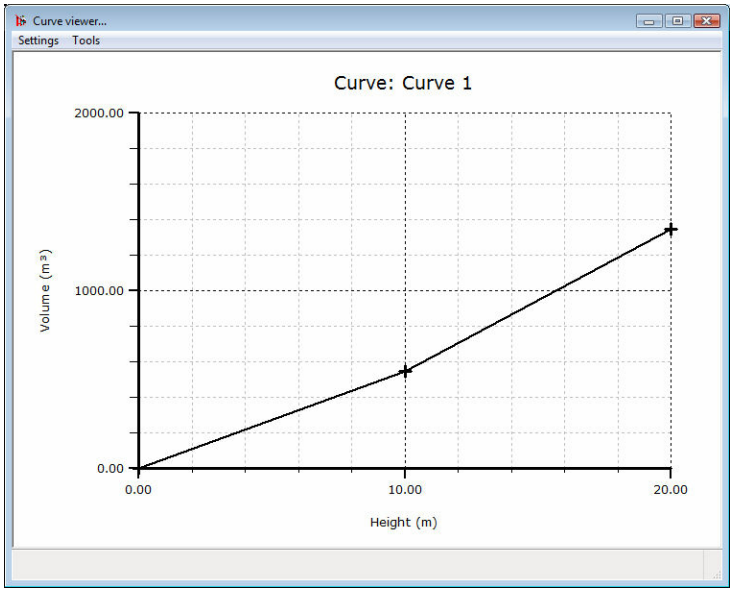
1. Press **Export.** The file selection dialog box appears:



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

To view a curve:

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



6.5.3 Delete

To delete an existing curve:

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

6.5.4 Edit

To edit an existing curve:

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

6.5.5 Move

To move an existing curve upwards in the list:

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

To move an existing curve downwards in the list:

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

6.5.6 Sort

To sort the curve list:

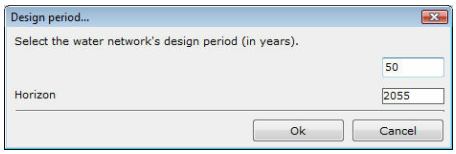
1. Press **Sort A->Z**.
2. The list is sorted alphabetically.

6.6 Design Period

With this option, you can enter the design period of the project. This is used for the calculation of the design population. If this is not of interest, you can safely ignore the design period.

To enter the design period of the project:

1. Select **Design Period** from the **Data** menu. The following form appears:



2. Enter the **design period** in years, by typing into the corresponding text box.
3. The **horizon** is calculated based on the start date and the design period.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

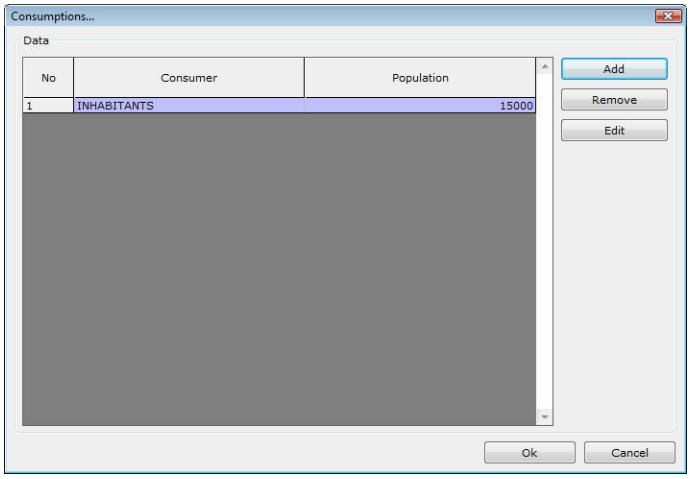
6.7 Consumptions

6.7.1 General

With this option, you can define and modify the various consumers. If the water demands are already known, you need not enter consumer data.

To define and modify the various water consumers:

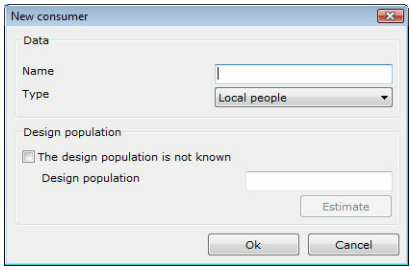
1. Select **Consumptions** from the **Data** menu. The following form appears:



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

To enter a new consumer:

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



2. Enter the **name** of the consumer by typing into the corresponding text box.
3. Select the **type** of the consumer from the drop-down list. This can be one of:
- Local people
- Long-term tourists
- Tourists
- Factories
- Green areas
- Special consumers
- Agricultural use
- Fire hydrant
- Losses

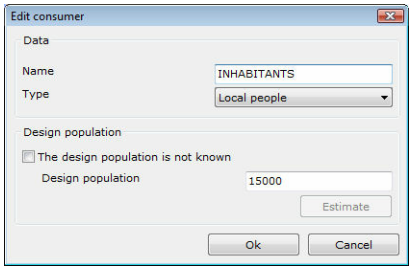
4. If the selected consumer type requires **population** then:
- If the population is known, it is entered in the corresponding text box.
- If the population is not known, check **The design population is not known** and select **Estimate...** to estimate it.

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

NOTE: For proper estimation of the population, the design period is required.

To modify an existing consumer:

1. Select the consumer from the list.
2. Select **Change.** The consumer data form appears:



3. Make the appropriate changes, as described above.

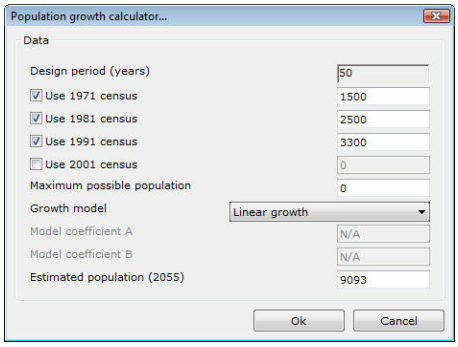
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

To delete an existing consumer:

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

6.7.2 Population Estimation

With this option, you can estimate the design population based on census records. At least two records are required. If the maximum population is known, it is entered in the corresponding text box:



Select the calculation method. This can be one of the following:

- **Linear increase**: the evolution of the population follows the linear model.
- **Geometric increase**: the evolution of the population increases geometrically. The coefficient is estimated but it can be modified by the user.
- **Decaying increase**: the evolution of the population follows the decaying increase model. The maximum population is required. The coefficient is estimated but it can be modified by the user.
- **S curve**: the evolution of the population follows the S curve model. The maximum population is required. The coefficients are estimated but can be modified by the user.

You can select the active census records by checking the appropriate check boxes.

Select **Ok** to close the dialog box and transfer the estimated population to the corresponding fields. Select **Cancel** to close the dialog box without transferring any data.

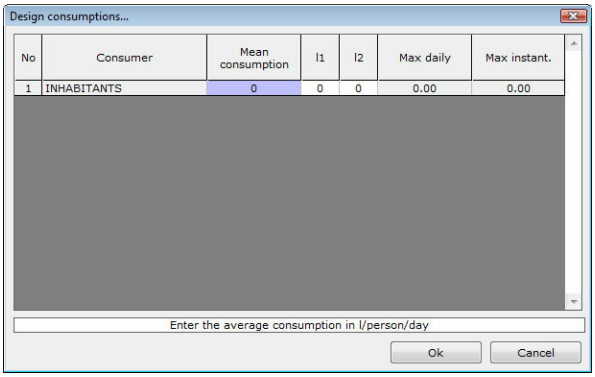
6.8 Design Consumptions

With this option, you can enter design consumptions for the consumers. A description is displayed according to the selected column of the grid.

To enter design consumptions for the consumers:

1. Select **Design consumptions** from the **Data** menu.
2. Enter the appropriate data based on the consumer type. For example, in case of

tourists, the mean consumption should be entered in L/tourist/day and the maximum daily and maximum instantaneous factors are required.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



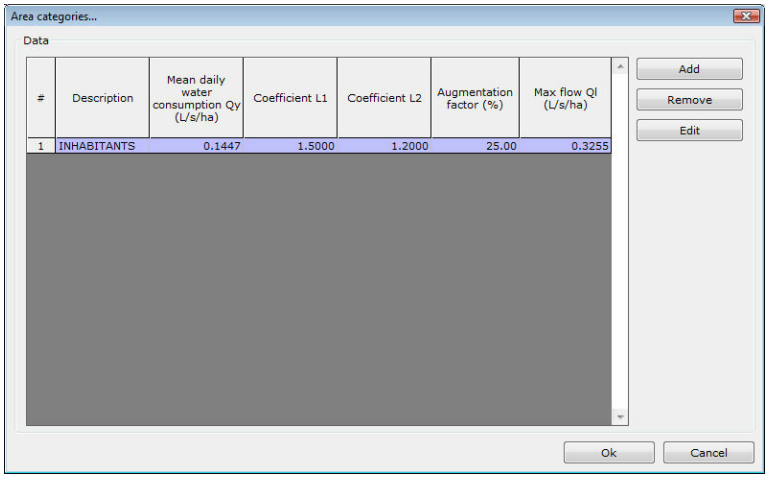
NOTE: The above consumption data are used in the **demand categories** property of the nodes.

6.9 Area categories

Area categories represent areas with different but constant instantaneous demands, in terms of flow units to area unit. Both flow and area units vary, depending on what the user has chosen as his flow units in the Hydraulics tab located in the general data section. While the consumptions are virtual objects, which cannot be represented visually, the areas are objects which can be drawn on the plane view and their areas can either be computed automatically from the coordinates of their vertices or specified by the user.

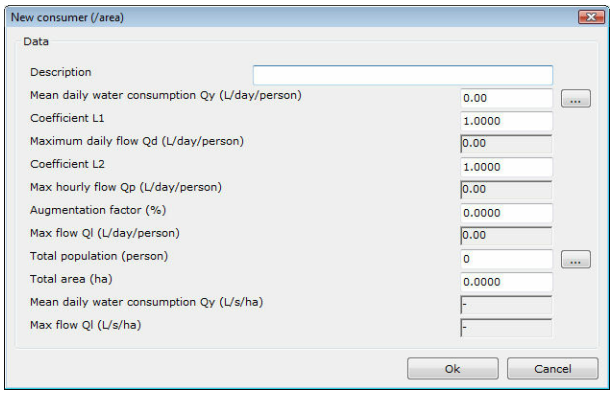
To manage the area categories:

1. Select Area Categories from the Data menu.
2. Make the appropriate changes, as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



To add a new area category:

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

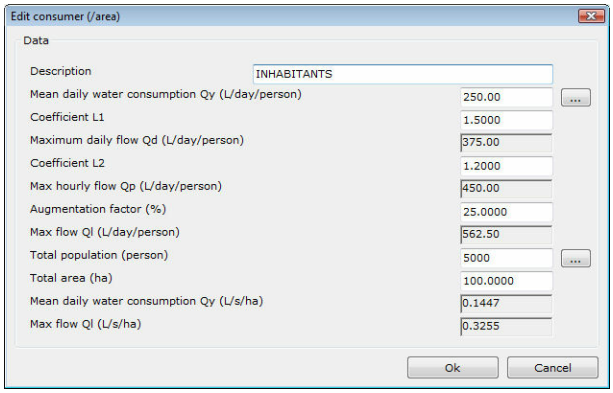


2. Enter the **description** of the area by typing into the corresponding text box.
3. Enter the mean daily consumption. If the average daily consumption is not known, you may estimate its value using the embedded database of daily consumption values by clicking on the ellipses button (...).
4. Enter the coefficient L_1. Usually, its value varies from 1.1 to 1.5. For Athens, the water aythorities suggest a value between 1.15 and 1.20.
5. Enter the instantaneous peak coefficient L_2. This value is used to convert the maximum daily peak to maximum instantaneous.
6. You can optionally enter an augmentation factor which will be used to increase the computed demand.
7. Enter the total population. If the population is unknown, you may estimate it by cliock on the ellipses button (...) which will open the population estimation wizard.
8. Enter the total area in hectares (metric) or acres (english units).
9. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

NOTE: For proper estimation of the population, the design period is required.

To modify an existing area:

1. Select the area from the list.
2. Select **Change.** The area data form appears:



3. Make the appropriate changes, as described above.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

To delete an existing area:

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

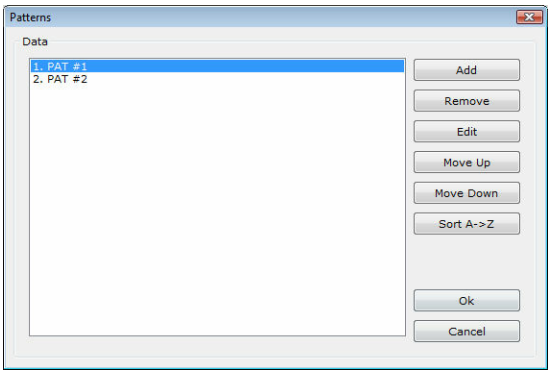
6.10 Patterns

6.10.1 Management

With this option, you can add, edit and delete variation patterns. If all components of the network are constant in time, you do not need to define variation patterns.

To add, edit and delete variation patterns:

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



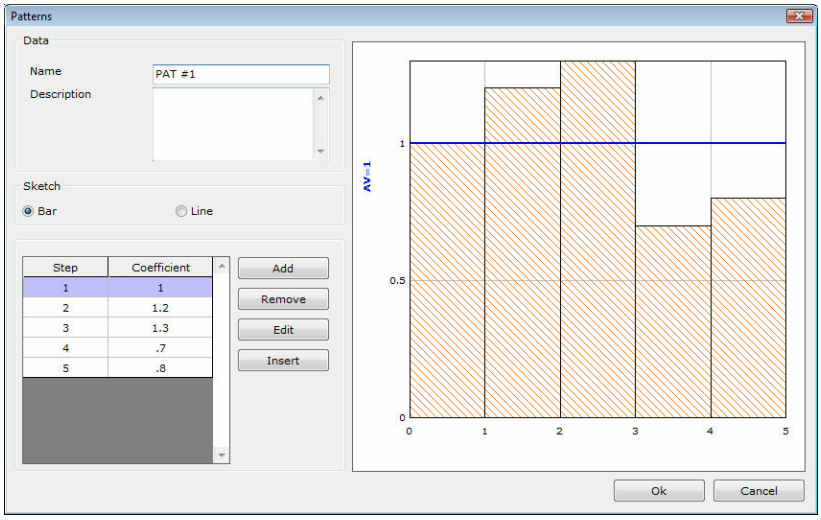
2. Select **Add** to add a new pattern.
3. Select **Remove** to delete the currently selected pattern.
4. Select **Edit** to modify the currently selected pattern.
5. Select **Move up** to move the currently selected pattern by one place.
6. Select **Move down** to move the currently selected pattern by one place.
7. Select **Sort A->Z** to sort the patterns alphabetically.
8. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.10.2 Add

With this option, you can add a new variation pattern. A pattern is a series of pairs of values that describe the variation of a factor. The first column refers to the time step and the second column refers to the corresponding multiplying coefficient. The time step is defined in the Times tab of the General Data. For example, if a factor has a nominal value of 100 then during a time step that has a multiplying coefficient of 1.2, the factor will take the value of 120.

To add a new variation pattern:

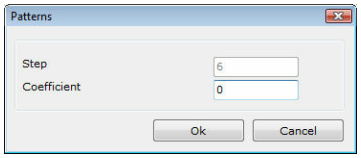
1. Select **Add** from the patterns form. The following form appears:



2. Enter a unique **name** and an optional **description** for the pattern.
3. Select the sketch type of the pattern. This can be one of **bar** or **line**, and it does not affect calculations. The pattern variation and the mean value are displayed in graphical form on the right.
4. Make the appropriate changes in the pattern data, as described next.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

To add a new entry at the end of the list:

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



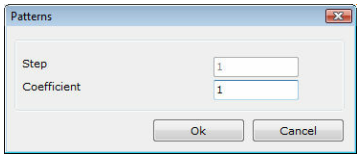
2. Enter the value of the coefficient.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

To delete an existing entry:

1. Select the entry from the list.
2. Select **Delete**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. If you select **Yes**, the entry is deleted.

To modify an existing entry:

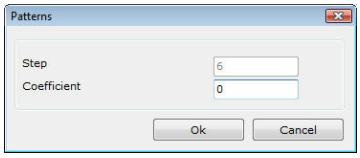
1. Select the entry from the list.
2. Select **Edit**. The following form appears:



3. Enter the new value of the coefficient.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

To insert a new entry:

1. Select the entry above which you wish to insert a new one.
2. Select **Insert**. The following form appears:



3. Enter the value of the coefficient.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.10.3 Delete

With this option, you can delete an existing variation pattern.

To delete an existing variation pattern:

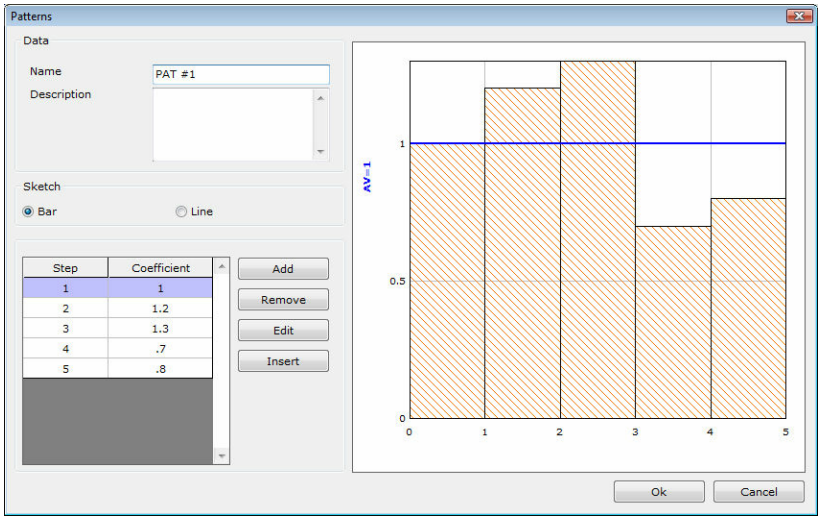
1. Select the pattern from the list in the patterns form.
2. Select **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. If you select **Yes**, the pattern is deleted.

6.10.4 Modify

With this option, you can modify an existing variation pattern.

To modify an existing variation pattern:

1. Select the pattern from the list in the patterns form.
2. Select **Change**. The following form appears:



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

6.10.5 Move

To move an existing curve upwards in the list:

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

To move an existing curve downwards in the list:

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

6.10.6 Sort

To sort the pattern list:

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

6.11 Rules

6.11.1 Management

Rules determine how pipes, pumps and valves in the drainage system will be adjusted over the course of a simulation.

Each rule is a series of statements of the form:

RULE ruleID

IF condition_1
AND condition_2

OR condition_3
AND condition_4
Etc.

THEN action_1
AND action_2
Etc.

ELSE action_3
AND action_4
Etc.

PRIORITY value

Only the **RULE**, **IF** and **THEN** portions of a rule are required; the **ELSE** and **PRIORITY** portions are optional.

When mixing **AND** and **OR** clauses, the **OR** operator has higher precedence than **AND**, i.e.,

IF A **OR** B **AND** C

is equivalent to:

IF (A **OR** B) **AND** C.

If the interpretation was meant to be

IF A **OR** (B **AND** C)

then this can be expressed using two rules as in

IF A **THEN** ...
IF B **AND** C **THEN** ...

The **PRIORITY** value is used to determine which rule applies when two or more rules require that conflicting actions be taken on a link. A rule without a priority value always has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.

Condition clauses

A condition clause of a rule has the following format:

object id attribute relation value

where **object** object type
id object name
attribute attribute or property of the object
relation relational operator (=, <>, <, <=, =>, >)
value property value

Some examples of condition clauses are:

```
NODE        N1     DEPTH       >      10
PUMP        A1     STATUS      =      OFF
SYSTEM      CLOCKTIME          =      22:45:00
```

The objects and attributes that can appear in a condition clause are as follows:

Object	Property	Value
Node	Depth Head Pressure	numerical value numerical value numerical value
Tank	Level Filltime Draintime	numerical value numerical value numerical value
Link	Status Flow Setting	Open/Closed/Active numerical value pump speed or valve setting
System	Demand Time Clocktime (00:00:00)	total system demand hours from start of simulation time of day in hr:min:sec

Action clauses

An action clause of a rule can have one of the following formats:

- PUMP id STATUS = ON/OFF
- PUMP id SETTING = value

where the meaning of SETTING depends on the object being controlled:

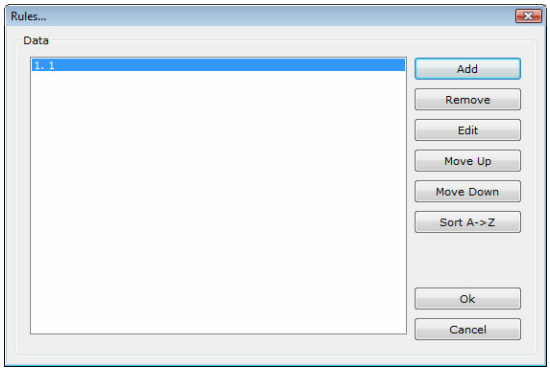
- for pumps it is a multiplier applied to the flow computed from the pump curve or a status condition.
- for pipes it is the status condition.
- for valves it is a status condition or a setting value.

Some examples of action clauses are:

```
PUMP P67 STATUS = OFF
VALVE V212 SETTING = 0.5
```

To manage rules:

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



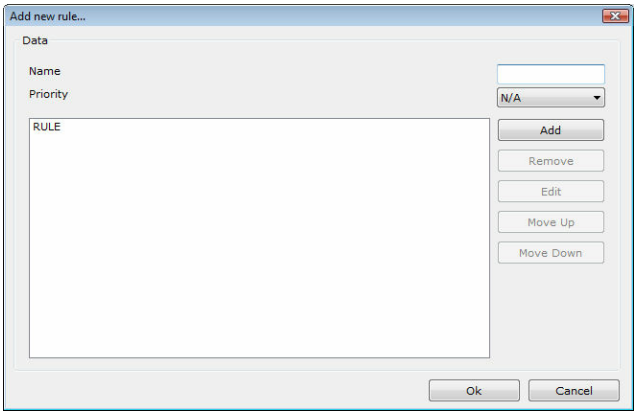
2. Make the necessary modifications.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

6.11.2 Add

With this option, you can add a new rule.

To add a new rule:

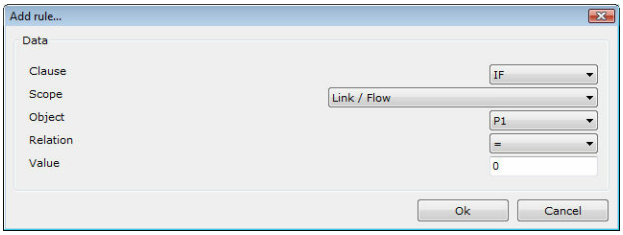
1. Press **Add**. The data form appears:



2. Enter a user-assigned **name**. This name cannot be null or used for another rule in the project.
3. Optionally, you can assign a **priority** level for the rule.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To enter a new logical statement to the rule:

1. Press **Add** to enter a new logical statement to the rule. The following form appears:



2. Select the **Clause**, **Scope**, **Object**, allowed **Relation** and **Value**. Value may be one of the following:

- Option 1: a boolean (YES/NO)
- Option 2: OPEN/CLOSED/ACTIVE if a status setting is required
- Option 3: a numeric value

Not all options are available for a selected scope. If more than one option is available, the user must specify which one will be used by activating the radio button on the left of the option.

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

To delete an existing logical statement of a rule:

1. Select the statement from the list.
2. Select **Delete**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. If you select **Yes**, the statement is deleted.

To modify an existing logical statement of a rule:

1. Select the statement from the list.
2. Select **Edit**.
3. Make the appropriate changes.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

To modify the position of an existing logical statement of a rule:

1. Select the statement from the list.
2. Select **Move Up** to move the statement up by one row. Select **Move Down** to move the statement down by one row.

6.11.3 Delete

To delete an existing rule:

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

6.11.4 Edit

To edit an existing rule:

1. Select the rule from the list on the left.

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

6.11.5 Move

To move an existing rule upwards in the list:

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

To move an existing rule downwards in the list:

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

6.11.6 Sort

To sort the rule list:

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

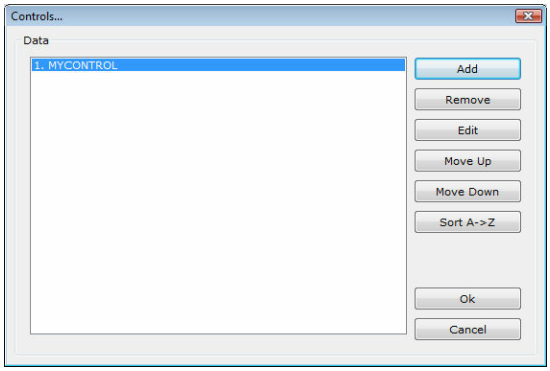
6.12 Controls

6.12.1 Management

With this option, you can define control rules that affect the behaviour of linear objects (pipes, pumps, valves) based on the status of a point object (node, tank, reservoir) or the time instant of the simulation.

To define control rules:

1. Select **Controls** from the **Data** menu.
2. Make the appropriate changes, as described below.
3. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

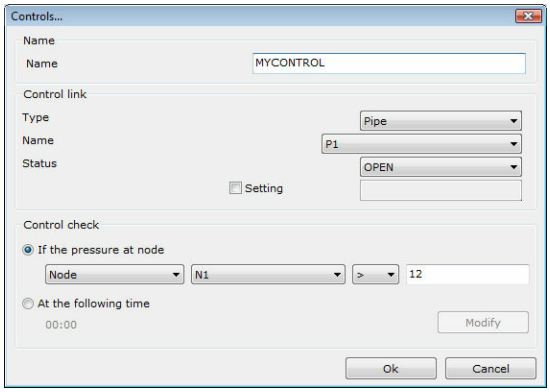


6.12.2 Add

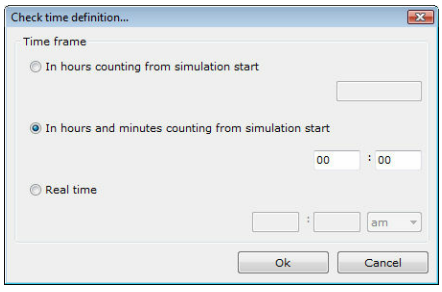
With this option, you can add a new rule.

To add a new rule:

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



2. Enter the name of the control.
3. Select the link type and depending on the type, the link name.
4. For the combinarion of link type and name, the status drop down list loads allowed options. Select one option from the list of available options.
5. Select the new status of the control object. Pipes can become **Open** or **Closed**. Pumps can become **Open** or **Closed** or obtain a new **Speed** property. Valves can become **Open** or **Closed** or obtain a new **Setting** property. For the **Speed** or **Setting** property, check the check box below the status and enter the new property value in the corresponding text box.
6. Select the control check that will trigger the modification. The check can be a **pressure relation** at a specific point object (node, tank, reservoir) or a specific **time instant**. Select the check type by clicking on the corresponding option button of the **Control check** frame.
7. In case of a pressure relation at a specific point object, select the **type** and **name** of the object, the **inequality sign** and the pressure value.
8. In case of a specific time instant, select the time by selecting **Modify.** The following form appears:



- Select the method of calculating the time instant. This can be in decimal hours or hours and minutes from simulation start, or a real time instant.
- Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

9. Select **Ok** to add the new control rule in the list or **Cancel** to discard any changes.

6.12.3 Delete

To delete an existing control rule:

1. Select the control rule from the list.
2. Select **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. If you select Yes, the control rule is deleted.

6.12.4 Edit

To modify an existing control rule:

1. Select the control rule from the list.
2. Make the appropriate changes.
3. Select **Change** to store the updated values.

6.12.5 Move

To move an existing control rule upwards in the list:

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

To move an existing control rule downwards in the list:

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

6.12.6 Sort

To sort the control list:

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

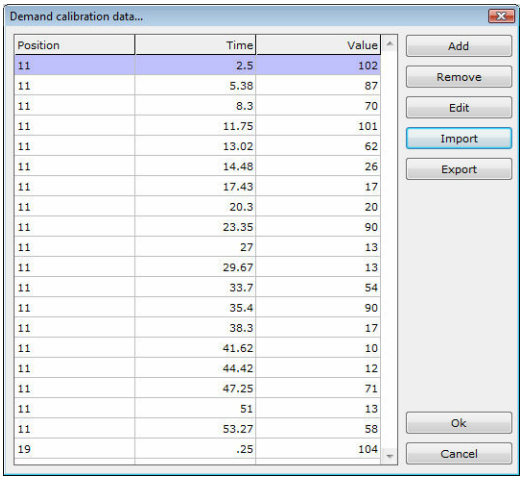
6.13 Calibration Data

6.13.1 General

With this option, you can enter calibration data from experiments to be compared with the results obtained from the model.

To enter calibration data:

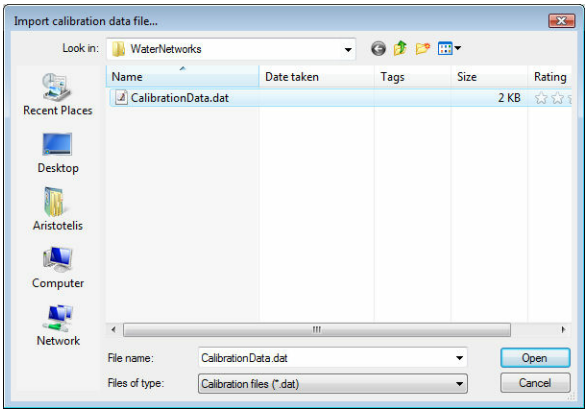
1. Select **Calibration Data** from the **Data** menu.
2. Select one of **Demand**, **Pressure Height**, **Pressure**, **Quality**, **Flow Rate**, **Velocity** from the **Calibration Data** menu. The corresponding data form appears:



3. Select **Add** to add a new entry.
4. Select **Remove** to delete the currently selected entry.
5. Select **Modify** to modify the currently selected entry .
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Alternatively, to add calibration data from an external file:

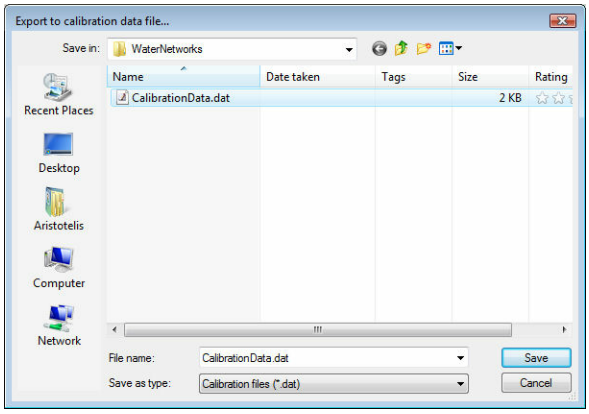
1. Select **Import**. The file selection dialog box appears:



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

To export the current calibration data to an external file:

1. Select **Export**. The following dialog box appears:



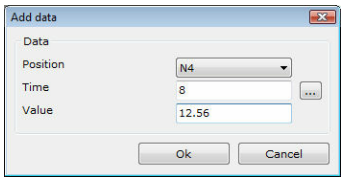
2. Select the path of the file.
3. Type the filename in the **File name** text box.
4. Select **Save** to save the file. Select **Cancel** to cancel the operation.

6.13.2 Add

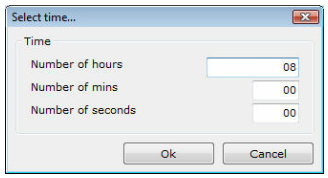
With this option, you can add a new calibration data entry.

To add a new calibration data entry:

1. Select **Add** from the calibration data form. The following form appears:



2. Enter the **position** of the object. In case of velocity or flow rate, the linear objects are available in the drop-down list. In all other cases, the point objects are available in the drop-down list.
3. Enter the decimal **time** corresponding to the measurement, in hours. Alternatively, select the button with the ellipses (...) to invoke the following form:



Select **Ok** to convert the hours, minutes and seconds to decimal hours and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

4. Enter the **value** of the measurement.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.13.3 Delete

With this option, you can delete an existing calibration data entry.

To delete an existing calibration data entry:

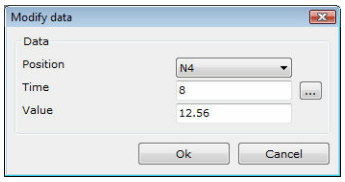
1. Select the calibration data entry from the list in the calibration data form.
2. Select **Remove**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. If you select **Yes**, the calibration data entry is deleted.

6.13.4 Modify

With this option, you can modify an existing calibration data entry.

To modify an existing calibration data entry:

1. Select the calibration data entry from the list in the calibration data form.
2. Select **Change**. The following form appears:



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

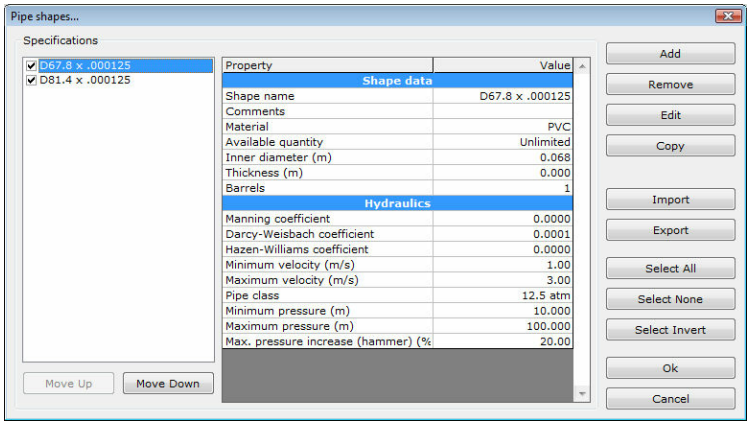
6.14 Conduit shapes

6.14.1 Management

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

To add or modify conduit shape specifications:

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



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

To change the order of pipe specifications:

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

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

The quick keys perform the following actions:

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

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

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

6.14.2 Add

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

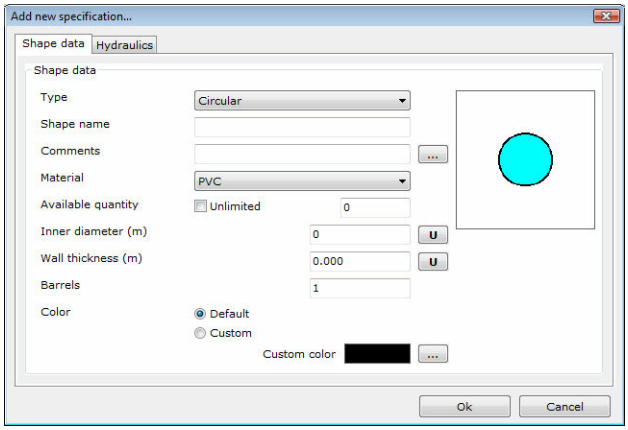
To add a new (empty) entry:

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

To add a new entry by copying an existing entry:

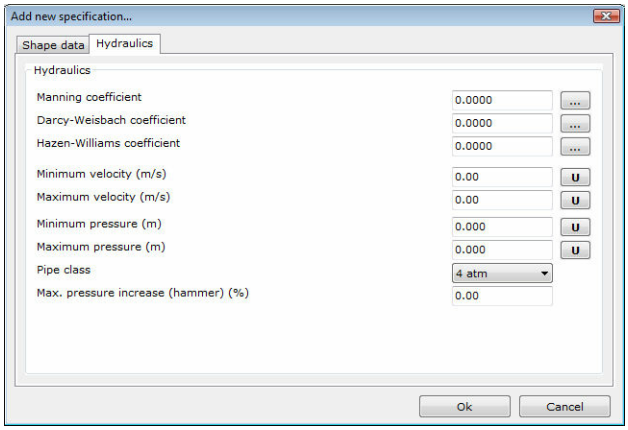
1. Select the existing (base) entry from the list.
2. Select **Copy**. A copy of the existing entry is created.
3. Select **Edit** to modify the properties of the clone entry, as described below.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without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

Shape Data Tab



- Select the **type** of the shape from the drop-down list.
- Enter a unique **shape name.**
- Optionally, you can enter some **comments.** Use the ellipsis button to edit multiline text.
- Select the **material** of the conduit from the drop-down list.
- Select whether the specified shape is available in **unlimited quantity.** If not, enter the available quantity in ft or m.
- Enter the **inner diameter** of the shape.
- Enter the **wall thickness** of the shape.
- Enter the number of **barrels**. In case of twin conduits, this equals 2. Normally, the number of barrels is equal to 1.
- Select the **color** that will be used to draw the conduit on the plane view. It can be either the default color or a custom color. This color will also be used when the plane view DXF drawing is created.

Hydraulics Tab



- Enter the Manning, Darcy - Weisbach, and Hazen - Williams friction coefficients. If these are not known, press the corresponding ellipsis button to invoke the corresponding database.
- Enter the **minimum velocity** in ft/s or m/s of the conduit shape. This is not a restriction but a calculation check. If violated, the velocity is printed in red. Additional checks can be defined. However, the checks in the specifications are dominant.

- Enter the **maximum velocity** of the conduit shape. This is not a restriction but a calculation check. If violated, the capacity of the conduit is printed in red. Additional checks can be defined. However, the checks in the specifications are dominant.
- Enter the **minimum pressure** in atm or psi for the junctions. This is not a restriction but a calculation check. If violated, the pressure head is printed in red. Additional checks can be defined. However, the checks in the specifications are dominant.
- Enter the **maximum pressure** in atm or psi for the junctions. This is not a restriction but a calculation check. If violated, the pressure head is printed in red. Additional checks can be defined. However, the checks in the specifications are dominant.
- Select the **pipe class** from the drop down list. This is used in the water hammer calculations.
- Select the **maximum pressure increase** for the water hammer calculcations. If the class of the pipe is 10 atm and 20% is entered, then a maximum of 120 m pressure head will be allowed.

6.14.3 Delete

To delete an existing specification:

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

6.14.4 Edit

To edit an existing specification:

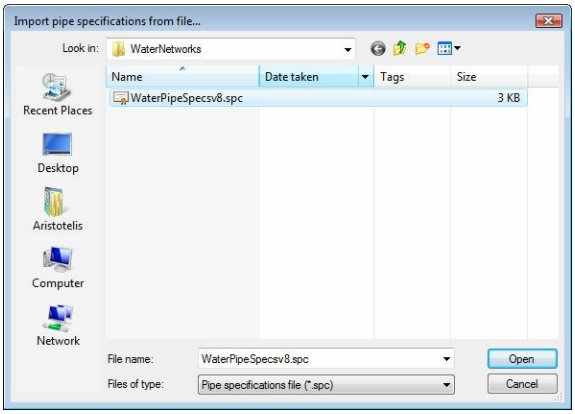
1. Select the specification from the list on the left.
2. Press **Edit**. The data form appears.
3. Make the appropriate selections as described in the add specification topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any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

6.14.5 Import

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

To import specifications from an external file:

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



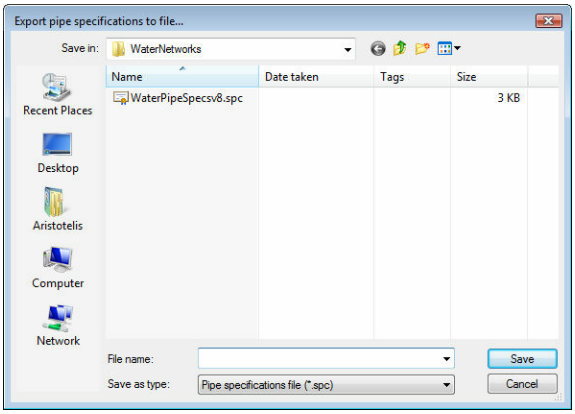
2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Pipe specification file" with the extension .spc.
4. Select the file by clicking on it.
5. Select **Open** to import the specifications.

6.14.6 Export

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

To export pipe specifications to an external file:

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



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

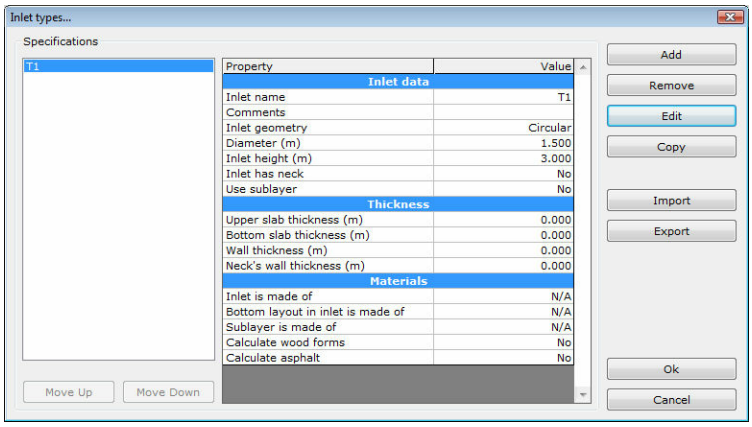
6.15 Manhole specifications

6.15.1 Management

With this option, you can add or modify inlet specifications.

To add or modify inlet specifications:

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



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

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

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

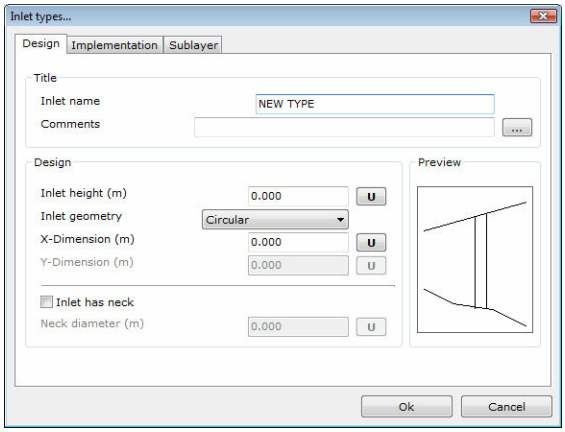
6.15.2 Add

With this option, you can add an inlet specification.

To add an inlet specification:

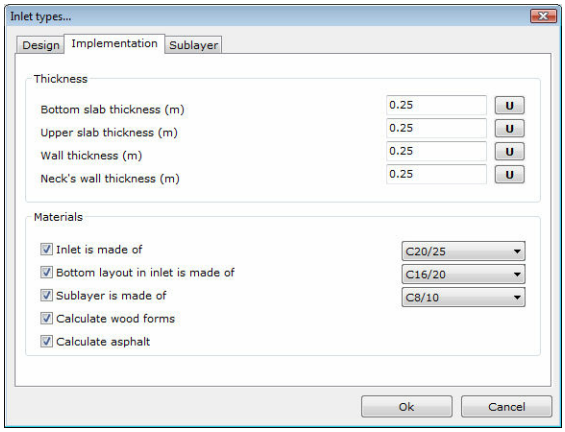
1. Select **Add**. A new entry is added to the list.
2. Enter the appropriate data as described below.
3. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Design Tab



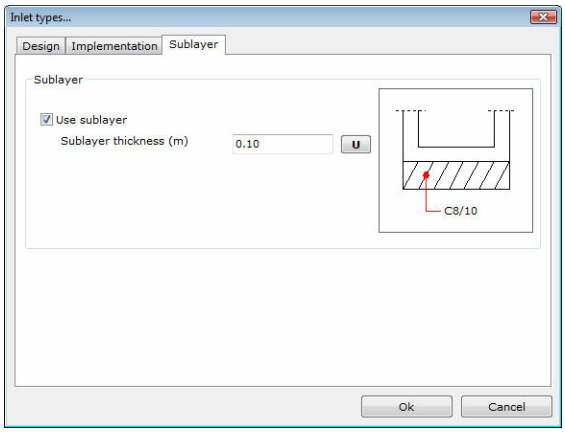
- Enter a unique **Name** for the specification.
- Optionally, you can add some **Comments.** Use the ellipsis button to edit multiline text.
- Enter the external **inlet height** in ft or m.
- Select tine **inlet geometry**. This can be one of **circular** or **rectangular.**
- Enter the external dimensions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

Implementation Tab



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

Sublayer Tab



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

6.15.3 Delete

To delete an existing specification:

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

6.15.4 Edit

To edit an existing specification:

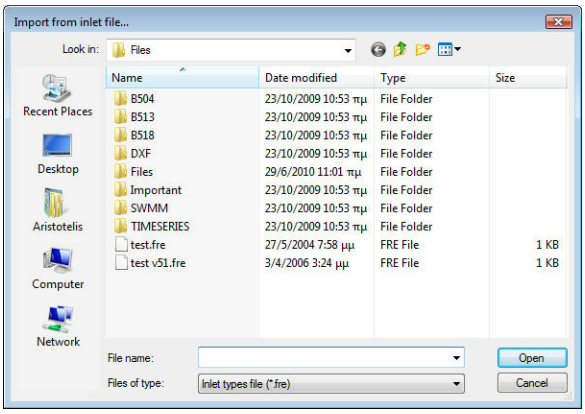
1. Select the specification from the list on the left.
2. Press **Edit**. The data form appears.
3. Make the appropriate selections as described in the add specification topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any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

6.15.5 Import

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

To import specifications from an external file:

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



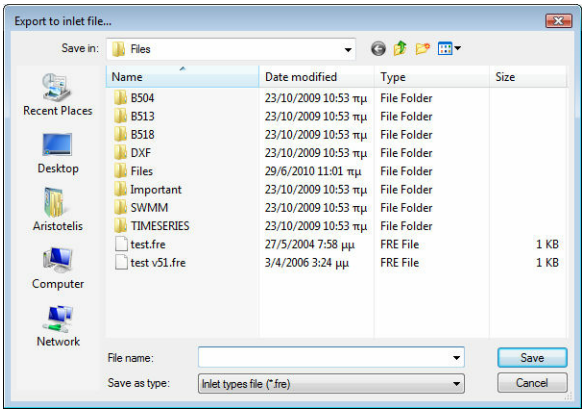
2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Inlet type file" with the extension .fre.
4. Select the file by clicking on it.
5. Select **Open** to import the specifications.

6.15.6 Export

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

To export pipe specifications to an external file:

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



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

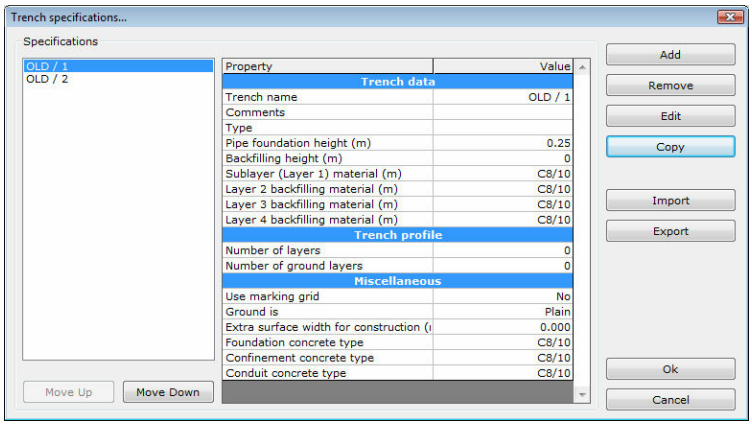
6.16 Trench specifications

6.16.1 Management

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

To add or modify trench specifications:

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



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

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

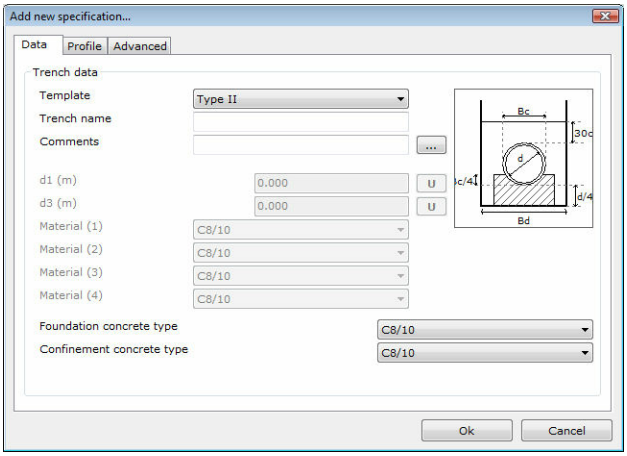
The next sections describe the process of adding, modifying or deleting a trench specification.

6.16.2 Add

With this option, you can add a trench specification.

To add a trench specification:

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

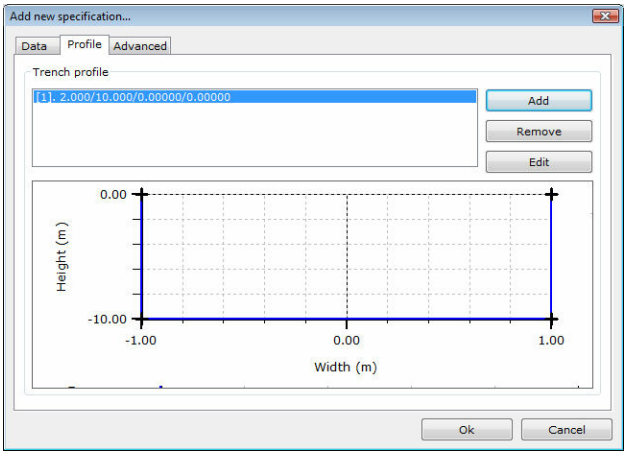


2. Make the appropriate selections as described below.
3. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

Data Tab

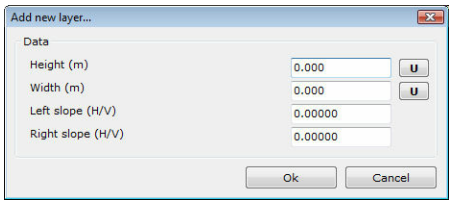
- Select a **template** for the trench.
- Enter a unique **Name** for the trench specification.
- Optionally, you can add some **Comments.** Use the ellipsis button to edit multiline text.
- Depending on the selected trench template, enter the required dimensions and / or material types. Unnecessary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

Profile Tab



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

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



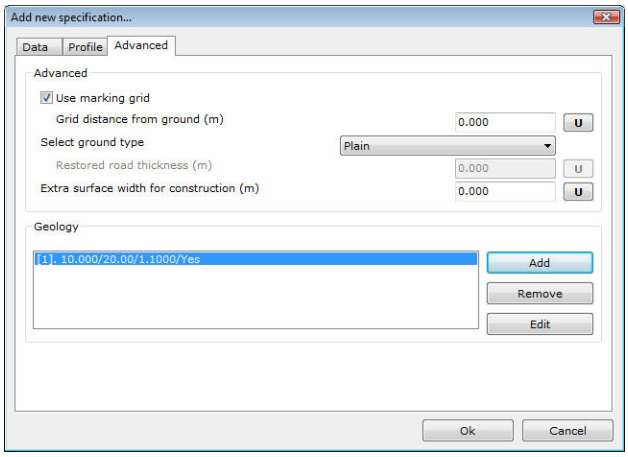
Enter the **height**, **width** and **slopes** of the layer. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

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

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

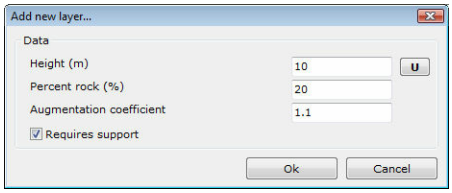
A sketch of the current trench profile is drawn in the form.

Advanced Tab



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

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



Enter the **height**, **rock percentage** and **augmentation factor** of the layer. Check **Requires support** if the specified layer requires support. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

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

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

6.16.3 Delete

To delete an existing specification:

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

6.16.4 Edit

To edit an existing specification:

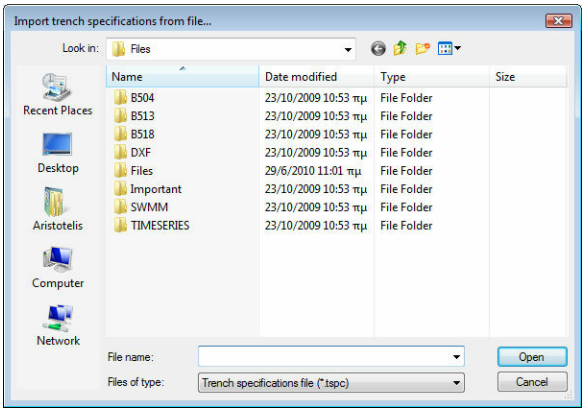
1. Select the specification from the list on the left.
2. Press **Edit**. The data form appears.
3. Make the appropriate selections as described in the add specification topic.
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any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

6.16.5 Import

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

To import specifications from an external file:

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



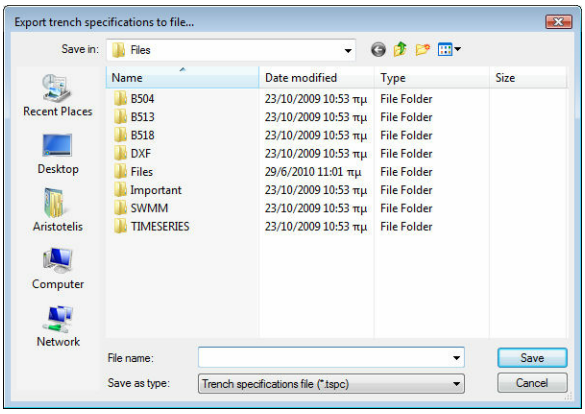
2. Select the path of the file.
3. Select the file type from the **Files of type** drop-down list. The default option is "Trench specifications file" with the extension .tspc.
4. Select the file by clicking on it.
5. Select **Open** to import the specifications.

6.16.6 Export

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

To export pipe specifications to an external file:

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



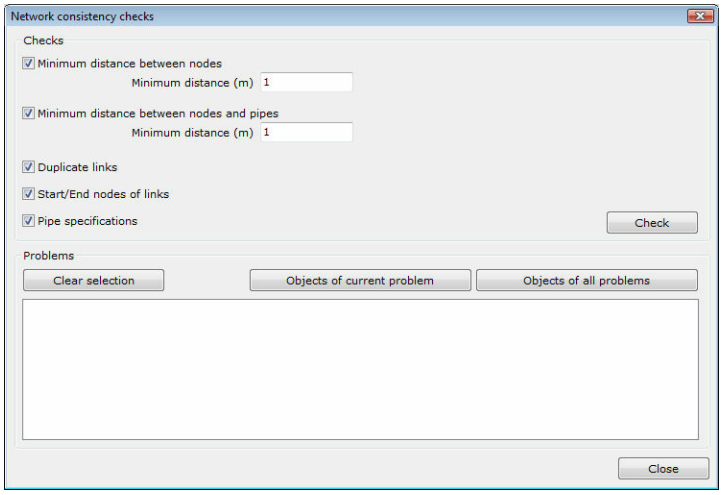
2. Select the path of the file.
3. Type the filename in the **File name** text box.
4. Select **Save** to export pipe specifications with the selected filename and path. Select **Cancel** to cancel the operation.

6.17 Network consistency

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

To use the network consistency check tool:

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



6.18 Options

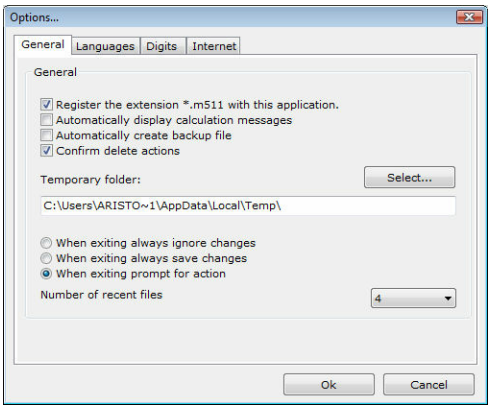
6.18.1 General preferences

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

To modify the general preferences:

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

General Tab



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

Check **Register the extension *.m511 with this application** to associate the extension .m511 with this program. This extension is used by the program when saving a project. In this way, you will be able to run the program and load a project by double-clicking on the project filename in Windows Explorer.

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

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

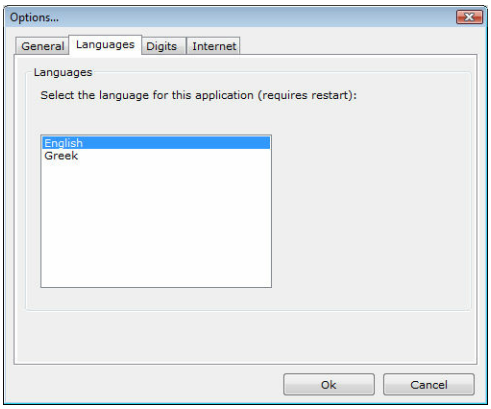
Check **Confirm delete actions** if you want to be asked for confirmation each time an object is about to be deleted. This setting affects the behaviour of all delete actions, for example the deletion of an object.

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

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

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

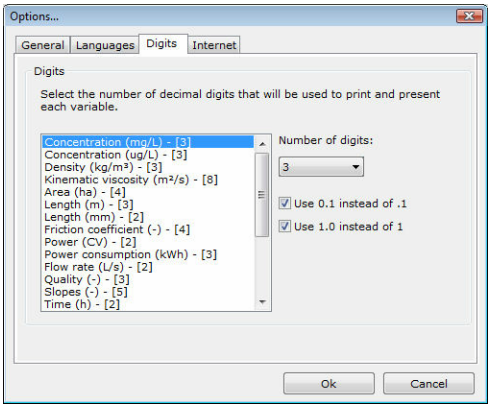
Languages Tab



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

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

Digits Tab



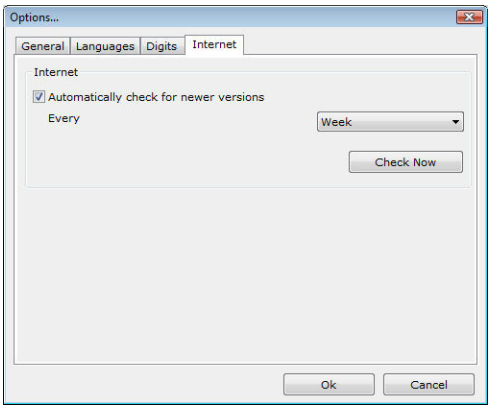
With this tab, you can modify the way the results are presented. All values used in the program are displayed in the list on the left.

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

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

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

Internet Tab



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

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

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

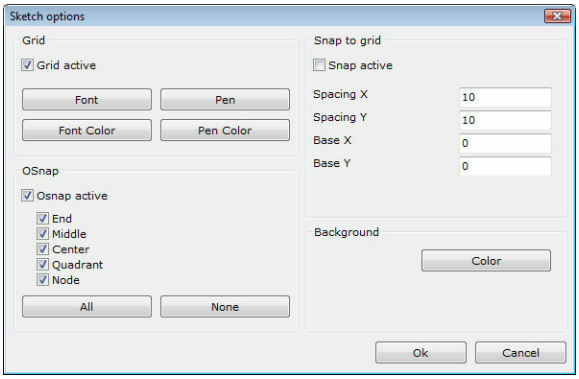
Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.18.2 Sketch

With this option, you can modify the profile sketch.

To modify the profile sketch:

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



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

Grid

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

OSnap

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

Snap to grid

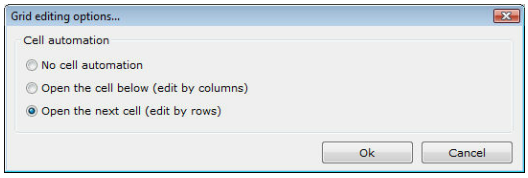
- Select **Snap active** if you want the snap to grid to be active.
- Select the appropriate **Spacing X** and **Spacing Y** values.
- Select the appropriate **Base X** and **Base Y** values.

Background

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

6.18.3 Grid editiing

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



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

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

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

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

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

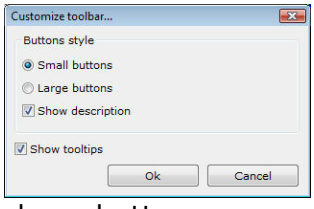
Select **Ok** to apply the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

6.18.4 Customize toolbar

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

To customize the toolbar of the main form:

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



The toolbar may contain small or large buttons.

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

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

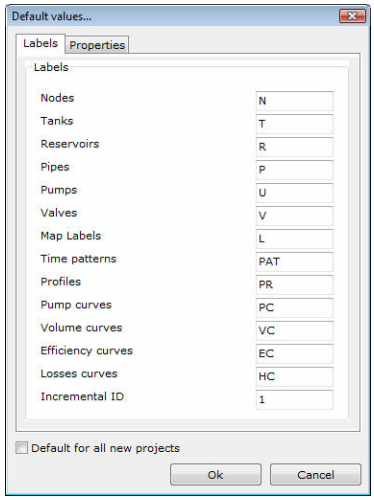
NOTE: These preferences affect all projects, both old and new.

6.18.5 Default values

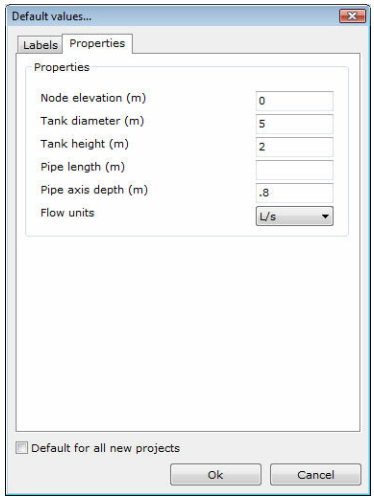
With this option, you can modify the default values for new linear or point objects, curves etc.

To modify the default values:

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



4. In the Properties tab select the default values for other new objects.



5. Select **Make default for all new projects** if you wish to use these settings for all new projects.
6. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the

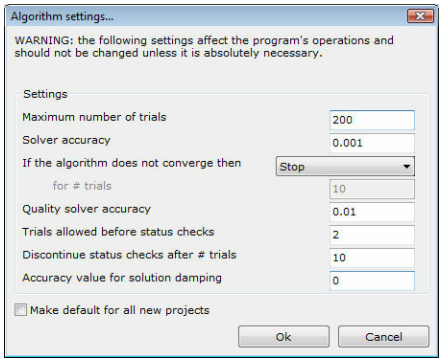
dialog box without saving any changes.

6.18.6 Algorithm

With this option, you can modify the settings of the solver. Note, however, that the use of the default values is highly recommended.

To modify the settings of the solver:

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



3. Enter the **maximum number of trials.** This ensures that the solver will not be trapped in endless loops. The use of 50 or more trials is sufficient for most cases.
4. Enter the **solver accuracy**. An accuracy of 0.001 is sufficient for most cases.
5. If the algorithm does not converge, then the process can be stop, continue for an additional number of trials, or the problem may be ignored.
6. Enter the **quality solver accuracy**. An accuracy of 0.01 is sufficient for most cases.
7. The option **Trials allowed before status checks** sets the number of solution trials that pass during hydraulic balancing before the status of pumps, check valves, flow control valves and pipes connected to tanks are once again updated. The default value is 2, meaning that status checks are made every other trial. A value equal to the maximum number of trials would mean that status checks are made only after a system has converged. (Whenever a status change occurs the trials must continue since the current solution may not be balanced.) The frequency of status checks on pressure reducing and pressure sustaining valves (PRVs and PSVs) is determined by the damping option (see #9).
8. The **discontinue status checks after # trials** is the number of solution trials after which periodic status checks on pumps, check valves flow control valves and pipes connected to tanks are discontinued. Instead, a status check is made only after convergence is achieved. The default value is 10, meaning that after 10 trials, instead of checking status every given number of trials, status is checked only at convergence.
9. The **accuracy value for solution damping** is the the accuracy value at which solution damping and status checks on PRVs and PSVs should begin. Damping limits all flow changes to 60% of what they would otherwise be as future trials unfold. The default is 0 which indicates that no damping should be used and that status checks on control valves are made at every iteration. Damping might be needed on networks that have trouble converging, in which case a limit of 0.01 is suggested.
10. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

Chapter

7 Objects

7.1 Objects menu

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

- Add
  - Junction
  - Tank
  - Reservoir
  - Pipe
  - Pump
  - Valve
  - Area
  - Profile
  - Label
- Properties
  - Node
  - Tank
  - Reservoir
  - Link internal vertices
  - Pipe
  - Pump
  - Valve
  - Area
  - Profile
  - Label
- Object conversion
- Add vertex
- Add vertex by distance
- Delete vertex
- Stretch vertex
- Convert vertex to junction
- Labels
- Swap link ends

7.2 Add

7.2.1 Node

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

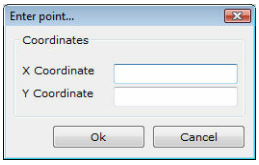
To add a junction:

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

To add more than one junctions:

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

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



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

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

7.2.2 Tank

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

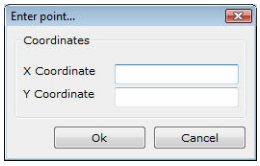
To add a tank:

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

To add more than one tanks:

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

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



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

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

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

7.2.3 Reservoir

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

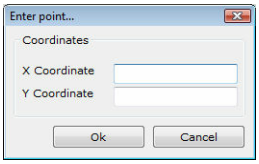
To add a reservoir:

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

To add more than one reservoirs:

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

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



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

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

7.2.4 Pipe

With this option, you can add one or more pipes. The ends of the pipe are always point objects (Junctions, Tanks, Reservoirs). Although there is no such restriction, it is recommended that the direction of pipes matches the flow direction.

To add a pipe:

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

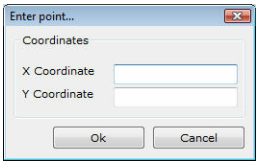
To add more than one pipes:

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

NOTE: At least two point objects are required.

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

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



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

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

7.2.5 Pump

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

To add a pump:

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

To add more than one pumps:

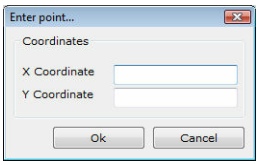
1. Select **Add** from the **Objects** menu.

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

NOTE: At least two point objects are required.

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

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



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

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

7.2.6 Valve

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

To add a valve:

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

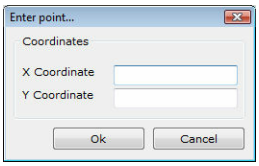
To add more than one valves:

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

NOTE: At least two point objects are required.

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

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



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

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

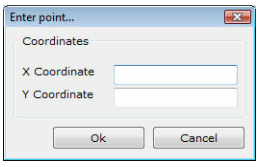
7.2.7 Area

With this option you can add a new area, which will be used to calculate the demands of the consumers.

To add a new area:

1. Select **Add** from the **Objects** menu.
2. Select **Area** from the **Add** menu.
3. With multiple clicks on the plane view you can define the vertices of the area's polygon.
4. Hit ENTER to create the area (the open polygon will be designed as closed automatically) or ESC to cancel your changes and stop the procedure without adding a new area.

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



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

NOTE: When selecting points graphically, you can use Snap and / or OSnap. These

options can be configured using the menu **Data > Options > Sketch** or by hitting **CTRL + 1**.

NOTE: You can use the areas from DXF tool to import closed polylines from an existing dxf file.

7.2.8 Profile

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

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

To add a profile:

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

To add more than one profiles:

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

7.2.9 Label

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

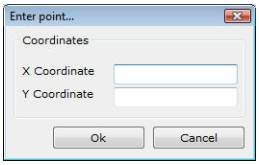
To add a label:

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

To add more than one labels:

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

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



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

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

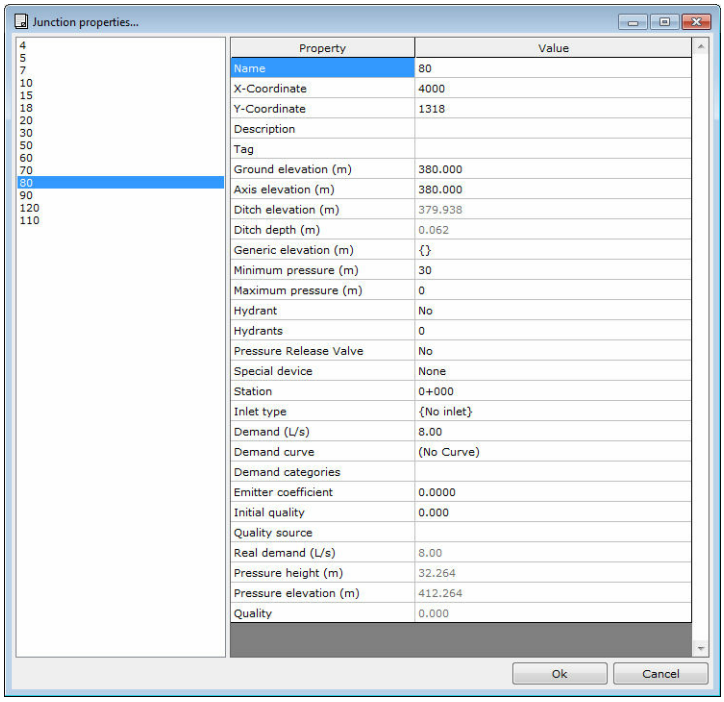
7.3 Properties

7.3.1 Node

With this option, you can view and modify the properties of node objects. You can also view some results regarding nodes; these are read-only fields.

To view and modify the properties of node objects:

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



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

Properties

- **Name**: enter the name of the node, for example N1. Two or more nodes may share the same name, but this is not recommended since there will be confusion in the results.
- **X coordinate**: enter the X coordinate of the node.
- **Y coordinate**: enter the Y coordinate of the node.
- **Description**: enter the description of the node, for example, "central node, no hydrant, no emitter".
- **Tag**: enter a tag for the node. This appears neither in the input data nor the results.
- **Ground elevation (ft or m)**: enter the ground elevation in ft or m.
- **Axis elevation (ft or m)**: enter the axis elevation in ft or m.
- **Ditch elevation (ft or m):** enter the ground elevation of the node.
- **Generic elevation (ft or m):** enter a generic elevation of the node. This elevation appears in the profile drawing only.
- **Minimum pressure (ft or m)**: enter the minimum pressure at the node. This is used during optimization.
- **Hydrant**: if set to YES then a hydrant sign is drawn. This property does not affect calculations. On the contrary, in case of fire scenarios, the demand must include

the additional flow rate.

- **Hydrants**: select the number of hydrants for this node. This property is used to evaluate correctly the flow based on Clement's formula in case of irrigation systems.
- **Pressure release valve**: if set to YES then a pressure release valve sign is drawn. This property does not affect calculations except for the water hammer analysis.
- **Special device**: may be set to NONE, vacuum valve or air release valve depending whether the location of the node coincides with a special device or not.
- **Station:** click the ellipsis button to enter the station of the node.
- **Inlet type:** select the inlet type for the junction from the drop-down list. The inlet types are entered as manhole specifications.
- **Demand (selected flow units)**: enter the demand (outflow) from the node in the selected flow units. If this is not known but consumption data are known, the use of demand categories is recommended.
- **Demand curve**: it is a curve that shows the fluctuation of demand with respect to time. If the demand is constant, set this property to (no curve). This property is used in extended period analyses.
- **Demand categories**: if consumer data are known, you can enter the consumer population instead of total demand. You can also combine outflow from direct demand and from demand categories.
- **Emitter coefficient**: for irrigation networks, you can enter emitter coefficients that describe the variation of flow with respect to pressure height at the node. You must have entered additional hydraulic data.
- **Initial quality**: enter the initial value of the qualitative index at the specific node. Additional quality data are required. The value and unit system of this property vary, depending on the selected quality check.
- **Quality source**: the quality source can be a constant concentration, additional mass in water inflow (for example, chlorinating), additional concentration in the water inflow or constant outflow concentration. The mass or concentration at the node is required. Also, a curve that shows the variation of mass or concentration with respect to time is required. This property is used in extended period analyses.

Results

Some basic results regarding the selected objects are also displayed in the object properties form. These fields are read-only and refer to the time instant that is displayed in the bottom right corner of the main form. These are:

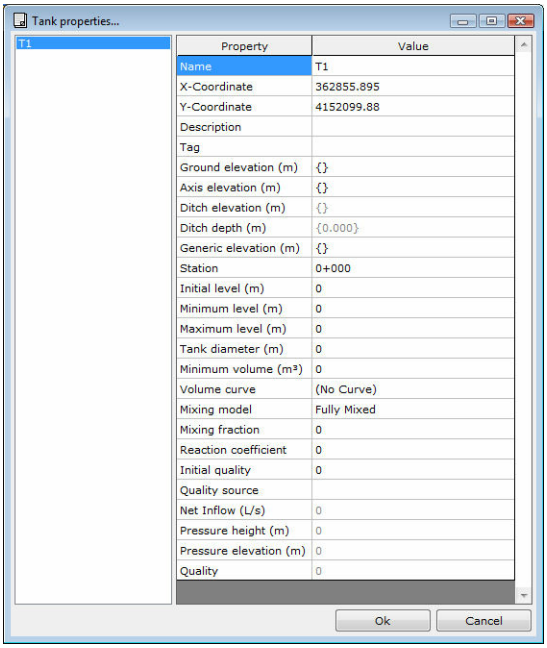
- **Real demand (flow units)**: this is the same as the demand in water networks, but it is different in irrigation networks because the actual outflow from the node is depended on the pressure height. Again, these values will differ only in case emitter coefficients are used.
- **Pressure elevation (ft or m)**: it is equal to the sum of the bottom elevation and the pressure height. If the bottom elevation is set to zero, it is equal to the pressure height.
- **Pressure height (ft or m)**: the pressure height at the node in ft or m.
- **Quality**: the final quality at the node.

7.3.2 Tank

With this option, you can view and modify the properties of tank objects. You can also view some results regarding tanks; these are read-only fields.

To view and modify the properties of tank objects:

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



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

Properties

- **Name**: enter the name of the tank, for example T1. Two or more tanks may share the same name, but this is not recommended since there will be confusion in the results.
- **X coordinate**: enter the X coordinate of the tank.
- **Y coordinate**: enter the Y coordinate of the tank.
- **Description**: enter the description of the tank.
- **Tag**: enter a tag for the tank. This appears neither in the input data nor the results.
- **Ground elevation (ft or m)**: enter the ground elevation of the tank base in ft or m.

- **Axis elevation (ft or m)**: enter the axis elevation in ft or m.
- **Ditch elevation (ft or m):** enter the ground elevation of the node.
- **Generic elevation (ft or m):** enter a generic elevation of the node. This elevation appears in the profile drawing only.
- **Initial level (ft or m)**: enter the initial level of water in the tank at the beginning of the simulation. This is calculated with the ground elevation as reference elevation.
- **Station:** click the ellipsis button to enter the station of the node.
- **Minimum level (ft or m)**: enter the minimum level of water in the tank in ft or m. If there is more demand for water, then the tank is considered empty and it is disconnected from the network.
- **Maximum level (ft or m)**: enter the maximum level of water in the tank in ft or m. If more water is available, it is ignored due to spillover.
- **Tank diameter (ft or m)**: if the tank is circular, enter the tank diameter in ft or m. You can also use this property for other prismatic tanks, if you calculate and enter the equivalent diameter.
- **Minimum volume (ft^3 or m^3).** alternatively to minimum level, you can enter the minimum volume in ft^3 or m^3. For non-circular tanks, it is easier to enter the minimum volume instead of the minimum level. If you use the minimum volume, you can safely ignore the minimum level.
- **Volume curve**: select the volume - height curve that describes the variation of volume with water level. In this way, any tank, whether circular or not, can be used.
- **Mixing model**: select one of fully mixed, 2 compartments, FIFO (first in, first out) or LIFO (Last in, first out). The mixing model is used for the calculation of the lingering time of chemicals in the tank. It is used only in quality analyses.
- **Mixing fraction**: if the 2 compartments mixing model is used, this field is the fraction of the whole tank volume that consists the input-output system of the tank.
- **Reaction coefficient**: it refers to the increase or decrease of the concentration of a chemical, in 1/day. Positive value means increase and vice versa. If a global reaction coefficient is defined in the reactions form, this value may be left empty. If this field is non-zero, it overrides the global reaction coefficient for the specific tank.
- **Initial quality**: enter the initial value of the qualitative index at the specific tank. Additional quality data are required. The value and unit system of this property vary, depending on the selected quality check.
- **Quality source**: the quality source can be a constant concentration, additional mass in water inflow (for example, chlorinating), additional concentration in the water inflow or constant outflow concentration. The mass or concentration at the tank is required. Also, a curve that shows the variation of mass or concentration with respect to time is required. This property is used in extended period analyses.

Results

Some basic results regarding the selected objects are also displayed in the object properties form. These fields are read-only and refer to the time instant that is displayed in the bottom right corner of the main form. These are:

- **Net inflow (flow units)**: it refers to the net inflow of water, in the selected flow units. This is positive if water flows into the tank and vice-versa.

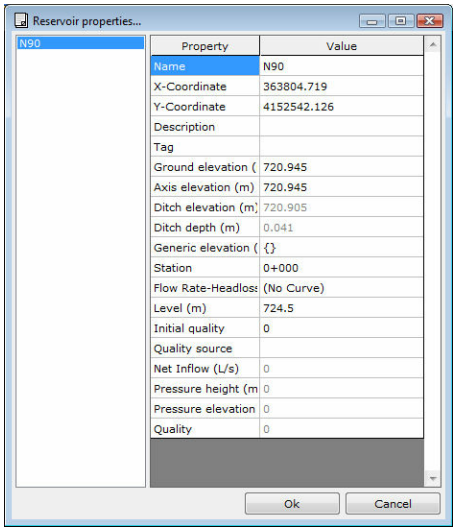
- **Pressure elevation (ft or m)**: it is equal to the sum of the bottom elevation and the pressure height. If the bottom elevation is set to zero, it is equal to the pressure height.
- **Pressure height (ft or m)**: the pressure height at the node in ft or m.
- **Quality**: the final quality at the tank.

7.3.3 Reservoir

With this option, you can view and modify the properties of reservoir objects. You can also view some results regarding reservoirs; these are read-only fields.

To view and modify the properties of reservoir objects:

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



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

Properties

- **Name**: enter the name of the reservoir, for example P1. Two or more reservoirs may share the same name, but this is not recommended since there will be confusion in the results.
- **X coordinate**: enter the X coordinate of the reservoir.
- **Y coordinate**: enter the Y coordinate of the reservoir.
- **Description**: enter the description of the reservoir.
- **Tag**: enter a tag for the reservoir. This appears neither in the input data nor the results.

- **Ground elevation (ft or m)**: enter the ground elevation of the reservoir in ft or m.
- **Axis elevation (ft or m)**: enter the axis elevation in ft or m.
- **Ditch elevation (ft or m):** enter the ground elevation of the node.
- **Generic elevation (ft or m):** enter a generic elevation of the node. This elevation appears in the profile drawing only.
- **Station:** click the ellipsis button to enter the station of the node.
- **Volume curve**: select the flow rate - headloss curve that describes the variation of the level of the reservoir with time. This can be used in case the reservoir is part of a network with varying pressure.
- **Level (ft or m)**: enter the absolute elevation of the free surface of the reservoir. This is assumed to be constant, except in the case a volume curve is used.
- **Initial quality**: enter the initial value of the qualitative index at the specific reservoir. Additional quality data are required. The value and unit system of this property vary, depending on the selected quality check.
- **Quality source**: the quality source can be a constant concentration, additional mass in water inflow (for example, chlorinating), additional concentration in the water inflow or constant outflow concentration. The mass or concentration at the reservoir is required. Also, a curve that shows the variation of mass or concentration with respect to time is required. This property is used in extended period analyses.

Results

Some basic results regarding the selected objects are also displayed in the object properties form. These fields are read-only and refer to the time instant that is displayed in the bottom right corner of the main form. These are:

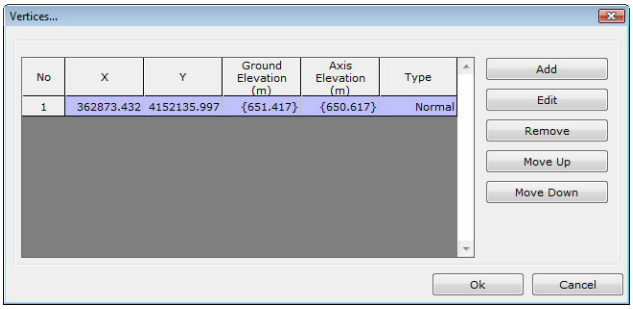
- **Net inflow (flow units)**: it refers to the net inflow of water, in the selected flow units. This is positive if water flows into the reservoir and vice-versa.
- **Pressure elevation (ft or m)**: it is equal to the level of the reservoir, unless a volume curve is used.
- **Pressure height (ft or m)**: this is always zero, as the reservoir has a free surface.
- **Quality**: the final quality at the reservoir.

7.3.4 Link internal vertices

All links may include internal vertices. These are used to describe a more complex plan view, so that calculation of quantities and profile drawings are more precise. There is no restriction in the number of intermediate vertices.

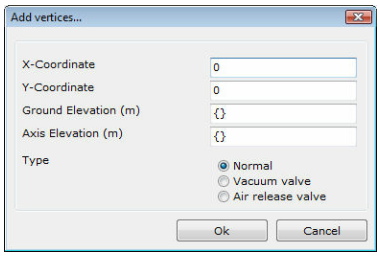
To manage the internal vertices, you can use the corresponding buttons of the toolbar or the internal vertices form:

1. Select the properties form of a link (pipe, pump or valve).
2. Double click on the **Internal vertices** property.
3. Click the ellipsis button.
4. Make the appropriate changes.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.



To add a new internal vertex:

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



2. Make the appropriate selections as described below.
3. Enter the **X-coordinate**, the **Y-coordinate**, the **ground elevation** (in ft or m), the **axis elevation** (in ft or m) and optionally select the type of the internal vertex. Leaving either elevation field empty activates auto-filling. If the specified link belongs to a profile in which upstream and downstream ground elevation data exist, the program calculates the elevation based on linear interpolation. In this case, the result appears within curly braces "{}".
4. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To edit an existing internal vertex:

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

To delete an existing internal vertex:

1. Select the internal vertex from the list on the left.
2. Press **Remove**. The snow pack is deleted from the list.

To move an existing internal vertex upwards in the list:

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

To move an existing internal vertex downwards in the list:

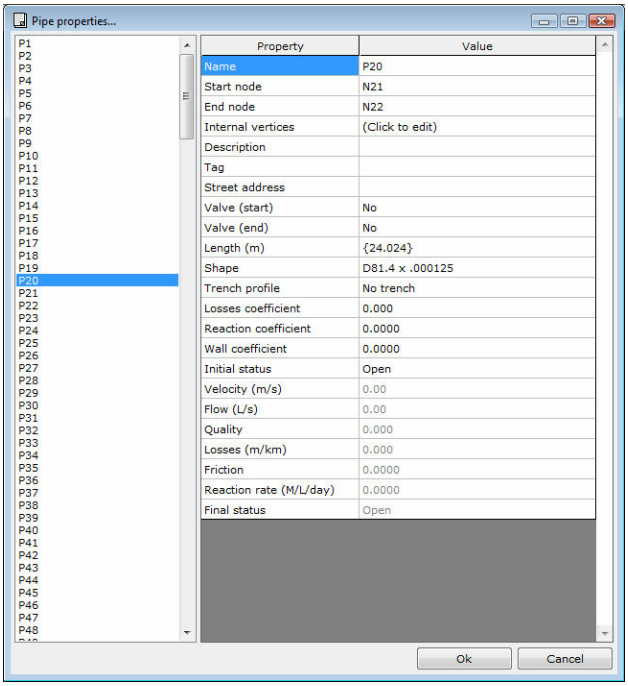
1. Select the internal vertex from the list on the left.
2. Press **Move Down**.
3. The internal vertex is moved one place downwards.

7.3.5 Pipe

With this option, you can view and modify the properties of pipe objects. You can also view some results regarding pipes; these are read-only fields.

To view and modify the properties of pipe objects:

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



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

Properties

- **Name**: enter the name of the pipe, for example P1. Two or more pipes may share the same name, but this is not recommended since there will be confusion in the results.
- **Start node**: select the start point object.
- **End node**: enter the end point object.

- **Internal vertices**: click the ellipsis button to modify the internal vertices of the conduit.
- **Description**: enter the description of the pipe.
- **Tag**: enter a tag for the pipe. This appears neither in the input data nor the results.
- **Street address**: optionally, enter the street name of the link.
- **Valve**: specify if the upstream end of the link is a check valve or not. This is used by the water hammer solver and appears on the plan view.
- **Length (ft or m):** leave this field empty to activate auto-fill and let the program calculate the length from the plan view. In this case, the result appears within curly braces "{}". Alternatively, specify an explicit value that will be used.
- **Shape**: select the appropriate conduit shape. If none is selected, the program uses the default shape or the shape that was explicitly defined upstream in a profile that includes the conduit. In the latter case, the name of the conduit shape appears within curly braces.
- **Pipe specification fixed**: set this to yes if you want the selected shape not to be altered during optimization.
- **Trench profile:** select the trench profile from the drop-down list.
- **Friction coefficient**: enter the friction coefficient of the pipe. The unit system is depended on the selected friction formula. In any case, a custom form appears that helps you enter the correct friction coefficient either directly or by invoking the corresponding database.
- **Losses coefficient**: optionally, enter the local losses coefficient. For example, this can be used in the case of pipe curvature.
- **Reaction coefficient**: it refers to the increase or decrease of the concentration of a chemical, in 1/day. Positive value means increase and vice versa. If a global reaction coefficient is defined in the reactions form, this value may be left empty. If this field is non-zero, it overrides the global reaction coefficient for the specific pipe.
- **Wall coefficient**: it refers to the increase or decrease of the concentration of a chemical, in 1/day. Positive value means increase and vice versa. If a global wall coefficient is defined in the reactions form, this value may be left empty. If this field is non-zero, it overrides the global wall coefficient for the specific pipe. This value is used for the reactions that take place on the pipe walls.
- **Initial status**: a pipe may be closed (non-existent), open (normal) or check valve. The check valve will enable the pipe when flow occurs from the start node to the end node, and vice versa.

Results

Some basic results regarding the selected objects are also displayed in the object properties form. These fields are read-only and refer to the time instant that is displayed in the bottom right corner of the main form. These are:

- **Length (ft or m)**: this is the length that will be used in the calculations. It coincides with the fixed length in case the latter is non-zero. In all other cases, the length is calculated from node coordinates, taking into account intermediate pipe vertices and elevations.
- **Velocity (ft/s or m/s)**: the flow velocity in ft/s or m/s.
- **Flow rate (flow units)**: the mean flow rate in the selected flow units.
- **Quality**: the final quality in the pipe.
- **Reaction rate (M/L/day or M/ft^3/day)**: the reaction rate both internally and

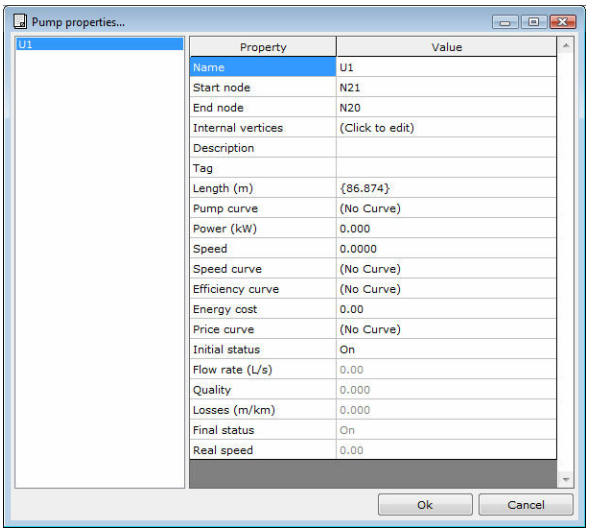
on the pipe walls.
- **Losses (ft/1000ft or m/km)**: the friction losses in ft/1000ft or m/km, taking into account local losses.
- **Friction**: the final friction coefficient. In case of the Darcy-Weisbach formula, this field is equal to f.
- **Final status**: the final status of the pipe (open or closed). This field is meaningful when the initial status of the pipe is set to check valve. In all other cases, this field coincides with the initial status.

7.3.6 Pump

With this option, you can view and modify the properties of pump objects. You can also view some results regarding pumps; these are read-only fields.

To view and modify the properties of pump objects:

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



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

Properties

- **Name**: enter the name of the pump, for example P1. Two or more pumps may share the same name, but this is not recommended since there will be confusion in the results.
- **Start node**: select the start point object.
- **End node**: enter the end point object.

- **Internal vertices**: click the ellipsis button to modify the internal vertices of the pump.
- **Description**: enter the description of the pump.
- **Tag**: enter a tag for the pump. This appears neither in the input data nor the results.
- **Length (ft or m):** leave this field empty to activate auto-fill and let the program calculate the length from the plan view. In this case, the result appears within curly braces "{}". Alternatively, specify an explicit value that will be used.
- **Pump curve**: select a pump curve that describes the relation of the flow rate and the pressure height.
- **Power (kW)**: for constant power pumps, enter the power in kW. This value is independent of the flow rate. This field is ignored if a pump curve is selected.
- **Speed** : the speed coefficient of the pump. This value shows the relation of the pump speed with respect to normal operation. For example, a speed coefficient of 1.5 shows that a pump has 50% more rpm with respect to normal operation.
- **Speed curve**: select a speed curve that describes the operation of the pump with respect to time. If at some time instants the coefficients are zero, the pump does not operate.
- **Efficiency curve**: select an efficiency curve that describes the relation between the efficiency of the pump and the flow rate. If you use a global efficiency factor, you do not need to select a curve.
- **Energy cost**: enter the mean or nominal cost of energy in currency units per kWh. This is used only in case of operational cost analyses. If you use a global energy cost then you do not need to enter this value.
- **Price curve**: select a price curve that shows the variation of energy cost with respect to time. This is used only in case of operational cost analyses, when energy cost varies, as in the case of pump operation during the night as compared to operation in peak hours.
- **Initial status**: a pump may be closed (non-existent) or open (normal).

Results

Some basic results regarding the selected objects are also displayed in the object properties form. These fields are read-only and refer to the time instant that is displayed in the bottom right corner of the main form. These are:

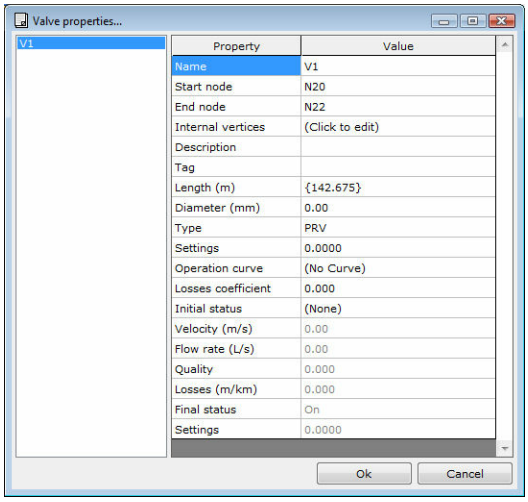
- **Flow rate (flow units)**: the mean flow rate in the selected flow units.
- **Quality**: the final quality in the pump.
- **Losses (ft/1000ft or m/km)**: the friction losses in ft/1000ft or m/km.
- **Final status**: the final status of the pump (open or closed). This depends on the operational parameters (flow rate, pressure height) and its specifications.

7.3.7 Valve

With this option, you can view and modify the properties of valve objects. You can also view some results regarding valves; these are read-only fields.

To view and modify the properties of valve objects:

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



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

Properties

- **Name**: enter the name of the valve, for example V1. Two or more valves may share the same name, but this is not recommended since there will be confusion in the results.
- **Start node**: select the start point object.
- **End node**: enter the end point object.
- **Internal vertices**: click the ellipsis button to modify the internal vertices of the valve.
- **Description**: enter the description of the valve.
- **Tag**: enter a tag for the valve. This appears neither in the input data nor the results.
- **Length (ft or m):** leave this field empty to activate auto-fill and let the program calculate the length from the plan view. In this case, the result appears within curly braces "{}". Alternatively, specify an explicit value that will be used.
- **Diameter (mm or in)**: enter the diameter of the valve in mm or in. This field is required.
- **Valve type**: Select the type of the valve. This can be one of the following:
  - PRV: Pressure reducing valve, setting value represents pressure height in m.
  - PSV: Pressure sustaining valve, setting value represents pressure height in m.
  - PBV: Pressure breaker valve, setting value represents pressure height in m. This is pressure height difference because of the valve.
  - FCV: Flow control valve, setting value represents flow rate in L/s. This is the maximum flow rate that can pass through the valve.
  - TCV: Throttle control valve, setting value represents friction coefficient. This valve simulates a partly open valve by modifying the local losses coefficient.

Usually, the relation between the local losses coefficient and the opening of the valve is provided by the manufacturer.
    - GPV: General purpose valve. This valve uses a flow rate - headloss curve that describes the relation between the flow rate and the headloss. The setting field is ignored and the operation curve is taken into account.
- **Setting**: enter a value for pressure, flow rate, friction coefficient depending on the valve type (see above).
- **Operation curve**: for GPV valves only, select a flow rate - headloss curve that describes the relation between the flow rate and the headloss.
- **Losses coefficient**: enter the losses coefficient. This is zero if the valve is fully open. Other values depend on the flow rate and are provided by the manufacturer.
- **Initial status**: a valve may be closed (non-existent), open (normal) or it may have no initial status, in which case its behavior is governed by the aforementioned properties.

Results

Some basic results regarding the selected objects are also displayed in the object properties form. These fields are read-only and refer to the time instant that is displayed in the bottom right corner of the main form. These are:

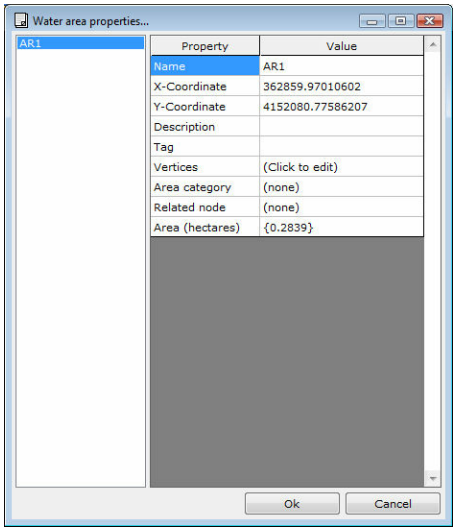
- **Velocity (ft/s or m/s)**: the flow velocity in ft/s or m/s.
- **Flow rate (L/s)**: the mean flow rate in L/s.
- **Quality**: the final quality in the valve.
- **Losses (m/km or ft/1000ft)**: the friction losses in m/km or ft/1000ft, taking into account local losses.
- **Final status**: the final status of the valve (open or closed).

7.3.8 Area

With this option, you can view and modify the properties of area objects.

To view and modify the properties of area objects:

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



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

Properties

- **Name**: enter the name of the area, for example A1. Two or more areas may share the same name, but this is not recommended since there will be confusion in the results.
- **X-Coordindate**: the X-Coordinate of the polygon's centroid.
- **Y-Coordindate**: the Y-Coordinate of the polygon's centroid.
- **Description**: enter the description of the valve.
- **Tag**: enter a tag for the area. This appears neither in the input data nor the results.
- **Vertices**: click the ellipsis button to modify the internal vertices of the area.
- **Area category**: select the area category that will be used to compute the water demands from the particular area.
- **Related node**: the node that will be used to supply the computed water depand of the particular area.
- **Area**: if you wish to override the automatically computed area, you can type-in the desired area in this text field in hectares or acres.

7.3.9 Profile

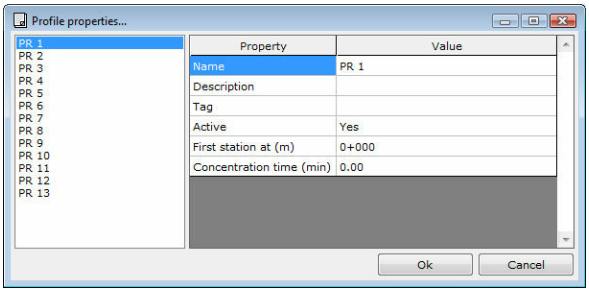
Profiles are not actual objects but rather a series of references to alternating nodes and links. Data input becomes very easy using profiles since:

- data are input in tabular form,

- the elevations are displayed in a visual and comprehensive way,
- the creation of profile DXF drawings is easy,
- a network may be defined without plan view data.

To view and modify the properties of profiles:

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



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

Properties

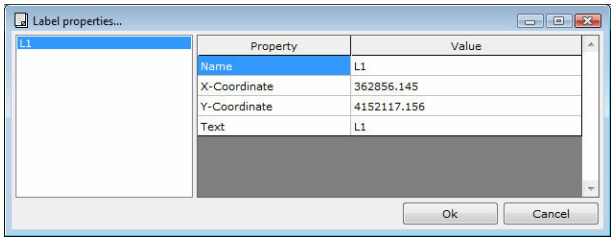
- **Name**: enter the name of the profile. Two or more profiles may share the same name, but this is not recommended since there will be confusion in the results.
- **Description**: enter the description of the profile.
- **Tag**: enter a tag for the profile. This appears neither in the input data nor the results.
- **Active**: select whether the profile will appear in the profile drawing. This setting can be changes using the profile drawing form.
- **First station**: enter the first station of the profile. This option is used with the semi-automatic data input of profile stations.
- **Concentration time (min)**: this property is NOT used.

7.3.10 Label

Labels are optional text objects that can be used to enrich the plan view with important data.

To view and modify the properties of labels:

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



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

Properties

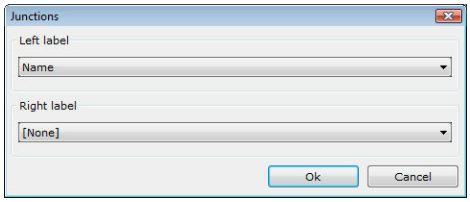
- **Name**: enter the name of the label (not the text).
- **X-coordinate**: enter the X-coordinate of the label.
- **Y-coordinate**: enter the Y-coordinate of the label.
- **Text**: enter the text that will be displayed.

7.4 Labels

With this option, you can select which object properties will be displayed. Up to two properties can be displayed simultaneously, one on the left label and one on the right.

To select which object properties will be displayed:

1. Select **Labels** from the **Objects** menu.
2. Select the object type from the **Labels** menu. The following form appears:



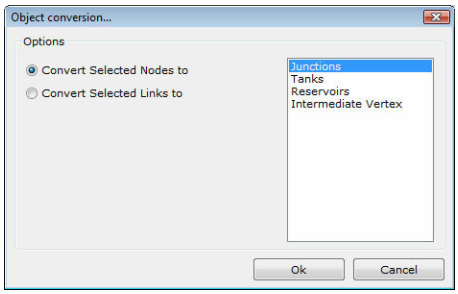
3. Select the property that will be displayed on the left.
4. Select the property that will be displayed on the right.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

7.5 Object conversion

With this option, you can convert objects from one type to another.

To convert objects from one type to another:

1. Select (in plan view) the objects you wish to convert.
2. Select **Object conversion** from the **Objects** menu. The following form appears:



3. Select if you wish to convert nodes (point objects) or links (linear objects).
4. Select the target object type from the list on the right.
5. Select **Ok** to save the changes and close the dialog box. Select **Cancel** to close the dialog box without applying any changes.

NOTE: Non-common properties are lost during object conversion.

NOTE: To convert a vertex back to a junction, see the convert vertex to junction function.

7.6 Add vertex

With this option, you can add an intermediate vertex to an existing link: pipe, pump, valve. This procedure is described also in the section of link internal vertices.

To add an intermediate vertex to an existing link:

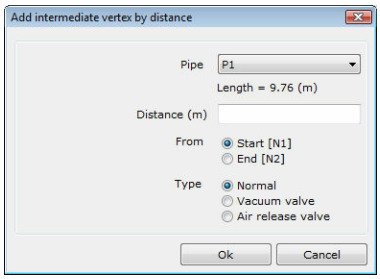
1. Select **Add vertex** from the **Objects** menu.
2. Click on the link to add a vertex.

7.7 Add vertex by distance

With this option, you add a vertex on a conduit, by specifying a distance from either end of the conduit. For example, if the conduit has a length equal to 20 m and you need to add a vertex that be located 5 m from its end (station 0+015) then select its ending node and enter 5 or its starting node and enter 15.

To add a vertex by distance:

1. Select **Add Vertex By Distance** from the **Objects** menu.
2. Select the **conduit** from the drop-down list. Its length appears on the form.



3. Enter the desired distance from either starting or ending node.
4. Select whether this distance is measured from the starting or ending node.
5. Select **Ok** to add a vertex on the specified conduit. Select **Cancel** to close the dialog box without applying any changes.

7.8 Delete vertex

With this option, you can delete an intermediate vertex of an existing link: pipe, pump , valve. This procedure is also described in the section of link internal vertices.

To delete an intermediate vertex of an existing link:

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

7.9 Stretch vertex

With this option, you can move an intermediate vertex of an existing link: pipe, pump, valve. This procedure is also described in the section of link internal vertices.

To delete an intermediate vertex to an existing link:

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

7.10 Convert vertex to junction

To convert one or more vertices back to junctions, one must you this function instead of the generic conversion form.

To convert one vertex to a junction:

1. From the **Objects** menu, click on **Convert Vertex To Junction**.
2. On the map, click on the vertex you wish to convert back to a junction.
3. The vertex is converted.

To convert multiple vertices to junctions:

1. From the **Objects** menu, click on **Convert Vertex To Junction**.
2. Hold **CTRL** down and click on the map the vertex you wish to convert back to a junction.
3. Repeat step 2 until all vertices are converted.

7.11 Swap link ends

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

To swap the ends of the selected links:

1. Select the links in the drawing.

2. Select **Swap link ends** from the **Objects** menu. The ends are swapped.

Chapter VIII

8 Profiles

8.1 Profiles menu

With this menu, you can perform various operations regarding profiles and network design. These are available in case at least one profile has been defined. In the **Profiles** menu you can select one of the following options:

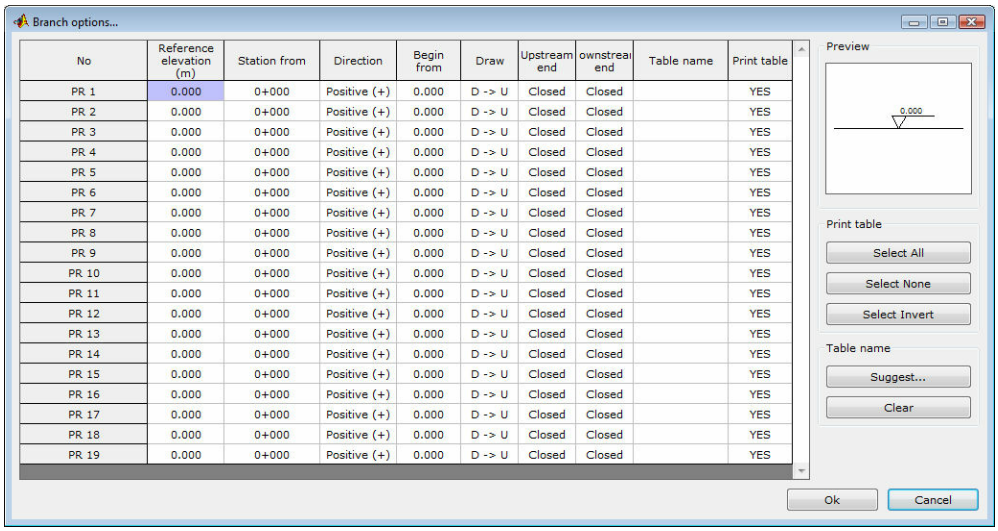
- Profile options
- Elevation calculations
- Stations
- Street addresses
- Vertical street addresses
- Other junctions
- Special devices

8.2 Profile options

With this option, you can modify the way the branches are displayed.

To modify the way the branches are displayed:

1. Select **Profile options** from the **Profiles** menu. The following form will appear:



No	Reference elevation (m)	Station from	Direction	Begin from	Draw	Upstream end	ownstreai end	Table name	Print table
PR 1	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 2	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 3	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 4	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 5	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 6	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 7	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 8	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 9	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 10	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 11	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 12	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 13	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 14	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 15	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 16	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 17	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 18	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES
PR 19	0.000	0+000	Positive (+)	0.000	D -> U	Closed	Closed		YES

2. Make the appropriate changes as described below. By clicking on each column, a sketch of the setting is displayed on the right.

- **Reference elevation**: Enter the reference elevation in m. However, it is recommended that you leave this field empty, in which case a label "AUTO" will appear and the reference elevation will be automatically calculated by the program.
- **Station from**: enter the initial station if this is not zero. This affects only the drawing of the profiles; it is not related with the stations of the data table.
- **Direction**: enter one of **positive**, **negative** if you want the stations to be increasing or decreasing, respectively.

- **Begin from:** enter the distance from the first station in m. Usually this value will coincide the **Station from** field, but it will be displayed in a different row in the profile.
- **Draw:** select one of **D->U, U->D** if you want the profile to be drawn downstream to upstream or vice versa, respectively. This affects the profile drawing; in the data table of the main form and the profile sketch, the branch is drawn upstream to downstream.
- **Upstream end**: select one of **open, closed** if you want the end to be drawn open or closed, respectively, in the profile drawing.
- **Downstream end:** select one of **open, closed** if you want the end to be drawn open or closed, respectively, in the profile drawing.
- **Table name**: enter the title of the branch. This is optional and it will be printed above the corresponding table in the profile drawing. When editing this value, the buttons **Suggest** and **Clear** of the **Title** frame become enabled. The former sets the name equal to the title of the branch; the latter clears all table names.
- **Print table**: select one of **Yes, No** if you want a table with row descriptions to be included for the specific branch in the profile drawing. When editing this value, the buttons **Select all, Select None** and **Select Invert** of the **Table** frame become enabled. With these, you can set **Yes** or **No** to all branches with one click.

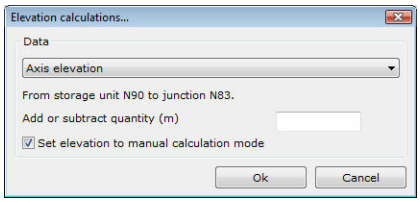
3. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

8.3 Elevation calculations

With this option, you can add or subtract a value to or from a specified elevation.

To add or subtract a value to or from a specified elevation:

1. Select the branch from the list of the main form.
2. Select two or more rows in the data table (when viewing the profile) by clicking and dragging the mouse. The first and last row signify the first and last station, respectively, of the part that the calculations will be applied.
3. Select **Elevation calculations** from the **Profiles** menu. The following form will appear:



4. The form displays the first and last station.
5. Select the type of elevation that will be modified from the drop-down list. The available options are:

- **Ground elevation**
- **Upstream bottom elevation**
- **Downstream bottom elevation**
- **Trench elevation**
- **Custom elevation**
- **Manhole bottom elevation**

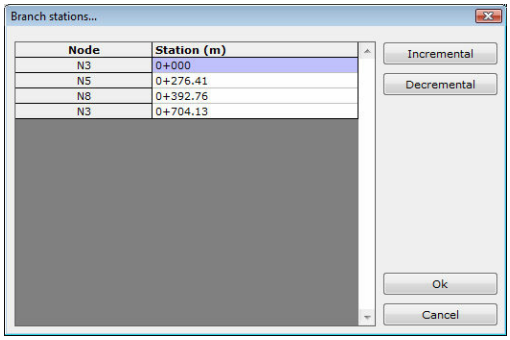
6. Enter the value to be added or subtracted by typing into the corresponding text box.
7. If some values are calculated automatically by the program using linear interpolation then, after this procedure, these will be recalculated, thus canceling the effect of the above modifications. Check **Convert automatically computed data** to fix the new values and prevent the program from recalculating them.
8. Select **Ok** to proceed with the operation and close the dialog box. Select **Cancel** to close the dialog box and cancel the operation.

8.4 Stations

With this option, you can enter station data for the selected profile. This option is very helpful if you want to enter the data of all stations, especially if these are uniformly spaced (e.g. every 20m). Alternatively, the station data can be input using the data table of the main form.

To enter station data for the selected profile:

1. Select the profile from the list.
2. Select **Stations** from the **Data** menu. The following form appears:



3. Enter the station data.
4. You can optionally use the following quick buttons:

- **Incremental**: beginning with the first station, add the distance between two stations.
- **Decremental**: beginning with the first station, subtract the distance between two stations.

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

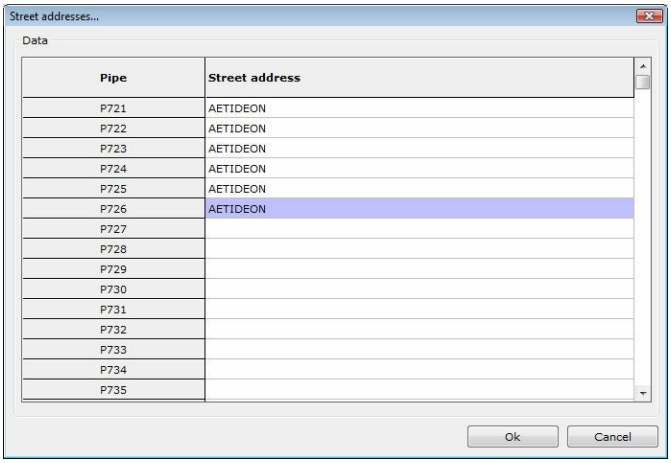
NOTE: The first station is defined in the profile options form.

8.5 Street addresses

With this option, you can add or modify street addresses to the profile drawing. This data does not affect calculations.

To add or modify street addresses:

1. Select the profile from the list of the main form.
2. Select **Street addresses** from the **Profiles** menu. The following form will appear:



3. Enter the street addresses by typing directly onto the grid.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

For your convenience, the following options are available from the **Edit** menu:

- **Select all:** All cells are selected.
- **Cut:** The data of the selected cells are deleted from the grid and copied to the clipboard.
- **Copy:** The data of the selected cells are copied to the clipboard.
- **Paste:** The data of the clipboard is pasted into the grid.
- **Clear selection:** The data of the selected cells are deleted.

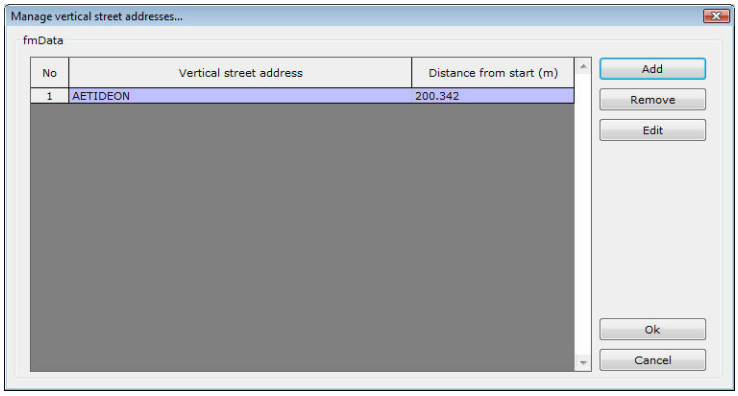
NOTE: If the same street address spans more than one sections, the program will merge the street addresses into one.

8.6 Vertical street addresses

With this option, you can add or modify vertical street addresses to the profile drawing. This data does not affect calculations.

To add or modify vertical street addresses:

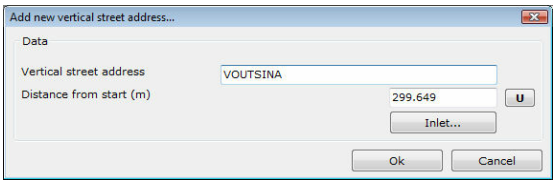
1. Select the profile from the list of the main form.
2. Select **Vertical street addresses** from the **Profiles** menu. The following form will appear:



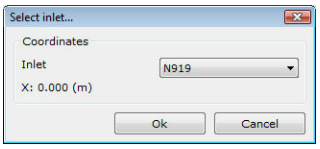
3. Make the appropriate changes.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a vertical street address:

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



2. Enter the **Street address** by typing in the corresponding text box.
3. In the **Coordinates** frame, enter the **distance from start** of the vertical street address. For your convenience, you can select **Inlet**. The following form appears:



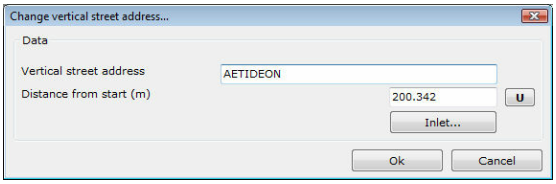
- Select the inlet from the drop-down list.

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

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

To modify an existing vertical street address:

1. Select the vertical street address from the list.
2. Select **Modify**. The following form will appear:



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

To delete an existing vertical street address:

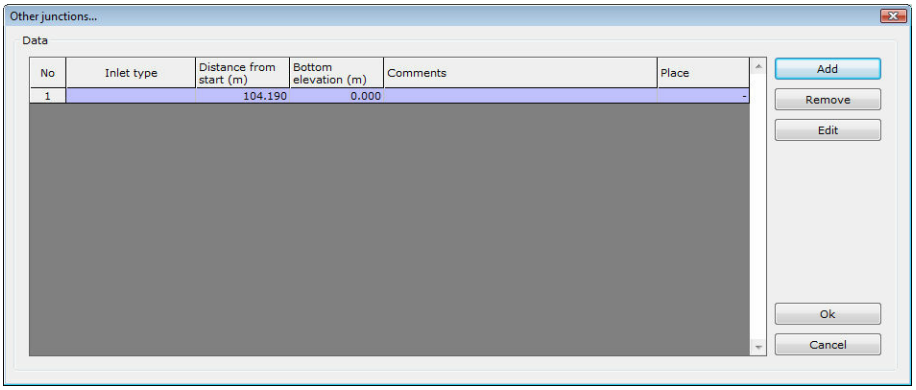
1. Select the vertical street address from the list.
2. Select **Delete**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The vertical street address is deleted.

8.7 Other junctions

With this option, you can add pipe junctions from other networks to the profile drawing and the profile sketch. These do not affect calculations.

To add pipe junctions to the profile drawing and the profile sketch:

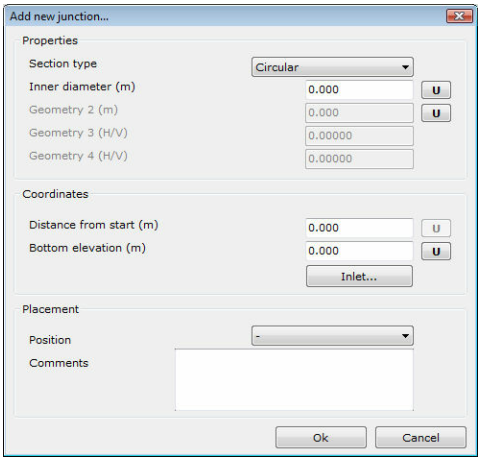
1. Select the profile from the list of the main form.
2. Select **Other junctions** from the **Profiles** menu. The following form will appear:



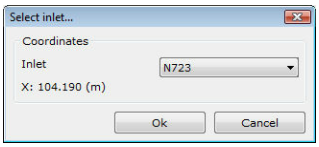
3. Make the appropriate changes.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a pipe junction:

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



2. Select the **Section type** and **Diameter** of the pipe in m.
3. In the **Coordinates** frame, enter the **Distance from start** of the pipe junction and the **Bottom elevation** in m. For your convenience, you can select **Inlet**. The following form appears:



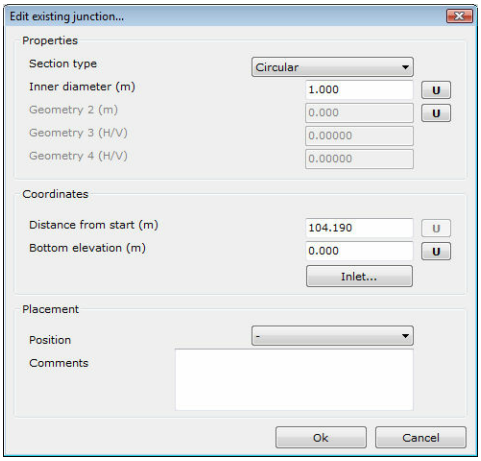
- Select the inlet from the drop-down list.
- Check **Match invert** if you wish to match the invert elevation of the specified inlet.

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

4. In the **Placement** frame, you can select the **position** of the pipe junction i.e. whether the junction comes from the left or right, and you can add comments to the pipe junction. This data is used only for the profile drawings.
5. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To modify an existing junction:

1. Select the junction from the list.
2. Select **Modify**. The following form will appear:



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

To delete an existing junction:

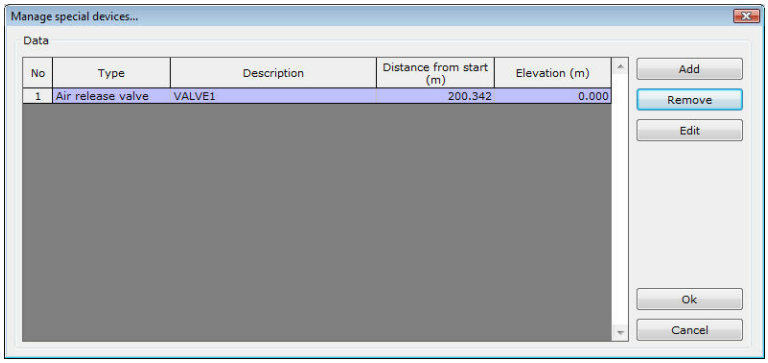
1. Select the junction from the list.
2. Select **Delete**. You will be asked for confirmation only if you have selected to confirm deletions in the General preferences tab.
3. The junction is deleted.

8.8 Special devices

With this option, you can add special devices to the profile drawing and the profile sketch. These do not affect calculations.

To add special devices to the profile drawing and the profile sketch:

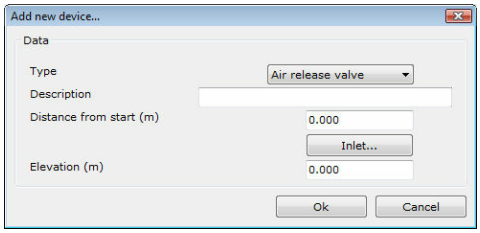
1. Select the profile from the list of the main form.
2. Select **Special devices** from the **Profiles** menu. The following form will appear:



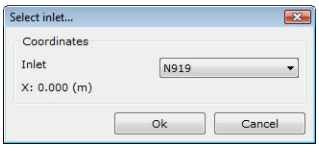
3. Make the appropriate changes.
4. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

To add a special device:

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



2. In the **Properties** frame, select one of **Air release valve, Vacuum valve** and **Valve** as the type of the special device. You can also provide a **Description** of the special device by typing into the corresponding text box.
3. In the **Coordinates** frame, enter the **Distance from start** of the special device and the **Elevation** in m. For your convenience, you can select **Inlet**. The following form appears:



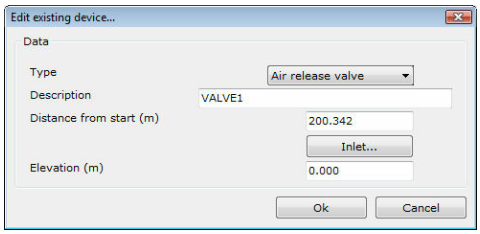
- Select the inlet from the drop-down list.
- Check **Match invert** if you wish to match the invert elevation of the specified inlet.

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

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

To modify an existing special device:

1. Select the special device from the list.
2. Select **Modify**. The following form will appear:



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

To delete an existing special device:

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

Chapter

9 Tools

9.1 Tools menu

With this menu, you can perform advanced operations regarding network's design and integrity design. In the **tools** menu you can select one of the following options:

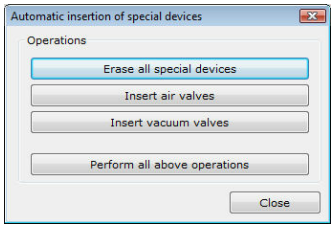
- Automatic insertion of special devices
- Object renaming
- Demand distribution
- Exact total water area
- Water areas from DXF
- Axis elevation initialization
- Axis elevation calculations
- Contours
- Water hammer

9.2 Automatic insertion of special devices

This dialog box allows the user to automatically add air valves and vacuum valves in a network.

To automatically insert special devices:

1. From the **Tools** menu select **Automatic Insertion Of Special Devices**.
2. If there are special devices present, you may wish to erase them first. Click on **Erase all special devices** to remove all existing special devices from the network.
3. To automatically add the required air valves, click the **Insert air valves** button.
4. To automatically add the required vacuum valves, click the **Insert vacuum valves** button.
5. To automatically add the required air and vacuum valves, click the **Perform all above operations** button.
6. Press **Close** when done.



NOTE: There are special network configurations where more than necessary special devices are added or when some necessary special devices are missing. It is the engineer's responsibility to ensure correct network design. This tool is provided as a convenient shortcut only.

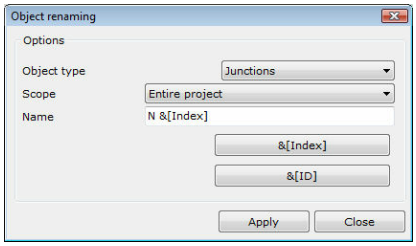
NOTE: This tool is available as a separate product. It is enabled once the "Water Networks Toolpack" product is purchased.

9.3 Object renaming

To conveniently rename objects of a certain type such as junctions, conduits, pumps, etc, you may use the object renaming tool.

To rename objects of a certain type:

1. Select **Object Renaming** from the **Tools** menu.
2. Select the object type from the drop-down list.
3. Select the **scope** from the drop-down list. The scope can either be the entire project or the current selection.
4. Enter the renaming command (see below).
5. Press **Apply** to rename the objects described above or **Cancel** to close the form and ignore all changes.



Renaming command

The renaming command is a string that dictates how the objects will be renamed. This command is consisted of keywords and strings. Keyword vary depending on the object type selected and appear as buttons for your convenience.

Examples:

J&[Index]: will rename all objects as J1, J2, J3, etc.
&[StartNodeName] -> &[EndNodeName]: will rename all pipes as J1 -> J2, J2 -> J3, etc.

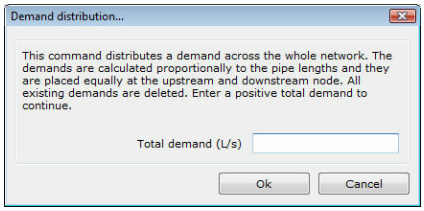
NOTE: This tool is available as a separate product. It is enabled once the "Water Networks Toolpack" product is purchased.

9.4 Demand distribution

This tool can be used to distribute a demand over the entire network, depending on the conduits' lengths. The demand is equally divided on each node's upstream and downstream ends.

To distribute a specified demand over the entire network:

1. Select **Demand Distribution** from the **Tools** menu.
2. Enter the total demand in flow units which will be distributed depending on the length of each conduit.
3. Press **Ok** to calculate and add the demand at each node depending on each conduit length or press **Cancel** to hide the tool.



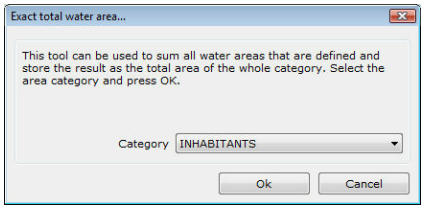
NOTE: This tool is available as a separate product. It is enabled once the "Water Networks Toolpack" product is purchased.

9.5 Exact total water area

This tool calculates the total area of the area objects for a selected area category and automatically transfers the result to the category's data.

To calculate the area of a specified area category:

1. Select **Exact Total Water Area** from the **Tools** menu.
2. Select the area category whose area will be computed.
3. Click **Ok** to compute the total area and transfer the result to the area category. Click **Cancel** to hide the tool.



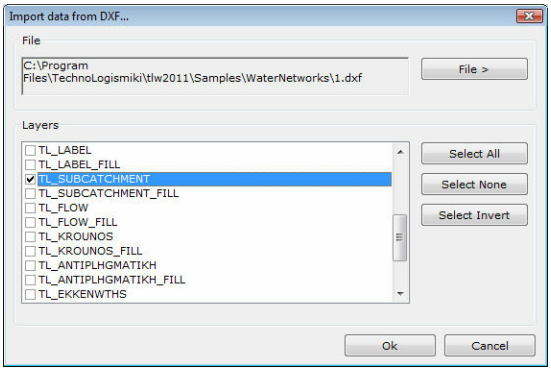
NOTE: This tool is available as a separate product. It is enabled once the "Water Networks Toolpack" product is purchased.

9.6 Water areas from DXF

If you do not wish to specify on screen each area separately via the program's user interface, it is possible to import one or more areas from external dxf files. The areas have to be closed polygons (polylines). This tool, apart from reading and plotting areas from external dxf files, scans for the nearest junction to every area and connects them.

To import water areas from DXF:

1. Select **Water Areas From DXF** from the **Tools** menu.
2. The following form appears, where the input file can be selected.



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

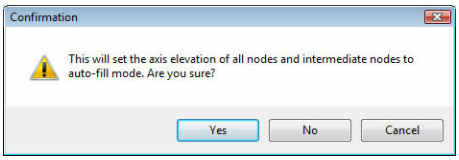
NOTE: This tool is available as a separate product. It is enabled once the "Water Networks Toolpack" product is purchased.

9.7 Axis elevation initialization

Use this tool to remove all axis elevation information and set them to their default values. This will apply to all nodes and intermediate vertices regardless any selection present.

To initialize all axis elevations:

1. From the **Tools** menu select **Axis Elevation Initialization**.
2. Click **Yes** to the following prompt:



NOTE: This tool is available as a separate product. It is enabled once the "Water Networks Toolpack" product is purchased.

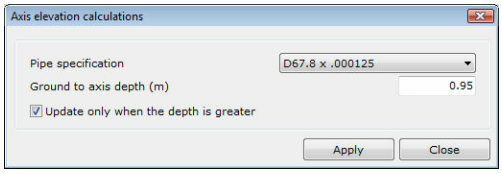
9.8 Axis elevation calculations

Depending on the pipe specification, you may relocate all pipes at once. To do so, the axis elevation calculations tool is provided.

To use this tool:

1. From the **Tools** menu click on **Axis Elevation Calculations**.
2. Select the **pipe specification** from the drop down menu.
3. Enter the desired **ground to axis depth** in m or ft.
4. If you wish this relocation to by applied only in cases where the depth is greater

than the amount entered above, enabled the **Update only when the depth is greater** option.
5. Press **Apply** to relocate all pipes or **Close** to hide the tool.



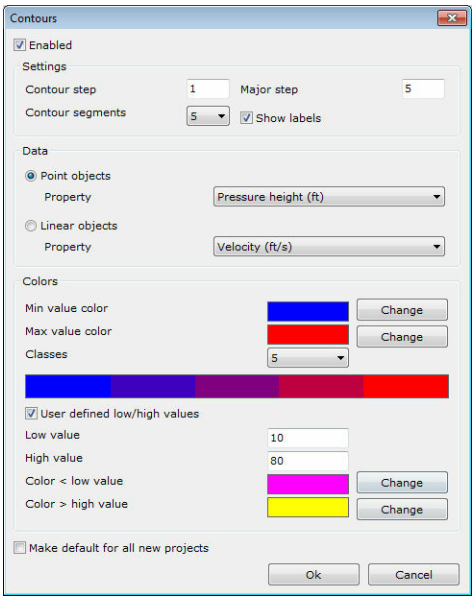
NOTE: This tool is available as a separate product. It is enabled once the "Water Networks Toolpack" product is purchased.

9.9 Contours

Use this tool to create contours for a specific property of point objects.

To create contours for a specific property of point objects:

1. From the **Tools** menu select **Contours**. The following form appears:



2. Check **Enabled** to enable the contours.
3. Select the **Contour step** (e.g. for the ground elevation of nodes, this is expressed in units of length), the step for the **major contours** (which are printed thicker and they are accompanied by their value) and the **Contour segments**, which controls the smoothness of curves.
4. Select **Point objects** and the corresponding property.
5. Select the colors that correspond to the minimum and maximum values by clicking on the corresponding **Change** button. If you want to use custom (user defined) high and low values, check the corresponding field. In this case you need to provide the colors for the values that are higher than the high value or lower than the low value.
6. Select the number of classes for the classification of objects.
7. Select **Ok** to close the dialog box and save changes. Select **Cancel** to close the

dialog box and ignore changes.

NOTES:

1. The settings for the contour coloring are the same (and affect) those of object coloring. To create contours though, you must select **Point objects**.
2. Each time you perform calculations, the contours are evaluated anew. This operation may take a lot of time.
3. The contours are exported, in color, when you perform File > Export > Plan view to DXF.

NOTE: This tool is available as a separate product. It is enabled once the "Sewer Networks Toolpack" product is purchased.

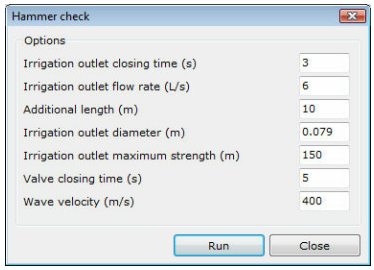
9.10 Water hammer

This tool checks for problems that may appear in the network's operation due to the water hammer phenomenon. Water hammer appears in a non-looped network when a demand stops or a valve closes. To prevent the pipes from breaking, pressure release valves are used.

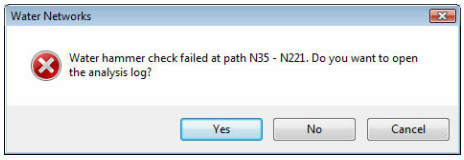
The solver can account for water hammer effects only outside loops. Pumps, reservoirs, tanks and pressure release valves are considered fail safe devices. Water hammer in pumps must be separately examined since the program considers them to be "safe by design".

To solve for water hammer problems:

1. Solve the network.
2. From the **Tools** menu click on **Water Hammer**.
3. Enter the **irrigation outlet closing time**. This could be the time required for a water supply demand to stop or an irrigation sprinkler to stop. The time cannot be zero as no closing can be considered instantaneous. A common approximation is to use a 3 seconds closing interval.
4. Enter the **irrigation outlet flow rate** in flow units. This is the flow rate of a single sprinkler or a single water demand.
5. Enter the **additional length** in m or ft. The additional length represents the length of the pipe that connects the sprinkler or the water consumer with the distribution pipe. This variable varies, however a typical value is 10 meters.
6. Enter the **irrigation outlet diameter** in m or ft, which is the diameter of the pipe mentioned above.
7. Enter the **irrigation outlet maximum strength** in m or ft. This is the maximum instantaneous strength of the aforementioned pipe. If the pipe is 12.5 atm, then a value 12.5*10+20%*12.5*10=150 should be used. 20% is the additional strength of the pipe for instantenous over-pressure, above its nominal strength. This percentage is arbitrary and could be set equal to 0 for maximum safety.
8. Enter the **valve closing time** in seconds. This closing time refers to valves that are used to shutoff the flow completely in a pipe.
9. Enter the **wave velocity** in m/s or ft/s. The wave velocity is the speed that the wave generated by the change in the boundary conditions (i.e. valve closing) travels in the pipe. A value of 400 m/s for PVC is recommended.
10. Click on **Run** to start the **Water Hammer** solver or Close to hide the form.



If the water hammer solver detects a problem then a message like the following will pop-up:



To correct this problem, the geometry of the network could be altered, i.e. increase the diameter or a pressure release valve could be added upstream of the node where the problem appears. A new run is required until no error messages are generated.

In any case a complete analysis report is generated and can be exported directly to Microsoft Word or the Printer (print preview).

NOTE: This tool is available as a separate product. It is enabled once the "Water Networks Toolpack" product is purchased.

Chapter

10 Irrigation

10.1 Irrigation menu

With this menu, the user obtains access to advanced tools for the design and optimization of irrigation networks, or in general, of branched networks (without loops) under pressure. In the **Irrigation** menu, the following options are available:

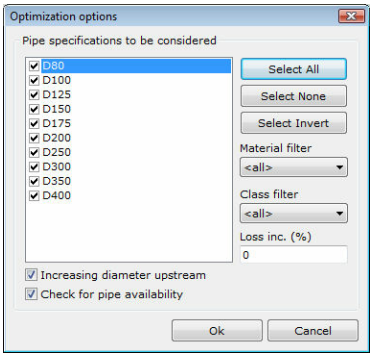
- Optimization options
- Optimization
- Characteristic curve options
- Characteristic curve
- Flow based on Clement's First Formula
- Clear Clement's Flow
- Check optimization constraints

10.2 Optimization options

This dialog box allows the user to set the optimization options for irrigation networks.

To set the optimization options for irrigation networks:

1. From the **Irrigation** menu, select **Optimization options**. The following form appears:



2. Select one or more pipe specifications which will be used for optimization. The **material** and **class filter** may assist you by limiting the available choices.
3. Check **Increased diameter upstream** if you want the program to ensure that the upstream pipes will have increasing (or the same) diameter.
4. Check **Check for pipe availability** if you want the program to take into account the availability of the pipes, as defined in conduit shapes**.**
5. Optionally, you can impose in **increase in head loss** in the flow through pipes, by setting an appropriate percentage in the relevant field.
6. Press **Ok** to close the form and save the changes. Press **Cancel** to close the form without saving any changes.

NOTE:

For the optimization to be successful and efficient, some requirement must be met. The following should be noted:

- The optimization algorithm is applied to branched networks without loops with exactly one reservoir.
- After the optimization, you can convert the reservoir to a tank and finalize the project.
- You should select as many pipe specifications as possible, which should cover all flow demands in your network.
- The conduit shapes must be fully defined, i.e. they must include the cost, the diameter, maximum velocity, minimum velocity, friction coefficient etc.
- The optimization algorithm is based on SIMPLEX.
- If the results indicate that two conduit shapes must be used for a single pipe, the largest shape is selected for the whole pipe.
- For the aforementioned reason, some minor violations of availability constraints may be observed, in the case the conduit shape availability is not unlimited.
- If you want to fix a specific conduit to a certain shape, produce its properties form by double clicking on the conduit, and set **Fixed pipe specification** to **Yes**.
- The optimization constraints that are taken into account are: minimum flow velocity, maximum flow velocity, minimum head at the junctions, pipe availability (optional), increasing diameter upstream (optional). The minimum head at the junctions is evaluated as **ground elevation** plus the **minimum pressure**. These properties are available in the junction properties form.

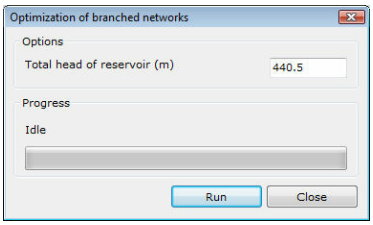
NOTE: This menu is available as a separate product. It is activated when you purchase the program "Irrigation optimization".

10.3 Optimization

This dialog box allows the user to optimize an irrigation network.

To optimize an irrigation network:

1. From the menu **Irrigation**, select **Optimization**. The following form appears:



2. Select the **total head of the reservoir**. This setting is saved into the reservoir's properties.
3. To optimize the network, press **Run**. The shape of the pipes are modified in the main project. An informative message appears, regarding the total cost of the network .
4. Press **Close** to close the form.

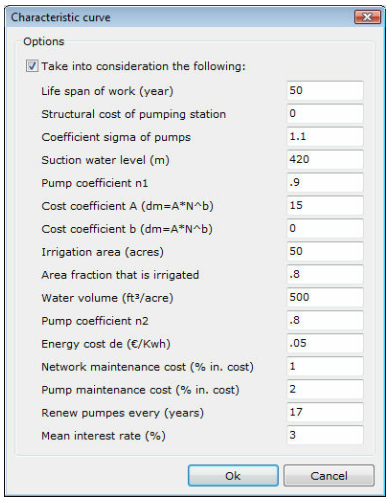
NOTE: This menu is available as a separate product. It is activated when you purchase the program "Irrigation optimization".

10.4 Characteristic curve options

With this dialog box, the user can set the options regarding the evaluation of the network cost.

To set the options regarding the evaluation of the network cost:

1. From the **Irrigation** menu, select **Characteristic curve** options. The following form appears:



2. Select **Take into account the following** to enable the following options:

- **Life span of work (year)**: Enter the expected life span of the project. Usual values are in the range of 50 years.
- **Structural cost of pumping station**: Enter the structural cost of the pumping station in € or any other consistently used currency.
- **Coefficient sigma of pumps:** Enter the sigma coefficient of the pump cost. Usual values are 1.10 - 1.15.
- **Suction water level**: Enter the suction level of the water.
- **Pump coefficient n1:** Enter the pump efficiency coefficient n1. Usual value is 0.9.
- **Cost coefficient A (dm=A*N^b)**: Enter the A coefficient which produces the cost per HP of the pumping station.
- **Cost coefficient A (dm=A*N^b):** Enter the b coefficient which produces the cost per HP of the pumping station. For b=0, the cost per HP is independent of the size of pumps (i.e. the total cost is a linear function of the number of HP).
- **Irrigation area (acres)**: Enter the irrigation area in acres.
- **Area fraction that is irrigated**: Enter the fraction of the above area that is irrigated. The fraction is assumed constant for the whole life span of the project.
- **Water volume (ft³/acre)**: Enter the required water volume for irrigation per acre.
- **Pump coefficient n2**: Enter the mean efficiency of the pump station, which is depended among others on its configuration.
- **Energy cost de (euro/KWh)**: Enter the energy cost in euros (or any other currency that is consistently used) per KWh.
- **Network maintenance cost (% initial cost)**: Enter the network maintenance

cost, as a percentage of the initial cost. Usually this is taken equal to 1%.
- **Pump maintenance cost (% initial cost)**: Enter the pump maintenance cost, as a percentage of the initial cost. Usually this is taken equal to 2%.
- **Renew pumps every (years)**: Enter the life span of the pumping station. Usual values are in the range of 17 years.
- **Mean interest rate (%)**: Enter the mean interest rate for the evaluation of present value of the investment.

3. Press **Ok** to close the form and save the changes. Press **Cancel** to close the form without saving any changes.

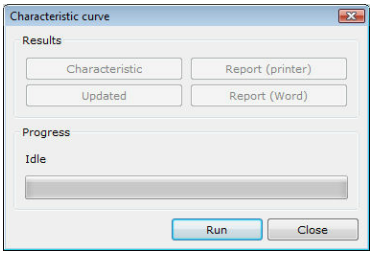
NOTE: This menu is available as a separate product. It is activated when you purchase the program "Irrigation optimization".

10.5 Characteristic curve

With this option, the user can evaluate the characteristic curve of the network. If additional financial data are given, then the evaluation of the present value of the investment is also possible.

To evaluate the characteristic curve of the network:

1. From the **Irrigation** menu, select **Characteristic curve**. The following form appears:



2. Press **Run** to evaluate the characteristic curve of the network. The buttons in the **Results** frame are enabled:

- **Characteristic**: Produces the characteristic curve of the network without taking into account additional financial data i.e. based on the pipe costs only.
- **Updated**: Produces the characteristic curve of the network taking into account additional financial data i.e. based on the present value of the cost of the whole investment.
- **Report (printer)**: Prepares and produces a calculation report in the print manager.
- **Report (Word)**: Prepares and produces a calculation report in Microsoft Word. Microsoft Word version 2003 or newer is required, with all available patches installed.

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

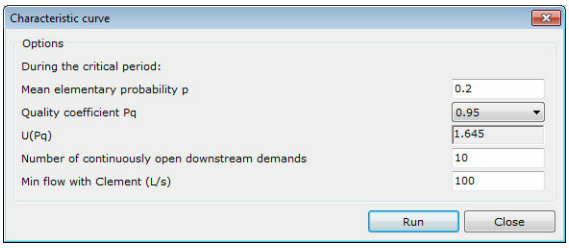
NOTE: This menu is available as a separate product. It is activated when you purchase the program "Irrigation optimization".

10.6 Flow based on Clement's First Formula

The program is able to evaluate and apply corrective demands in the junction demands, so that flow in the pipes is set based on Clement's first formula.

For the flow based on Clement's first formula:

1. Form the **Irrigation** menu, select **Flow based on Clement's first formula**. The following form appears:



2. Enter the following data:

- **Mean elementary probability p**: Enter the mean elementary probability of each hydrant to be open. During the analysis, each demand in the network is assumed to be a hydrant.
- **Quality coefficient Pq**: Enter the cumulative probability Pq that is related to the quality of the network (usually >=95%).
- **Number of continuously open downstream demands**: Enter the number of downstream hydrants that will be assumed to be continuously open.
- **Min flow with Clement**: set the minimum flow that can be used when evaluating the flow based on Clement's formula.

4. Press **Run** to evaluate and apply the flow based on Clement's first formula. Press **Close** to close the form.

NOTE: The flow based on Clement's first formula can be erased with the menu Clear Clement's Flow that is described in the following paragraph. In this way, the program is restored to its normal mode of analysis.

NOTE: This menu is available as a separate product. It is activated when you purchase the program "Irrigation optimization".

10.7 Clear Clement's Flow

With this options, the flows based on Clement's first formula are erased. The program is restored to its normal mode of analysis.

To clear Clement's flow:

1. From the **Irrigation** menu, select **Clear Clement's flow**.

NOTE: This menu is available as a separate product. It is activated when you purchase the program "Irrigation optimization".

10.8 Check optimization constraints

With this option, the user can check whether the optimization constraints are fulfilled. The constraints that are checked are: minimum velocity, maximum velocity in the pipes, minimum pressure at the junctions. The minimum head at the junctions is evaluated as **ground elevation** plus the **minimum pressure**. These properties are available in the junction properties form. The associated cost for the current state is also evaluated and displayed. This cost refers to the pipes only.

To check whether the optimization constraints are fulfilled:

1. From the **Irrigation** menu, select **Check optimization constraints**.
2. A relevant informative message appears.

NOTE: This menu is available as a separate product. It is activated when you purchase the program "Irrigation optimization".

Chapter

11 Optimization

11.1 Optimization menu

With this menu, the user obtains access to advanced tools for the design and optimization of water networks. In the **Optimization** menu, the following options are available:

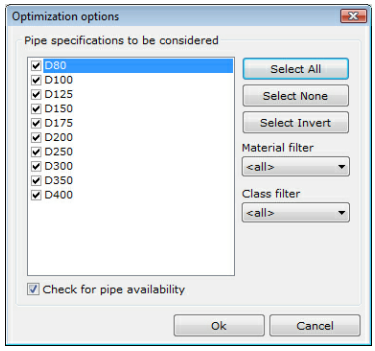
- Optimization options
- GA options
- Cost function
- Optimization
- Check optimization constraints

11.2 Optimization options

This dialog box allows the user to set the optimization options for networks.

To set the optimization options for networks:

1. From the **Optimization** menu, select **Optimization options**. The following form appears:



2. Select one or more pipe specifications which will be used for optimization. The **material** and **class filter** may assist you by limiting the available choices.
3. Check **Check for pipe availability** if you want the program to take into account the availability of the pipes, as defined in conduit shapes**.**
4. Press **Ok** to close the form and save the changes. Press **Cancel** to close the form without saving any changes.

NOTE:

For the optimization to be successful and efficient, some requirement must be met. The following should be noted:

- You should select as many pipe specifications as possible, which should cover all flow demands in your network.
- The conduit shapes must be fully defined, i.e. they must include the cost, the diameter, maximum velocity, minimum velocity, friction coefficient etc.
- The minimum and maximum allowed pressure must be defined for all junctions.

- The optimization algorithm is based on Genetic Algorithms (GAs). A local optimizer is invoked at the end of the GA analysis.
- If you want to fix a specific conduit to a certain shape, produce its properties form by double clicking on the conduit, and set **Fixed pipe specification** to **Yes**.
- The optimization constraints that are taken into account are: minimum flow velocity, maximum flow velocity, minimum/maximum pressure at the junctions, pipe availability (optional). The minimum head at the junctions is evaluated as **ground elevation** plus the **minimum/maximum pressure**. These properties are available in the junction properties form.

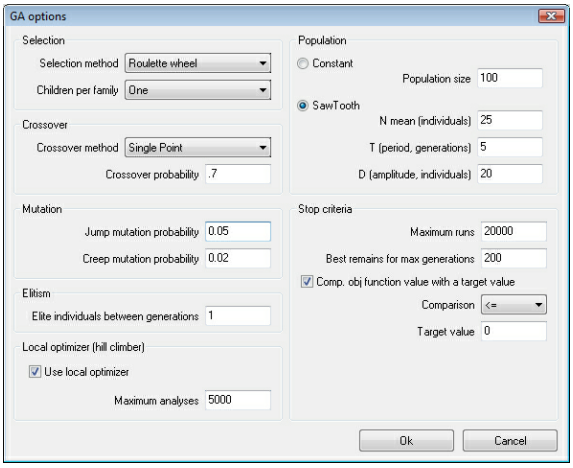
NOTE: This menu is available as a separate product. It is activated when you purchase the program "Network optimization".

11.3 GA options

With this option, the user can set the Genetic Algorithm (GA) options.

To set the Genetic Algorithm (GA) options:

1. From the **Optimization** menu, select **GA options**:



2. Make the desired changes.
3. Press **Ok** to close the form and save the changes. Press **Cancel** to close the form without saving any changes.

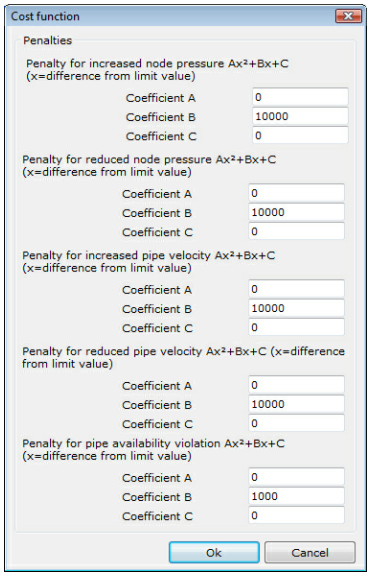
NOTE: This menu is available as a separate product. It is activated when you purchase the program "Network optimization".

11.4 Cost function

With this option, the user can set options for the cost function.

To set the options for the cost function:

1. From the **Optimization** menu, select **Cost function**:



2. Make the desired changes.
3. Press **Ok** to close the form and save the changes. Press **Cancel** to close the form without saving any changes.

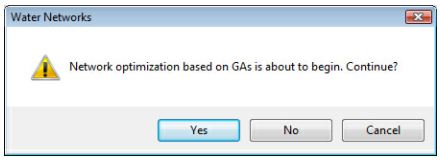
NOTE: This menu is available as a separate product. It is activated when you purchase the program "Network optimization".

11.5 Optimization

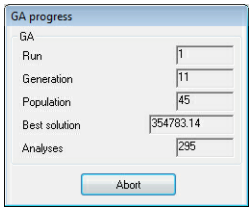
With this option, you can optimize a water network.

To optimize a water network:

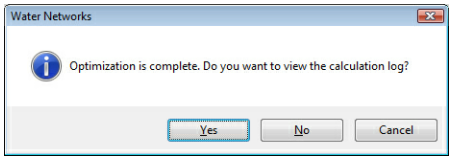
1. From the **Optimization** menu, select **Optimization**. The following form appears:



2. Select **Yes** to proceed with optimization:



3. By letting the GA to finish, or by pressing **Abort** once, the local optimizer is invoked. Press the **Abort** button again, or let the local optimizer to finish, to conclude the optimization:



4. Press **Ok** to view the calculation log. Press any other button to close the form. The network has been updated with the best solution found by the optimization process.

NOTE: This menu is available as a separate product. It is activated when you purchase the program "Network optimization".

11.6 Check optimization constraints

With this option, the user can check whether the optimization constraints are fulfilled. The constraints that are checked are: minimum velocity, maximum velocity in the pipes, minimum pressure, maximum pressure at the junctions. The minimum/ maximum head at the junctions is evaluated as **ground elevation** plus the **minimum/maximum pressure**. These properties are available in the junction properties form. The associated cost for the current state is also evaluated and displayed. This cost refers to the pipes only.

To check whether the optimization constraints are fulfilled:

1. From the **Optimization** menu, select **Check optimization constraints**.
2. A relevant informative message appears.

NOTE: This menu is available as a separate product. It is activated when you purchase the program "Network optimization".

Chapter

12 Results

12.1 Results menu

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

- Perform calculations
- Results report
- Animation
- Grid
- Filters
- Colors
- Calibrate Point Objects
  - Demand
  - Pressure Height
  - Pressure
  - Quality
- Calibrate Linear Objects
  - Flow Rate
  - Velocity
- Energy consumption
- Reaction report
- Total conduit lengths
- Special devices count
- Profiles
  - Options
  - Design
- Quantities

12.2 Perform calculations

With this option, you can perform calculations.

To perform calculations:

1. Select **Perform calculations** from the **Results** menu.
2. The calculations are performed.

12.3 Results report

After the completion of calculations, a report is prepared that contains a list with possible issues.

To show this report:

1. Select **Perform calculations** from the **Results** menu.
2. Select **Results report** from the **Results** menu. If an error report is available, it is

displayed.
3. Hit **ESC** to close the form.

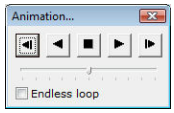
NOTE: The error codes are described in detail in the Appendix. For each error code, common troubleshooting options are provided.

12.4 Animation

If extended period analysis has been performed, then with this option you can view the results in animated form.

To view the results in animated form:

1. Select **Animation** from the **Results** menu. The following form appears:



2. Select to move to the first time instant.
3. Select to view the results in reverse.
4. Select to stop animation.
5. Select to start animation.
6. Select to move to the last time instant.

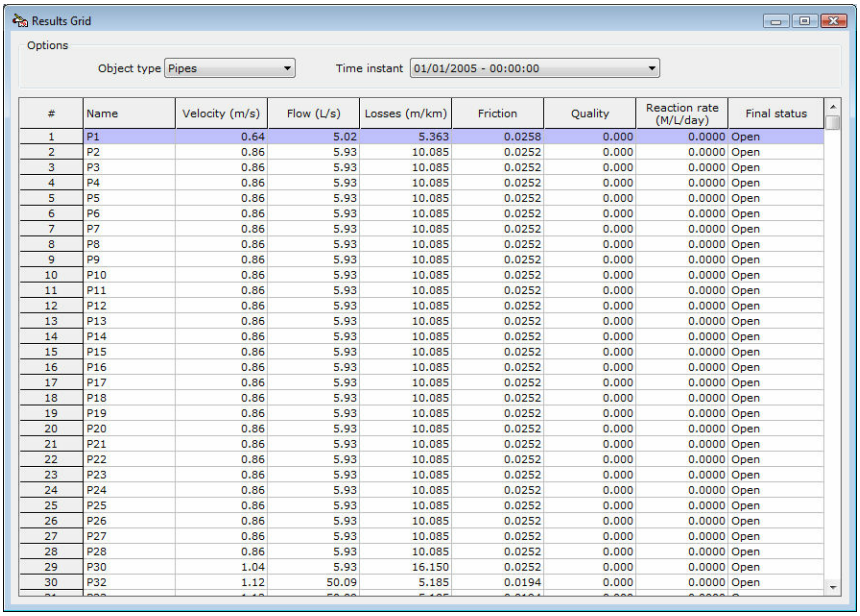
7. Check **Endless loop** if you wish to view the animation with endless loop.

12.5 Grid

With this option, the results of the calculations are displayed in tabular form an a per-object and time-instance basis.

To display the results of the calculations in tabular form an a per-object basis:

1. Make sure that the calculations have been completed successfully.
2. Select **Grid** from the **Results** menu.
3. The following form appears:



#	Name	Velocity (m/s)	Flow (L/s)	Losses (m/km)	Friction	Quality	Reaction rate (M/L/day)	Final status
1	P1	0.64	5.02	5.363	0.0258	0.000	0.0000	Open
2	P2	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
3	P3	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
4	P4	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
5	P5	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
6	P6	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
7	P7	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
8	P8	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
9	P9	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
10	P10	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
11	P11	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
12	P12	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
13	P13	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
14	P14	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
15	P15	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
16	P16	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
17	P17	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
18	P18	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
19	P19	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
20	P20	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
21	P21	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
22	P22	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
23	P23	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
24	P24	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
25	P25	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
26	P26	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
27	P27	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
28	P28	0.86	5.93	10.085	0.0252	0.000	0.0000	Open
29	P30	1.04	5.93	16.150	0.0252	0.000	0.0000	Open
30	P32	1.12	50.09	5.185	0.0194	0.000	0.0000	Open

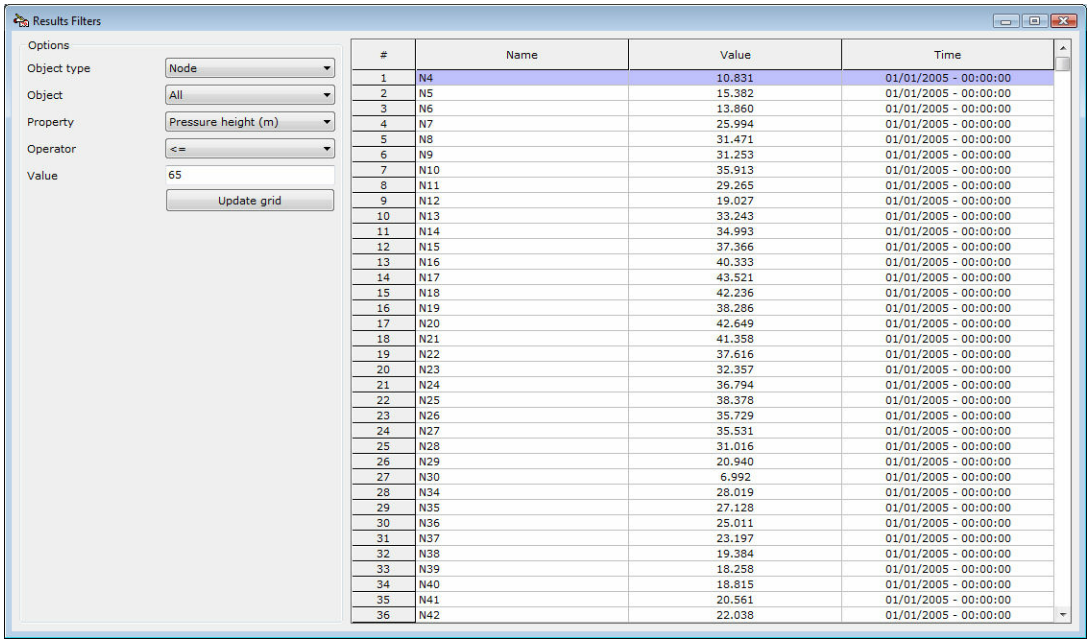
4. Select the **Object Type** from the drop-down list.
5. Select the **Time Instance** from the drop-down list.
6. Hit **ESC** to close the form.

12.6 Filters

With this option, the results meeting certain criteria are displayed in tabular form.

To display the filtered results of the calculations in tabular form:

1. Make sure that the calculations have been completed successfully.
2. Select **Filters** from the **Results** menu.
3. The following form appears:



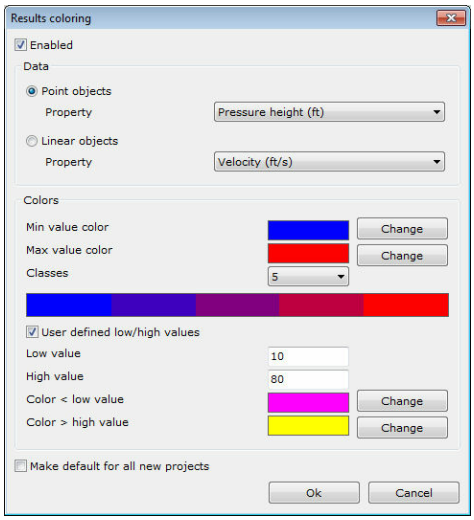
4. Enter the filtering criteria.
5. When done, click **Update Grid** to view the filtered results.
6. You may repear steps 4 and 5 indefinitely.
7. Hit **ESC** to hide the form.

12.7 Colors

With this option, you can modify the settings for the coloring of objects depending on their properties.

To modify the settings for the coloring of objects:

1. Select **Colors** from the **Results** menu. The following form appears:



2. Check **Enabled** to enable the coloring of objects.

3. Select whether you want to color the point or the linear objects.
4. Select the colors that correspond to the minimum and maximum values by clicking on the corresponding **Change** button. If you want to use custom (user defined) high and low values, check the corresponding field. In this case you need to provide the colors for the values that are higher than the high value or lower than the low value.
5. Select the number of classes for the classification of objects.
6. Select **Ok** to close the dialog box and save changes. Select **Cancel** to close the dialog box and ignore changes.

NOTE: In some cases, such as when the calculations have not been completed or when there is no variation of property values, the coloring will not be applied. In these cases, the default coloring is used.

12.8 Calibrate Point Objects

12.8.1 Demand

With this option, you can compare observed calibration data with results obtained from the model. Statistical data and an autocorrelation graph are displayed, along with the observed and calculated data.

To calibrate point objects with respect to demand:

1. Select **Calibrate Point Objects** from the **Results** menu.
2. Select **Demand** from the **Calibrate Point Objects** menu. The calibration results are displayed.

NOTE: This option is disabled if no observed calibration data are available.

NOTE: An example of calibration is given in the paragraph regarding quality.

12.8.2 Pressure Height

With this option, you can compare observed calibration data with results obtained from the model. Statistical data and an autocorrelation graph are displayed, along with the observed and calculated data.

To calibrate point objects with respect to pressure height:

1. Select **Calibrate Point Objects** from the **Results** menu.
2. Select **Pressure Height** from the **Calibrate Point Objects** menu. The calibration results are displayed.

NOTE: This option is disabled if no observed calibration data are available.

NOTE: An example of calibration is given in the paragraph regarding quality.

12.8.3 Pressure

With this option, you can compare observed calibration data with results obtained from the model. Statistical data and an autocorrelation graph are displayed, along with the observed and calculated data.

To calibrate point objects with respect to pressure:

1. Select **Calibrate Point Objects** from the **Results** menu.
2. Select **Pressure** from the **Calibrate Point Objects** menu. The calibration results are displayed.

NOTE: This option is disabled if no observed calibration data are available.

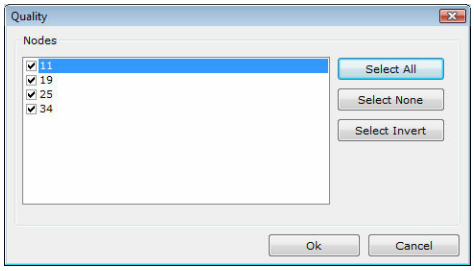
NOTE: An example of calibration is given in the paragraph regarding quality.

12.8.4 Quality

With this option, you can compare observed calibration data with results obtained from the model. Statistical data and an autocorrelation graph are displayed, along with the observed and calculated data.

To calibrate point objects with respect to quality:

1. Select **Calibrate Point Objects** from the **Results** menu.
2. Select **Quality** from the **Calibrate Point Objects** menu. The following form appears:



3. Select the point objects that will be taken into account in the calibration. The quick keys (**Select all, Select None, Invert Selection**) can be used to quickly select all objects, deselect all objects and invert the current selection.
4. Select **Ok** to proceed with calibration. Select **Cancel** to cancel the operation and close the dialog box.
5. Select the **Statistics** tab to view the results of the statistical analysis.
6. Select the **Correlation** tab to view the autocorrelation graph.
7. Select the **Mean Comparison** to view the results of the comparison of mean values.
8. Select **Ok** or **Cancel** to close the form.

For your convenience, the following options are available from the **Functions** menu:

- **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 option is disabled if no observed calibration data are available.

12.9 Calibrate Linear Objects

12.9.1 Flow Rate

With this option, you can compare observed calibration data with results obtained from the model. Statistical data and an autocorrelation graph are displayed, along with the observed and calculated data.

To calibrate linear objects with respect to flow rate:

1. Select **Calibrate Linear Objects** from the **Results** menu.
2. Select **Flow Rate** from the **Calibrate Linear Objects** menu. The calibration results are displayed.

NOTE: This option is disabled if no observed calibration data are available.

NOTE: An example of calibration is given in the paragraph regarding quality.

12.9.2 Velocity

With this option, you can compare observed calibration data with results obtained from the model. Statistical data and an autocorrelation graph are displayed, along with the observed and calculated data.

To calibrate linear objects with respect to velocity:

1. Select **Calibrate Linear Objects** from the **Results** menu.
2. Select **Velocity** from the **Calibrate Linear Objects** menu. The calibration results are displayed.

NOTE: This option is disabled if no observed calibration data are available.

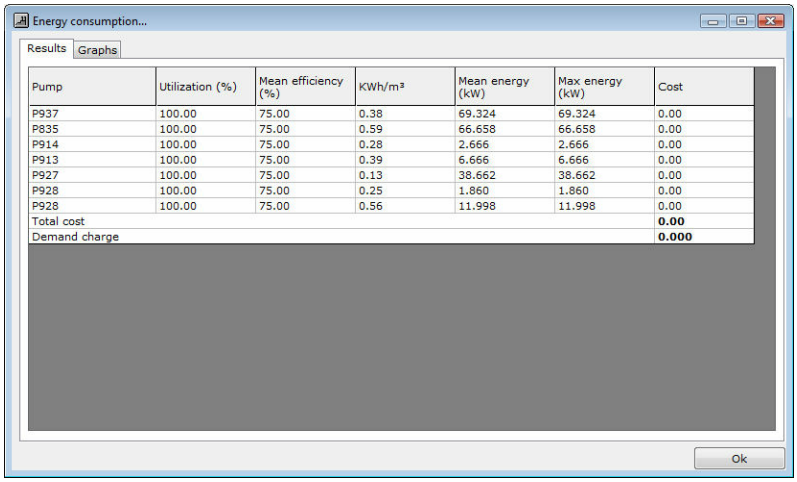
NOTE: An example of calibration is given in the paragraph regarding quality.

12.10 Energy Consumption

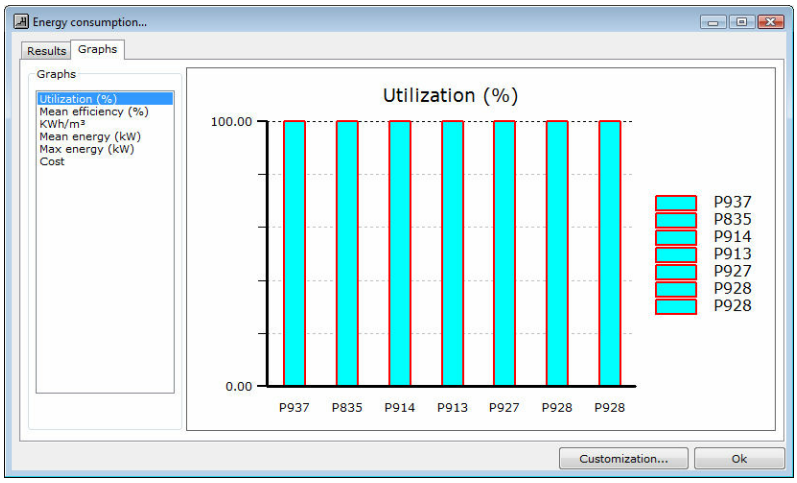
If one or more pumps are included in the network, with this option you can view a report regarding energy consumption and operational costs. Note that all data regarding energy consumption and operational costs must have been provided.

To view a report regarding energy consumption and operational costs:

1. Select **Energy Consumption** from the **Results** menu.
2. Select the **Results** tab to show an overview of the results:



3. Select the **Graphs** tab to view the results in a visual way:



Select the value from the list on the left. The graph is updated automatically. Optionally, you can modify the graph by selecting **Customization...**.

4. Select **Ok** to close the form.

12.11 Reaction Report

If all quality data are provided, with this option you can view a report regarding the mean reaction (in kg/h) on the pipe walls, in the pipe interior and in tanks. The results are displayed in both analytical and graphical form.

To view a report regarding the mean reaction:

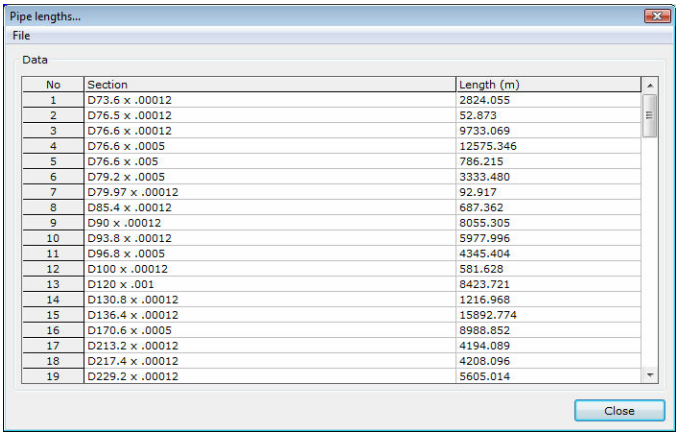
1. Select **Reaction Report** from the **Results** menu.
2. Optionally, you can modify the graph by selecting **Customization...**
3. Select **Ok** or **Cancel** to close the form.

12.12 Total conduit lengths

With this option, you can calculate the conduit lengths per profile that is currently used in the network. The report is based on conduit shapes.

To calculate the conduit lengths:

1. Select **Total conduit lengths** from the **Results** menu. The following form appears:



No	Section	Length (m)
1	D73.6 x .00012	2824.055
2	D76.5 x .00012	52.873
3	D76.6 x .00012	9733.069
4	D76.6 x .0005	12575.346
5	D76.6 x .005	786.215
6	D79.2 x .0005	3333.480
7	D79.97 x .00012	92.917
8	D85.4 x .00012	687.362
9	D90 x .00012	8055.305
10	D93.8 x .00012	5977.996
11	D96.8 x .0005	4345.404
12	D100 x .00012	581.628
13	D120 x .001	8423.721
14	D130.8 x .00012	1216.968
15	D136.4 x .00012	15892.774
16	D170.6 x .0005	8988.852
17	D213.2 x .00012	4194.089
18	D217.4 x .00012	4208.096
19	D229.2 x .00012	5605.014

2. The total conduit lengths per specification are displayed in the list.
3. Select **Close** to close the dialog box.

From the **File** menu, the following options are available:

Print, Print to Word, Print to Excel

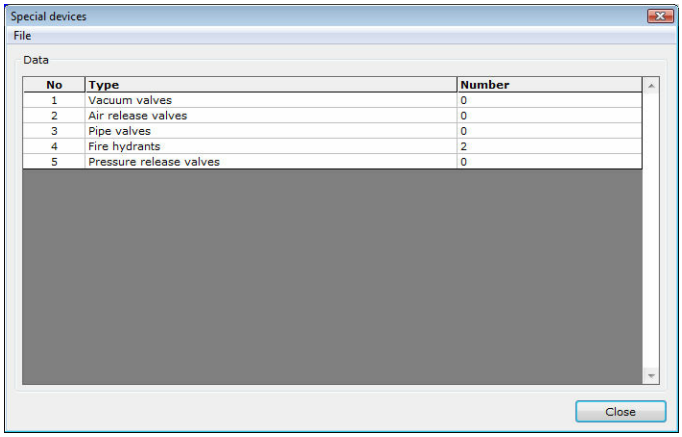
Select the appropriate option to create a report and sent it to the corresponding recipient.

12.13 Special devices count

With this option, you can count the special devices, i.e. air release valves, vacuum valves and valves, that are currently used in the network.

To count the special devices:

1. Select **Special devices count** from the **Results** menu. The following form appears:



2. The total number of special devices are displayed in the list.
3. Select **Close** to close the dialog box.

From the **File** menu, the following options are available:

Print, Print to Word, Print to Excel

Select the appropriate option to create a report and sent it to the corresponding recipient.

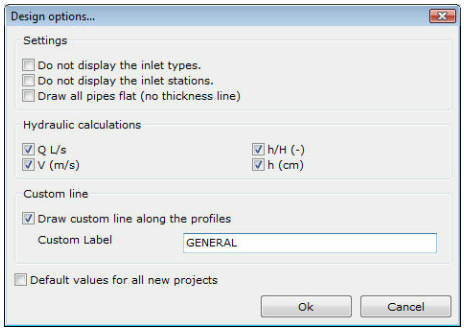
12.14 Profiles

12.14.1 Options

With this option, you can modify the settings of the profile drawings.

To modify the settings of the profile drawings:

1. Select **Profiles** from the **Results** menu.
2. Select **Options** from the **Profiles** menu. The following form will appear:



2. In the **Settings** frame:

- Check **Do not display the inlet types** if you do not want to display the inlet types.
- Check **Do not display the inlet stations** if you do not want to display the inlet stations.

- Check **Draw all pipes flat (no thickness line)** if you do not want to display the pipe thickness in the profile drawing. This does not affect the sketch of the main form where the pipe thickness is not drawn.

3. In the **Hydraulic calculations** frame, select one or more values that you wish to be included in the profile drawing.
4. In the **Custom line** frame:

- Check **Draw custom line along the profiles** if you want a custom line to be drawing along the profiles. In this case, you can provide the title of the custom line in the corresponding text box.

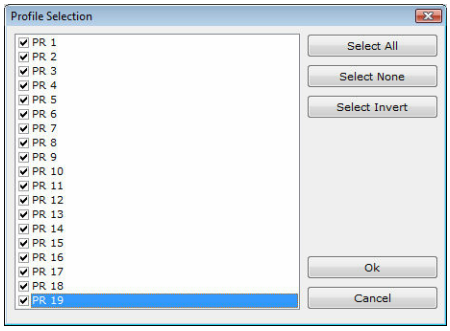
5. Check **Default values for all new projects** if you wish to make the settings default for all new projects. This does not affect existing projects.
6. Select **Ok** to save changes and close the dialog box. Select **Cancel** to close the dialog box without saving any changes.

12.14.2 Design

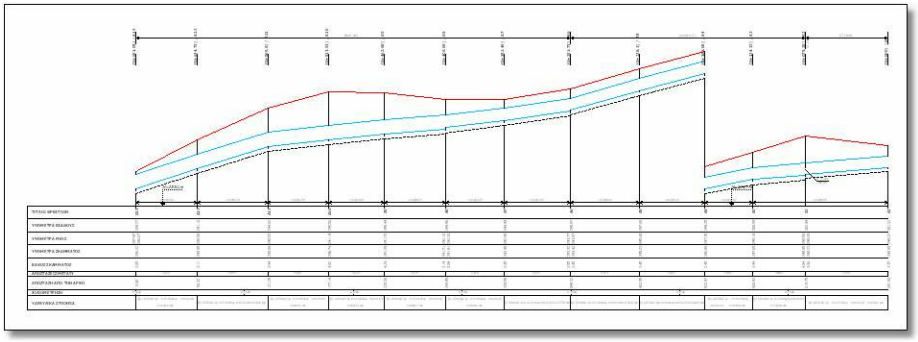
With this option, you can create the profile drawing. The data are prepared and sent to the **Profile designer**. A complete user manual on the capabilities of **Profile designer** can be found in the corresponding help file.

To create a profile:

1. Select **Draw profiles** from the **Results** menu. The following form appears:



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



12.15 Quantities

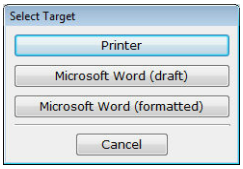
With this option, a report on the network quantities is assembled and prepared for preview.

In the current version of the program, this report includes:

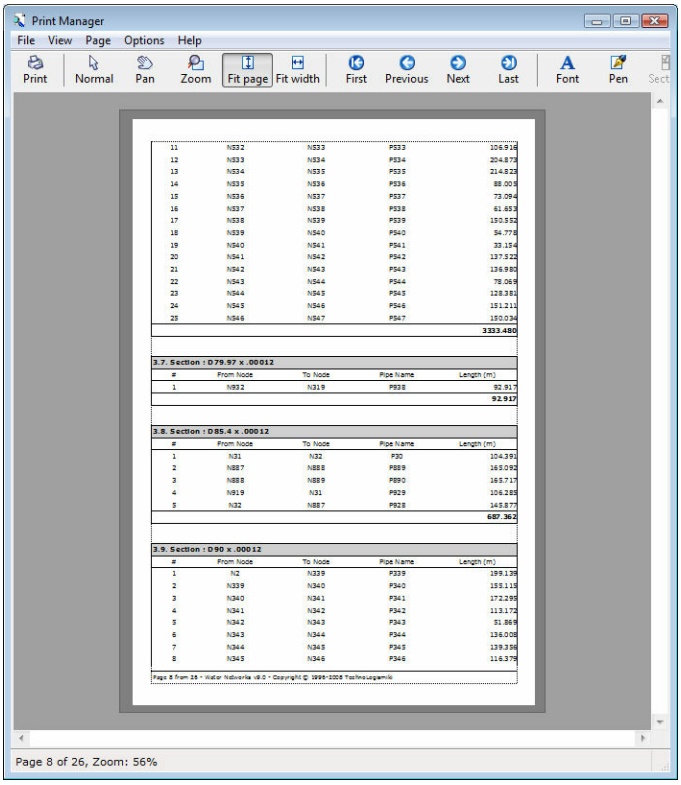
- Excavation tables based on trench profiles
- Excavation/backfill volumes
- Pipe quantities based on conduit shapes
- Manhole quantities based on manhole specifications
- Trench quantities based on trench specifications

To create a report on the network quantities:

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



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



NOTE: The formatted Microsoft Word file requires the use of the clipboard. During the creation of the file, you should not use the clipboard.

Chapter

13 Help

13.1 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

13.2 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.

13.3 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.

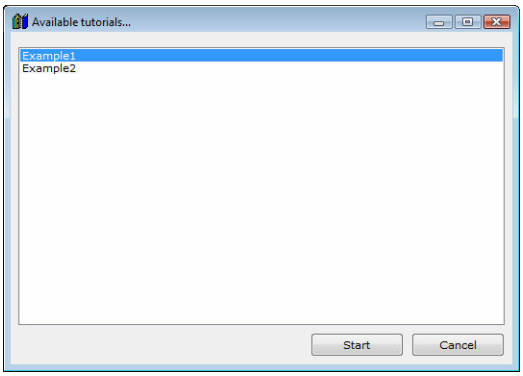
NOTE: Adobe Acrobat Reader or a similar program that can display pdf files is required in order to view or print the user guide.

13.4 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.



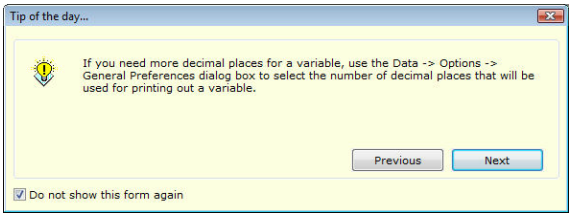
NOTE: The number and content of the tutorials is changed frequently. Use the live update system of TechnoLogismiki's products to download the latest tutorials.

13.5 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.



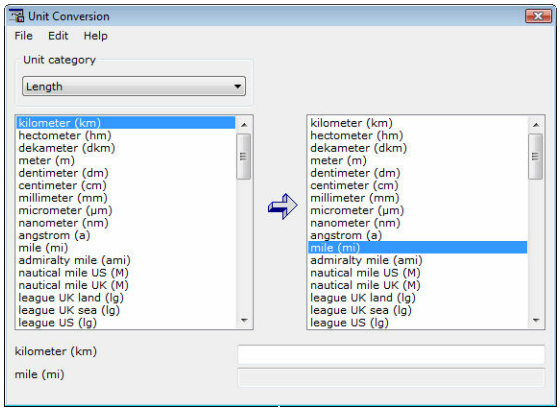
NOTE: The number and content of the tips is changed frequently. Use the live update system of TechnoLogismiki's products to download the latest tips.

13.6 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.

13.7 TechnoLogismiki website

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

13.8 Buy products

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

13.9 TechnoLogismiki NOMOS

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

13.10 TechnoLogismiki Live!

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

13.11 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

14 Appendix

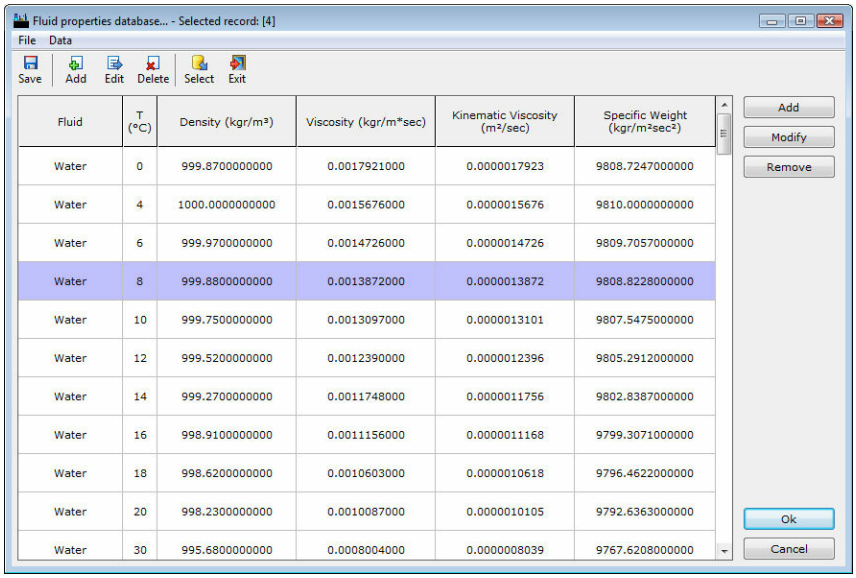
14.1 Unit system

Unit	Metric system	English system
Area	hectares m^2	acres ft^2
Length	m mm	ft in
Rainfall intensity	mm/h	in/h
Flow rate	m^3/s L/s ML/day	ft^3/s g/m Mg/day
Pollutant concentration	mass/hectare	mass/acre
Volume	m^3	ft^3

14.2 Fluid database

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

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

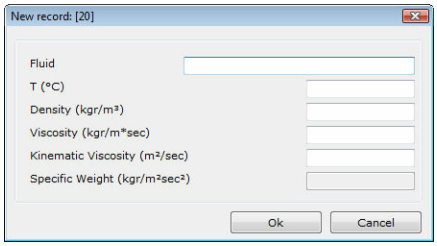


Fluid	T (°C)	Density (kgr/m³)	Viscosity (kgr/m*sec)	Kinematic Viscosity (m²/sec)	Specific Weight (kgr/m²sec²)
Water	0	999.8700000000	0.0017921000	0.0000017923	9808.7247000000
Water	4	1000.0000000000	0.0015676000	0.0000015676	9810.0000000000
Water	6	999.9700000000	0.0014726000	0.0000014726	9809.7057000000
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:

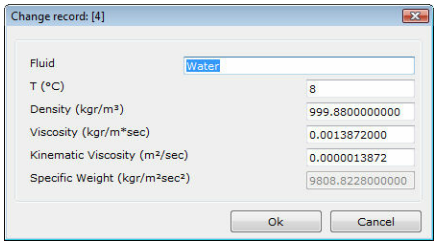
1. Click **Add** to open the new record dialog box.
2. Type the name of the fluid. This field is required.

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.



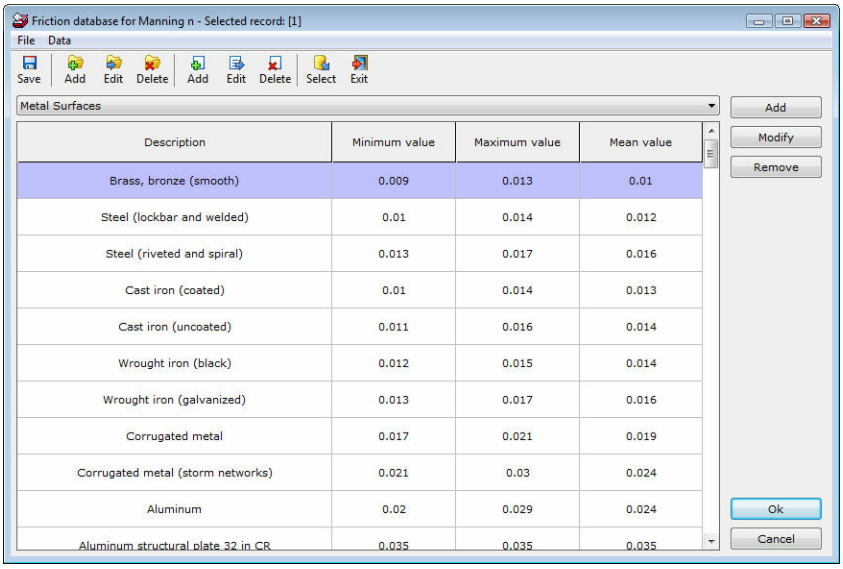
To remove an existing record:

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.

14.3 Friction database

For your convenience, a fully customizable friction database is embedded in the program. The friction database is invoked in various cases within the program. By selecting an appropriate friction record (which is depended on the selected friction formula) and clicking **Ok**, the data is transferred to the corresponding fields. Select **Cancel** to close the database without transferring any data.

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

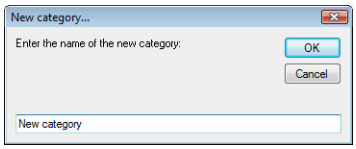


Description	Minimum value	Maximum value	Mean value
Brass, bronze (smooth)	0.009	0.013	0.01
Steel (lockbar and welded)	0.01	0.014	0.012
Steel (riveted and spiral)	0.013	0.017	0.016
Cast iron (coated)	0.01	0.014	0.013
Cast iron (uncoated)	0.011	0.016	0.014
Wrought iron (black)	0.012	0.015	0.014
Wrought iron (galvanized)	0.013	0.017	0.016
Corrugated metal	0.017	0.021	0.019
Corrugated metal (storm networks)	0.021	0.03	0.024
Aluminum	0.02	0.029	0.024
Aluminum structural plate 32 in CR	0.035	0.035	0.035

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

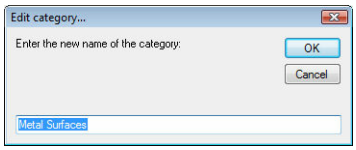
To add a new category:

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.



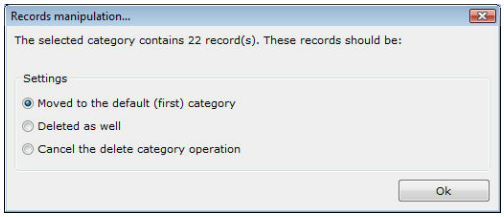
To modify the name of an existing category:

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:

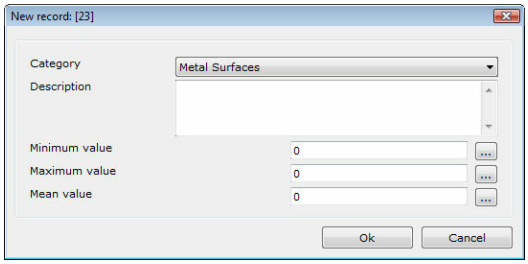


4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

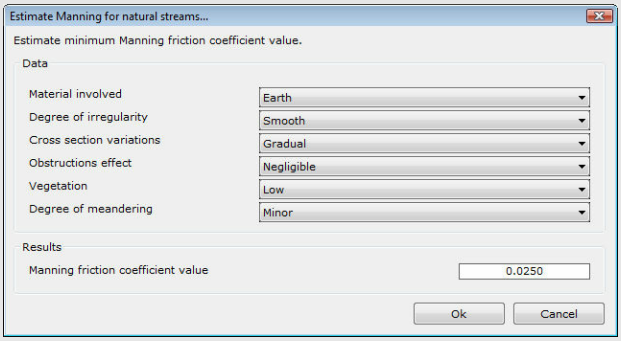
NOTE: The database must contain at least one category.

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.



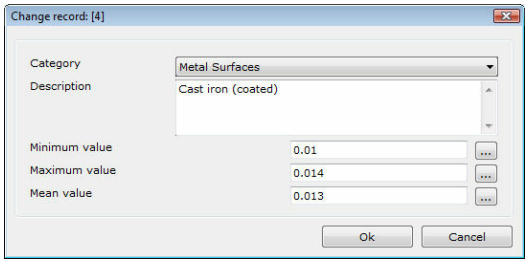
NOTE: In case of Manning friction coefficients in natural streams, you can estimate the values based on several characteristics of the stream. Click on the buttons with the ellipses (...) next to the text boxes to invoke the following dialog box:



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.

14.4 Manning friction coefficients

Surface / Material	Mean Value
Aluminum	0.024
Asbestos cement	0.013
Asphalt ditch	0.016
Asphalt pavement	0.016
Asphalt smooth	0.013
Asphalted cast iron	0.012
Natural ground	0.020
Best concrete	0.010
Brick in mortar	0.015
Brick sewer	0.015
Cast iron	0.012
CMP	0.024
Concrete	0.013
PVC	0.010
Centrifugal SPUN	0.013
Concrete (steel forms)	0.011
Concrete (wood forms)	0.015
Concrete gutter (broom finish)	0.016
Concrete gutter (troweled finish)	0.012
Copper	0.011

Fiber glass roving	0.011
Gravel riprap (D=25)	0.033
Gravel riprap (D=50)	0.041
Grouted riprap	0.030
Natural stream (clean)	0.030
Natural stream (stone)	0.050
Natural stream (weedy)	0.035

14.5 Bazin friction coefficients

Surface / Material	Max value	Min value	Mean value
Rough concrete	0.5	0.4	0.46
Smooth concrete	0.08	0.04	0.06
Brick in mortar	0.018	0.014	0.016
Sewer pipes (Greek regulations 696/74)	0.25	0.25	0.25
Storm pipes (Greek regulations 696/74)	0.46	0.46	0.46

14.6 Hazen - Williams friction coefficients

Surface / Material	Mean value
Asbestos cement	140
Asphalted cast iron	130
Best concrete	150
Centrifugal SPUN	135
Concrete (wood forms)	120
Concrete (steel forms)	140
Copper	135
Ductile iron	130
Galvanized iron	120
Glass	140
PVC	150
Riveted steel (new, rough)	80
Riveted steel (new, smooth)	110
Steel	120
Wood (new)	140

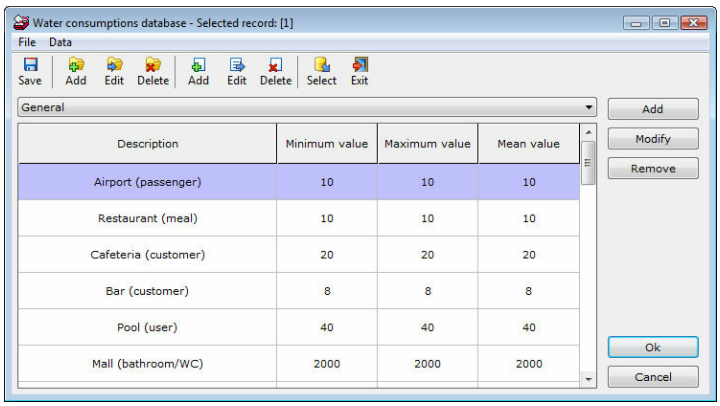
14.7 Darcy - Weisbach friction coefficients

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

14.8 Water consumption

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

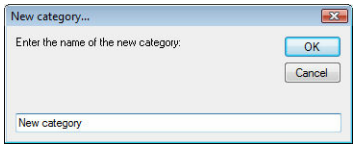
You will be asked to confirm any changes you have made to the database when exiting. The changes will be instantly available to other programs using the same database.



The database consists of several categories.

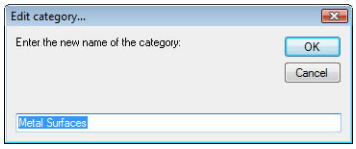
To add a new category:

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



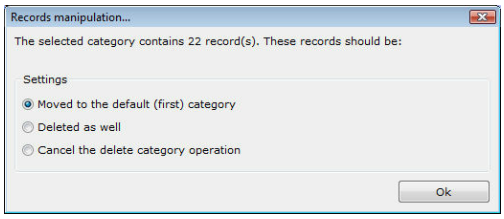
To modify the name of an existing category:

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



To remove an existing category:

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



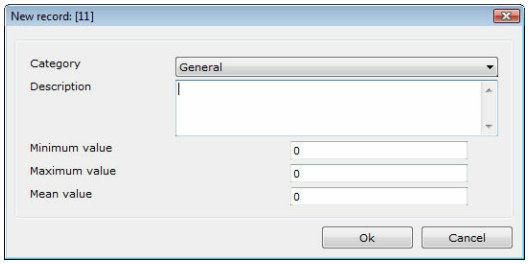
4.1. Select the first option to move the records of the category to the default (first category).
4.2. Select the second option to delete the records.
4.3. Select the third option to cancel the deletion.
5. Click **Ok** to proceed.

NOTE: The database must contain at least one category.

To add a new record:

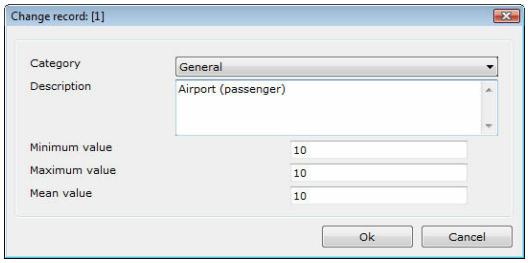
1. Click **Add** to open the new record dialog box.

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



To modify an existing record:

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



To remove an existing record:

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

14.9 Local energy losses friction coefficients

Fitting	K-value	Fitting	K-value
Pipe entrance		**90° smooth bend**[1]	
Bellmouth	0.00 ~ 0.05	r/D=4 (r radius)	0.16 ~ 0.18
Rounded	0.18	r/D=2	0.19 ~ 0.25
Sharp edged	0.50	r/D=1.5	0.26 ~ 0.34
Projecting	0.80 ~ 1.00	r/D=1	0.35 ~ 0.40

Fitting	K-value	Fitting	K-value
Contraction - Sudden		**Mitered bend**	
$D_2/D_1 = 0.20$	0.41 ~ 0.50	15° angle	0.05
$0.20 < D_2/D_1 = 0.40$	0.30 ~ 0.41	30° angle	0.10
$0.40 < D_2/D_1 = 0.60$	0.18 ~ 0.30	45° angle	0.20
$0.60 < D_2/D_1 = 0.80$	0.06 ~ 0.18	60° angle	0.35
$D_2/D_1 = 0.80$	0.00 ~ 0.06	90° angle	0.80
Contraction - Conical		**Tee [2]**	
15° angle	0.02	Line flow	0.30 ~ 0.40
22.5° angle	0.04	Branch flow	0.60 ~ 2.10
45° angle	0.07	**45° Wye [2]**	
Expansion - Sudden		Line flow	0.20 ~ 0.35
$D_2/D_1 = 0.20$	0.92 ~ 1.00	Branch flow	0.45 ~ 0.55
$0.20 < D_2/D_1 = 0.40$	0.71 ~ 0.92	**Cross [2]**	
$0.40 < D_2/D_1 = 0.60$	0.41 ~ 0.71	Line flow	0.40 ~ 0.60
$0.60 < D_2/D_1 = 0.80$	0.13 ~ 0.41	Branch flow	0.60 ~ 0.90
$D_2/D_1 = 0.80$	0.00 ~ 0.13	**Spherical valves [3]**	
Expansion - Conical		90° angle	0.05
15° angle	0.03	60° angle	1.20
22.5° angle	0.07	45° angle	10.00
45° angle	0.14	30° angle	50.00

NOTES:

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

14.10 Error messages

14.10.1 Error codes

The error codes that are displayed in the results report consist of a three-digit number and a brief description. In the following, a more detailed description as well as common troubleshooting options are provided.

- Codes 100-199
- Codes 200-299
- Codes 300-399

14.10.2 Codes 1XX

[101] insufficient memory.
There is not enough physical memory in the computer to analyze the study area.

[102] no network data to process.
There is no network present.

[103] Hydraylics solver not initialized.
The internal hydraulics solver has stopped because of insufficient input. Please make sure that you have entered all necessary information and retry.

[104] No hydraulic results available.
The time steps are such that no results are reported. You must correct the time steps and retry to solve the network.

[105] water quality solver not initialized.
The internal water quality solver has stopped because of insufficient input. Please make sure that you have entered all necessary information and retry.

[106] no results to report on.
The time steps are such that no hydraulic or quality results are reported. You must correct the time steps and retry to solve the network.

[110] cannot solve water quality transport equations.
The network quality stopped before successful termination. This could be caused be erroneous data or time steps that lead to an unstable system.

14.10.3 Codes 2XX

[200] one or more errors in input file.
This message appears when one or more input file parsing errors (the 200-series errors) occur.

[202] illegal numeric value in function call.
A variable has an illegal numeric value. Please correct the problem and retry.

[203] undefined node in function call.
A reference was made to a node that was never defined. An example would be if node 123 is used in a rule yet no such node was ever defined in the study area.

[204] undefined link in function call.
A reference was made to a link that was never defined. An example would be if link 123 is used in a rule yet no such link was ever defined in the study area.

[205] undefined time pattern in function call.
A reference was made to a time pattern that was never defined. An example would be if time pattern 123 is used in a rule yet no such time pattern was ever defined in the study area.

[207] attempt made to control a check valve.
Check valves cannot be controlled by a rule or a control. Please remove or correct the control or rule that is trying to set the status of a check valve.

[223] not enough nodes in network.
There are not enough nodes (junctions, tanks and reservoirs) in the network. Please add the missing nodes.

[224] no tanks or reservoirs in network.
For every network at least one tank or reservoir is required.

[240] undefined source in function call.
A quality source has been used which has not been defined first.

[250] function argument has invalid format.
The format of the function is not accepted by the solver. Please correct your input and retry.

[251] illegal parameter code in function call.
An illegal value has been used and must be corrected.

14.10.4 Codes 3XX

[301] files share same names.
The input, report, and binary output files specified on the command line cannot have the same names.

[302] cannot open input file.
The input file either does not exist or cannot be opened (e.g., it might be in use by another program).

[303] cannot open report file.
The report file cannot be opened (e.g., it might reside in a directory to which the user does not have write privileges).

[304] cannot open binary results file.
The binary output file cannot be opened (e.g., it might reside in a directory to which the user does not have write privileges).

[305] cannot open hydraulics file.
There was an error in trying to open the hydraulics file. This is caused because of lack of privileges. Log-in as administrator and try again.

[306] invalid hydraulics file.
There was an error accessing the hydraulics file. Disable any anti-virus software

and retry.

[307] cannot read hydraulics file.
There was an error accessing the hydraulics file. Disable any anti-virus software and retry.

[308] cannot save results to file.
There was an error in trying to save the results to a file. This is caused because of lack of privileges or the disk is full. Log-in as administrator or free some hard disk space and try again.

[309] cannot write report to file.
There was an error in trying to save the results to a file. This is caused because of lack of privileges or the disk is full. Log-in as administrator or free some hard disk space and try again.

Keyword index

- 6 -

- A -

- B -

- C -

- D -

- E -

- F -

- G -

- H -

- I -

- J -

- K -

- L -

- M -

- N -

- O -

- P -

- Q -

- R -

- S -

- T -

- U -

- V -

- W -

- X -

- Y -

- Z -

This page was intentionally left blank.