WaterNET-CAD

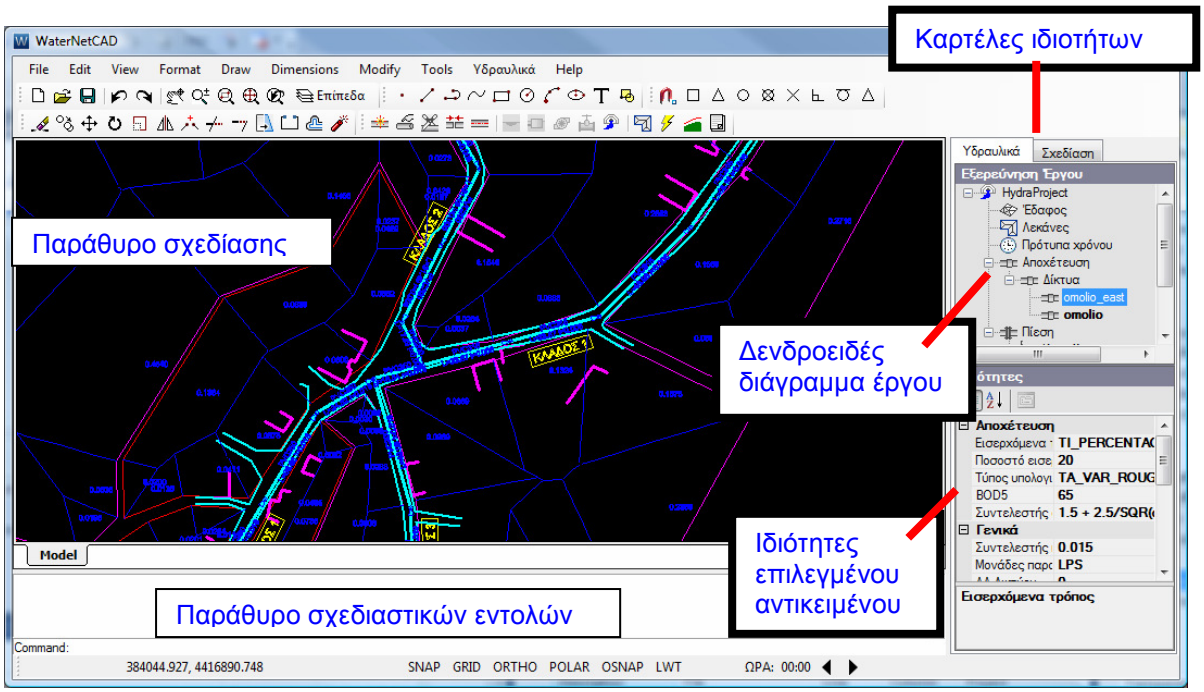
Pipe Networks Design Software

User's Guide

CONTENTS

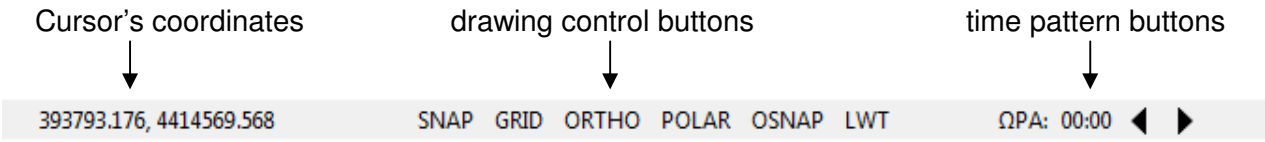
BASICS

The **WaterNET-CAD** is software product that can be used in order to create, edit and analyze any kind of sewer, water distribution, storm or vacuum sewer network facility. Include a number of powerful functions for hydraulic network analysis integrated in a rich CAD environment involving all well known (2D) CAD commands for drawing production.



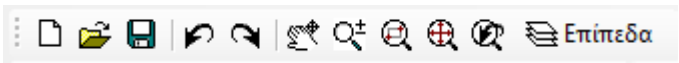
Main window

The main window of the application is divided in three parts. In the central area you will find the drawing canvas, in the area below you will find the drawing commands window and in the right side you will find the properties window. Below the commands window the program have the drawing status toolbar including: the cursor coordinates, the drawing control buttons (SNAP, GRID, ORTHO, POLAR, OSNAP, LWT) and finally the buttons controlling the current time pattern of hydraulic analysis.



Drawing status toolbar

Above the drawing canvas you will find toolbars for graphical entities creation and modification as well as toolbars for hydraulic networks design and analysis.



Basic toolbar

Create graphical entities

Snap to graphical entities

Modify graphical entities

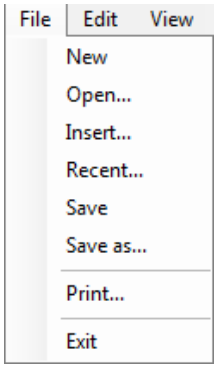
Hydraulic analysis toolbar

The CAD platform of the program have the following features:

- Graphics objects: Point, Line, Polyline, Circle, Arc, Ellipse, Rectangle, Text (single, multiline, arc), Hatch, Dimension (rotated, aligned, angular, ordinate, radial, diametic), Leader, Block Insertion, Viewport.
- Raster images of various formats: BMP, JPG, GIF, TIF, PNG, PCX, etc., and also huge ER Mapper Compressed Wavelet (ECW) and Jpeg2000 images. With TWAIN support you can acquire images directly from scanners and digital cameras..
- Named objects: Block, Layer, Linetype, Text Style, Dimension Style, Multiline style, Layout..
- Supports Windows TTF fonts and AutoCAD SHP/SHX fonts.
- Supports AutoCAD hatch patterns (PAT) and linetypes (LIN).
- Edit operations with graphic objects: Copy, Move, Rotate, Scale, Mirror, Explode, Erase, Join, Offset, Trim, Extend. Single objects can be edited by moving their control points (grips).
- Various snap modes allows you to draw/edit objects accurately without tedious calculations. Snapping modes use ID markers to enhance operator speed and vision. Superior Polar Tracking capability. Command line interface offers exact placement of objects. You can enter values for coordinates, lengths and angles from keyboard.
- Zoom capabilities allows you to view the entire drawing or any desired part of it.
- Mouse Wheel supports real-time Zoom and Pan.
- Export / import of drawings with various file format via plugins.
- Print Preview. Also you can use Paper space layouts to prepare paper sheets ready for printing.
- Clipboard copy and paste capabilities.
- Undo-Redo buffer saves 100 commands.

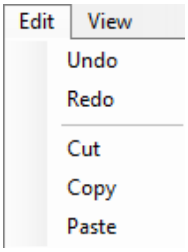
In the following pages you will find a brief description for all commands of the CAD platform.

FILE MENU



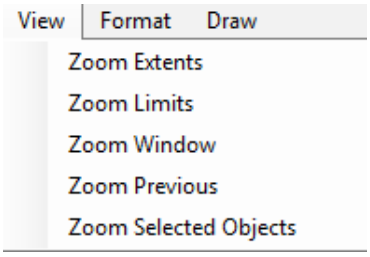
- **New.** Creates a new drawing.
- **Open...** With this command you can load an existing drawing. If together with the *.dwg file the folder contain the hydraulic *.xml file, the hydraulic projects will be loaded also.
- **Insert…** With this command you can insert a drawing file in the existing drawing.
- **Recent…** With this command you can load the recently opened files.
- **Save.** Saves the current drawing. If you have defined hydraulic networks in the project the program will create a secondary file with same name having *.xml extension. This file contains all data used in hydraulic analysis. Without this file the *.dwg file is simple drawing file.
- **Save as…** Saves the current drawing in a new directory. If you have defined hydraulic networks in the project the program will create a secondary file with same name having *.xml extension. This file contains all data used in hydraulic analysis. Without this file the *.dwg file is simple drawing file.
- **Print…** By executing this command the program shows the plot dialog.
- **Exit.** This command close the program.

EDIT MENU



- **Undo.** Undo command.
- **Redo.** Redo command.
- **Cut.** Cut graphic objectss.
- **Copy.** Copy graphic objects.
- **Paste.** Paste graphic objects.

VIEW MENU



- **Zoom Extents.** By executing this command the program zooms to display the maximum extents of all objects in the drawing.

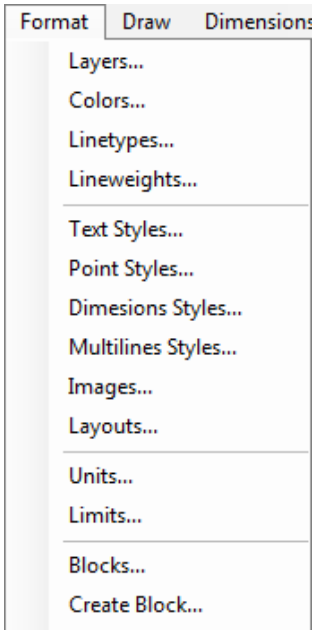
- **Zoom Limits.** (This command is not available in this version)

- **Zoom Window.** By executing this command the program zooms to display an area specified by a rectangular window. With the mouse cursor, you can define an area of the drawing to fill the main window.

- **Zoom Previous.** Zooms to display the previous view.

- **Zoom Selected Objects.** Zooms to display one or more selected objects as large as possible and in the center of the view.

FORMAT MENU



- **Layers...** If you execute this command, the program shows the *Layers properties manager* dialog box. Use layers to control the visibility of objects and to assign properties such as color, linetype or lineweight.

- **Colors...** By executing this command the program shows the *Select color* dialog box in which you can set the color of graphic objects.

- **Linetypes...**
  This command shows the *Linetype Manager* dialog in which you can set the linetype for graphic objects. Also, in this dialog you can load LIN files.

- **Lineweights...**
  This command shows the *Lineweight Settings* dialog box. This dialog box sets the current lineweight. Controls the display scale of lineweights and sets the default lineweight value for layers.

- **Text Styles...**
  If you execute this command the program shows the *Text Styles Manager* dialog box in which you can create, modify or specify text styles. In this dialog you can specify the font, height, width and other parameters for text styles.

- **Points Style...**
  This command shows the *Point style* dialog box in which you can specify the style and size of point objects.

- **Dimesions Styles...**
  This command shows the *Dimension styles* manager. In this dialog you can create new styles, set the current style and set overrides on the current dimension style.

- **Multiline Styles...**
  This command shows the *Multiline Styles* dialog box. In this dialog you can create, modify, save and load multiline styles.

- **Images...**
  This command shows the *Image Manager* dialog box. In this dialog box you can manage available images in the current drawing.

- **Layouts...**
  This command shows the print *Layout* dialog box.

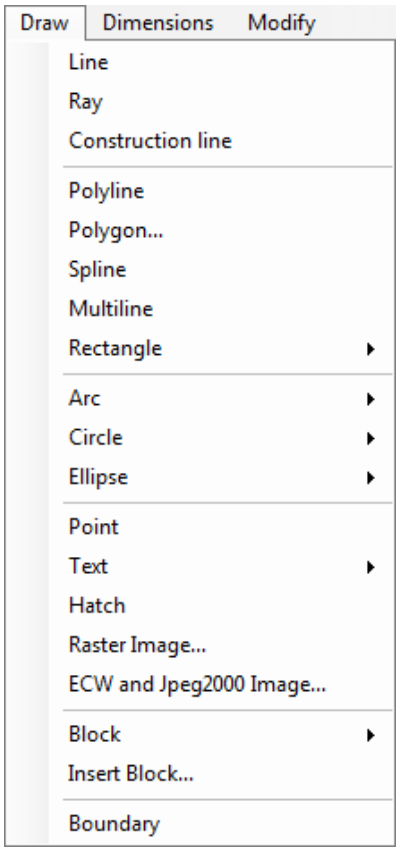
- **Units...** This command shows the *Drawing units* dialog box. In this dialog you can control, for the current drawing, the displayed precision and format for coordinates and angles.

- **Limits...** With this command you can define the canvas limits for the current drawing.

- **Blocks...** If you execute this command the program shows the *Blocks* manager dialog box in which you can manage the available blocks in the drawing.

- **Create Block...**
  By using the *Block definition* dialog box you can define blocks in the current drawing.

DRAW MENU



- **Line.** With this command, you can create a series of contiguous line segments. Each segment is a line object that can be edited separately.

- **Ray.** This command creates a linear object that starts at a point and continues to infinity.

- **Construction Line.**
  This command creates a linear object that continues to infinity.

- **Polyline.**
  With this command, you can create a 2D polyline, a single object that is composed of line and arc segments.

- **Polygon.**
  This command creates an equilateral closed polyline.

- **Spline.**
  Creates a smooth curve that passes through or near a set of fit points, or that is defined by the vertices in a control frame.

- **Multiline.**
  This command creates multiple parallel lines.

- **Rectangle.**
  With this command you can create a rectangular polyline. You have three options available: 1) by defining the diagonal cross 2) by defining one side and the height of the rectangle 3) by defining the center of the rectangle, the mid-length of the diagonal cross and the angle from the X-axis.

- **Arc.**

  With this command you can draw an arc in the current drawing. You have eleven (11) options to create the arc: by inserting 1) the start, mid and end points 2) the start, center of the circle and end point 3) the start point, center and included angle 4) the start point, center and chord length 5) the start point, end point and included angle 6) the start point, end point and tangent direction in the beginning 7) the start point, end point and radius of the arc 8) the center, start point and end point 9) the center, start point and included angle 10) the center, start point and chord length 11) contiguous arcs by defining two (2) points for each arc.

- **Circle.**

  With this command you can draw a circle in the current drawing. You have four (4) available options to define the circle: by inserting 1) the center and radius 2) the center and diameter 3) two points defining the diameter 4) three points

- **Ellipse.**

  With this command you can draw an ellipse in the current drawing. You have two (2) available options to define the ellipse: 1) major axis, minor axis 2) center, major axis, minor axis.

- **Point.**

  By using this command you can add a 3d point in the current drawing.

- **Text.**

  By executing this command, you can add a text in the current drawing. You have four (4) options to do this: 1) one line text 2) multiple lines text 3) text on arc 4) text with true type font.

- **Hatch.**

  With this command you can select the object on which the hatch will be applied. After object selection the program shows a dialog box with the available hatch patterns.

- **Raster Image….**

  If you execute this command the program shows a dialog box with which you can select an image to insert in the current drawing.

- **ECW and Jpeg2000….**

  If you execute this command the program shows a dialog box with which you can insert the ECW and Jpeg2000 file types in the current drawing.

- **Block.**

  For blocks of graphical objects, the program has three (3) available options: 1) make a block definition by selecting graphical objects in the screen. Ορισμός block γραφικών αντικειμένων 2) define insertion point 3) define attributes for the block.

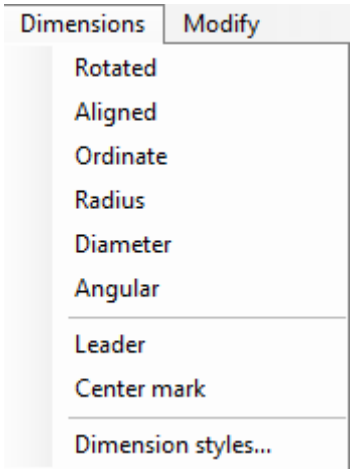
- **Insert Block...**

  By executing this command the program shows a dialog box in which you can select the block that you want to insert in the current drawing as well as the insertion point, the scale factor and the rotation angle.

- **Boundary.**

  With this command you can draws a boundary polyline.

DIMENSIONS MENU



- **Rotated.**
  This command inserts dimension in the drawing having vertical or horizontal direction.
- **Aligned.**
  This command inserts dimension in the drawing having arbitrary direction.

- **Ordinate.**
  This command inserts a level dimension in the drawing.

- **Radius.**
  This command inserts radius dimension in circles or arcs.

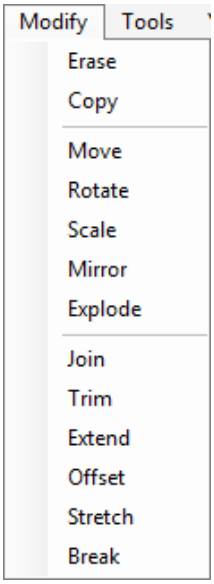
- **Diameter.**
  This command inserts diameter dimension in circles or arcs.

- **Leader.**
  This command inserts a line that connects annotation to a feature.

- **Center mark.**
  When you execute this command the program prompts you to select a circle or arc in order to mark its center point.

- **Dimensions styles.**
  (see in the Format menu)

MODIFY MENU



- **Erase.**
  If you have already objects selected, the program erases these objects. If you don't have selected objects the program prompts you to select objects in the screen in order to erase them from the current drawing.

- **Copy.**
  You can use this command to copy graphical objects. After the objects selection the program asks for the start and end points of the moving vector.

- **Move.**
  You can use this command to move graphical objects. After the objects selection the program asks for the start and end points of the moving vector.

- **Rotate.**
  This command rotates graphical objects in the drawing. After the objects selection you must insert the rotation point.

- **Scale.**
  This command enlarges or reduces the selected objects, keeping the proportions of the object the same after scaling. After the objects selection you must insert the base point for the scale operation and the corresponding scale factor.

- **Mirror.**
  This command creates a mirrored copy of the selected objects. After the object selection you must specify the mirroring axis.

- **Explode.**
  This command breaks a compound object into its components.

- **Join.**
  This command joins the endpoints of linear and curved objects to create a single object.

- **Trim.**
  Trim objects to meet the edges of other objects. To trim objects, select the boundaries. Then select the objects that you want to trim.

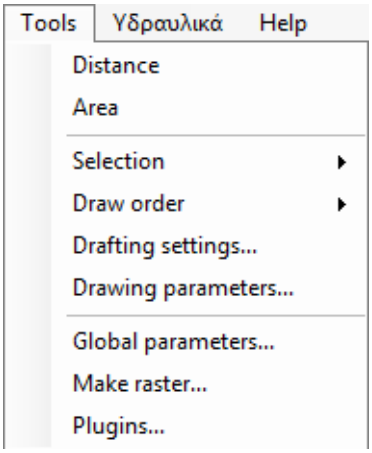
- **Extend.**
  Extents objects to meet the edges of other objects. To extend objects, first select the boundaries. Then select the objects that you want to extend.

- **Offset.**
  Offset an object to create a new object whose shape is parallel to the original object. To complete this command, first you must select the object that you want to offset and then you must specify either the trough point of the new object or the offset distance. The offset option can be changed by clicking the right mouse button and select the desired option.

- **Stretch.**
  This command stretches objects crossed by a selection window or polygon. Objects that are partially enclosed by a crossing window are stretched. Objects that are completely enclosed within the crossing window, or that are selected individually, are moved rather than stretched.

- **Break.**
  This command breaks the selected object between two points. You can create a gap between two specified points on an object, breaking it into two objects.

TOOLS MENU



- **Distance.**
  This command measures the distance between two points in the plane.

- **Area.**
  This command calculates the area of a defined region.

- **Selection.**
  Commands group for objects selection. The command group has four (4) commands: 1) Select All. Select all objects in the design window. 2) Unselect All. Deselect all the selected objects in the design window. 3) Select by Current Layer. Select all objects of the current layer. 4) Select by Polygon. Select all objects that are within a given polygon.

- **Draw order.**
  Commands group defining the draw order for the graphical objects in the screen. The commands group contains four (4) commands: 1) Bring to Front. Move the selected objects to the top of the order of objects in the drawing. 2) Send to Back. Move the selected objects to the bottom of the order of objects in the drawing. 3) Bring Above Object. Move the selected object above the specified reference

objects. 4) Send Under Object. Move the selected objects below the specified reference objects.

- **Drafting settings...**
  When you execute this command the program shows the *Drafting settings* dialog box. In this dialog you can specify the grid to snap, the polar tracking and the object snap properties.

- **Drawing parameters...**
  By executing this command the program shows the *Drawing's Parameters* dialog box including details and statistics for the current drawing.

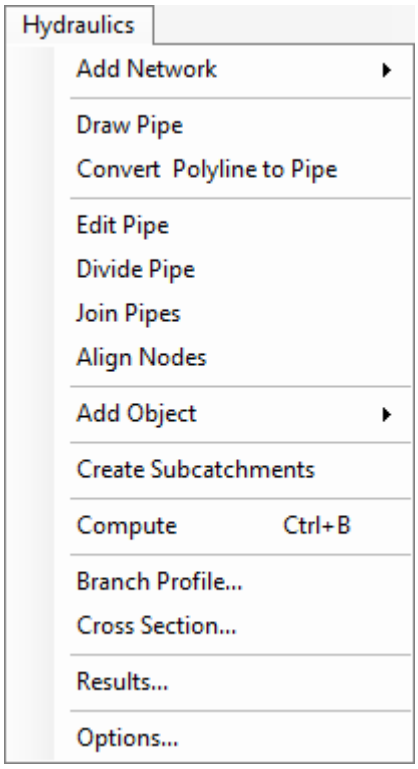
- **Global parameters...**
  By executing this command the program shows the *Global parameters* dialog box including the general settings of the CAD platform.

- **Make raster...**
  With this command you can create an image file from the current drawing in the main window.

HYDRAULICS MENU

«Hydraulics» menu

The "Hydraulics" menu contains all commands used to design a pipe network (water distribution, sewer, vacuum sewer or storm water). This menu is special designed in order to include all the basic commands for the hydraulic analysis as well as the commands that frequently used in pipe networks design and analysis.

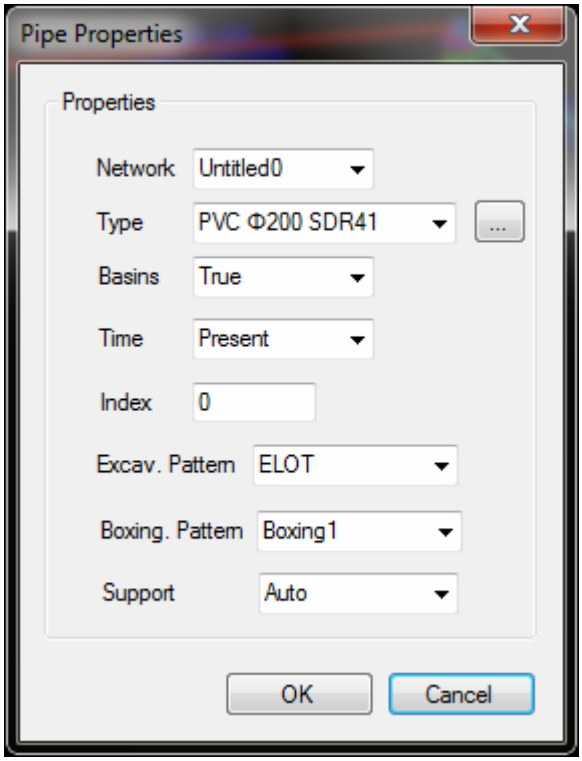


Hydraulics Menu

- ***Add Network.*** This command adds a new network in the project. The program supports four (4) kinds of network: sewerage networks, pressure networks, vacuum sewer networks, storm water networks.

To begin the design of a pipe network, first you must add a network object in the project. Alternative you can add a network in the project by right clicking in the root label of the project's explorer tree (placed in the upper right portion of the main window)

- ***Draw Pipe.*** With this command you can add a pipe branch in the current network. The pipe branch is inserted like a simple polyline specifying point-by-point its nodes. When the pipe insertion is completed you must press either the ESC key or press the right mouse button and select *Enter* from the popup menu. After that the program shows a dialog box in which you can define the properties for the inserted pipe.



The *Pipe Properties* dialog box

The *Pipe Properties* dialog box has eight fields. In the first field you can select the network in which the pipe belongs (by default filled with the current network name). Below you can find more details about the term *current layer*. In the second field you can define the pipe details (the material, external diameter and thickness). The program fills this drop down list depend on the network type.

You can modify the default contents of this drop down list by selecting the *Hydraulic Project* label in the project explorer and changing the contents of *Sew Pipes* or *Pressure Pipes* arrays depending on the kind of the current network.

The third field defines if the pipe is associated with its neighbor subcatchments. For example, if a water transfer pipe passes through a village without distributing water then you must set this parameter to *false* in order to avoid link subcatchments to the pipe.
In fourth parameter you can define the pipe age. If the pipe is existing you must set this parameter to 'Past', if the pipe is to be contracted in the future you must set this parameter to 'Future' in all other cases set this parameter to 'Present'. In the fifth field, you

define the index of the inserted pipe branch. This parameter is used only in pressure networks defining the branch order in the network.
In sixth field you can define the excavation pattern for the current pipe. The program fills this field with the active excavation pattern.
In seventh parameter you can define the boxing pattern.
Finally in eight field you set the supporting type. If you select *Auto* in this field, the program will compute the supporting type automatically. If you choose the *No* option the program will not apply support in the current pipe.
By pressing *OK* in the *Pipe Properties* dialog, the program assign identifiers to nodes and draw the pipe branch in the screen.
When you select the polyline, the program shows the polyline's grips with which you can move the nodes of the pipe. By selecting a vertex of the polyline and press the right mouse button, the program shows a popup menu having commands for inserting and deleting vertex.

In the case of pressure networks, when you select the pipe's polyline, the program colors the total branch path which can have more than one polylines. The total branch path marked with cyan color and the current selected pipe segment with red.

When you select a pipe polyline the program shows the properties for the current selected pipe segment. In the same way, in order to display the node's properties you must select the circle surrounding the node.

- ***Convert Polyline to Pipe:***
  Convert a 2D polyline to pipe. When you execute the command the program prompts you to select an object in the drawing window. After selecting the polyline the program shows the Pipe properties dialog. By pressing the 'OK' button the program complete the pipe creation.

Note: The current network is marked with bold font under the *Networks* label in the *Project Explorer*. You can turn a network to current by selecting a pipe or node of the specific network. Also, you can turn a network to current by clicking on the name label of the network and executing the corresponding command from the popup menu showed by the program.

- ***Edit Pipe:***
  By executing this command the program prompts you to select a pipe from the drawing window (the mouse cursor changes to small square). After selecting the pipe, the program shows the *Pipe Properties* form filled with the pipe's properties..

- ***Divide Pipe:*** Divides a pipe into two new pipes having the properties of the parent pipe. When you execute this command, the program prompts you to select objects in the drawing window. The command is completed in two steps. First you must select the pipe and then the node (more precisely the circle marking the node) in which you want to divide the pipe.

- ***Join Pipes:*** Joins two pipes into one. When you execute this command, the program prompts you to select objects in the drawing window. The command is completed in two steps. First you must select the 1st pipe and then the 2nd pipe. The order in which you select the polylines determines the nodes order of the new pipe. If the selected pipes have different properties (diameter, material,

thickness etc.) then the new pipe will get the properties of the first selected pipe.

- ***Align Nodes:***

  Align all nodes between two selected nodes according to the line defined by the two selected nodes. When you execute this command the program prompts you to select the nodes in the drawing window. The command is completed in three steps. First you must select the 1st node then the 2nd node and finally you must press the right mouse button in order to complete the command.

- ***Add Object:***

  Sewer networks, Storm water networks:

  Outlet: With this command, you can define the outlet node for the current network. When you start the pipe insertion in a network, the program set the first inserted node as the outlet point of the network and draws an arrow block on this point. If you want to change the default outlet point for the network, you can use this command. By executing this command the program prompts you to select the outlet node (more precisely the circle surrounding the node).

  Vacuum sewer networks:

  Valve: With this command you can add a vacuum valve in a network node. When you execute this command the program prompts you to select the node (the circle surrounding the node) on which the program will add the vacuum valve.

  Pump: With this command you can add a vacuum pump in a network node. When you start the pipe insertion in a network, the program assumes that the vacuum pump of the network is placed on the first inserted node and draws a relative block on this point.
  When you execute this command the program prompts you to select the node (the circle surrounding the node) on which the vacuum pump station will be placed.

  Pressure networks:

  Reservoir: 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.
  By executing this command the program prompts you to select the node (the circle surrounding the node) on which the reservoir will be placed.

  Tanks: Tanks are nodes with storage capacity, where the volume of stored water can vary with time during a simulation. The primary input properties are: i) the offset distance between the foundation plate of the tank and the ground level ii) the initial, minimum and maximum water levels iii) the diameter of the tank. If the tank has rectangular shape then you must specify a diameter for the tank resulting area equal with the rectangular shape.
  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.

When you execute this command the program prompts you to select the node (the circle surrounding the node) on which the tank will be placed.

Valves: Valves are links that limit the pressure of flow at a specific point in the network. If you execute this command, the program prompts you to select the pipe segment on which the valve will be placed.

There are seven different valve types:

1) Pressure Reducing Valves (PRV). The PRV limit the pressure at a point in the pipe network. The program computes in which of three states a PRV can be in: i) partially opened (i.e., active) to achieve its pressure setting on its downstream side when the upstream pressure is above the setting. ii) fully open if the upstream pressure is below the setting. iii) closed if the pressure on the downstream side exceeds that on the upstream side (i.e., reverse flow is not allowed).
2) Pressure Sustaining Valves (PSV). The PSV 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: i) partially opened (i.e., active) to maintain its pressure setting on its upstream side when the downstream pressure is below this value. ii) fully open if the downstream pressure is above the setting iii) closed if the pressure on the downstream side exceeds that on the upstream side (i.e., reverse flow is not allowed).
3) Pressure Break Valves (PBV). The PBV force a specified pressure loss to occur across the valve. Flow through the valve can be in either direction. PBVs are not true physical devices but can be used to model situations where a particular pressure drop is known to exist.
4) Flow Control Valves, (FCV). The FCV 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).
5) Throttle Control Valve, (TCV). The TCV 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.
6) General Purpose Valves, (GPV). The GPV 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.
7) Non Return Valves, (CV). The CV (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 segment in which they are placed.

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

- The pressure reduce valves (PRV), the pressure sustaining valve (PSV) and the flow control valve (FCV)

cannot be directly connected to a reservoir or tank (use a length of pipe to separate the two).
- The pressure reducing valves (PRVs) cannot share the same downstream node or be linked in series.
- Two pressure sustaining valves (PSVs) cannot share the same upstream node or be linked in series.
- A pressure sustaining valve cannot be connected to the downstream node of a pressure reducing valve (PRV).

Set begin node: Every pressure network has a starting point when the program gives identifies to the nodes. By default the program set the first inserted node as the begin point of the network and draws an arrow block on this point.
When you execute this command the program prompts you to select a node (the circle surrounding the node) that is to be the starting point of the network. The begin node affects only the way in which the program gives ids to the nodes.

When you add hydraulic objects in the drawing the program draws the corresponding blocks on the given positions. The program reads these blocks from the blocks.dwg drawing which can be fount in the program's folder. You can change, if you prefer, the default blocks defined for the hydraulics objects in the blocks.dwg.
You can erase a hydraulic object from the project as if it were a simple graphic object. Thus, If you want to delete an hydraulic object from a network simply select the corresponding block and then press the *Del* key or press the right mouse button and select the command *Erase* or execute the *Modify->Erase* command.

- ***Create Subcatchments:***

This command divides the subcatchment polygons into smaller parts according to the bisector criterion (like roofs). The program computes and draws all the edges separating the above parts as well as the texts labeling the area or the corresponding number of citizen of each part (depending on the option selected in the *Subcatchments* object).
After processing the subcatchment polygons, the program links these subcatchment parts to the neighbor pipes or nodes and computes the new hydraulic properties for all pipes in the network.
The layer of subcatchment polygons as well as details about the population densities can be defined in the *Subcatchment* object listed in the project's explorer tree. For more details about this object see the section the *Subcatchment* object.

- ***Compute:***

When you execute this command the program compute the geometric definition, the hydraulic properties and the bill of quantities for all networks in the project.

- ***Branch Profile…:***

With this command you can select the pipe branch that you want to draw in the secondary drawing window. First you must select a pipe from the network and then you must press the right mouse button.
If you want to show a specific path in a network you must select the start and end nodes of the path and then press the right mouse button to complete the command. Note that in this case the two selected nodes must be in the same branch.

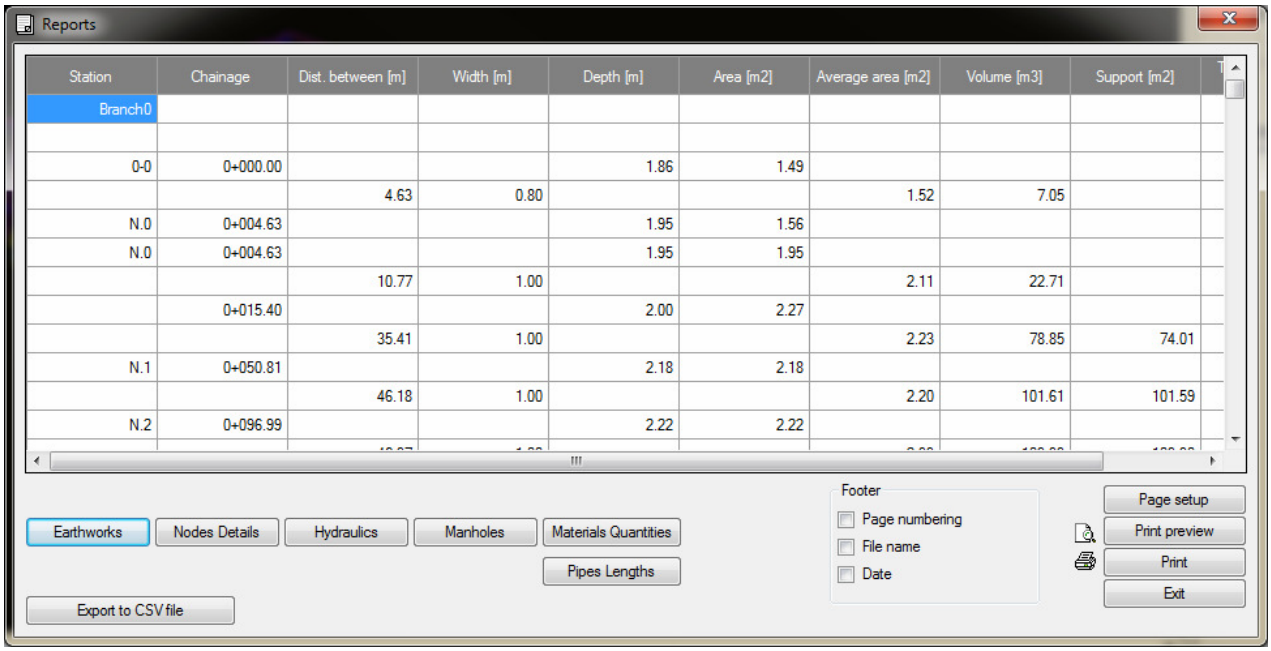
Thus, the program shows the pipe branch if you select only one pipe and a specific part of the profile by selecting two nodes of the branch.
The profile window is a secondary drawing window including a number of commands that can be used in order to manipulate the vertical alignment of the branch. Any modification maiden in the profile window modifies also the elevation data of nodes stored in the network's database. When the profile design is completed you can save the drawing directly to a DWG file. In the "Profile window" section you will find more details about the *Profile Window* and the commands included in this window.

- ***Cross section…:***

When you execute this command the program shows in the drawing window a temporary line with the current branch chainage waiting to select a specific section on the branch.
By clicking on a specific position, the program displays the *Sections window* showing the configuration of the trench in the selected position.

- ***Reports…:***

If you execute this command the program shows a dialog box including the reports for the current network.



Reports dialog box

The *Reports* dialog box includes six report tables (Earthworks, Nodes Details, Hydr. Calc., Manholes, Materials Quantities, Pipes Lengths) showed by clicking the corresponding button on the down-left part of the form.
Earthworks*:* If you execute this command, the program shows the earthworks table analyzing the earthworks quantities branch-by-branch. After the final node of each branch, the table shows the total quantities of the corresponding branch and in the end of the table the total quantities of all branches in the current network.
Nodes details*:* If you execute this command the program shows all branches included in the network. For each node the table contains the chainage, X, Y, Z coordinates as well as the ground elevation in the corresponding position.

***Hydraulics**:* When you execute this command the program shows the hydraulic properties table for the current network. The format of this table changes depending on the network type.

Thus, in the case of *sewerage (sanitary) network* the hydraulics table includes, for the nodes: the name, the elevation of the manhole bottom, the ground elevation and the flow elevation inside the manhole while for the pipe segments: the subcatchment areas, the total subcatchment areas, the population, the waste-water flow, the infiltration flow, the total flow, the external diameter of the pipe, the internal diameter, the slope, the loading ratio, the flow velocity, the flow velocity for Q/10, the length of the pipe segment, the flow when the pipe is full (Q0), the velocity when the pipe is full, the Q/Q0 ratio, the V/V0 ratio, the flow height, the critical flow height, the ZPomeroy value and the concentration (mg/l) of H_2S.

In the case of *storm water network* the hydraulics table includes: the name, the elevation of the manhole bottom, the ground elevation and the flow elevation inside the manhole while for the pipe segments: the subcatchment areas, the total subcatchment areas, the storm water flow, the external diameter of the pipe, the internal diameter, the slope, the loading ratio, the flow velocity, the flow velocity for Q/10, the length of the pipe segment, the flow when the pipe is full (Q0), the velocity when the pipe is full, the Q/Q0 ratio, the V/V0 ratio, the flow height and the critical flow height.

In the case of *pressure network* the table includes: details about the network connectivity, the pressure on nodes as well as the lengths, internal diameter, flow, velocity and loss height in the pipe segments

In the case of *vacuum networks* the table is created according to the ATV-DVWK-A 116 German norm and includes: details about the network connectivity, the population of each pipe segment, the total population till section end, the local LWV at section end, the medium LWV, the nominal width section, the section length, the maximum static pressure in section and the maximum static pressure height difference till section end. Note that the section end is the part of the pipe segment downstream (nearest to the vacuum pump).

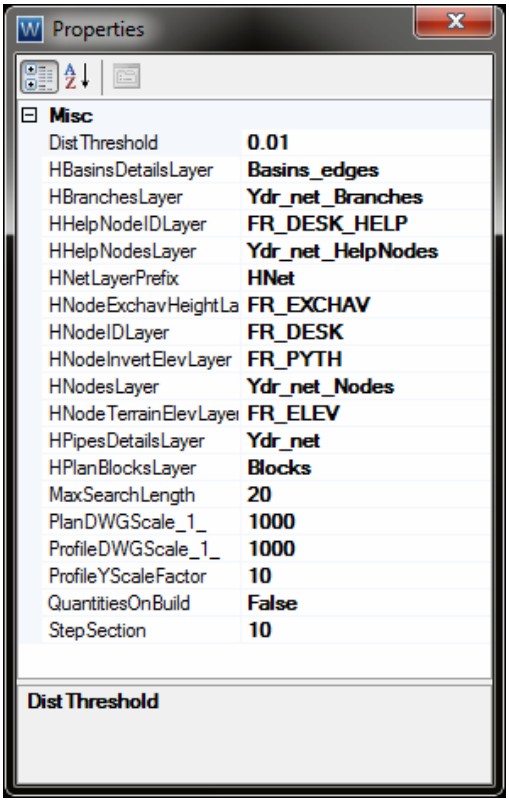
***Manholes**:* When you execute this command, the program shows the *Manholes* table. This table includes details about manhole construction quantities like concrete volumes, formworks areas, ground treatment volumes etc.

***Materials Quantities**:* If you execute this command, the program shows the *Materials Quantities* table. This table includes details about the material quantities used in the trench restoration.

***Pipes Lengths**:* This command shows the table including the total pipe lengths for each pipe type used in the current network.

- ***Options:***

This command shows the *Options* dialog box which includes the global parameters of the hydraulic module of the program.



Global parameters

DistThreshold: This property defines the dist threshold above which the program considers two or more nodes as one. By default the program sets this value equal with 0.01.

HBasinsDetailsLayer: This property defines the name of layer in which the program draws the subcatchment edges.

HBranchesLayer: This property defines the name of layer in which the program draws the branch names.

HHelpNodeIDLayer: This property defines the name of layer in which the program draws the help (intermediate) nodes identifiers.

HHelpNodesLayer: This property defines the name of layer in which the program draws the circles surrounding the help (intermediate) nodes.

HNetLayerPrefix: This property defines the layer in which the program draws the pipes of all networks in the project.

Note that you must be careful to avoid change the properties (like color, lineweight etc) of the objects included in this layer because the program uses internally this data in order to create the networks.

HNodeExchavHeightLayer: This property defines the layer in which the program draws the excavation depth texts.

HNodeIDLayer: This property defines the layer in which program draws the node identifier text.

HNodeInvertElevLayer: This property defines the layer in which program draws the node invert elevation texts.

HNodesLayer: This property defines the layer in which program draws the circles surrounding the nodes.

HNodeTerrainElevLayer: This property defines the layer in which program draws the node terrain elevation texts.

HPipesDetailsLayer: This property defines the layer in which program draws the pipe segment texts.

HPlanBlocksLayer: This property defines the layer in which program draws the blocks used in hydraulic projects.

MaxSearchLength: This property defines the maximum allowed searching distance during linking the subcatchment areas to pipe segments. This value must be at least equal with the semi-width of widest road in the region.

PlanDWGScale_1_: Defines the scale factor for hydraulic networks in the plan drawing.

ProfileDWGScale_1_: Defines the scale factor for the pipe profile drawing.

ProfileYScaleFactor: Defines the height/length ratio in the profile window.

QuantitiesOnBuild: If this property is equal with *true* then when you execute the *Hydraulics->Compute* command, the program will calculate also the quantities for all networks in the project.

StepSection: Defines the length step between consecutive cross sections. When you press the *Previous* or *Next* buttons in the *Sections* window (or if you request to show all sections in a specific branch) the program will project sections per constant distance, equal with this property.

HELP MENU

- ***About…:***

  With this command the program shows a form including details about the version and the author of the program.

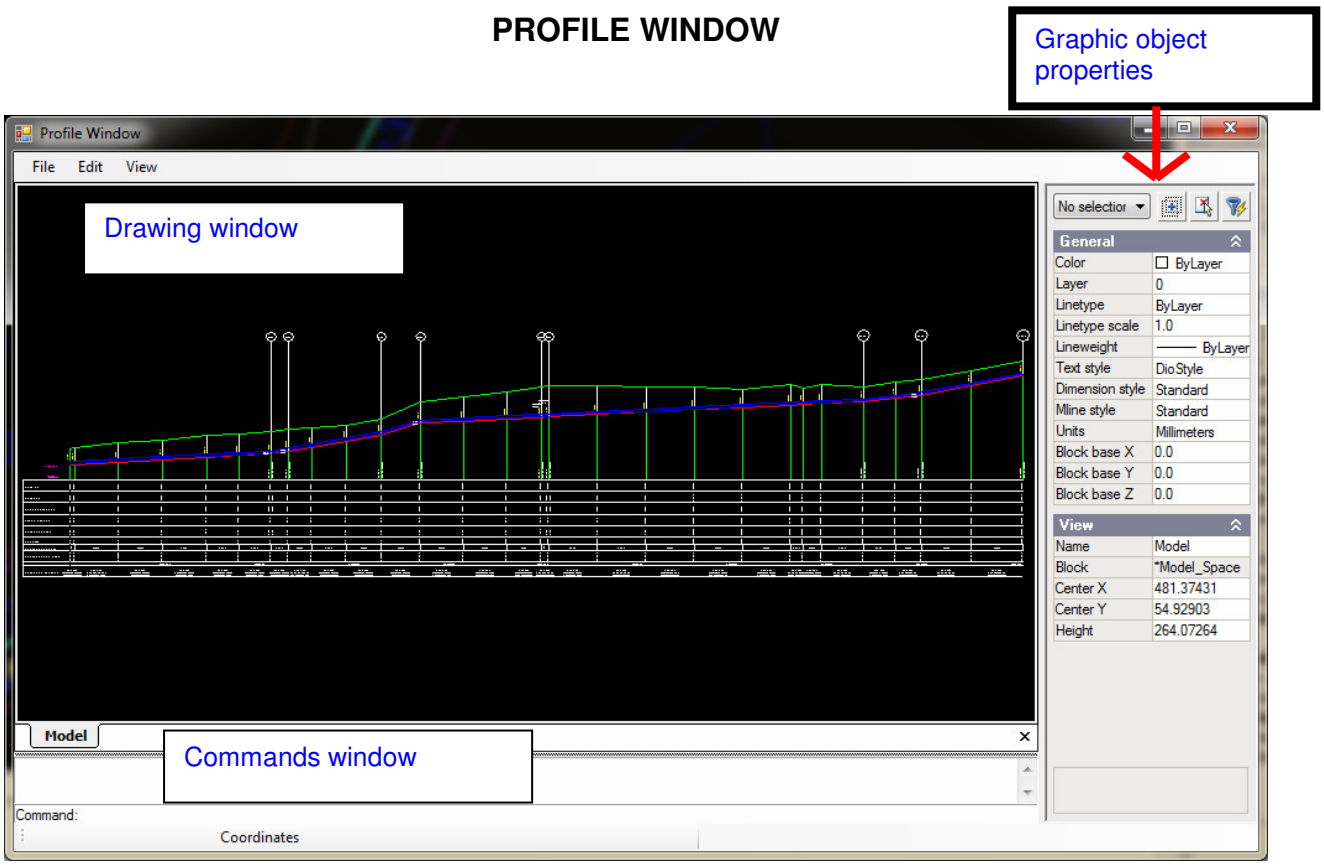
- ***Help:***

  This command shows the help dialog box.

- ***Check for Updates…:***

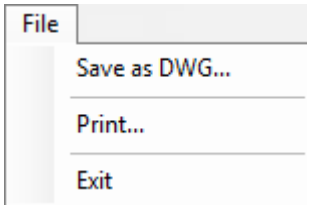
  When you execute this command the program check, via internet connection, if there is available update for the software. This command can be successfully completed if your PC is connected to internet.
  When the update process is completed the program restarts automatically.

PROFILE WINDOW



Profile window

File menu



Menu «File»

- **Save as DWG…**

  With this command you can save the current drawing in a DWG file. If you execute this command, the program shows a dialog box in which you can define the file name and the saving path for the drawing.

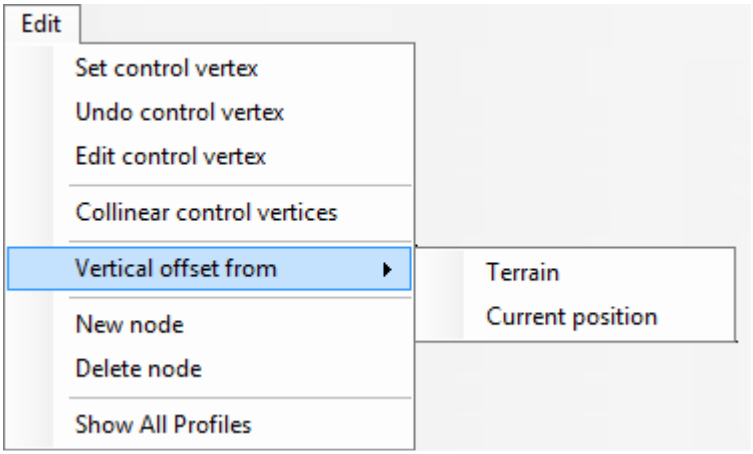
- **Print**

  By executing this command the program shows the plot dialog.

- **Exit**

  This command closes the *Profile* window.

Edit menu



Edit menu

- **Set control vertex**

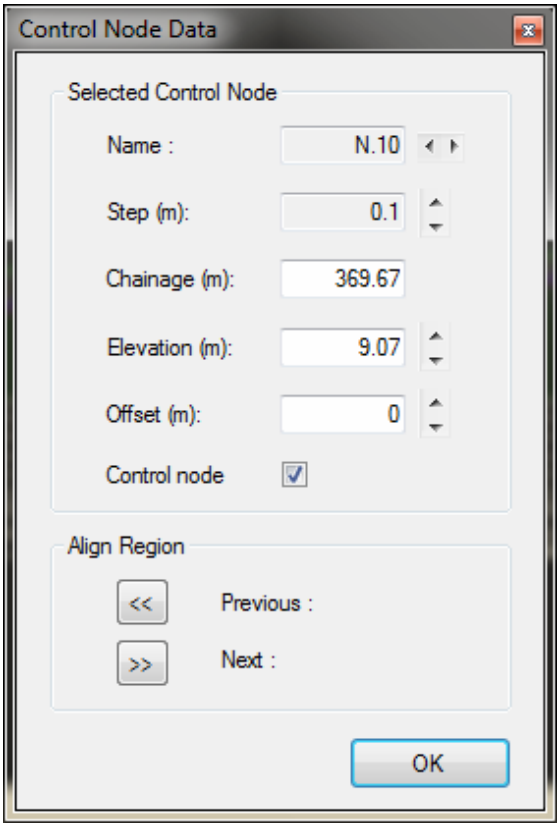
  With this command you can convert a simple node to a control node. The control node has its own certain elevation and all nodes between control nodes take elevation by interpolating the elevation values of previous and next control nodes. When you execute this command the program prompts you to select the node's vertical line in the profile drawing. By default the program set the first and last points of each pipe as control nodes.

- **Undo control vertex**

  With this command you can convert a control node back to simple node. By executing this command the program prompts you to select the node's vertical line. To complete the command you must press the right mouse button. You can make this conversion to a series of control nodes by selecting the first and last control nodes. In the same way, to complete the command, you must press the right mouse button.
  To define the vertical alignment of the pipe, the program needs to have control nodes at its starting points. Thus, the program does not allow you to undo these control nodes.

- **Edit control vertex**

  With this command, you can edit the elevation of a control node. If you execute this command the program prompts you to select the node's vertical line. When you select the line the program shows the *Control Node Data* dialog box and paints the selected control node with orange color.



Control Node Data dialog box

The first field (with label *Name*) contains the node's identifier. By pressing the left and right arrow buttons, in the right side, you can go to the previous and next control nodes.
In the second field (with label *Step*) you can define the modification step.
In the third field (with label *Chainage*), the program shows the chainage of the current selected control node.
In the fourth field (with label *Elevation*), the program shows the elevation value.
In the fifth field (with label *Offset*) you can define a vertical discontinuity for the pipe segment that starts from the selected control node.
By using this feature you can define a **back-drop manhole** to gravity driven pipe.
If you uncheck the sixth field, the program converts the current selected control node to simple node.
Finally, in the *Alignment Region* you can define the previous and next alignment control nodes. All nodes (simple or control) within this region are aligned according to the straight lines defined by these control nodes. By pressing the "<<" button the *Control Node Data* dialog is hiding and the program prompts you to select the previous alignment control node. In the same way, by pressing the ">>" button the *Control Node Data* dialog is hiding and the program prompts you to select last alignment control node.

- **Collinear control vertices**

  With this command, you can put a series of nodes on to straight line defined by two selected control nodes. If you execute this command, the program prompts you to select the first and last control nodes defining the straight line. To complete the command you must press the right mouse button.

- **Vertical offset from**

Terrain

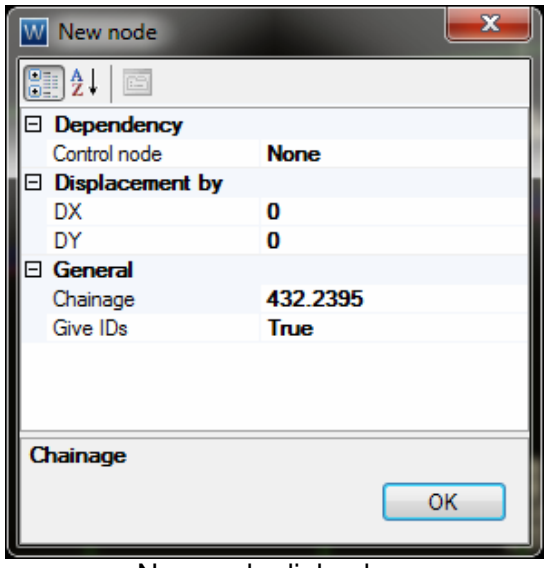
When you execute this command, the program prompts you to select graphic objects in the screen. Then, you must select the first and last nodes that you want to offset from terrain. By pressing the right mouse button, the program shows an input box waiting from you to define the offset distance. When you press the *OK* button all nodes between the two selected nodes are placed in the specified offset distance from terrain.

Current position

When you execute this command, the program prompts you to select graphic objects in the screen. Then, you must select the first and last nodes that you want to offset from the current position. By pressing the right mouse button, the program shows an input box waiting from you to define the offset distance. When you press the *OK* button all nodes between the two selected nodes are placed in the specified offset distance from the current position.

- **New node**

With this command you can insert a new node in a selected position in the profile drawing. By executing this command the program draws temporary lines between the previous and next control nodes.
By pressing the right mouse click the program shows the *New node* dialog box.



New node dialog box

In the *Dependency* group you can define the “relatively to” point for the new created node. There are three options for this property: *None*: insert without dependency. *Previous*: insert relatively to previous control node. The new node will be inserted in DX, DY distances from the previous control node. *Next*: insert relatively to next control node. The new node will be inserted in DX, DY distances from the next control node. In the *General* group you can define the exact chainage in which the new node will be inserted and if the new node will have a normal identifier or not.

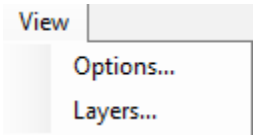
- **Delete node**

With this command you can erase a node from the network (the node is erased from the plan drawing also).
This command must be used carefully because it is possible to change the plan definition also and you don't have the ability to undo.

- **Show all profiles**

When you execute this command the program shows the profile window filled with the profiles of all branches in the network. The order in which the program shows the profiles follows the numbering of nodes in the plan drawing. As with the single branch profile, you can save the current drawing in a DWG file.

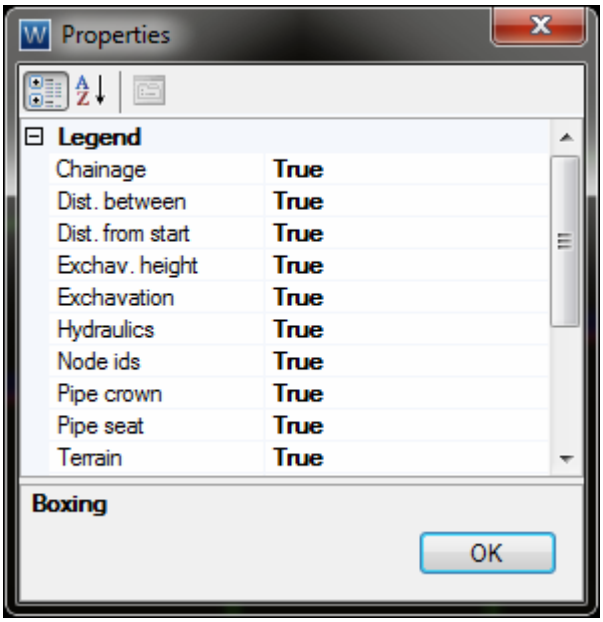
Menu View.



View menu

- **Options…**

When you execute this command the program shows the *Display Properties* dialog box. In the legend group you can select which of the available rows will be showed in the legend of the profile drawing.



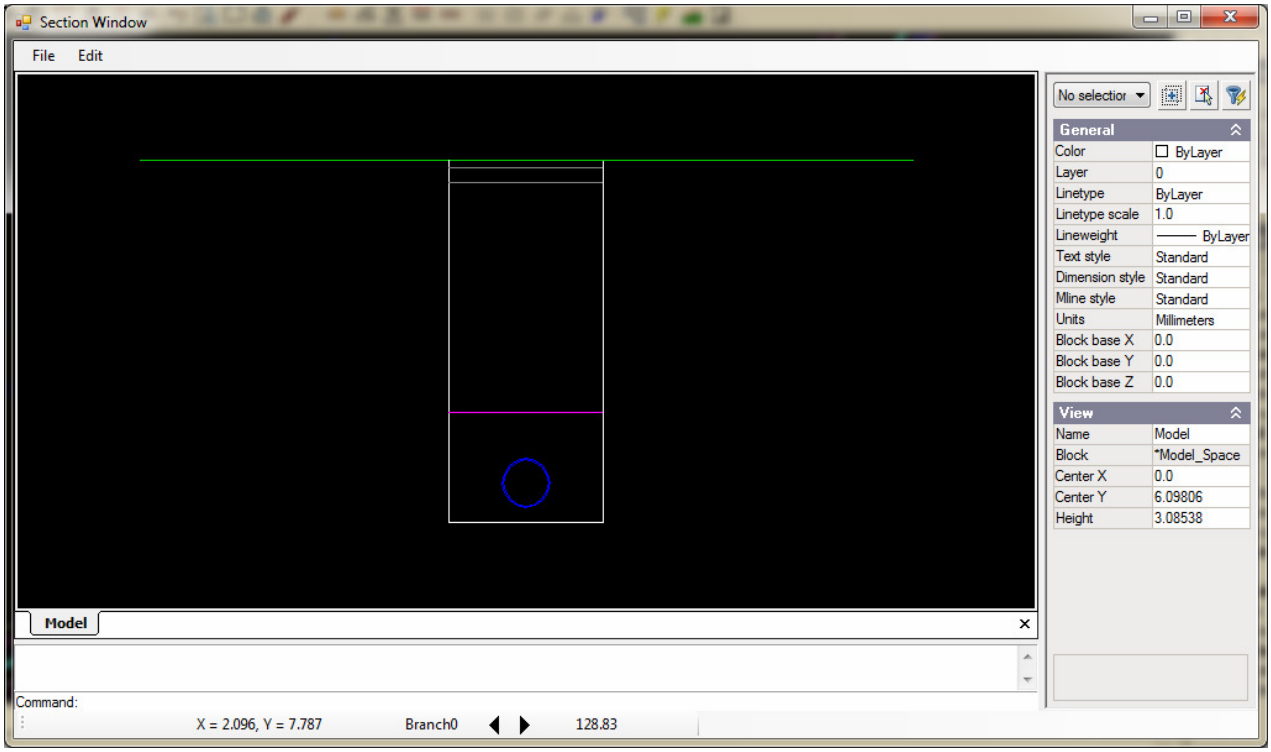
Legend properties dialog box

In the Plines group you can select which of the available poly-lines will be showed in the profile drawing.

- **Layers…**

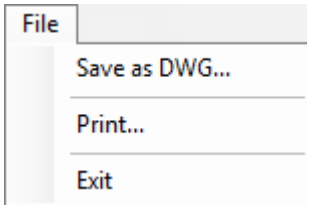
When you execute this command the program shows the layers dialog box of the profile drawing.

CROSS SECTION WINDOW



Cross Section window

Menu "File".



Menu «File»

- **Save as DWG…**

  With this command you can save the current drawing in a DWG file. If you execute this command, the program shows a dialog box in which you can define the file name and the saving path for the drawing.

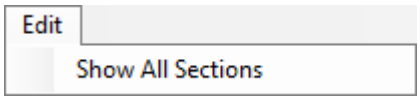
- **Print**

  By executing this command the program shows the plot dialog.

- **Exit**

  This command closes the *Section* window.

Menu "Edit".



Menu «Edit»

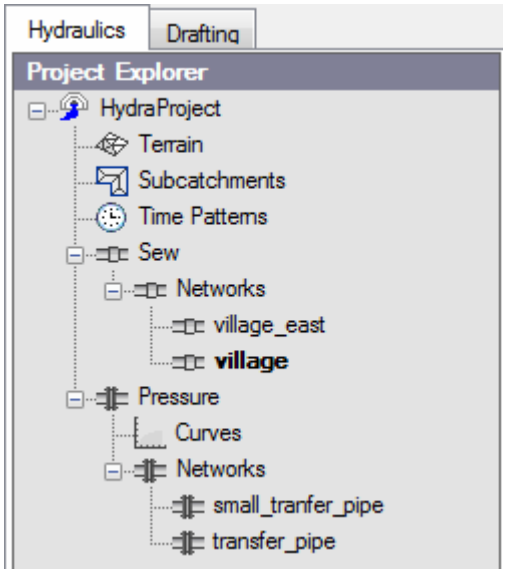
- **Show all sections**

  If you execute this command the program shows all sections included in the current branch. The chainage interval is taken from *StepSection* property of the global parameters of the program.

PROJECT PROPERTIES

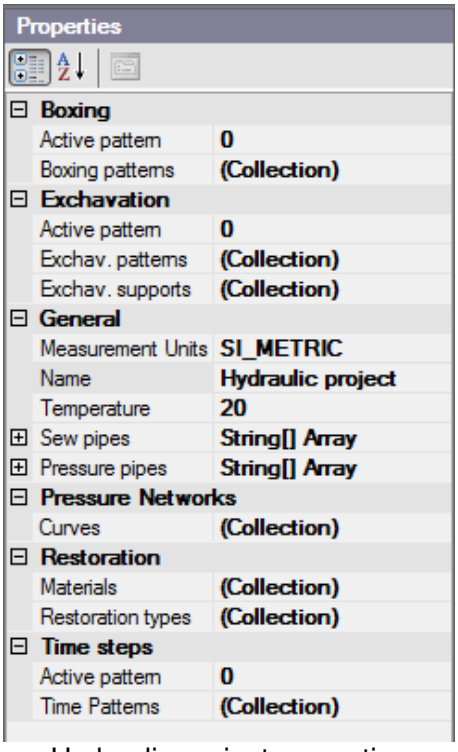
"*Hydraulics*" tab

The hydraulic properties tab is divided in two parts. On top, with label named "Project explorer", you will find a tree control including all objects of the hydraulic project.



Hydraulics tab

When you select an object in the *Project Explorer* tree the program shows on the bottom of the *Hydraulics* tab the corresponding data grid view control filled with the properties of the selected object.
Additionally when you select a node or a pipe segment in the plan drawing the program fills the data grid view control with the corresponding properties of the selected object. Note that the program draws the nodes in the plan drawing as circles.



Hydraulic project properties

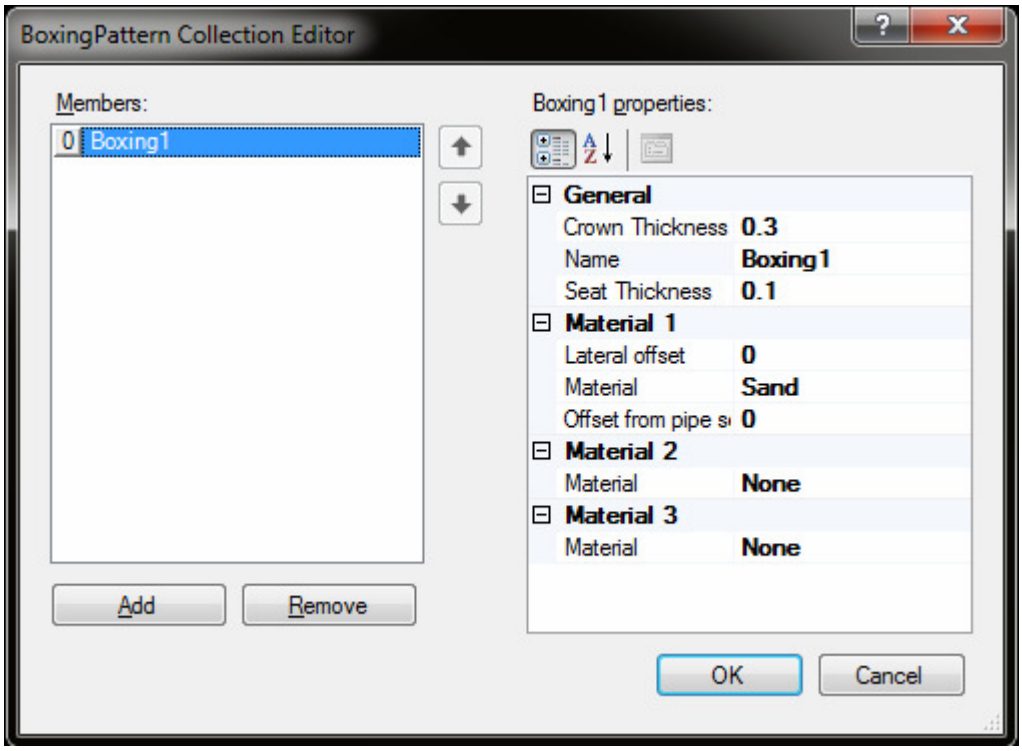
The properties of the hydraulic object (root in the tree control) are the following:

Active Boxing Pattern:

In this property you can define the active index for the boxing pattern collection. By default the program sets this property to zero (0). Thus, when you insert a new pipe in the network the *Boxing Pattern* field of the *Pipe Properties* dialog box is filled with the first boxing pattern of the boxing patterns collection.

Boxing patterns:

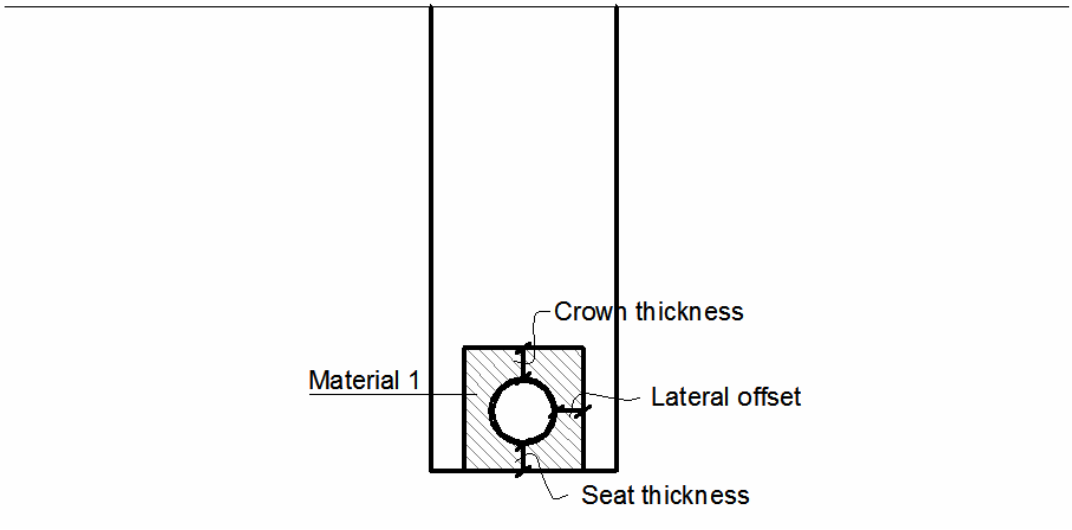
This collection contains all the available boxing patterns in the project. The boxing pattern defines the pipe boxing configuration in the trench.



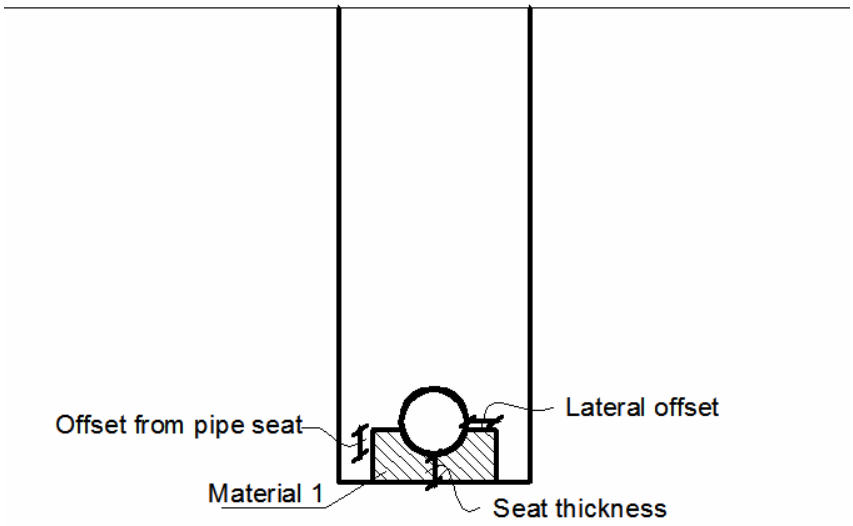
Boxing pattern dialog box

For each boxing pattern you can define: i) the name of the pattern ii) the crown thickness iii) the seat thickness.
Στις παραμέτρους του υλικού 1 ορίζουμε: i) το πάχος της άνω στρώσης έδρασης (αν υπάρχει). ii) το πλευρικό offset του υλικού 1 σε σχέση με τον αγωγό (αν υπάρχει). iii) το όνομα του υλικού.
If the field "*Offset from pipe seat"* is equal to zero (0) and the other two materials are not used then the material 1 will be extended until the top of the crown thickness.
If the field "Lateral offset" is equal to zero (0) and the other two materials are not used then the material 1 will be extended until the trench limit.

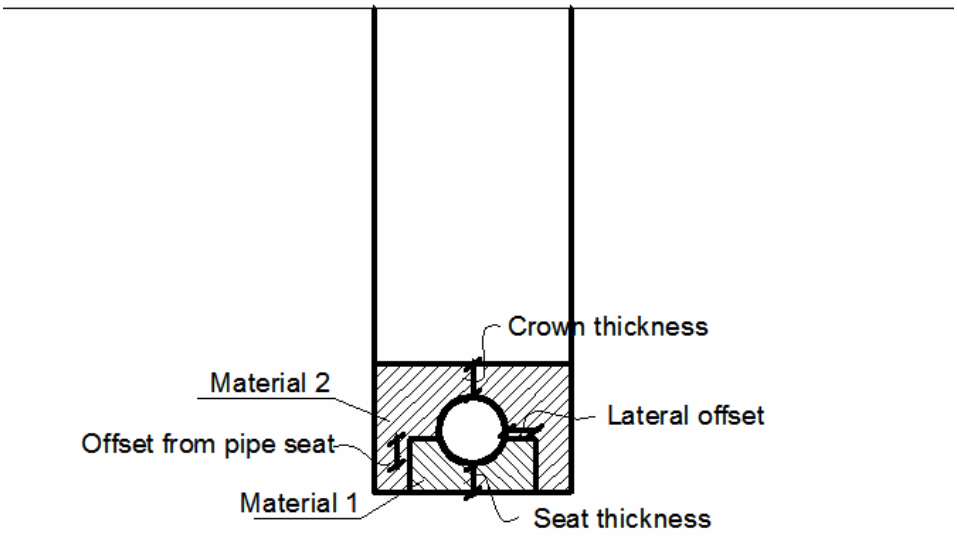
In the following figures you can see the meaning of the above described properties of the boxing configuration.



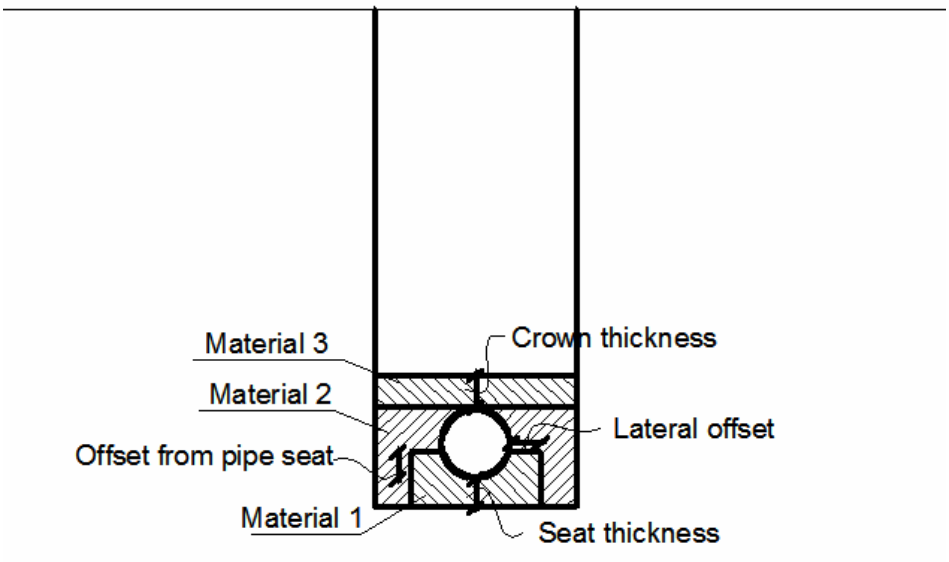
Only material 1 without value in the "Offset from pipe seat" property



Only material 1 with value in the "Offset from pipe seat" property



Materilas 1 & 2



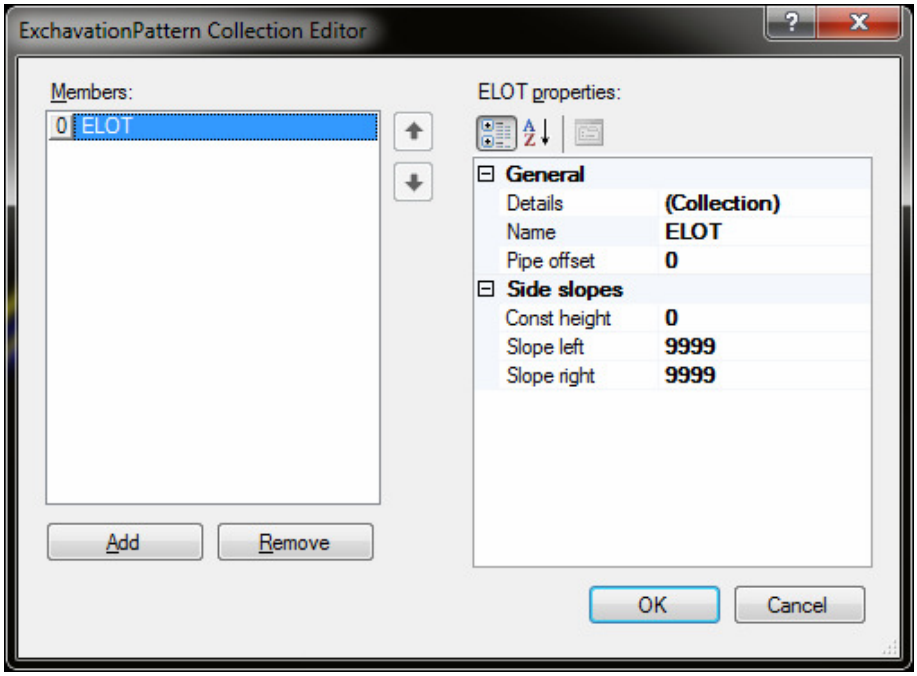
Materials 1 & 2 & 3

Active excavation pattern:

In this property you can define the active index for the excavation pattern collection. By default the program sets this property to zero (0). Thus, when you insert a new pipe in the network the *Excavation Pattern* field of the *Pipe Properties* dialog box is filled with the first excavation pattern of the excavation patterns collection.

Excavation patterns:

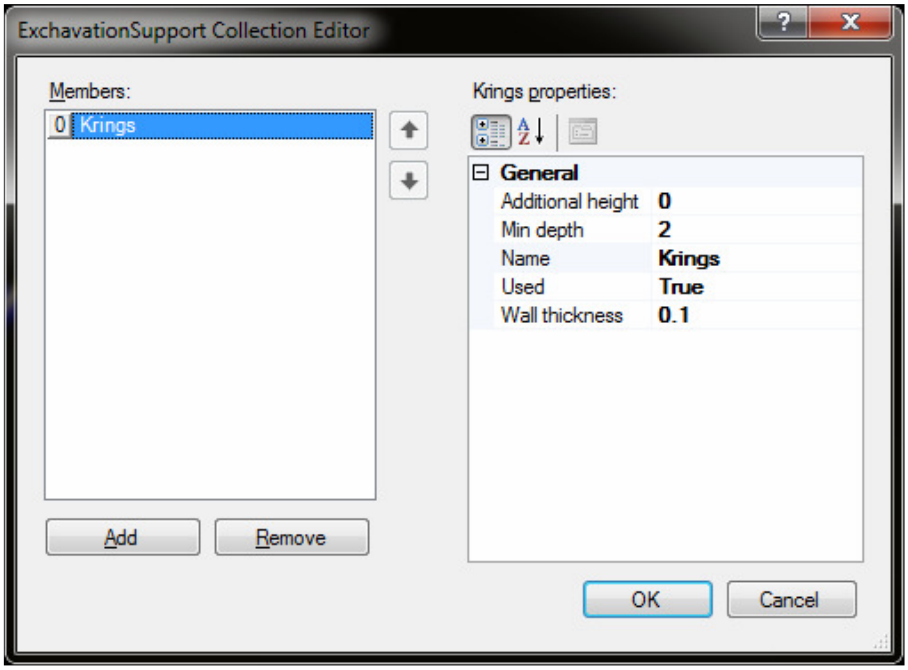
This collection contains all the available excavation patterns in the project. The boxing pattern defines the trench width escalation depending on excavation depth and pipe diameter. Additionally in the excavation patterns you can insert details about the side-slopes geometric definition.



Excavation patterns collection.

In the general group you can define i) details about the trench width escalation depending on the excavation depth and pipe diameter ii) the name of the excavation pattern iii) the offset of pipe axis from the midpoint of the trench. In the side-slopes group you can define the grade of side-slopes as well as the height in which the side-slopes will be vertical. If you insert big values (like 9999) in the slope fields, the program creates vertical side-slopes.

Excavation supports:



Excavation support collection

For each excavation support you can define i) the additional height above trench crown level until which the support will be extended ii) the minimum depth above which the program will apply the support iii) the support name iv) if the support will be available for use or not v) the support wall thickness

Measurement units: In this property you can define the active unit system for the project. There are two (2) available option for this field i) SI_METRIC. This is the international system of units. ii) US_CUSTOMARY. This is the United States system of measurements (most units of this system are virtually identical to the British imperial units).

Name: The name of the project.

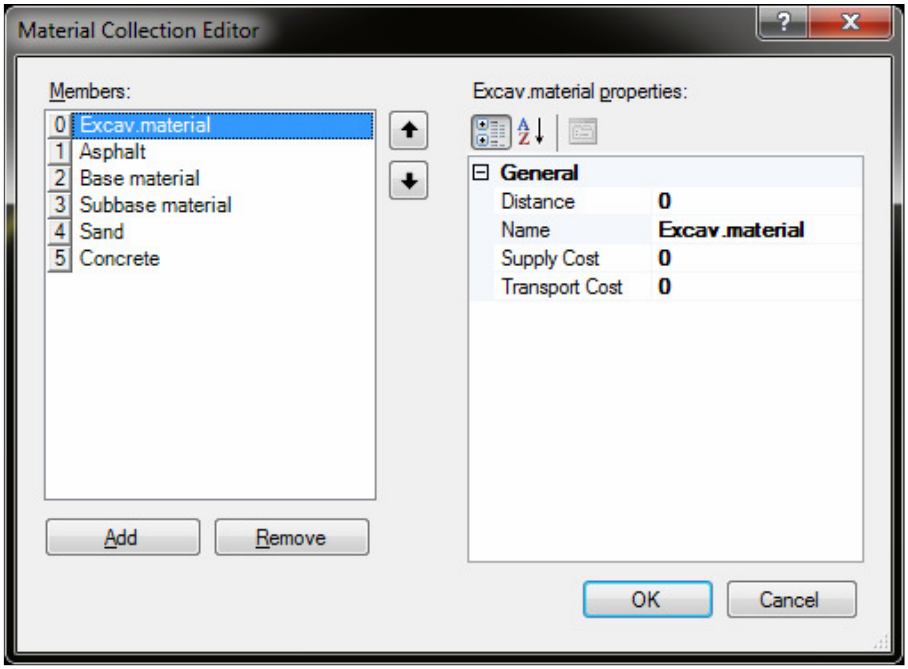
Temperature: The temperature in the project's region.

Sew Pipes: Array with the available pipes used in the gravity driven networks.

Pressure Pipes: Array with the available pipes used in pressure networks.

Pressure networks curves: Array with the available curves (functions) used in pressure networks.

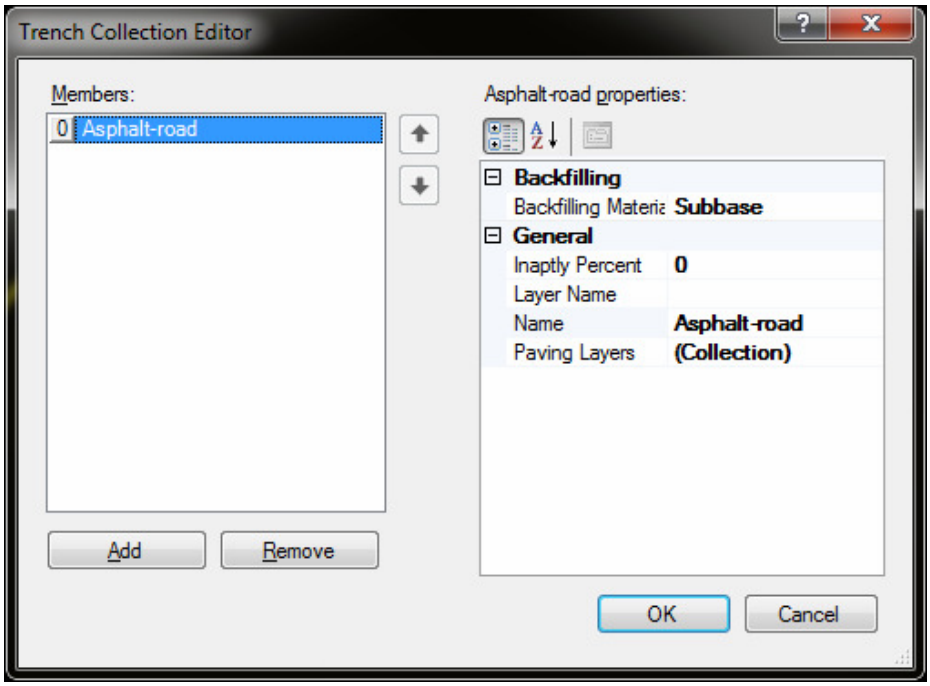
Materials: Collection with available restoration materials.



Restoration materials collection.

The *Distance, Supply Cost* and *Transport cost* are not available in the current version of the program.

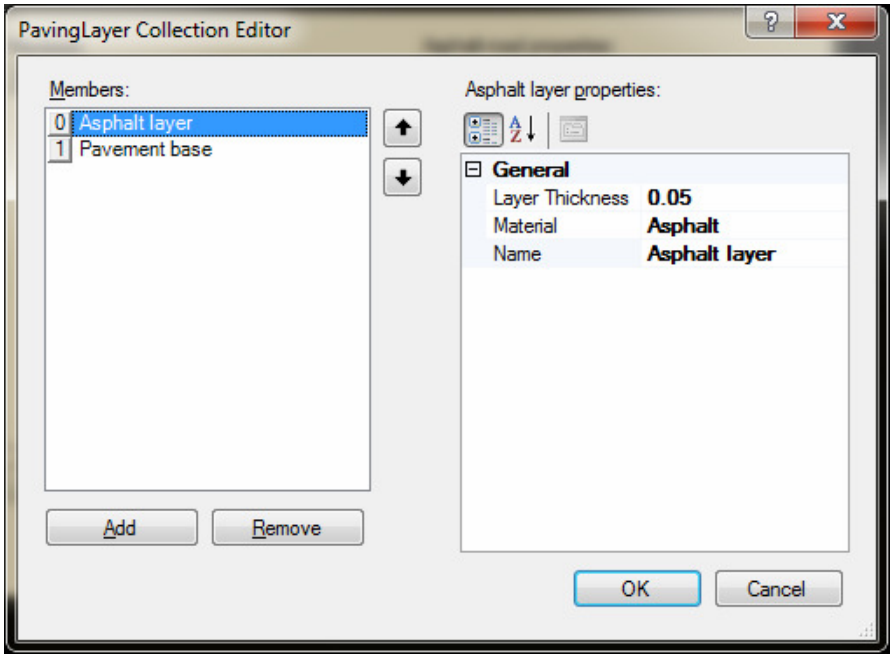
Restoration types: Collection with the available restoration types in the project.



Πλαίσιο διαλόγου διαθέσιμων διαμορφώσεων αποκαταστάσης.

In the *Backfilling* group you can define the backfill material used in the selected restoration type. In the *General* group you can define i) the layer containing the closed polylines surrounding the specific restoration type ii) the name of the restoration type iii) the paving layers. The *Inaptly percent* parameter does not used in the current version of the program.

Note that You can define a specific restoration type in the plan drawing by surrounding the specific region with closed polylines placed in a specific layer.
The *Paving Layers* collection defines the paving layers sequence for the corresponding restoration type. The sequence begins from the layer placed in the trench crown to layer placed in the bottom. For each pavement layer you can define the thickness, the material and the layer's name. The sequence of the pavement layers in the collection defines the sequence of the pavement layers in the trench restoration beginning from the top layer.



The *Paving layers collection*

Τέλος ορίζουμε το υλικό επανεπίχωσης του σκάμματος για το επιλεγμένο είδος αποκατάστασης.

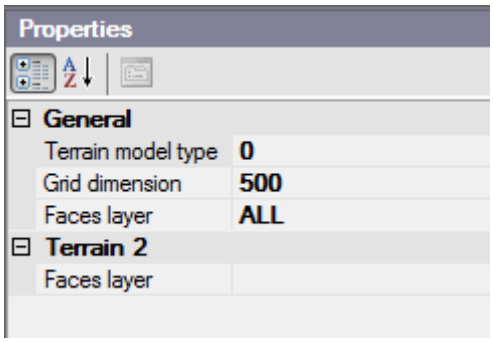
Time steps: This group includes two properties i) the index of the active time pattern ii) the collection of available time patterns (for more details see the *Time Patterns* section).

PROJECT EXPLORER

The objects included in the project's explorer are the following:

- ***Terrain.*** This object stores details about the terrain model of the project.

Note. The program creates the terrain model of the project based on 3d faces placed in a specific layer of the plan drawing.



Properties of the "Terrain" object

Terrain model type: This must be zero (0).

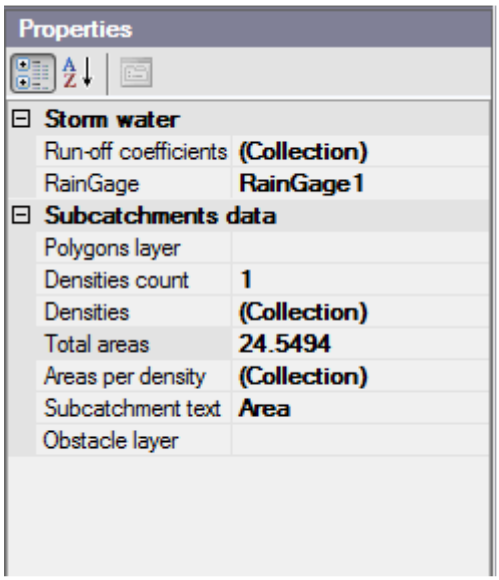
Grid dimension: The grid dimension of the terrain database. This value must be greater than the greatest edge of all 3d faces placed in the *Faces layer*. By default the program sets this property equal with 200 μ.

Faces layer: The layer in which you have placed the 3d faces of the terrain model. If you select the 'ALL' option then the program will create the terrain model based on the 3d faces included in all layers of the plan drawing.

Terrain 2:

Faces layer: The layer in which you have placed the 3d faces of a secondary terrain model. This property can be used in design cases when the final surface of roads will be completed in the future. In this case it is useful to see during the design the final level of the road surface in profile drawing.

- ***Subcatchments.*** This object includes details about the subcatchments of the project.



Properties of the *Subcatchments* object

Storm water

Run-off coefficients:

In this collection you can define the run-off coefficients for different kind of subcatchments. In the last position of this collection you must define the Run-off coefficient used for the roads.
e.g. if the collection have the following form

0.8
0.6
0.7

then the program sets for the subcatchments included in the first region (for more details about regions look the notes bellow) the Run-off coefficient equal to 0.8, for the second region equal to 0.6 and for the roads equal to 0.7.

Rain gage With this property you can define the active rain gage of the subcatchments.

Subcatchments data

Polygons layer: In this drop-down list, you can select one of the layer (if you have more than one regions) containing the subcatchment polygons. The polygons can be composed from closed polylines, lines or both poly-lines and lines together. You are freely to insert the polygons with order both clockwise and counter-clockwise.

Densities count: The number of regions with different population densities (or different run-off coefficients in the case of storm water networks).

Note. The program uses the same subcatchment polygons both for sewerage networks and for storm water networks.

Densities: Collection which includes the density of each region in the design area. The size of this collection is equal with the number of different regions in the design area.

Total areas: This property is equal with the total subcatchments area and is filled automatically by the program when you execute the *Hydraulics->Create Subcatchments* command.

Areas per density: Collection which includes the area of each separate region in the project. This collection is filled automatically by the program when you execute the *Hydraulics->Create Subcatchments* command.

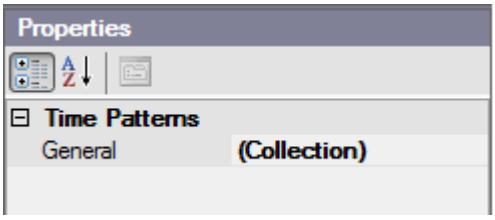
Subcatchment text: With this property you can choose to draw in the subcatchment polygons a text label with the corresponding population or area.

Obstacle layer: In this property you can define the layer containing the obstacle lines. When you have between a subcatchment and a pipe segment an obstacle line then the program cannot link the subcatchment to the specific pipe segment.
This feature can be used in order to insert an obstruction (like rivers, channels etc.) that pipes cannot cross.

Note The polygons layer names have two parts. The first part is the layer name and the second part is the index of the corresponding subcatchment region.

For example in a sewer network, suppose you have inserted in *Polygons layer* property the "Rymot X" layer (where X is an integer value). If you want to define two population densities in the design area having densities 80 and 50 respectively you must fill the first position (0 index) of the densities collection with 80 and the second position (1 index) with 50. Then you must put all the subcatchment polygons of the first region in the "Rymot 0" layer and the polygons of the second area in the "Rymot 1" layer.
Correspondingly in a storm network, if you want to define two different regions in the design area having two different run-off coefficients equal to 0.8 and 0.5 respectively, you must insert 0.8 in the first position of the densities collection and 0.5 in the second position. Then you must put all the subcatchment polygons of the first region in the "Rymot 0" layer and the polygons of the second area in the "Rymot 1" layer.

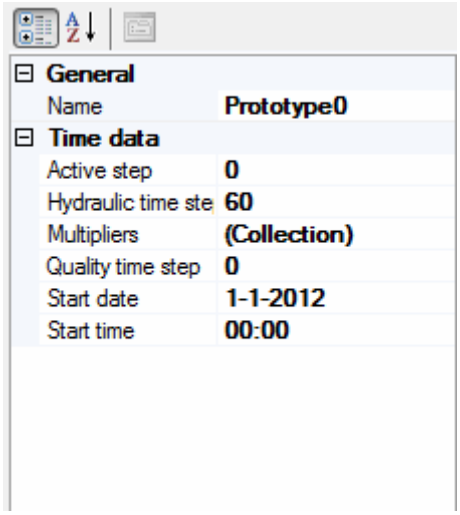
- ***Time patterns.*** This object keeps details about the time patterns in the project.



The "Time patterns" collection

General: This collection contains all the time patterns used in the project.
By default the program uses the first object of this collection as the active time pattern for the project. If you want to change the active pattern you must select the root object (named *Hydraulic project*) in the project's tree control and then change the index in *Active pattern* field of the *Time steps* group.

Time pattern:



Properties of the "Time pattern" object

Name: The name of the pattern.

Active step: This property is filled automatically by the program.

Hydraulic time step: The time step, expressed in minutes, used in the hydraulic analysis.

Multipliers: This collection contains the demand multiplier of each time step. This property is used only in pressure networks.

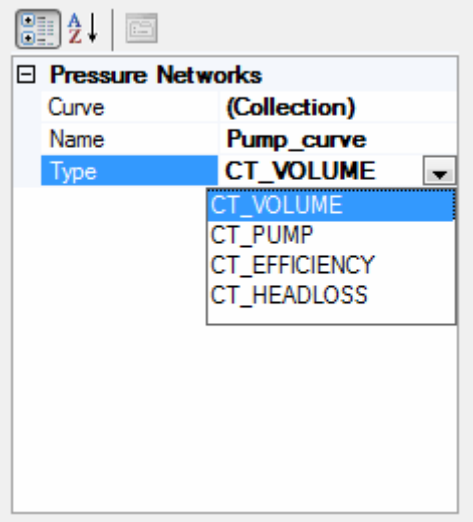
Quality time step: The time step, expressed in minutes, used in the quality analysis of pressure networks.

Start date: The start date of the analysis.

Start time: The start time of the analysis

- ***Pressure.*** This group contains all the objects used only in pressure networks. The program adds this group in the project's tree only if you have at least one pressure network in the project.

Curves: Collection with all curves used in pressure networks.



"Curve" object of pressure networks

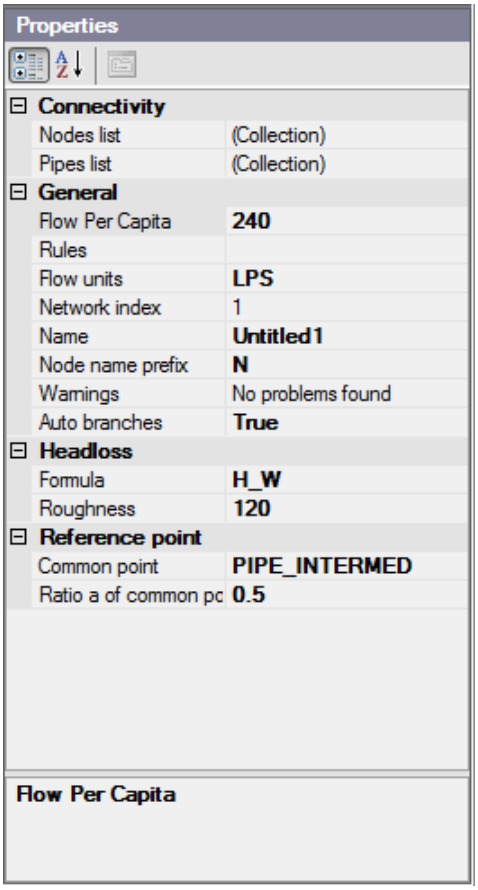
Curve: Collection containing the coordinates of the curve.

Name: The name of the curve.

Type: This property defines the curve type. There are four (4) available curve types in pressure networks:
Tank volume curve (CT_VOLUME). 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.
Pump curve (CT_PUMP). 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. If you insert one point in the curve's collection the program creates a parabola curve that passes through the point. If you insert more than one points in the collection the program take the curve as it is.
Pump efficiency curve (CT_EFFICIENCY). An Efficiency Curve determines pump efficiency (Y in percent) as a function of pump flow rate (X in flow units). 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 fixed global pump efficiency will be used.
Head loss curve (CT_HEADLOSS). A Headloss Curve is used to describe 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 behaviour.

- ***Networks.*** In this group you will find all the pressure networks included in the project.



Pressure network object

Connectivity:

Nodes list: This collection contains all the node objects of the selected network. This collection is created automatically from the program.

Pipes list: This collection contains all the pipe segment objects of the selected network. This collection is created automatically from the program.

General:

Flow per capita: In this field you can define the value of flow per capita in the current selected network.

Rules: In this field you can define rule-based controls that modify links based on a combination of conditions. These rules can be formed using the logical commands IF, THEN and the logical operators AND, OR.

Below you can find two examples of rule definition:

```
RULE 1
IF TANK 1 LEVEL ABOVE 3.0
THEN PUMP 1 STATUS IS CLOSED
AND PIPE 55 STATUS IS OPEN
```

The above rule define that if the water level in the TANK 1 is above 3 then the pump 1 closes and the pipe segment 55 opens.

```
RULE 2
IF TANK 1 LEVEL BELOW 0.5
THEN PUMP 1 STATUS IS OPEN
AND PIPE 55 STATUS IS CLOSED
```

The above rule define that if the water level in the TANK 1 drops below 0.5 then the PUMP 1 opens and the pipe segment 55 opens.

Flow units: The flow units used in the hydraulic design analysis of the current network. The list of available options for this field depends from the selected system in the *Measurement units* property of the root object in the *Project Explorer* tree.

Network index The network index. This property is filled automatically and cannot be changed by the user.

Name: The name of the current network.

Node name prefix:

The nodes prefix for the current network.

Warnings:

Warnings report describing problems in the geometric definition of the network. By pressing the button of this field the program shows the error report of the current network.

Auto branches:

If you choose *true* in this field the program will compute the pipe branches automatically beginning from the starting point. If you choose *false* in this field, the program will set each pipe as branch. The branches order will follow the pipes input order.

Headloss:

Formula: The method used in hydraulic calculations. There are three available options for this property: Hazen-Williams (H_W), Darcy-Weisbach (D_W) και Chezy-Manning (C_M).

Roughness: The corresponding roughness factor for the above selected method.

Reference point:

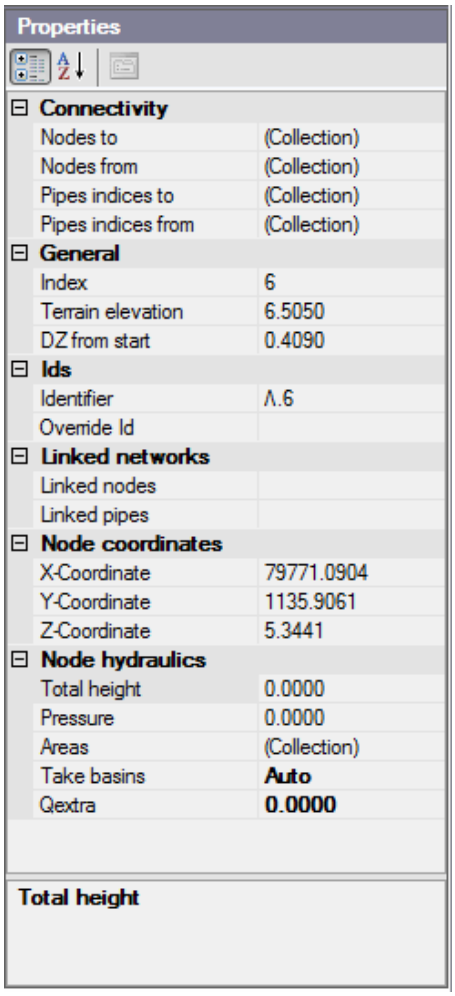
Common point: There are available three options for this field. PIPE_CROWN: The program uses as reference point the pipe crown point. Thus, when the diameter differs between pipes, the program keeps the pipe crown as common point between the two pipes.

PIPE_SEATING: The program uses as reference point the pipe seating point. Thus, when the diameter differs between pipes, the program keeps the pipe seating point as common point between the two pipes.
PIPE_INTERMED: The program uses as reference point the point abstaining a*D from the pipe seating level. Where a is the common ration value (see below) and D the diameter of the pipe.
By default the program identifies the pipes in center lines.

Ratio a of common point:

The common point ratio a. This parameter ranges between 0 and 1.

- ***Pressure network node.*** When you click on a node in the plan drawing (more precisely the circle surrounding the node) the program fills the property grid control with the properties of the selected node.
  The properties with the bold text can be modified while the properties with normal text cannot be modified by the user.
  For pressure networks the properties of the selected node are listed below:



Pressure node object

Connectivity:

Nodes to: Collection with the node indices placed in the end points of the pipe segments having as start point the selected node.

Nodes from: Collection with the node indices placed in the start points of the pipe segments having as end point the selected node.

Pipes indices to: This collection contains the pipe segment indices having as start point the selected node.

Pipes indices from: This collection contains the pipe segment indices having as end point the selected node.

General:

Index: The index of the selected node.

Terrain elevation: The terrain elevation in the node's position.

DZ from start: The elevation difference between the current selected node and the starting point.

Ids:

Identifier: The node identifier. This field is filled automatically.

Override id: Overrides the automatically created identifier.

Linked networks:

Linked nodes: Collection with the node indices of other network linked to the current selected node. Each line in this collection consists of two parts. The first part is an integer value equal with the index of the node and the second part is an integer value equal with the index of the network. For example if you insert the following lines in the collection:

2 0
5 1
…….

then you will connect the node 2 of network 0 and the node 5 of network 1 to the current selected node. Note that the above integer values are the corresponding indices.

Linked pipes: Collection with the pipe segment indices of other network linked to the current selected node. Each line in this collection consists of two parts. The first part is an integer value equal with the index of the pipe and the second part is an integer value equal with the index of network. For example if you insert the following lines in the collection:

2 0

5 1
…….

Then you will connect the pipe segment 2 of network 0 and the pipe segment 5 of network 1 to the current selected. Note that the above integer values are the corresponding indices.

Coordinates:

X-Coordinate: The X-coordinate of the node.

Y-Coordinate: The Y-coordinate of the node.

Z-Coordinate: The Z-coordinate of the node.

Node hydraulics:

Total height: The elevation of the hydraulic grade line in the node.

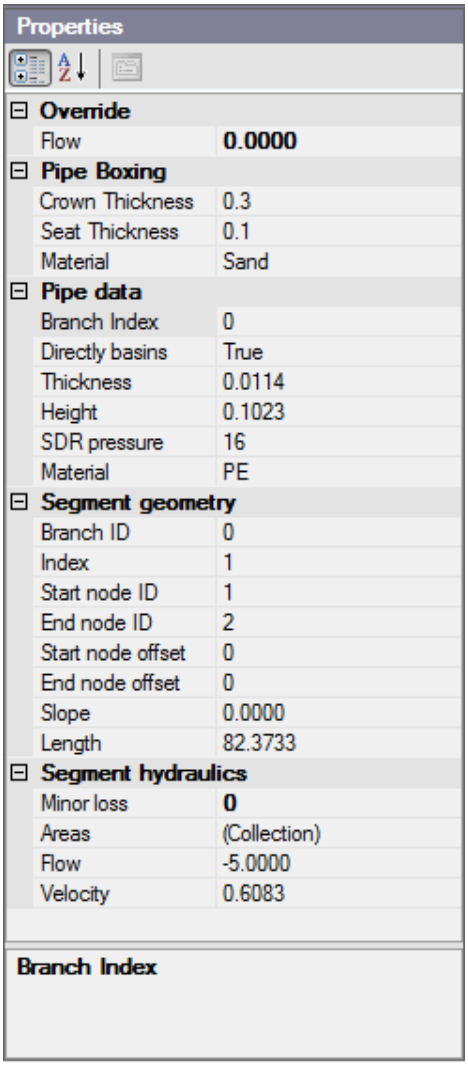
Pressure: The pressure on the current node (expressed in meters or feets)

Areas: Collection containing the subcatchment areas per region.

Take basins: This property controls where subcatchment areas can be assigned to selected node or not. There are three options available for this property:
i)*Auto*. Assign subcatchment areas according to the option *Basins* of the *Pipe Properties* dialog box. The program shows this dialog when you insert or edit a pipe.
ii)*TakeBasins*. If you choose this option, the program assigns subcatchment areas to the selected node.
iii)Not*TakeBasins*. If you choose this option, the program does not assign subcatchment areas to the selected node.

Q extra: Extra flow demand for the selected node. By using this property, you can insert additionally flow demand to the selected node.
Note that this value does not affected by the multipliers of the time patterns.

- ***Pressure network pipe segment.*** When you click on a pipe segment in the plan drawing the program fills the property grid control with the properties of the selected pipe segment. The properties with bold text can be modified while the properties with normal text cannot be modified by the user.
  For pressure networks the properties of the selected pipe segment are listed below:



Pressure network pipe segment object

Override:

Flow: If you insert value in this field then the program will use this value as the flow value in the specific pipe segment.

Pipe boxing:

Crown thickness: The thickness of the crown boxing layer.

Seat thickness: The thickness of the seat boxing layer.

Material: The boxing material.

Pipe data:

Directly basins: If this property is *true* then the program can link subcatchment areas to the pipe segment while if is *false* then the program cannot link subcatchment areas to the pipe segment.

Thickness: The thickness of the pipe segment.

Height: The height of the selected pipe segment (internal diameter in the case of circular pipes).

SDR pressure: The nominal pressure of the pipe segment.

Material: The pipe material of the selected pipe segment.

Segment geometry:

Branch ID: The branch index that belongs the selected pipe segment.

Index: The pipe segment index.

Start node ID: The index of the starting node of the segment.

End node ID: The index of the end node of the segment.

Start node offset: Vertical offset from the node in the starting point of the segment.

End node offset: Vertical offset from the node in the end point of the segment.

Slope: The slope of the pipe segment.

Length: The length of the pipe segment..

Segment hydraulics:

Minor loss: The minor losses in the current pipe segment.

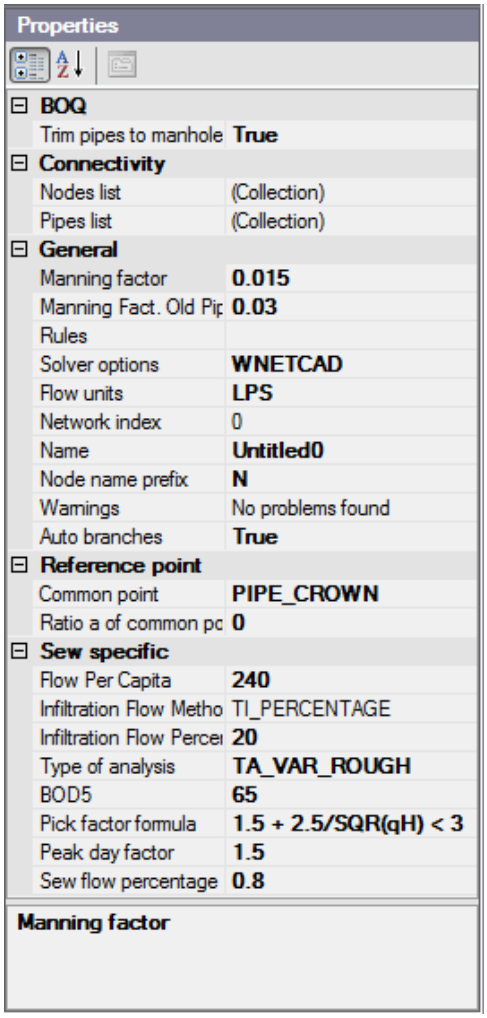
Areas: Collection containing the subcatchment areas per region.

Flow: The flow across the selected pipe segment.

Velocity: The velocity across the selected pipe segment.

SEWERAGE NETWORKS

Sewerage. This group contains all the objects used only in sewerage networks. The program adds this group in the project's tree only if you have at least one sewerage network in the project.



Sewerage network object

BOQ (Bill of quantities):

Trim pipes to manholes :

If this property is equal with *true* then the program computes and shows in the report the pipe segment lengths until the external surface of the manhole wall. If this property is equal with *false* then the program computes the pipe segment lengths until the center point of the manhole.

Connectivity:

Nodes list: This collection contains all the node objects of the selected network. This collection is created automatically from the program.

Pipes list: This collection contains all the pipe segment objects of the selected network. This collection is created automatically from the program.

General:

Manning factor:

Manning roughness factor used in hydraulic calculations.

Manning factor old pipes:

Manning roughness factor used in hydraulic calculations of existing (old) pipes.

Rules: This property does not used in the current version of the program.

Solver options: The algorithm type used in hydraulic calculations.

Flow units: The flow units used in the hydraulic design analysis of the current network. The list of available options for this field depends from the selected system in the *Measurement units* property of the root object in the *Project Explorer* tree.

Network index: The network index. This property is filled automatically and cannot be changed by the user.

Name: The name of the current network.

Node name prefix:

The nodes prefix for the current network.

Warnings:

Warnings report describing problems in the geometric definition of the network. By pressing the button of this field the program shows the error report of the current network.

Auto branches:

If you choose *true* in this field the program will compute the pipe branches automatically beginning from the starting point. If you choose *false* in this field, the program will set each pipe as branch. The branches order will follow the pipes input order.

Reference point:

Common point: There are available three options for this field.
PIPE_CROWN: The program uses as reference point the pipe crown point. Thus, when the diameter differs between pipes, the program keeps the pipe crown as common point between the two pipes.
PIPE_SEATING: The program uses as reference point the pipe seating point. Thus, when the diameter differs between pipes, the program keeps the pipe seating point as common point between the two pipes.
PIPE_INTERMED: The program uses as reference point the point abstaining a*D from the pipe seating level. Where a is the common ration value (see below) and D the diameter of the pipe.
By default the program identifies the pipes crown.

Ratio a of common point:

The above described ratio “a”. This parameter ranges between 0 and 1.

Sew specific:

Flow per capita: In this field you can define the value of flow per capita in the current selected network.

Infiltration flow method: Method to estimate the infiltration flow. There are available two methods in this property. With the first method (TI_PERCENTAGE) the infiltration flow is computed as a percentage of the main flow in the pipe segment. With the second method (TI_AREA_UPSTREAM) the infiltration flow is computed depending on total subcatchments area upstream.

Infiltration Flow Percentage:
The percentage of main flow that is equal with the infiltration flow. This property is used together with the TI_PERCENTAGE option.

Type of analysis: With this property you can define the way in which the program computes the hydraulic parameters. There are available two options in this field. If you choose the TA_VAR_ROUGH option then the program will use variable roughness factor in hydraulic computations (this option gives greater values for flow height). If you choose the TA_CONST_ROUGH option then the program will use constant roughness factor in hydraulic calculations.

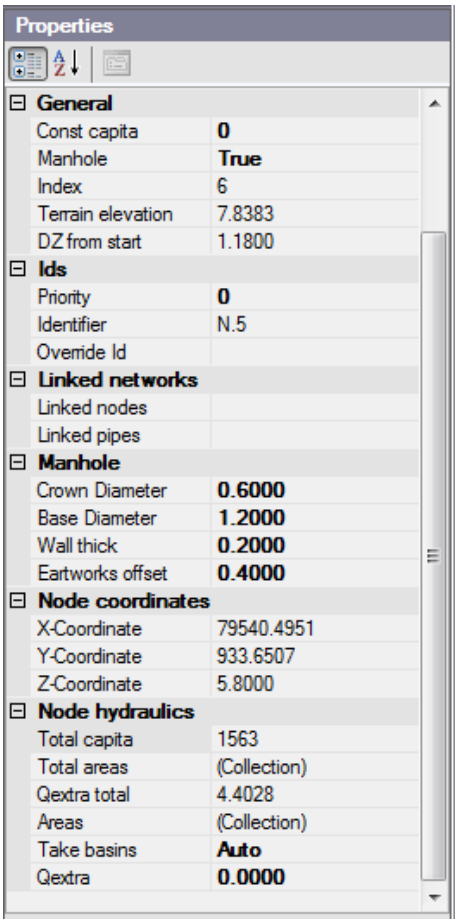
BOD5: The biochemical oxygen demand of wastewater during decomposition occurring over a 5-day period (gr/day/capita).

Pick factor formula: Formula to compute the flow pick factor. By default the program uses the French formula to find the peak flow coefficient.

Peak day formula:
This coefficient converts the average daily flow to maximum daily flow.

Sew flow percentage:
The percentage of drinking water enters to the sewerage network.

- ***Sewer network node.*** When you click on a node in the plan drawing (more precisely the circle surrounding the node) the program fills the property grid control with the properties of the selected node.
  The properties with the bold text can be modified while the properties with normal text cannot be modified by the user.
  For sewer (sanitary) networks the properties of the selected node are listed below:



Sewer node object

Connectivity:

Nodes to: Collection with the node indices placed in the end points of the pipe segments having as start point the selected node.

Nodes from: Collection with the node indices placed in the start points of the pipe segments having as end point the selected node.

Pipes indices to: This collection contains the pipe segment indices having as start point the selected node.

Pipes indices from: This collection contains the pipe segment indices having as end point the selected node.

General:

Const capita: With this property you can set a specific population for the current selected node. This feature can be used when you want to solve a network by defining directly

the population of each node. By default the program computes the population by multiplying the subcatchment areas and the corresponding subcatchment population densities.

Manhole: If this property is equal with *true* then the program will assign a manhole construction in the selected node. If this property is equal with *false* then the program will leave the node without manhole.

Index: The index of the selected node.

Terrain elevation: The terrain elevation in the node's position.

DZ from start: The elevation difference between the current selected node and the starting point.

Ids:

Priority: This property can be used when you want to change the default order of the branches resulting automatically by the program. Each branch in the network has a leaf node (end) node. By changing the *Priority* value on leaf node you can change also the priority of the entire branch.
For example, if you set this property to 1 in a leaf node then the entire branch until the outlet will be the 1st branch.
Similarly, if you set this property to 2 in another leaf node the entire branch until the 1st branch will be the 2nd branch etc.
If this property has value zero (0) then the branch order results automatically by the program.

Identifier: The node identifier. This field is filled automatically.

Override id: Overrides the automatically created identifier.

Linked networks:

Linked nodes: Collection with the node indices of other network linked to the current selected node. Each line in this collection consists of two parts. The first part is an integer value equal with the index of the node and the second part is an integer value equal with the index of the network. For example if you insert the following lines in the collection:

2 0
5 1
…….

then you will connect the node 2 of network 0 and the node 5 of network 1 to the current selected node. Note that the above integer values are the corresponding indices.

Linked pipes: Collection with the pipe segment indices of other network linked to the current selected node. Each line

in this collection consists of two parts. The first part is an integer value equal with the index of the pipe and the second part is an integer value equal with the index of network. For example if you insert the following lines in the collection:

2 0
5 1
…….

Then you will connect the pipe segment 2 of network 0 and the pipe segment 5 of network 1 to the current selected. Note that the above integer values are the corresponding indices.

Manhole:

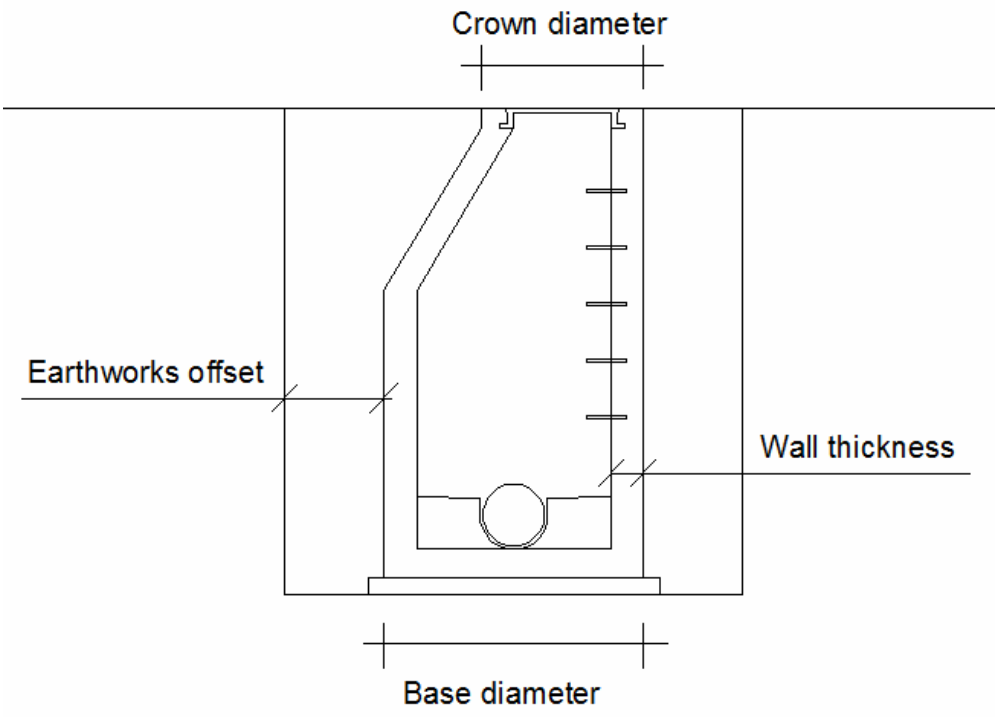
Crown diameter: The external diameter of the manhole on the crown.

Base diameter: The external diameter of the manhole in the base.

Note that sewerage manholes are cylindric.

Wall thick: The thickness of the manhole walls.

Earthworks offset: Offset distance from the excavation surface.



Sewer manhole figure

Coordinates:

X-Coordinate: The X-coordinate of the node.

Y-Coordinate: The Y-coordinate of the node.

Z-Coordinate: The Z-coordinate of the node.

Node hydraulics:

Total capital: The total population corresponding to the current selected node.

Total areas: The total subcatchment areas corresponding to the current selected node.

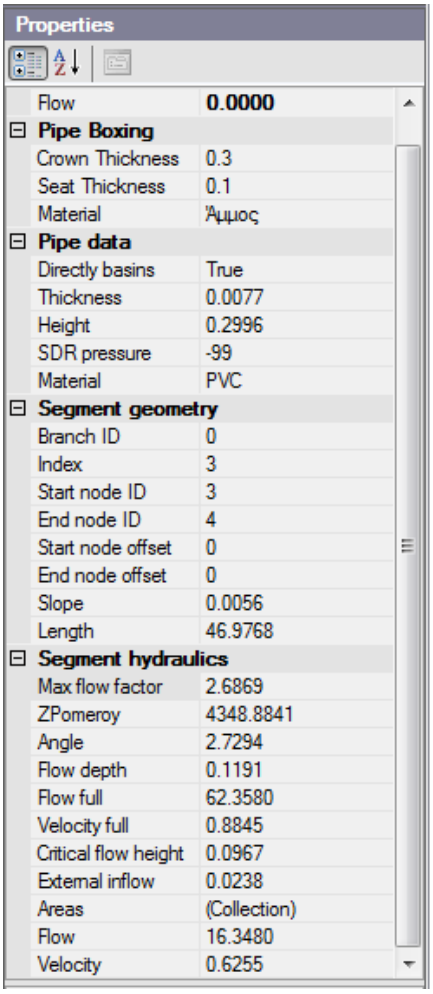
Qextra total: The total extra flow resulting by the sum of Qextra values of all nodes placed higher in the network's tree than the current selected node.

Areas: Collection containing the subcatchment areas per region corresponding to the selected node.

Take basins: This property controls where subcatchment areas can be assigned to selected node or not. There are three options available for this property:
i) *Auto*. Assign subcatchment areas according to the option *Basins* of the *Pipe Properties* dialog box. The program shows this dialog when you insert or edit a pipe.
ii) *TakeBasins*. If you choose this option, the program assigns subcatchment areas to the selected node.
iii) Not *TakeBasins*. If you choose this option, the program does not assign subcatchment areas to the selected node.

Q extra: Extra flow to the selected node. By using this property, you can insert additionally flow to the selected node.
Note that this value does not affected by the peak flow factors.

- ***Sewer network pipe segment.*** When you click on a pipe segment in the plan drawing the program fills the property grid control with the properties of the selected pipe segment. The properties with bold text can be modified while the properties with normal text cannot be modified by the user.
  For sewer (sanitary) networks the properties of the selected pipe segment are listed below:



Sewer pipe segment object

Override:

Flow: If you insert value in this field then the program will use this value as the flow value in the specific pipe segment.

Pipe boxing:

Crown thickness: The thickness of the crown boxing layer.

Seat thickness: The thickness of the seat boxing layer.

Material: The boxing material.

Pipe data:

Directly basins: If this property is *true* then the program can link subcatchment areas to the pipe segment while if is *false* then the program cannot link subcatchment areas to the pipe segment.

Thickness: The thickness of the pipe segment.

Height: The height of the selected pipe segment (internal diameter in the case of circular pipes).

SDR pressure: This value is always negative in gravity driven networks.

Material: The pipe material of the selected pipe segment.

Segment geometry:

Branch ID: The branch index that belongs the selected pipe segment.

Index: The pipe segment index.

Start node ID: The index of the starting node of the segment.

End node ID: The index of the end node of the segment.

Start node offset: Vertical offset from the node in the starting point of the segment.

End node offset: Vertical offset from the node in the end point of the segment.

Slope: The slope of the pipe segment.

Length: The length of the pipe segment.

Segment hydraulics:

Pick flow factor: Pick flow factor for the current selected segment.

ZPomeroy: Value describing how likely is the H_2S presence in the pipe segment.

Angle: Value of the central angle of flow in the pipe segment.

Flow depth: The flow depth in the pipe segment.

Flow full: The flow when the pipe segment is full with water.

Velocity full: The velocity when the pipe segment is full with water.

Critical flow height: The depth in the pipe segment.

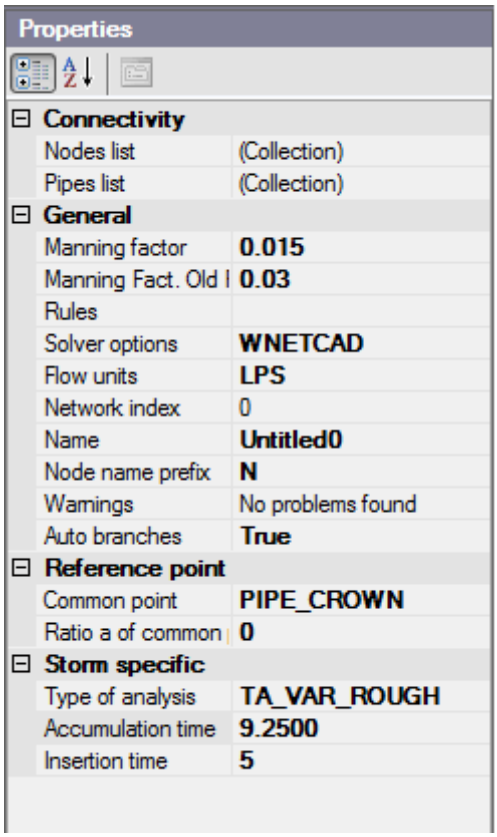
Infiltration flow: The infiltration flow in the pipe segment.

Areas: Collection containing the subcatchment areas per region corresponding to the selected node.

Flow: The flow across the selected pipe segment.

Velocity: The velocity across the selected pipe segment.

- ***Storm water.*** This group contains all the objects used only in storm water networks. The program adds this group in the project's tree only if you have at least one storm water network in the project.



Storm water network object

Storm water:

Connectivity:

Nodes list: This collection contains all the node objects of the selected network. This collection is created automatically from the program.

Pipes list: This collection contains all the pipe segment objects of the selected network. This collection is created automatically from the program.

General:

Manning factor: Manning roughness factor used in hydraulic calculations.

Manning factor old pipes:

Manning roughness factor used in hydraulic calculations of existing (old) pipes.

Rules: This property does not used in the current version of the program.

Solver options: The algorithm type used in hydraulic calculations.

Flow units: The flow units used in the hydraulic design analysis of the current network. The list of available options for this field depends from the selected system in the *Measurement units* property of the root object in the *Project Explorer* tree.

Network index: The network index. This property is filled automatically and cannot be changed by the user.

Name: The name of the current network.

Node name prefix:

The nodes prefix for the current network.

Warnings:

Warnings report describing problems in the geometric definition of the network. By pressing the button of this field the program shows the error report of the current network.

Auto branches:

If you choose *true* in this field the program will compute the pipe branches automatically beginning from the starting point. If you choose *false* in this field, the program will set each pipe as branch. The branches order will follow the pipes input order.

Reference point:

Common point: There are available three options for this field.
PIPE_CROWN: The program uses as reference point the pipe crown point. Thus, when the diameter differs between pipes, the program keeps the pipe crown as common point between the two pipes.
PIPE_SEATING: The program uses as reference point the pipe seating point. Thus, when the diameter differs between pipes, the program keeps the pipe seating point as common point between the two pipes.
PIPE_INTERMED: The program uses as reference point the point abstaining a*D from the pipe seating level. Where a is the common ration value (see below) and D the diameter of the pipe.
By default the program identifies the pipes crown.

Ratio a of common point:

The above described ratio “a”. This parameter ranges between 0 and 1.

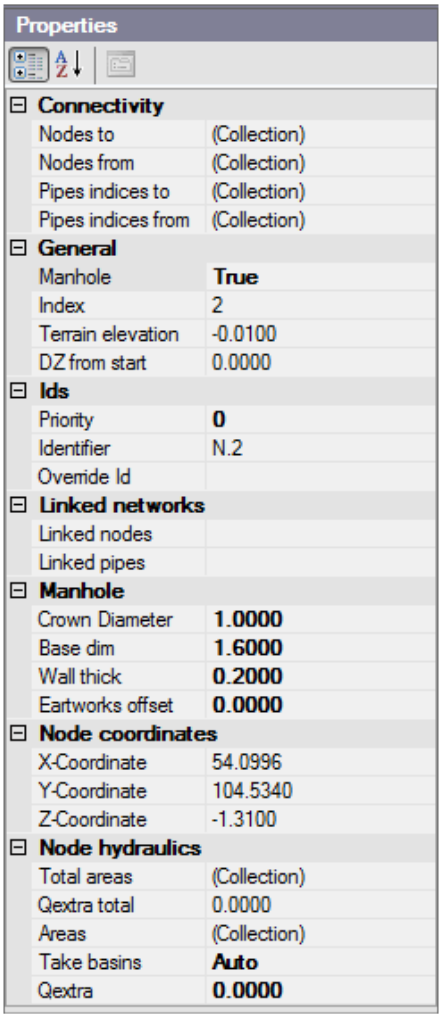
Storm specific:

Type of analysis: With this property you can define the way in which the program computes the hydraulic parameters. There are available two options in this field. If you choose the TA_VAR_ROUGH option then the program will use variable roughness factor in hydraulic computations (this option gives greater values for flow height). If you choose the TA_CONST_ROUGH option then the program will use constant roughness factor in hydraulic calculations.

Accumulation time: The time needed to travel the water from the farest point of the farest subcatchment to the outlet node of the network.

Insertion time: The time that water flows on subcatchments before enter to the network.

- ***Storm water network node.*** When you click on a node in the plan drawing (more precisely the circle surrounding the node) the program fills the property grid control with the properties of the selected node.
  The properties with the bold text can be modified while the properties with normal text cannot be modified by the user.
  For storm water networks the properties of the selected node are listed below:



Storm water node object

Connectivity:

Nodes to: Collection with the node indices placed in the end points of the pipe segments having as start point the selected node.

Nodes from: Collection with the node indices placed in the start points of the pipe segments having as end point the selected node.

Pipes indices to: This collection contains the pipe segment indices having as start point the selected node.

Pipes indices from: This collection contains the pipe segment indices having as end point the selected node.

General:

Manhole: If this property is equal with *true* then the program will assign a manhole construction in the selected node. If this property is equal with *false* then the program will leave the node without manhole.

Index: The index of the selected node.

Terrain elevation: The terrain elevation in the node's position.

DZ from start: The elevation difference between the current selected node and the starting point.

Ids:

Priority: This property can be used when you want to change the default order of the branches resulting automatically by the program. Each branch in the network has a leaf node (end) node. By changing the *Priority* value on leaf node you can change also the priority of the entire branch.
For example, if you set this property to 1 in a leaf node then the entire branch until the outlet will be the 1st branch.
Similarly, if you set this property to 2 in another leaf node the entire branch until the 1st branch will be the 2nd branch etc.
If this property has value zero (0) then the branch order results automatically by the program.

Identifier: The node identifier. This field is filled automatically.

Override id: Overrides the automatically created identifier.

Linked networks:

Linked nodes: Collection with the node indices of other network linked to the current selected node. Each line in this collection consists of two parts. The first part is an integer value equal with the index of the node and the second part is an integer value equal with the index of

the network. For example if you insert the following lines in the collection:

2 0
5 1
…….

then you will connect the node 2 of network 0 and the node 5 of network 1 to the current selected node. Note that the above integer values are the corresponding indices.

Linked pipes: Collection with the pipe segment indices of other network linked to the current selected node. Each line in this collection consists of two parts. The first part is an integer value equal with the index of the pipe and the second part is an integer value equal with the index of network. For example if you insert the following lines in the collection:

2 0
5 1
…….

Then you will connect the pipe segment 2 of network 0 and the pipe segment 5 of network 1 to the current selected. Note that the above integer values are the corresponding indices.

Manhole:

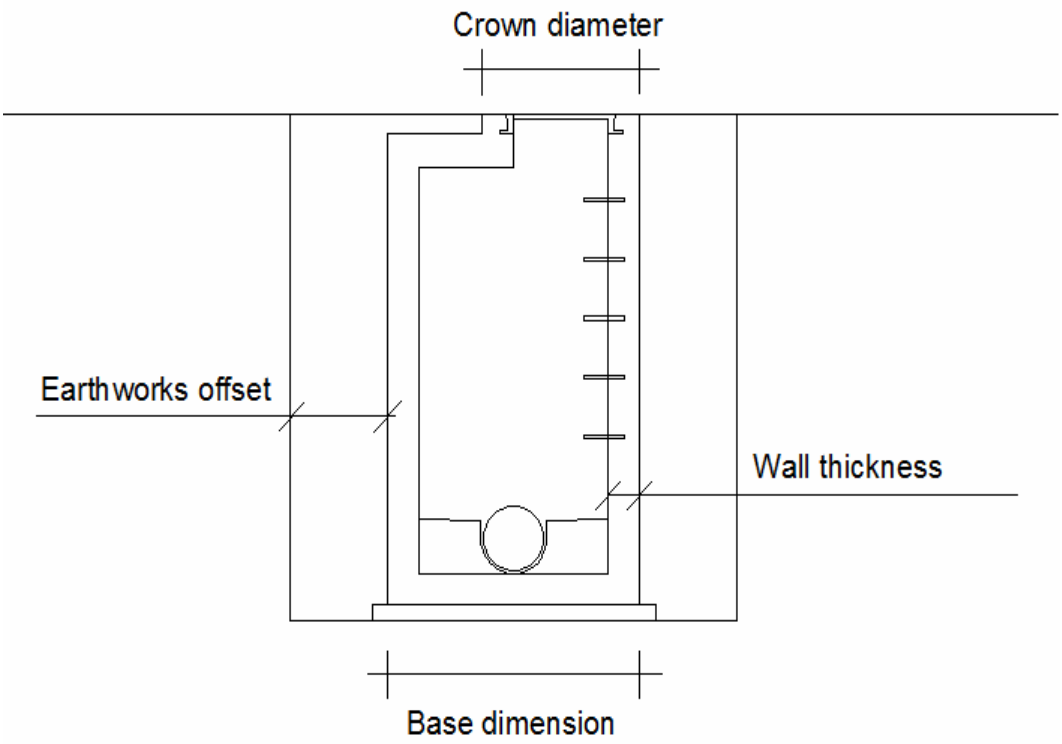
Crown diameter: The external diameter of the manhole on the crown.

Base dim: The external dimension of the manhole in the base.

Note that storm water manholes are rectangular.

Wall thick: The thickness of the manhole walls.

Earthworks offset: Offset distance from the excavation surface.



Storm water manhole figure

Coordinates:

X-Coordinate: The X-coordinate of the node.

Y-Coordinate: The Y-coordinate of the node.

Z-Coordinate: The Z-coordinate of the node.

Node hydraulics:

Total areas: The total subcatchment areas corresponding to the current selected node.

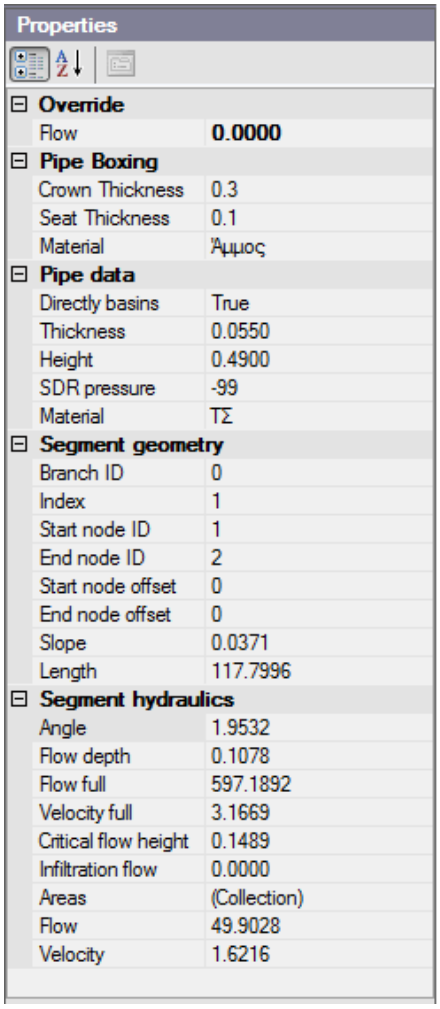
Qextra total: The total extra flow resulting by the sum of Qextra values of all nodes placed higher in the network's tree than the current selected node.

Areas: Collection containing the subcatchment areas per region corresponding to the selected node.

Take basins: This property controls where subcatchment areas can be assigned to selected node or not. There are three options available for this property:
i) *Auto*. Assign subcatchment areas according to the option *Basins* of the *Pipe Properties* dialog box. The program shows this dialog when you insert or edit a pipe.
ii) *TakeBasins*. If you choose this option, the program assigns subcatchment areas to the selected node.
iii) Not *TakeBasins*. If you choose this option, the program does not assign subcatchment areas to the selected node.

Q extra: Extra flow to the selected node. By using this property, you can insert additionally flow to the selected node.

- ***Storm water pipe segment.*** When you click on a pipe segment in the plan drawing the program fills the property grid control with the properties of the selected pipe segment. The properties with bold text can be modified while the properties with normal text cannot be modified by the user.
  For storm water networks the properties of the selected pipe segment are listed below:



Storm water network segment object

Override:

Flow: If you insert value in this field then the program will use this value as the flow value in the specific pipe segment.

Pipe boxing:

Crown thickness: The thickness of the crown boxing layer.

Seat thickness: The thickness of the seat boxing layer.

Material: The boxing material.

Pipe data:

Directly basins: If this property is *true* then the program can link subcatchment areas to the pipe segment while if is *false* then the program cannot link subcatchment areas to the pipe segment.

Thickness: The thickness of the pipe segment.

Height: The height of the selected pipe segment (internal diameter in the case of circular pipes).

SDR pressure: This value is always negative in gravity driven networks.

Material: The pipe material of the selected pipe segment.

Segment geometry:

Branch ID: The branch index that belongs the selected pipe segment.

Index: The pipe segment index.

Start node ID: The index of the starting node of the segment.

End node ID: The index of the end node of the segment.

Start node offset: Vertical offset from the node in the starting point of the segment.

End node offset: Vertical offset from the node in the end point of the segment.

Slope: The slope of the pipe segment.

Length: The length of the pipe segment.

Segment hydraulics:

Angle: Value of the central angle of flow in the pipe segment.

Flow depth: The flow depth in the pipe segment.

Flow full: The flow when the pipe segment is full with water.

Velocity full: The velocity when the pipe segment is full with water.

Critical flow height: The depth in the pipe segment.

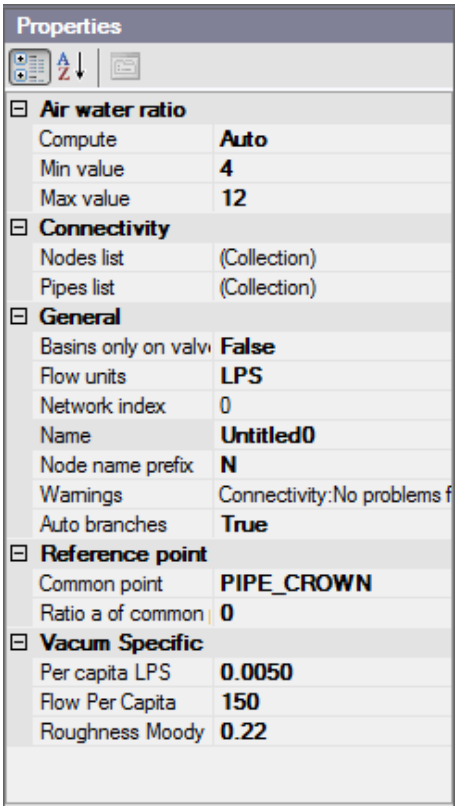
Infiltration flow: This property does not used in storm water networks

Flow: The flow across the selected pipe segment.

Velocity: The velocity across the selected pipe segment.

VACUUM SEWER NETWORKS

Vacuum. This group contains all the objects used only in vacuum sewer networks. The program adds this group in the project's tree only if you have at least vacuum sewer network in the project.



Vacuum sewer network object

Air water ratio:

Compute:

If you choose the *manual* option in this field then the air-water ratio must be defined manually for all nodes in the network. If you choose the *Auto* option in this field then the program will compute automatically the air-water for all nodes in the network. The automatically computed values will be between the *Min value* and *Max value* (see below properties)

Min value: The minimum value of air-water ratio in the network.

Max value: The maximum value of air-water ratio in the network.

Connectivity:

Nodes list: This collection contains all the node objects of the selected network. This collection is created automatically from the program.

Pipes list: This collection contains all the pipe segment objects of the selected network. This collection is created automatically from the program.

General:

Basins only on valves:

If you choose *true* in this option then the program will assign subcatchment areas only in the valve nodes.

Flow units: The current version of the program supports only the LPS (litter per second).

Network index The network index. This property is filled automatically and cannot be changed by the user.

Name: The name of the current network.

Node name prefix:

The nodes prefix for the current network.

Warnings:

Warnings report describing problems in the geometric definition of the network. By pressing the button of this field the program shows the error report of the current network.

Auto branches:

If you choose *true* in this field the program will compute the pipe branches automatically beginning from the starting point. If you choose *false* in this field, the program will set each pipe as branch. The branches order will follow the pipes input order.

Reference point:

Common point: There are available three options for this field.
PIPE_CROWN: The program uses as reference point the pipe crown point. Thus, when the diameter differs between pipes, the program keeps the pipe crown as common point between the two pipes.
PIPE_SEATING: The program uses as reference point the pipe seating point. Thus, when the diameter differs between pipes, the program keeps the pipe seating point as common point between the two pipes.
PIPE_INTERMED: The program uses as reference point the point abstaining a*D from the pipe seating level. Where a is the common ration value (see below) and D the diameter of the pipe.
By default the program identifies the pipes in center lines.

Ratio a of common point:

The common point ratio a. This parameter ranges between 0 and 1.

Vacuum:

Per capita LPS:

In this field you can define pick flow per capita.

Flow per capita:

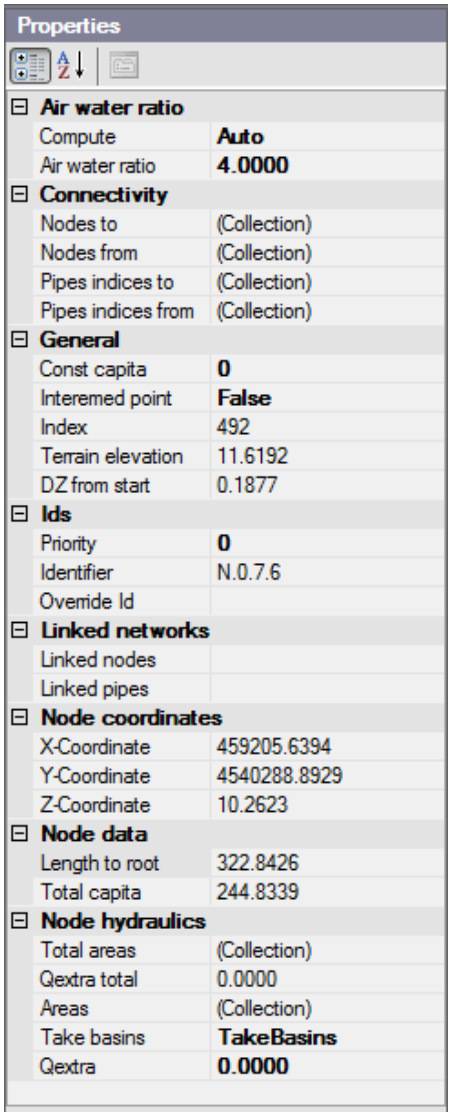
In this field you can define the mid flow per capita per day.

Note that program fill these fields with values based on the ATV-DVWK-A 116 norm. The hydraulic calculation is executed based on the corresponding tables of this norm which have been created for the above default values. If you make any change to these values you must adjust the hydraulic calculation accordingly.

Roughness Moody:

The Moody roughness coefficient. This coefficient is used to compute the loss height in the branches.

- ***Vacuum sewer network node.*** When you click on a node in the plan drawing (more precisely the circle surrounding the node) the program fills the property grid control with the properties of the selected node.
  The properties with the bold text can be modified while the properties with normal text cannot be modified by the user.
  For vacuum sewer networks the properties of the selected node are listed below:



Vacuum sewer node object

Air water ratio:

Compute:

If you choose the *manual* option in this field then the air-water ratio value will be taken by the *Air water ratio* property (see below). If you choose the *Auto* option in this field then the program will compute automatically the air-water ratio value. The automatically computed values will be between the *Min value* and *Max value* (see below properties)

Air water ratio: The value of the *Air water ratio* property. This property is used when you choose the *Manual* option in the above field.

Connectivity:

Nodes to: Collection with the node indices placed in the end points of the pipe segments having as start point the selected node.

Nodes from: Collection with the node indices placed in the start points of the pipe segments having as end point the selected node.

Pipes indices to: This collection contains the pipe segment indices having as start point the selected node.

Pipes indices from: This collection contains the pipe segment indices having as end point the selected node.

General:

Const capita: With this property you can set a specific population for the current selected node. This feature can be used when you want to solve a network by defining directly the population of each node. By default the program computes the population by multiplying the subcatchment areas and the corresponding subcatchment population densities.

Intermediate: If this property is equal with *true* then the program will convert the current selected node to an intermediate node. You can use the intermediate nodes if you want to put extra points between normal nodes. Note that intermediate nodes are collinear with the previous and next normal nodes.

Index: The index of the selected node.

Terrain elevation: The terrain elevation in the node's position.

DZ from start: The elevation difference between the current selected node and the starting point.

Ids:

Priority: This property can be used when you want to change the default order of the branches resulting

automatically by the program. Each branch in the network has a leaf node (end) node. By changing the *Priority* value on leaf node you can change also the priority of the entire branch.
For example, if you set this property to 1 in a leaf node then the entire branch until the outlet will be the 1st branch.
Similarly, if you set this property to 2 in another leaf node the entire branch until the 1st branch will be the 2nd branch etc.
If this property has value zero (0) then the branch order results automatically by the program.

Identifier: The node identifier. This field is filled automatically.

Override id: Overrides the automatically created identifier.

Linked networks:

Linked nodes: Collection with the node indices of other network linked to the current selected node. Each line in this collection consists of two parts. The first part is an integer value equal with the index of the node and the second part is an integer value equal with the index of the network. For example if you insert the following lines in the collection:

2 0
5 1
…….

then you will connect the node 2 of network 0 and the node 5 of network 1 to the current selected node. Note that the above integer values are the corresponding indices.

Linked pipes: Collection with the pipe segment indices of other network linked to the current selected node. Each line in this collection consists of two parts. The first part is an integer value equal with the index of the pipe and the second part is an integer value equal with the index of network. For example if you insert the following lines in the collection:

2 0
5 1
…….

Then you will connect the pipe segment 2 of network 0 and the pipe segment 5 of network 1 to the current selected. Note that the above integer values are the corresponding indices.

Node coordinates:

X-Coordinate: The X-coordinate of the node.

Y-Coordinate: The Y-coordinate of the node.

Z-Coordinate: The Z-coordinate of the node.

Node data:

Length to root: This property is equal with the path length from the current selected node to the outlet node (vacuum pump).

Total capita: The total population until the selected node.

Node hydraulics:

Total capital: The total population corresponding to the current selected node.

Total areas: The total subcatchment areas corresponding to the current selected node.

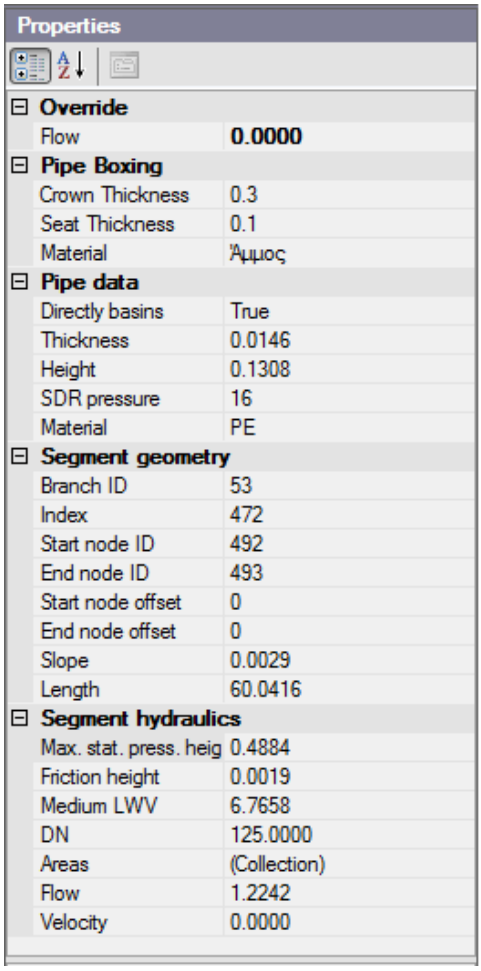
Qextra total: The total extra flow resulting by the sum of Qextra values of all nodes placed higher in the network's tree than the current selected node.

Areas: Collection containing the subcatchment areas per region corresponding to the selected node.

Take basins: This property controls where subcatchment areas can be assigned to selected node or not. There are three options available for this property:
i)*Auto*. Assign subcatchment areas according to the option *Basins* of the *Pipe Properties* dialog box. The program shows this dialog when you insert or edit a pipe.
ii)*TakeBasins*. If you choose this option, the program assigns subcatchment areas to the selected node.
iii)Not*TakeBasins*. If you choose this option, the program does not assign subcatchment areas to the selected node.

Q extra: Extra flow to the selected node. By using this property, you can insert additionally flow to the selected node.
Note that this value does not affected by the peak flow factors.

- ***Vacuum sewer pipe segment.*** When you click on a pipe segment in the plan drawing the program fills the property grid control with the properties of the selected pipe segment. The properties with bold text can be modified while the properties with normal text cannot be modified by the user.
  For vacuum sewer networks the properties of the selected pipe segment are listed below:



Vacuum sewer pipe segment object

Override:

Flow: If you insert value in this field then the program will use this value as the flow value in the specific pipe segment.

Pipe boxing:

Crown thickness: The thickness of the crown boxing layer.

Seat thickness: The thickness of the seat boxing layer.

Material: The boxing material.

Pipe data:

Directly basins: If this property is *true* then the program can link subcatchment areas to the pipe segment while if is *false* then the program cannot link subcatchment areas to the pipe segment.

Thickness: The thickness of the pipe segment.

Height: The height of the selected pipe segment (internal diameter in the case of circular pipes).

SDR pressure: The nominal pressure of the pipe segment.

Material: The pipe material of the selected pipe segment.

Segment geometry:

Branch ID: The branch index that belongs the selected pipe segment.

Index: The pipe segment index.

Start node ID: The index of the starting node of the segment.

End node ID: The index of the end node of the segment.

Start node offset: Vertical offset from the node in the starting point of the segment.

End node offset: Vertical offset from the node in the end point of the segment.

Slope: The slope of the pipe segment.

Length: The length of the pipe segment.

Segment hydraulics:

Max. stat. press. height:
This is the maximum static pressure height difference until section end.

Loss DH: The total loss height until until section end.

Medium LWV: Medium air/water ratio at section end.

DN: Nominal width section.

Flow full: The flow when the pipe segment is full with water.

Velocity full: The velocity when the pipe segment is full with water.

Critical flow height: The depth in the pipe segment.

Infiltration flow: The infiltration flow in the pipe segment.

Areas: Collection containing the subcatchment areas per region corresponding to the selected node.

Flow: The flow across the selected pipe segment.

Velocity: The velocity across the selected pipe segment.