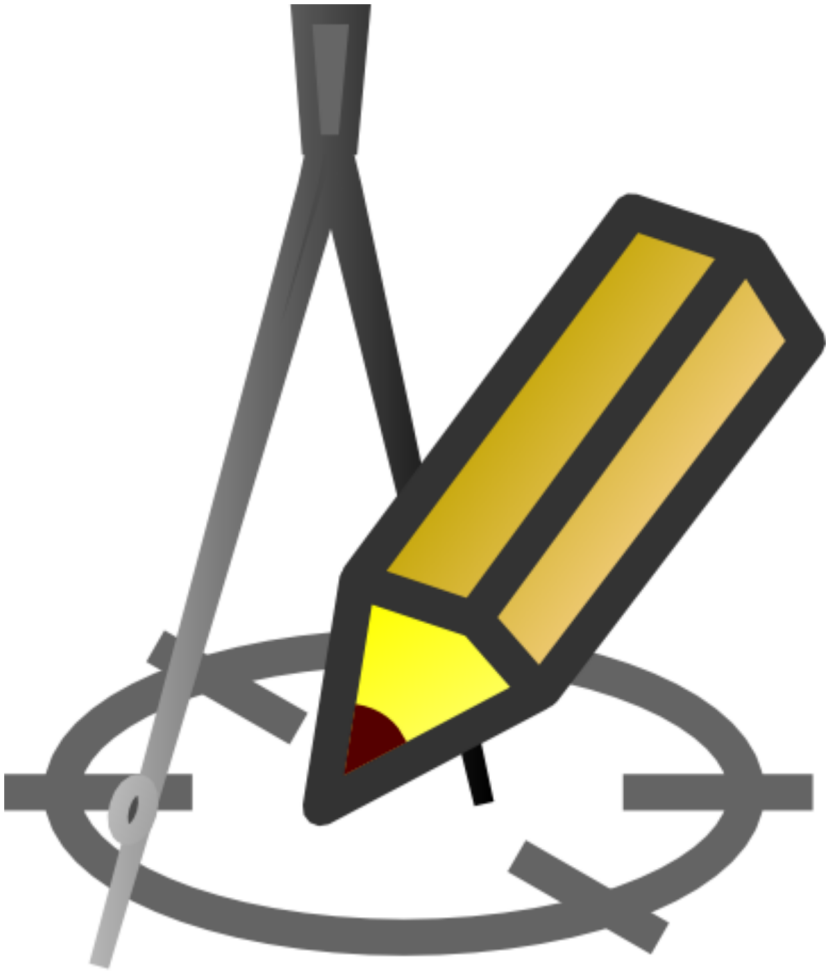
DRAFTER

simply and fast profile creation



user manual

 created: 2022-08-12

About

Version: **3.91**
Copyright: **Grzegorz Wisowski**

Updates: www.drafter.wisart.eu
E-mail: info@wisart.eu
Tel.: **+48 503 587 648** hrs. 9:00-16:00 GMT

Order form:
To register Drafter you need to buy license: http://www.drafter.wisart.eu/order.php

The contents of this book coincides with the contents of the **Drafter** help file. Presented in this guide, book form, is for some users a more user-friendly and easier to print.

Drafter - features

Drafter automatically creates drawing of longitudinal cross-sections and schemas of pipe networks, pipelines and so on. It was developed to work especially with sewage networks, however it can be successfully used for other systems like: water supply, gas or drainage.

Drafter offers help from entering the data of nodes, automatically calculating some values, until a pipe network drawing is generated. The drawing can be modified in most CAD applications or printed directly from Drafter.

Drafter:

- automatically determines characteristic values,
- allows to create drawings: longitudinal and vertical cross-sections, schemas,
- generates drawings in the vector format DXF (possibility of further processing in CAD type applications),
- allows to import of nodes and crossings from DXF files (maps),
- allows direct printing generated drawings without any additional software,
- includes objects, e.g.: septic tanks, manholes, dampers, pipes,
- generates pipe networks statistics (excavation volume and area, minimal and maximal: depth and drop, etc.),
- allows to export of data to applications allowing among other further analysis of flows: Epanet and EPA SWMM,
- greatly speeds up the creation and modification of drawings,
- is flexible - has many configuration options.

Project parameters

Phrases used in the manual:

- **Project** - file saved with .kre extension which can contain one or more pipe profiles of various type of networks,
- **Profile** - a part of the project which refers to two or more following nodes (junctions), one profile is assigned to one tab in **Data** table,
- **Active node (junction)** - node (which is also one row in **Data** table), which contain the active (highlighted) cell.

In the **Project parameters** window (Projects > Parameters), you can enter data describing the projected object and the personal data of the investor (the **Main data** tab), and designers involved in design work (the **Designers** tab). For rows **User ...** you can assign any, not provided values. Once entered the data may be used:

- to documents creating from **Templates**,
- in **Infotable**

It is possible to copy your data from another project. To do this, press the **Paste from file** button and select the file from which data will be copied. Before copying the data, they will be listed in the table and you will be asked to confirm them.

Working with projects

You can save your project by pressing the  button, which can be found on the toolbar of the main window of program. To load an already existing project just press the  button and choose the file. You can also press the arrow on the right, and choose from four recently used projects.

In case of moving .kre files between different PC's, remember to make sure that version of the Drafter is the same on both of them. Otherwise, some data from a newer file can be lost and will not be visible on older version. You can check a version of the program by choosing from menu (Help > About)

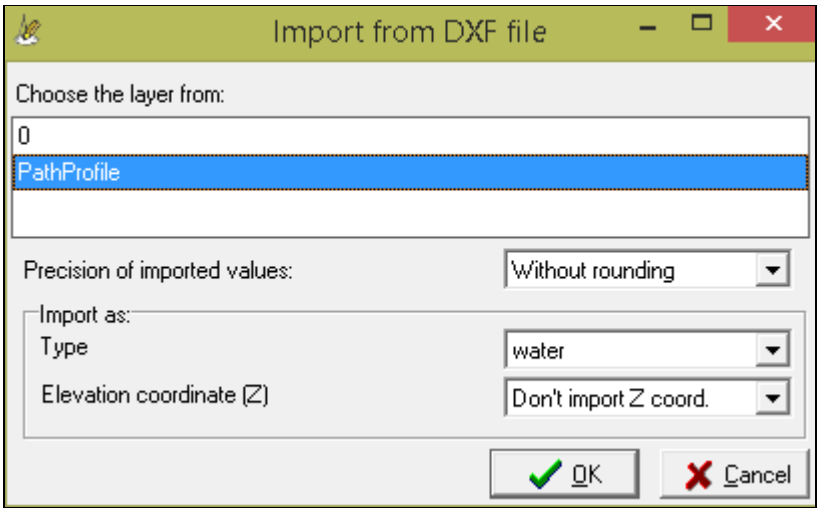
WARNING! Every time you press the **Preview**, **Print** or **Stats** buttons "rysunek.dxf" file will be saved in the folder where the Drafter is installed. This file contains last generated drawing. In case of problems with functioning of button **Open and edit drawing in the CAD program**  you can open that file directly in CAD program that you use. It is not recommended to work on the "rysunek.dxf" file in Drafter and CAD program at the same time.

Correct functioning of the button **Editing of project made with CAD program** depends on assigning the .dxf extension in the Windows system to the app which such files are meant to be opened in. For pressing the button bring the desired effect (opening the CAD app and generated expected drawing in it), a program that reads .dxf files must be installed on the computer. However, that program has to be set as a default app opening .dxf files. Most of programs using .dxf extension make themselves a default one during the installation process. If it did not happen then, you can do it by clicking on .dxf file with your right mouse button, then "open with" and choose your CAD app from the list of apps. Remember to check the "Always use this app to open .dxf files" option.

Data import from digital maps saved as .dxf file

In Drafter it is possible to import from a digital map coordinates of junctions of the pipeline that you are currently projecting. The file containing the map should be defined as .dxf and saved as text type document (it is sometimes called as ASCII with ANSI character encoding). In case of having your digital map saved as .dwg, you should export it as .dxf using your CAD app.

To import your file, choose in menu (Profile > Import of coordinates from DXF file) and next choose a proper file, from which junctions will be imported from.

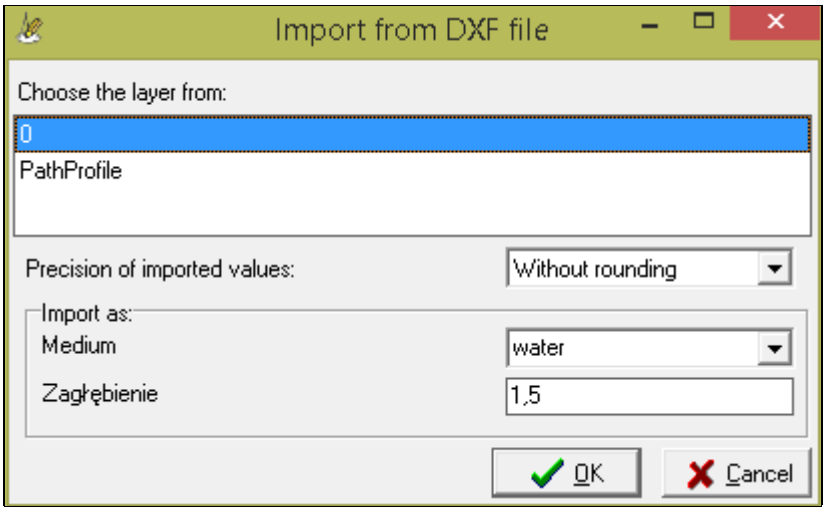


Next, in **Import of coordinates from DXF file** window, choose a layer from which junctions will be imported from. Junctions/points in .dxf file must be defined as lines or polylines and they cannot be located inside a BLOCK section. When the layout you imported to the Drafter will not belong together (will be consisted of separated fragments), then try to retry a importing procedure with limited decimal places of imported numbers.

You can also define type of imported pipeline in dropdown list **Type**. If imported junctions have their height coordinate "Z" defined in .dxf file, you can choose as which argument it will be imported (for example as ordinate of ground, ordinate of the pipe etc.). Not importing a "Z" coordinate is set as a default option. "Z" coordinate can be attributed in most of CAD programs. It can be done only to line type objects - it is not possible to polylines.

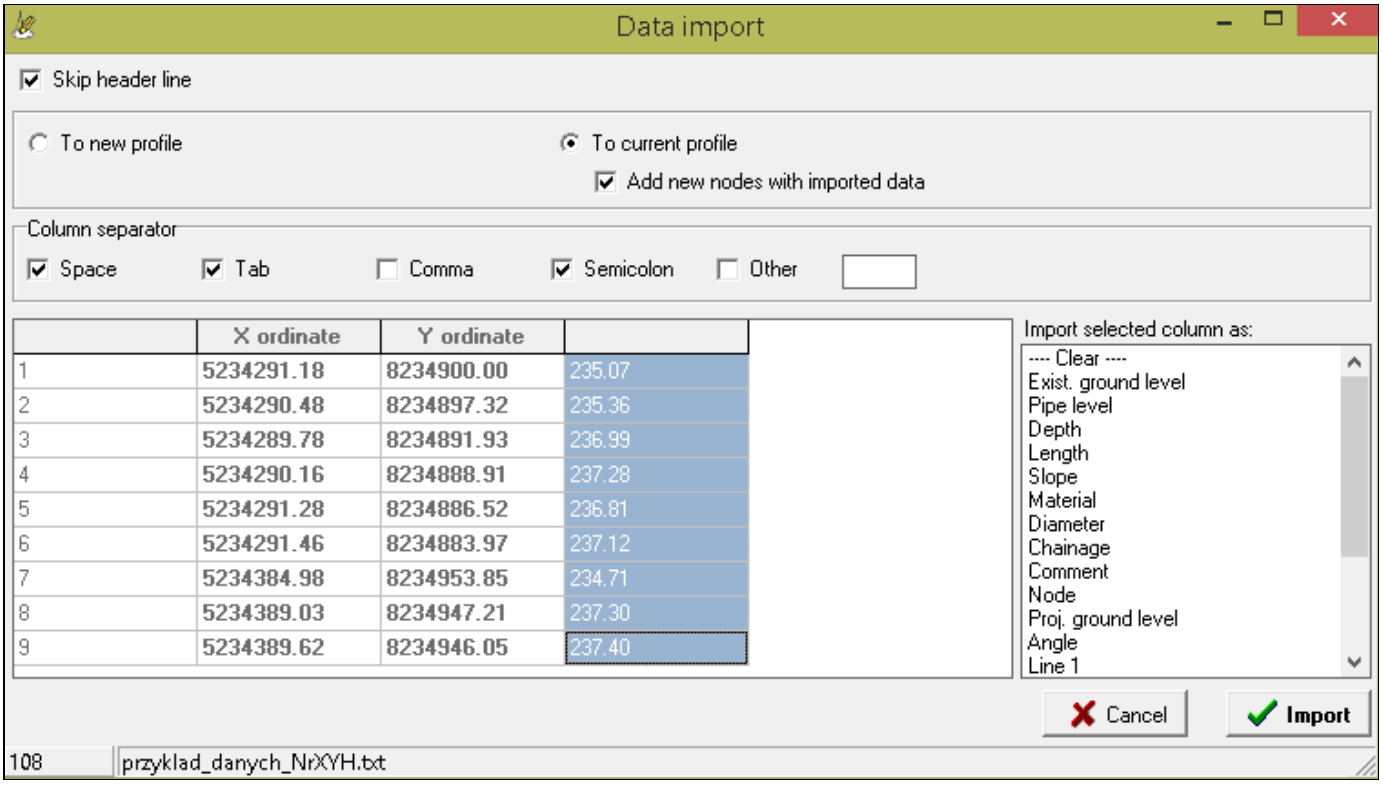
If **Calculate length and angle from coordinates** option is checked (Tools > Settings > "Data" table > Table parameters), the Drafter will automatically calculate distances between junctions and angles between segments, based on the entered data. Other parameters are needed to be entered by the user (ordinate of ground, ordinate of the pipe etc.).

If imported .dxf file includes another network/pipeline which is not the one that you are working on, but colliding with it, you can optionally import that collisions to the Drafter. To do that choose from menu: (Profile > Import collision from DXF file) and select proper file and layer including a colliding network. You can also define a network type and depth of the imported collisions. You need to check in **Settings** window **Crossing's depth is constant** option by choosing menu: (Tools > Settings > Others > Crossings Editor) to make the entered depth of collision applied. Then you can enter some missing properties of collision in **Crossings editor**.



Data import from text files

In the Drafter, you can import some junctions data from a text type file (.txt, .csv files etc.). To do this, choose from main window menu: **Project > Import...** and select a proper file to import data from.



Next you have to define if the data will be imported to a current profile or to a new one. If you choose import to the current one and check **Add new nodes with imported data** option, the Drafter will add new junctions and assign imported data to them. **If you do not check that option, data will be imported into already existing junctions, overwriting the data in them**.

In the **Column separator** groupbox, specify what character is used to separate the data on each line of the text file.

Under the above group of options there is a table containing a preview of the first few lines of the selected file, including the settings you have made. If the first line of the table shows headers of data to be imported, it is advisable to check the **Skip header line** option. This will prevent headers from being imported.

After selecting a data column, you must indicate as which data it is to be imported. This is done by double-clicking with the left mouse button on the list of data types **Import selected column as**, located on the right side of the table. The first item in the list named **---- Clear ----** allows to delete a previously defined assignment. The assigned columns have a specific title, and the values in them are written in bold. Only columns assigned to data types from the above list will be imported.

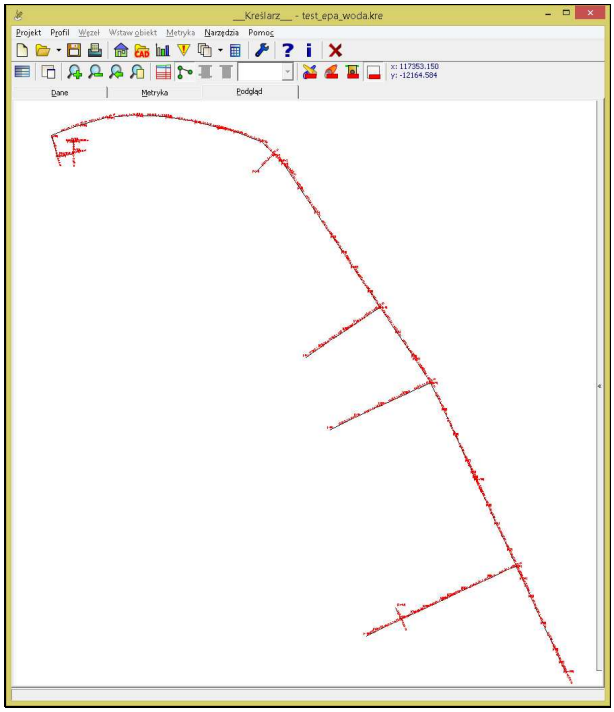
The number to the left of the status bar at the bottom of the window determines the number of lines that will be imported.

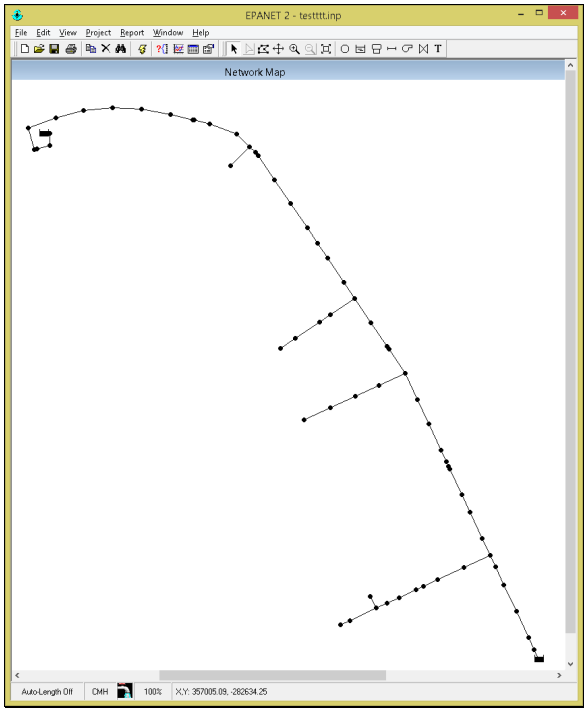
When you are importing a pipe ordinate, the ordinate of the bottom of the pipe is imported.

Data export

To export data for use in the other apps, select **Project > Export** in the menu. Note that the complete project is not exported. You can perform an export to the following file types:

- **Epanet** (.inp extension)- the exported file can be used in Epanet to analyse, among other things, flows and pressure losses in water networks. This is a free tool developed and provided by the U.S. Environmental Protection Agency. More information about the program can be found on its website. Below is a schematic of the water distribution network in the Drafter and in Epanet, loaded from a file exported form the Drafter.



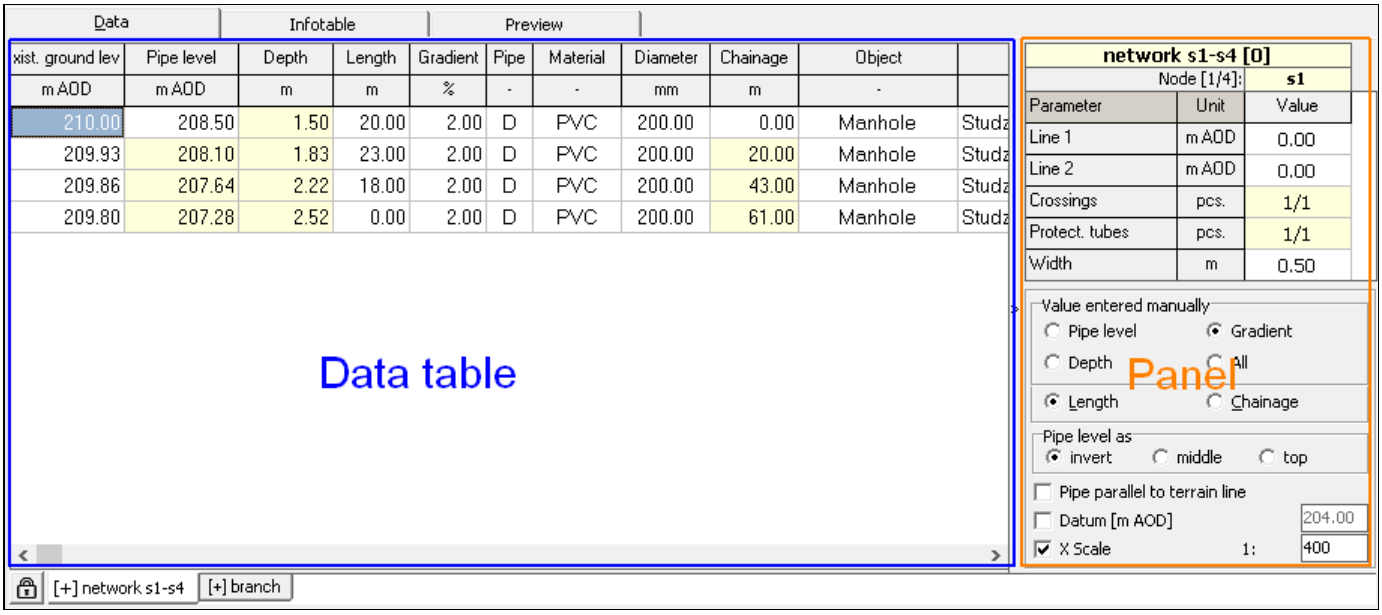


Data is exported from profiles for which **Type** (in the **Panel**) has been specified as **Water Supply**, to the extent that they are in the Drafter. Some data may need to be completed in Epanet to perform a full analysis.

- **Epa SWMM** (.inp extension) - the exported file can be used in the Epa SWMM program to analyse, among other things, flows and pressure losses in sewer networks. This is a free tool developed and provided by the U.S. Environmental Protection Agency. More information about the program can be found on its website. Data is exported from profiles for which the **Type** (in the **Panel**) has been specified as **Sewer** or **Storm water** , to extent that they are in the Drafter. To perform a complete analysis in Epa SWMM some data may need to be supplemented in Epa SWMM.

- **HTML** - allows to display the Data table with its content in the Web browser.
- **CSV** - allows to display the Data table with its content in a spreadsheet.

Data table and Panel



The **Data** tab is the most important part of the Drafter. It is where most of the data describing the individual nodes of the profile are entered. The tab is divided into three parts:

1. The largest one, the **Data** table with most frequently used parameters that characterize the pipeline.
2. **Panel** ? flyout in the right part of the tab, with less frequently used columns of the **Data** table and parameters of the created profile.
3. Tabs bar ? in the lower part, with names of profiles included in the project

The **Panel** can be extended/retracted in several ways:

- Clicking the  button on the toolbar
- Pressing **F4** button on the keyboard
- Choosing in menu: **Tools> Panel**
- Clicking on the right edge of the screen (while using program in full screen mode)

In addition, you can make another settings in the **Panel** and they are described in **Profile parameters** section.

To change the order of columns in the **Data** table or rows in the **Panel** , left-click on the column/row and hold down the button to move and drop it to another location.

You can move individual parameters between the **Data** table and the **Panel**. To move a column from the **Data** table to the **Panel**, right-click on any value in the table and select **Move column ? to Panel**. Similarly, you can move a parameter from the **Panel** to the **Data** table.

For most columns in the **Data** table it is possible to bulk change the values for selected nodes (ways of selecting multiple nodes are described in the chapter **Data entry rules**). To do this select the nodes for which you want to assign a new value. Then right-click on the selected area and select **Change value?** option. In the **Change value** window you can enter a new value and choose the way of using it:

- **As below** - the existing values in the selected cells in the Data table will be replaced by the new value
- **Multiply by** - the existing values in the selected cells in the Data table will be multiplied by the defined value (e.g. change the sign to the opposite when entering the value -1)
- **Change by** - the existing values in selected cells in the Data table will be incremented/decremented by the specified value (e.g. raise the ordinate of a pipe bottom by the specified value)

Confirm the data you have entered with the **OK** button or the **Enter** key.

In the tabs bar below the **Data** table the individual tabs correspond to the successive profiles included in the project. The tabs contain the names of the profiles. When you right-click on a profile name, a menu unfolds:

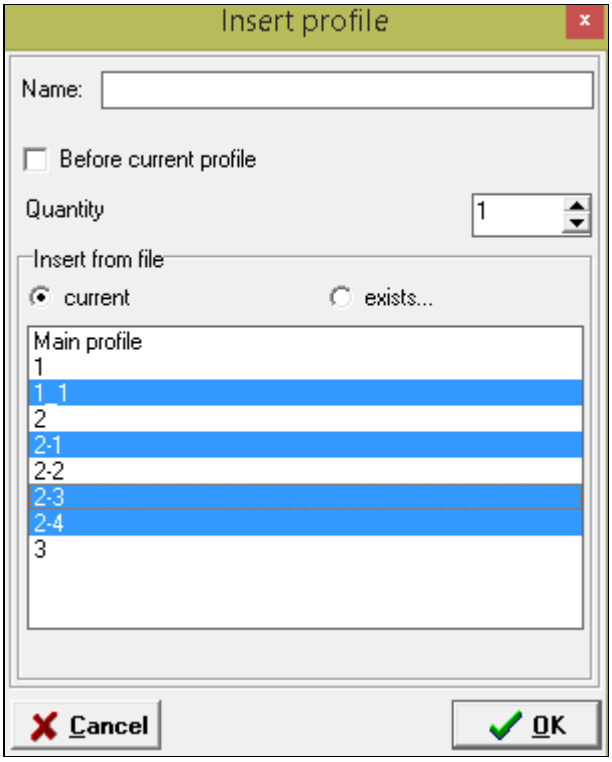
- **Include > Current profile (on/off)** - enables/disables the current profile

- **Include > Only current profile** - disables all profiles, leaving only the current profile enabled
- **Include > Only profiles of Type?** - leaves only profiles of the type of the current profile enabled
- **Include > All profiles except the current one** - enables all profiles except the current one
- **Include > All profiles** - enables all profiles
- **Insert** - inserts a clean profile into a profile
- **Duplicate** - adds a new profile based on the current one
- **Delete** - removes selected profile (project must contain at least 1 profile)
- **Mark as main** - option useful when automatically created scheme drawing does not meet expectations
- **Rename** - invokes the window allowing you to give a new name to the profile
- **Attach to the profile** - invokes the window allowing to select the profile to which the current profile will be attached.

A disabled profile, among other things, is not included in generated drawings and lists of materials. The names of enabled profiles are preceded by [+] and disabled by [-]. The tab order can be changed by ?grabbing? it with the mouse and dragging it to a new place. Change of tab order is not possible when padlock icon on the left side of the bar is activated.

A new profile can be inserted into a project in several ways:

- By selecting the **Insert** or **Duplicate** option in the aforementioned menu,
- By selecting menu: **Profile> Insert** or **Profile> Duplicate**,
- By clicking a  icon on the toolbar



Performing any of the above actions will cause the **Insert Profile** window to appear. In the **Name** textbox you can enter the name for the created profile. Leaving the field empty will cause the program to assign the name automatically. When inserting a profile that already exists in the current file or another file it is recommended to leave the **Name** field empty. Then the name of the imported profile will be assigned to the inserted profile (or multiple profiles).

By default the new profile is inserted after the currently selected profile in the list of tabs under the **Data** table. If the option **Before current profile** is checked then the new profile will be inserted before the current tab.

When inserting a profile from a file, after selecting the file, a list with the names of profiles existing in it becomes active. In the list you can select one or several (e.g. by holding down the **Ctrl** key) profiles.

By means of number specified as **Quantity** you can impose number of inserted profiles. Default value is 1. Eg. if you have defined connection/branch template in some file, you can immediately insert required number of connections into new project, by changing **Quantity** parameter.

Data table - shortcuts

Table navigation:

Cursors - move to the next cell;
Home - activates the first cell in the current node;
End - activates the last cell in the current node;
Ctrl + **Home** - activates the first cell in the table;
Ctrl + **End** - activates the last cell in the table;

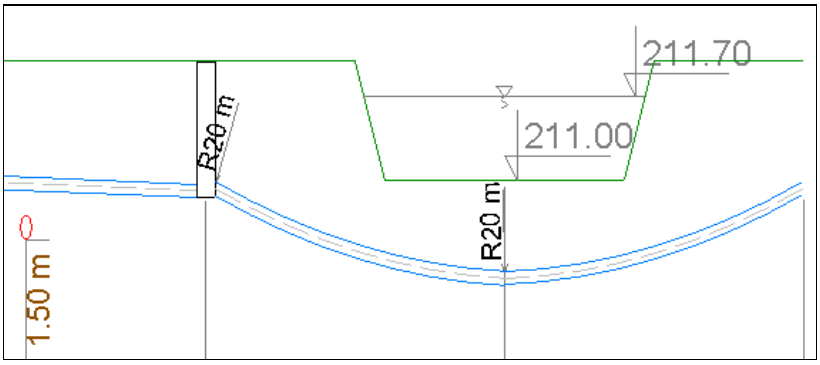
Data editing:

F2 - goes to edit cell;
F3 - opens the window for editing object parameters (if any is inserted in the node);
Insert - inserts a new node (row) above the currently selected node;
Delete - deletes the currently selected node;
Cursor down ↓ - if the last node is currently selected, adds a new one at the end of the table;
Shift + **Cursor down ↓** - copies the contents of the selected cell to the cell below;
Shift + **Cursor up ↑** - copies the contents of the selected cell to the cell above;
Shift + **Alt** + **Cursor down ↓** - copies the current node's data to the next node;
Shift + **Alt** + **Cursor up ↑** - copies the current node's data to the previous node;
Ctrl + **C** - copies the selected cell to the clipboard;
Ctrl + **V** - pastes the clipboard contents to the selected cell;
Ctrl + **Shift** + **C** - copies data from the selected node to the clipboard;
Ctrl + **Shift** + **V** - pastes the clipboard contents to the selected node;
Ctrl + **Z** - undoes the last action;
Ctrl + **Y** - restores the last retracted operation;
Enter - confirms entered data and closes the cell editing mode;
Ctrl + **[mark the range with the mouse with the left button pressed]** - selecting and ability to copying (e.g. to the spreadsheet) the area with data;
[double-click on the column title] - switches the column to manual entry mode (instead of automatic enumeration). This applies to columns: Pipe level, Depth, Slope;
[double-click in cell] - adjusts the column width so that the longest text in the column is visible;

Entering data to the table

In the Data-table one should enter details about nodes and sections of the considered profile. In the following table a characterization of individual data was compared:

Column title	Sample	Unit	Description
Ground level	212,50	m AOD	in order to implement ordinates of the ground level different for left and right side of the node type e.g. : "212,11/210,50". Options the one it is possible to use at drawing e.g. the dodge, of terrace.
Invert level	211,30	m AOD	Using record like mentioned above for ordinate pipe bottom it is possible to get the cascade well (when the ordinate of the intake into the well is larger than of take-off) or pump station - implementing the ordinate from left smaller than from right.
Depth	1,25	m	
Length	5,50	m	
Gradient	2	%	enter the fall pipe preceded by a minus sign "-" (e.g. -2) we get the opposite drop pipe, eg pumping cord.
Material	PCV	-	choice between: PCV, PE, PP (how to add other material)
Diameter	110	mm	record e.g.: "110/160" will be interpreted by the program as "pipe in the pipe" (applying e.g.: thermal insulation of the pipe, sleeve protective tube) .
Chainage	5,50	m	value determined automatically. (Except for the first node where you can insert any value)
Object	-	-	Available objects and how to enter them in chapter Objects
Comment	any text value	-	placed below the profile at a given node typed text
Node	any text value	-	placed above the table name of the node
Line	210,50	m AOD	allows you to put on an additional line in profile, e.g. ground water level, the proposed land line, etc. Recording format analogous in "Ground level" column. (how to specify line name)
Angle	45	°	inserts the value of the angle of refraction, together with the symbol of the direction. The introduction of a negative value will change the direction of collapse.
Trench width	0,50	m	value used to determine the volume of excavation and the creation of cross-sectional drawing. The use of recording such as: "1.20 / 0.80" allows you to get a sloping trench walls (the first value defines the "top" width of the trench and the second width measured at the bottom).
Section flow	1,50	m3/h	The flow that originates at a given section and at the beginning node of the section (with flow direction from left to right) or at the end node of the section with flow direction from right to left. The default flow unit [m3/h] can be changed in the **Configuration** window on the Data table tab.
Flow	10,50	m3/h	Total flow on a given section, with automatic consideration of preceding sections and inflows from connections. The value is calculated automatically and cannot be changed by the user.
X scale	1000	-	Horizontal scale (X) for a segment. This option is useful e.g. when the profile contains a long segment with no collisions. For such a segment a smaller scale can be set than for the whole profile. This will make the printout of the profile drawing shorter.
X coordinate	1	-	The X coordinate of the node. Length and angle values can be automatically calculated from the coordinates. The coordinates can be imported from a map in dxf format (menu: Profile > Import coordinates). If you need to scale coordinates after import, you can do it in bulk by clicking on the column heading in the table and selecting "Change value".
Y coordinate	1	-	The X coordinate of the node.
Bending radius	1	-	Pipe bending radius with value different from zero allows to have pipe in curved form on the profile drawing (instead of default straight section). This form can be useful e.g. when designing steerable jackings. Below is an example.



Data exchange between columns of the table

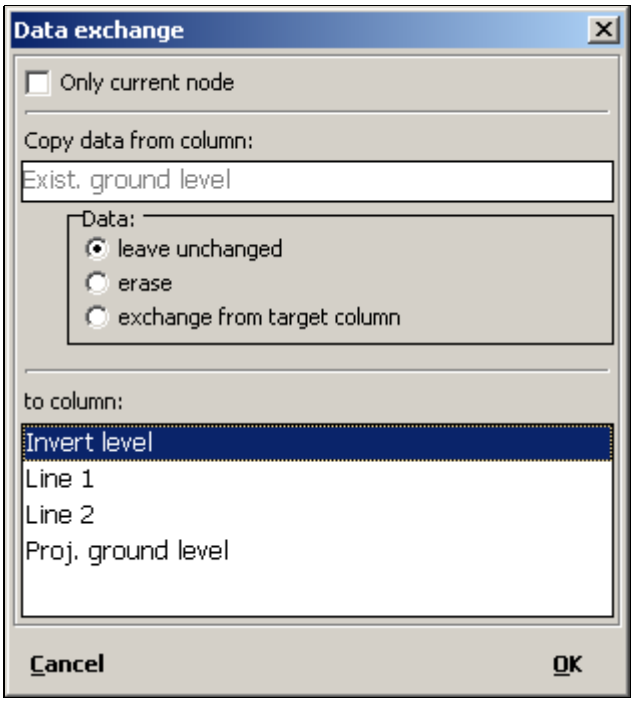
It is possible to quickly move data between columns in the **Data** table, where the values are expressed in m AOD. To do this, left-click the column from which the data will be moved an then right-click and select **Move data to column...**

In the window that opens, you can decide by selecting the appropriate option whether the data in the source (clicked) column will remain after the operation:

- left unchanged (only copied to the target column),
- deleted - null values will be assigned,
- replaced by data from the target column.

The target column can be selected from among the columns where the data is expressed in m AOD (except from the column that was selected as the source column).

The above operations can only apply to the selected node (and not the whole column) if the **Only current node** option is checked.



Network creating rules

The Drafter automatically creates a network scheme drawing. If a project consists of many profiles, nodes with identical names are connected (the program is not case-sensitive). To achieve a correct linking, the following rules must be followed:

- The main profile (e.g. network) should be placed in the project before shorter profiles (e.g. connections) that will be connected to it.

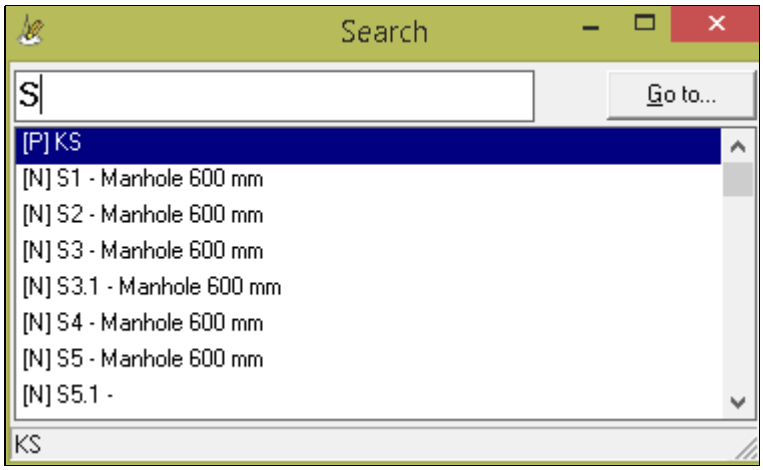
  If the shape of the network scheme differs from that expected, you can manually indicate main profile by right-clicking on the profile name and selecting **Mark as main**.

- The profiles that are to be connected must be of the same type. The **Type** is specified in the **Panel**
- The node that will be used to connect a profile to the network should be an extreme one (the first or the last)

The angle at which the profile is to be connected to the network must be entered in the **Angle** column of the **Data** table. By changing the sign to minus "-" you can get a connection from the other side of the network. There is a possibility of automatic shortening of too long sections in a schematic. The length above which the sections will be shortened may be defined in the **Settings** window, in the Parameters section.

Searching data

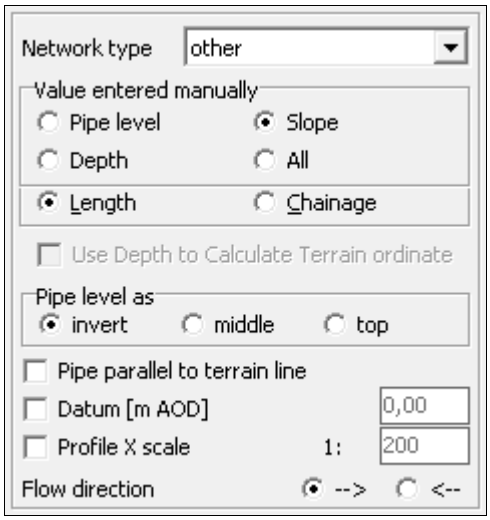
In multi-profile projects, it can be difficult to move efficiently between profiles and nodes. In order to make it easier, the ability to quickly search for a profile or node has been introduced. To access the **Search** window, press **Ctrl** + **F** or select **Profile > Search** from the menu.



To find a profile or node, type the desired name or part of it in the **Search** field. As you type, the list of found items updates dynamically. Found items for profiles start with **[P]** and for nodes with **[N]**. Double-clicking on a search entry or pressing the **Go to...** button closes the search window and moves the cursor to the selected profile or node.

The search only works within the currently open project/file and is case sensitive (e.g. "Profile" and "profile" are two different names).

Profile parameters



The basic parameters of each profile contained in the project/file can be changed on the **Panel** on the **Data** tab (see the **Data** table chapter for ways to activate the Panel).

In the **Type** drop-down list, you can select the type of pipeline to be designed from:

- water supply,
- sewage system,
- storm water,
- gas pipeline,
- heat pipeline,
- melioration,
- other.

The selected pipeline type affects, among other things: the way flows are summed up, creation of scheme drawing (pipelines of the same type are connecting automatically), automatic updating of pipe ordinates in the same named nodes.

By default, the automatically determined values in the **Data** table are: pipe ordinate, pipe depth. The value that needs to be entered manually is pipe slope.

These settings can be changed by marking as **Specify column with manually inserted values**:

- **Pipe level (ordinate)** - the program automatically determines the value for pipe depth and slope,
- **Pipe depth** - the bottom ordinate and slope are calculated automatically,
- **Slope** - the program automatically determines the ordinate and the bottom depth,
- **All**- only the distance or length is determined automatically, depending on the settings.

The manually entered value can also be changed by double-clicking in the **Data** table on the column heading whose values are to be manually changed.

When **Use depth to calculate terrain ordinate** checkbox is checked (can only be selected when Depth is selected as the manual input value) determines the ground ordinate based on the Depth and Pipe ordinate values (Pipe ordinate must be specified in the first node).

The Pipe level column in the **Data** table can indicate one of the following values:

- The bottom of the pipe (invert),
- The centerline of the pipe (middle),
- Top of the pipe (top).

The choice should be made in the **Pipe level** options group. The choice also affects the way of joining pipes on the profile, e.g. if there is a reduction in diameter at a node, selecting the option **Pipe top** will align the tops of pipes.

When the **Pipe parallel to terrain line** checkbox is checked, after entering the **Depth** value in the first node and entering, subsequent **Terrain ordinates**, the program automatically determines the characteristic values of subsequent sections so that the pipe line runs parallel to the terrain line.

By default, the comparison level is determined automatically by the program. Sometimes, there is a need to specify a different value, then you should select **Datum** checkbox and enter an appropriate value into edit field. If project contains more than one profile, after confirming entered value you can assign it to other profiles by choosing appropriate button in displayed dialog window.

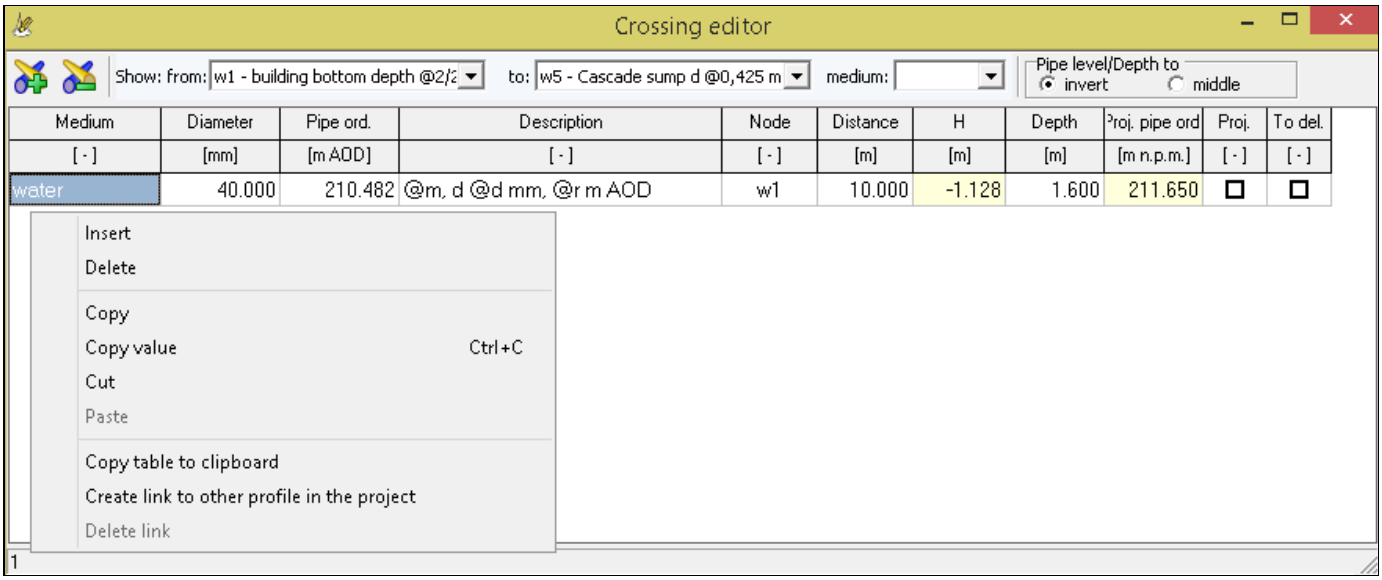
By default the X (horizontal) scale of the profile drawing is assumed as entered in the table (or calculated by the program) in the **Preferences** window. If there is a need to specify a different value of **X scale** for the selected profile, then select the X scale option and enter an appropriate value into the edit field. If project contains more than one profile, after confirming entered value it is possible to assign it to other profiles, selecting appropriate button in displayed dialog window.

By clicking on the appropriate mark, you can specify the **Flow direction**, from:

- **-->** left to right, that is, from the first node/row in the **Data** table to the last,
- **<--** right to left, which is the opposite of the above.

The flow direction can be plotted on drawings. To do so, select the checkbox **Mark flow direction on longitudinal section and scheme drawings...** in the **Preferences** window on the Drawing tab.

Inserting crossings



Methods of activation:

icon: menu: (Tools > Protecting Pipe Editor)

keyboard shortcut: **Ctrl** + **K**

Crossings editor used to enter the crossings occurring in the profile. Crossings should be introduced after entering data in the **Data** table in the main window. The precision of the numbers placed in the window can be set in **Configuration > Data Table > Decimal places**. The user can filter the amount of crossings presented by selecting the nodes between which they are situated.

Editor columns description:

- **Network Type** - name of the cable/pipe causing a collision (eg water, gas, phone, etc.).
- **Diameter** - diameter of pipe causing a collision given in millimeters. If the pipe has e.g. sleeve protecting tube or insulation **diameter** column needs to be filled as follows: "crossing_diameter**/**protecting_tube_diameter", where: crossing_diameter - diameter of the collision, protecting_tube_diameter - diameter casing pipe of a collision. Protecting pipe is also taken when generating the drawing.
- **Pipe bottom ord.** - bottom ordinate of pipe causing a collision given in m AOD
- **Desripiton** - Description of the collision, which will be shown in the drawing. The description can be used **variables**. Thanks to them, most of all collisions may have the same desription. User has access to the following variables:
  - **@m** - network type
  - **@d** - crossing diameter
  - **@o** - sleeve protecting tube diameter
  - **@r** - crossing pipe bottom ordinate
  - **@h** - crossing distance from the designed pipeline
  - **@z** - crossing pipe bottom depth
  - **@t** - ground level at the point of collision
  - **@p** - ordinate the designed pipeline at the point of collision (axis, bottom or top depending on the state **Invert level** on the **Data** tab)
  - **@pd** - ordinate the bottom designed pipeline at the point of collision
  - **@po** - ordinate the axis designed pipeline at the point of collision

  A sample entry in the first collision in the screenshot above "**@m fi @d mm @r**" while creating a drawing will be replaced by text "**water fi 40 mm 211.80**" and as such placed on it.

- **Node** - node name (from **Data** table), which will be given the distance to collision. When you edit this value you must select a node from a list of existing ones. Crossing is assigned to the selected node, ie, the user can freely change its position in the table each time without the need to update the position of the collision.
- **Distance** - distance from node from **Node** column to axis crossing pipe. This value can be negative if it refers to an existing node for the collision.
- **H** - projected pipeline outline distance from the outline pipe/cable causing a collision, specified in meters.
  This value can not be modified by the user. It is continuously updated by the program. Values preceded by "**+**" means that

a collision is **over**, the designed line, while the sign "**-**" - **under** it. Marked in red are designed collisions passing through the pipeline, then column value is "**0**". If, however, one tube line (projected or conflicting) "shall" in the pipe casing of the second (or an aspect only shielded pipes), the text cells turn red, and the value in it is different from zero.

- **Depth** - collision depth expressed in meters above sea-level. If user knows collision depth and does not know the ordinate may in this column, enter the value. Then the column **Pipe bottom ord.** will be updated automatically. Entering data in this column is only necessary when there has been supplemented by a column **Pipe bottom ord.**.

Columns: **Network Type** and **Description** have the ability to be quickly populated with user-defined names in the **Configuration** window under **Most-used values**.

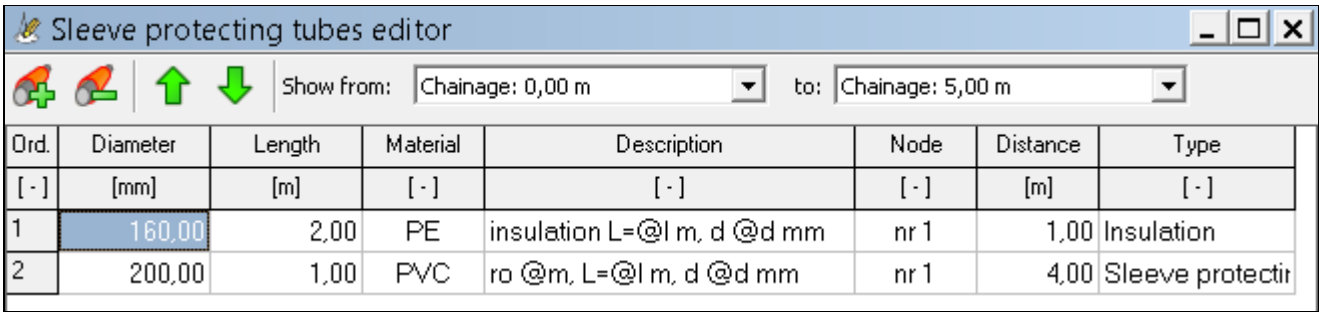
If **"error"** appears in the **H** or **Depth** column, it may mean that a collision was entered outside the profile (before the first or after the last node). Collisions added automatically are described as italic formatted text.

If the collision is a pipeline located in the same project (file), then the collision can be "linked" with it by right-clicking on it and selecting **Create link with other project profile** (the option is available after selecting a node and a distance from it to the collision). After selecting the option, in the window that opens, choose the profile, node and the distance from it where the collision occurs. The associated collision has automatically updated values for the network type, ordinate, depth and diameter. This means that if the depth of the colliding pipe changes, it will automatically be included in the collision parameters, without the need for manual adjustments. To remove the connection, right click on the mouse and select **Remove connection**.

Keyboard shortcuts:

F2 - goes to edit cell;
Insert - inserts a new node (row) above the currently selected one;
Delete - deletes the currently selected node;
Cursor down ↓ - if the last node is currently selected, adds a new one at the end of the table;
Shift + **cursor down ↓** - copies the contents of the selected cell to the cell below;
Shift + **cursor up ↑** - copies the contents of the selected cell to the cell above;
Shift + **Alt** + **cursor down ↓** - copies the data of the current collision to the collision below;
Shift + **Alt** + **cursor up ↑** - copies the data of the current collision to the collision above;
Ctrl + **cursor down ↓** - moves the current collision one position down;
Ctrl + **cursor up ↑** - moves current collision one position up;
Ctrl + **C** - copies the selected cell to the system clipboard;
Enter - validates the entered data and closes the cell editing mode;
[click on column title] - selects entire column;

Protecting tubes editor



Ord.	Diameter	Length	Material	Description	Node	Distance	Type
[-]	[mm]	[m]	[-]	[-]	[-]	[m]	[-]
1	160,00	2,00	PE	insulation L=@l m, d @d mm	nr 1	1,00	Insulation
2	200,00	1,00	PVC	ro @m, L=@l m, d @d mm	nr 1	4,00	Sleeve protectir

Methods of activation:

icon: menu: (Tools > Protecting Pipe Editor)

keyboard shortcut: **Ctrl** + **R**

The protecting pipe editor is used to enter the protecting pipes, pipe insulation, etc. that appear in the profile. Protecting pipes must be entered after entering the data in the **Data** table in the main program window. The precision of entered numbers can be set in Settings > Data table > Rounding. The user can narrow down the number of presented protecting pipes by selecting the nodes between which they are located.

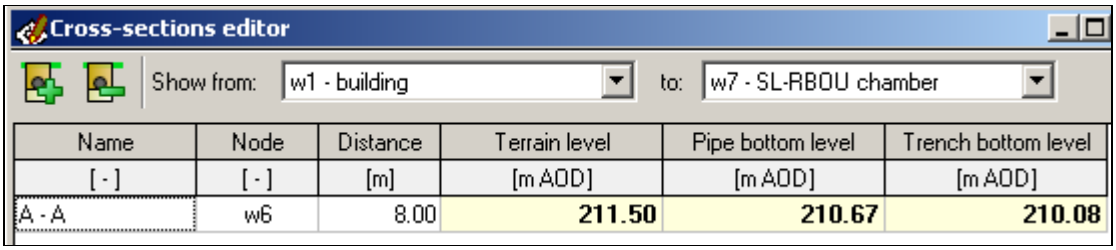
Description of individual columns of the Editor:

- **Diameter** - sleeve protecting tube diameter in [mm],
- **Length** - sleeve protecting tube length in [m],
- **Material** - material from which made is sleeve protecting tube,
- **Description** - description of the protecting pipe that will appear on the drawing. You can use **variables** in the description. Thanks to them practically all protecting pipes can have the same description. The user has the following variables available:
  - **@m** - material
  - **@d** - diameter
  - **@l** - length

  The example notation in the first protecting pipe in the above drawing "**sleeve protect. tube, L=@l m, d @d mm**" will be changed to "**sleeve protect. tube, L=6 m, d 200 mm**" and will be placed there as such.
- **Node** - name of the node (from **Data** table) from which distance to the beginning of protecting pipe will be given. When editing this value the user must select a node from the list of already existing ones. The protecting pipe is assigned to selected node so the user can freely change its position in **Data** table without the need to update the protecting pipe position each time.
- **Distance** - the distance from the node in the **Node** column to the beginning of the protecting pipe. This value can be negative if it refers to the node existing behind the sleeve protecting tube.
- **Type** - type of protecting tube, e.g.: insulation, protective pipe, etc. Depending on the selected type, you can, for example, assign corresponding cost estimate items when generating cost estimate.

Columns: **Material**, **Description** and **Type** have the ability to quickly selecting names defined by the user in a window **Settings > Most-used values**. (more »).

Entering cross-sections into the profile



icon: menu: (Tools > Section Editor)

keyboard shortcut: Ctrl + T

Cross-section editor is used to define cross-sections of designed network. The cross-sections should be entered after entering data in **Data** table in the main window of Drafter. The user can narrow down the number of presented cross-sections by selecting nodes between which they are located.

Cross-sections are marked on profile and scheme drawings. Cross-sections can also be generated as separate drawings.

Description of each column in the Editor:

- **Name** - the name of the cross section (e.g. A-A).
- **Node** - name of the node (from the **Data** table) to which the distance to the section will be given. When editing this value, the user must select a node from the list of already existing ones. The cross-section is assigned to the selected node, i.e., the user can freely change its position in the Data table without having to update the section position each time.
- **Distance** - the distance from the node in the **Node** column to current cross-section. The value can be negative if it refers to a node that exists after the section.
- **Terrain level** - the ordinate of the terrain in the selected section (read-only value).
- **Pipe bottom level** - the ordinate of the pipe bottom in the selected section (read-only value).
- **Trench bottom level** - the ordinate of the bottom of the trench in the selected section (read-only value).
- **Geo** - clicking in **Geo** column allows to add/edit geotechnical cross-section of soil layers in the cross-section area.

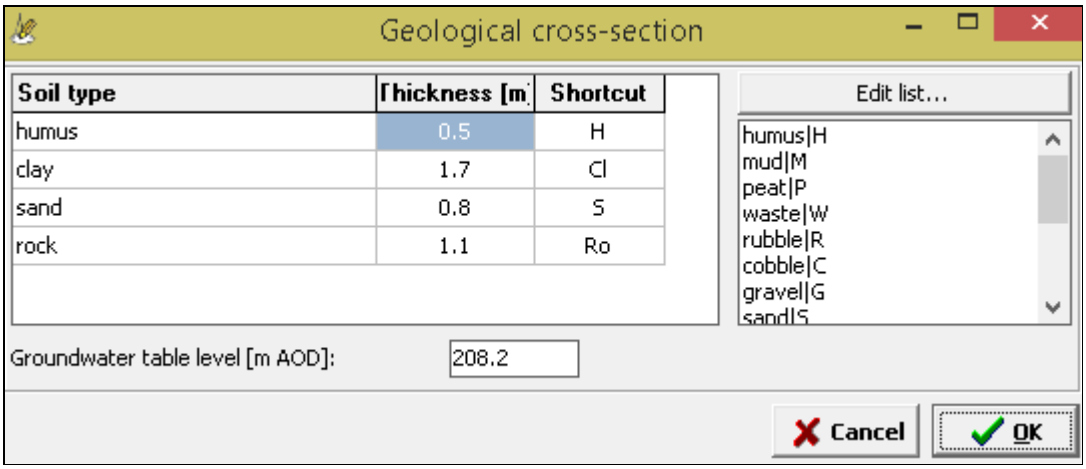
Geological cross section

You can add a geotechnical section to any section added in **Section Editor** by clicking on "+" sign in **Geo** column. When you click on it, the **Geotechnical cross-section** window will open. In it, you should enter successive soil layers specifying for each of them: **Soil type**, **Thickness** expressed in meters and optionally **Shortcut**. If an abbreviation is given, it will appear on the cross-section instead of proper description given in **Soil type** column. To add new row in the table press **Insert** key or while on the last row press **Down cursor ↓** key. To delete the current row, press **Delete** key.

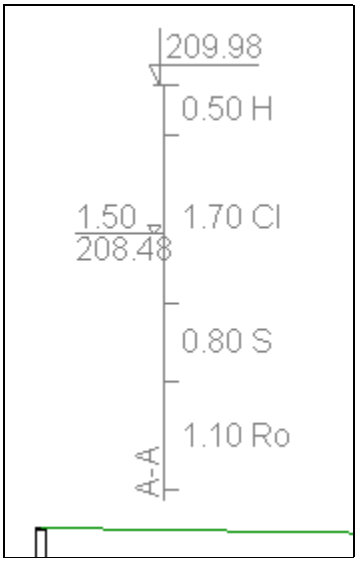
To speed up filling the table, you can use predefined soil types listed in the list on the right side of the window. This list can be customized by clicking the **Edit list** button.... To move an item selected in the list to the table, double-click it with the left mouse button or press **Enter** on the keyboard after hovering over it. If you hold down the **Shift** key, a new row will be added to the table at the end, and the item from the list will be inserted into it. If you hold down the **Ctrl** key, the list item will be appended to the current table row without deleting the existing contents.

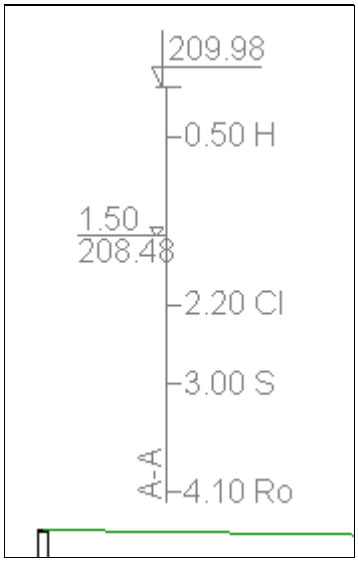
In the **Groundwater table depth** field you can enter the groundwater table depth expressed in meters. Leaving this field empty the water level will not be marked.

The defined geotechnical profile will be plotted on the profile drawing, at the location specified in the**Cross-section editor**.

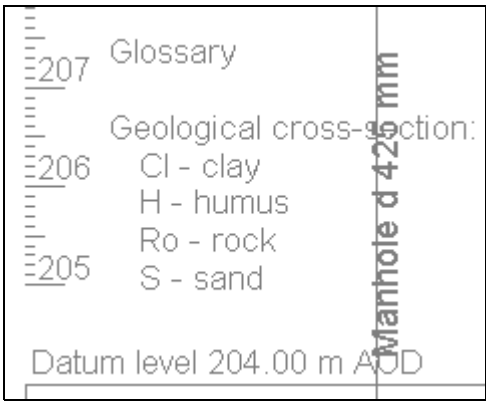


Depending on the state of the **In geological cross-section place descriptions in the middle of layers** checkbox in **Settings** window the cross-section on the profile drawing can look two ways:

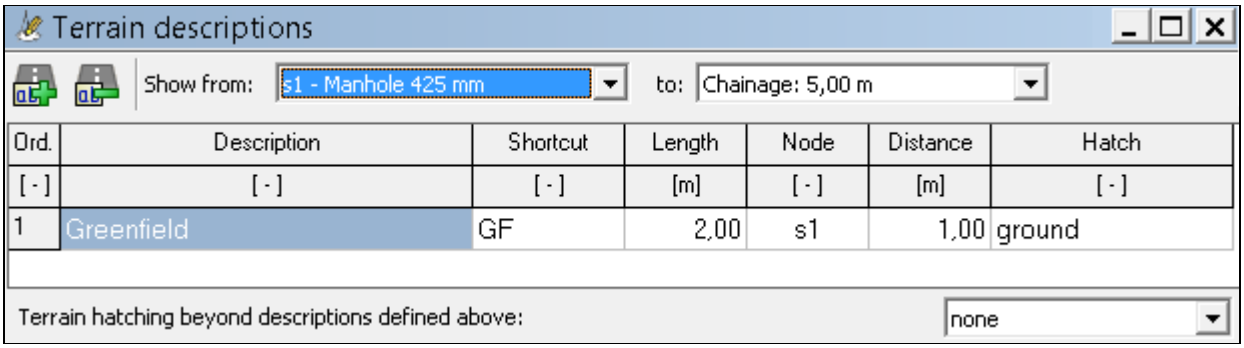




Additionally, if at least one **Shortcut** has been entered, a legend will be placed on the profile drawing:


Glossary
207
Geological cross-section:
206
Cl - clay
H - humus
Ro - rock
205
S - sand
Datum level 204.00 m AOD
Manhole d 425 mm

Terrain description



Methods of activation:

icon: menu: (Tools > Terrain descriptions)

In the **Terrain descriptions** window, you can enter descriptions of the type of terrain under which the designed pipeline passes. In this way, you can mark e.g. roads, lawns, pavements, etc. on the profile drawing.
Terrain descriptions should be entered after entering data in **Data** table in program's main window. The accuracy of the entered numbers can be set in the **Settings > Data table > Rounding window**. The user can narrow down the number of descriptions presented in the editor by selecting the nodes between which they are located.

A description of each column in the table:

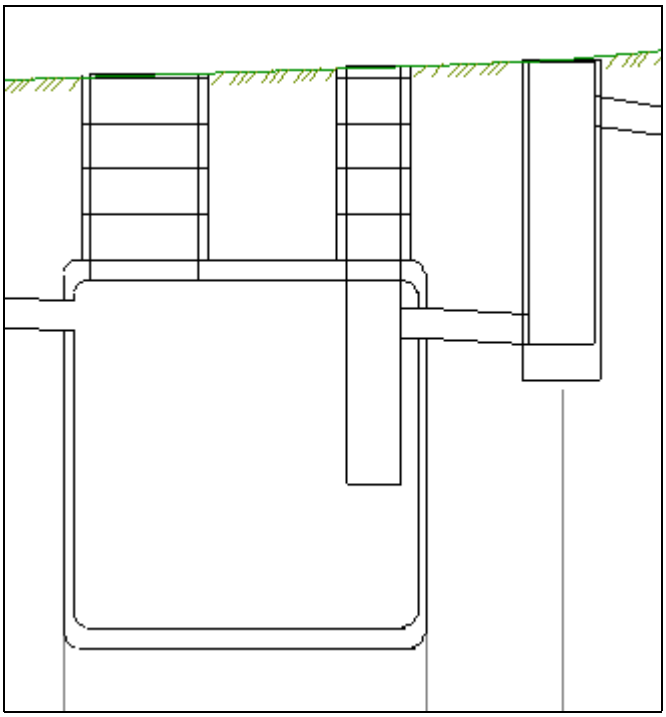
- **Description** - terrain description. **Variables** can be used in the description::
  - **@s** - shortcut (**Shortcut** column)
  - **@l** - length of described terrain (**Length** column)
- **Shortcut** - abbreviated site description, e.g.: GF for greenfield (if provided, appears in the table instead of the proper description). It is recommended to use it when the table cannot accommodate the proper description. If at least one abbreviation is entered, a legend explaining the abbreviation will be automatically placed on the drawing.
- **Length** - length of the described area.
- **Node** - name of the node (from the **Data** table), relative to which the distance to the beginning of the described terrain will be given. When editing this value the user must select a node from the list of already existing nodes. The description will be assigned to the selected node, i.e. the user can freely change its location in the Data table without having to update the location of the described terrain each time.
- **Distance** - the distance from the node in the **Node** column to the beginning of the described terrain. This value may be negative if it refers to a node that exists after the described terrain.
- **Hatching** - the hatch pattern of the terrain type applied under the existing terrain line.

The **Description** column has the ability to be quickly populated with user-defined names in the **Settings** window in the **Most-used values** section. By defining a description in the list, e.g. "Greenfield |GF" and selecting it, the **Description** ("Greenfield") and **Shortcut** ("GF") columns will be filled automatically.

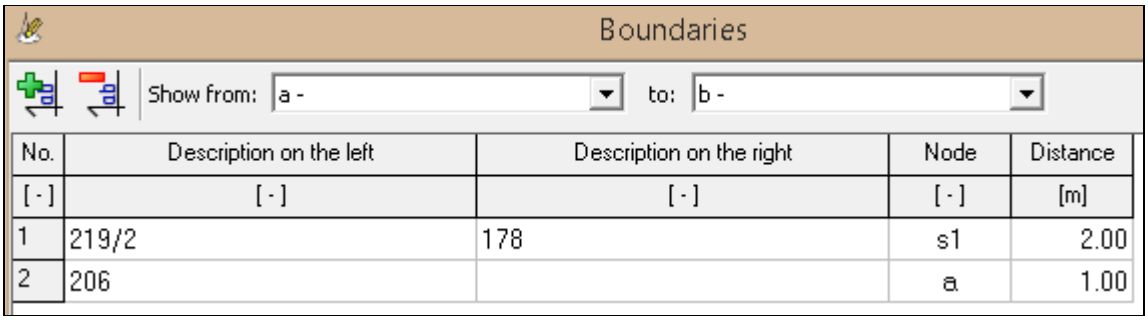
From the list named **Terrain hatching beyond descriptions defined above**, you can select the hatching pattern that will appear under the line of the existing terrain in the entire profile (without definying in the table terrain descriptions), except for: sections defined as terrain descriptions, manholes, and manholes for septic tanks/tanks. From the list you can select:

- none
- ground (pattern: /// /// ///)

Example of a profile fragment with "ground" hatching:



Marking parcels boundaries



No.	Description on the left	Description on the right	Node	Distance
[-]	[-]	[-]	[-]	[m]
1	219/2	178	s1	2.00
2	206		a	1.00

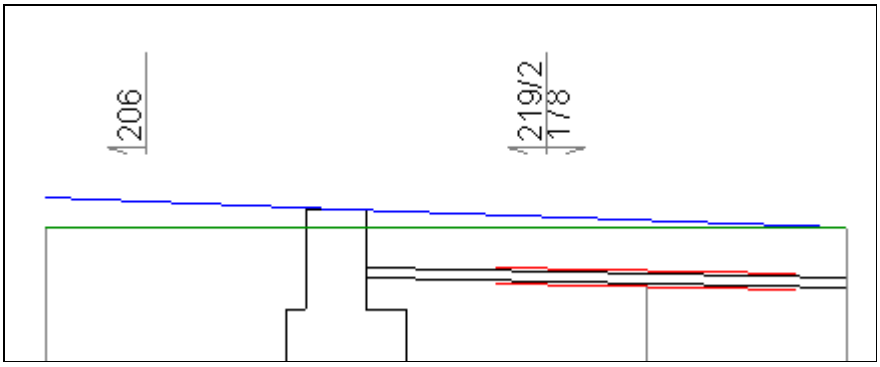
In **Boundaries** window you can enter parameters of parcel boundaries, which are crossed by designed pipeline on its route. The markings are placed on profile and scheme drawings, above the terrain line (example below). On a schematic drawing land plot boundaries are marked symbolically, always at right angle to a pipeline (which is not necessarily reflected in reality).

Boundaries descriptions should be entered after entering data in the **Data** table in the main program window. The user can narrow down the number of boundaries presented in the table by selecting nodes between which they are located.

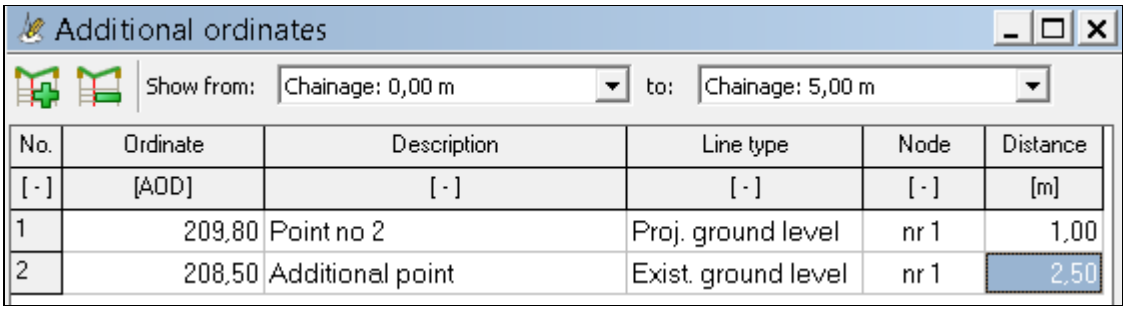
Columns description:

- **Description on the left** - Description of the plot placed on left side of boundary,
- **Description on the right** - Description of the plot placed on right side of boundary,
- **Node** - node name (from **Data** table), which will be given the distance to boundary. When you edit this value you must select a node from a list of existing ones. Boundary is assigned to the selected node, ie, the user can freely change its position in the table each time without the need to update its position.
- **Distance** - distance from node from **Node** column to boundary. This value can be negative if it refers to node placed behind boundary.

Sample:



Additional ordinates



No.	Ordinate	Description	Line type	Node	Distance
[-]	[AOD]	[-]	[-]	[-]	[m]
1	209,80	Point no 2	Proj. ground level	nr 1	1,00
2	208,50	Additional point	Exist. ground level	nr 1	2,50

Methods of activation:

icon: menu: (Tools > Additional ordinates)

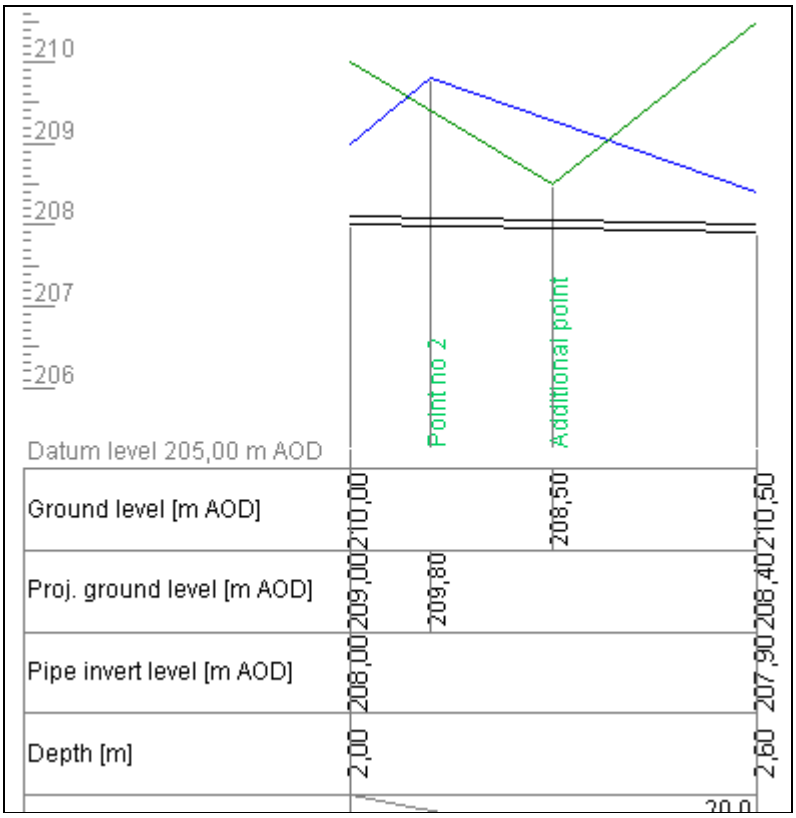
In the **Additional ordinates** window, additional characteristic elevation points can be entered that lie outside the nodes entered in the **Data** table. These are usually locations through which the pipeline to be designed passes, with changing terrain, and at the same time do not require the selection of objects, i.e. the insertion of nodes in the **Data** table. Additional ordinates can be inserted for:

- existing ground level,
- projected ground level - the prerequisite for including the ordinates of the projected ground is to uncheck the option of autofilling the ordinates of the Projected ground level in the menu: **Tools > Settings > Data table > Autofilling > Projected ground level**,
- line 1,
- line 2.

Description of the individual columns of the table:

- **Ordinate** - the ordinate of an additional point expressed in m above sea level (a notation for a vertical step, e.g. 212.30/211.00, is also acceptable),
- **Description** - description to appear on the profile on a vertical point reference,
- **Line** - the line on which the additional point is to appear (to be selected from the above-mentioned list),
- **Node** - name of the node (from the **Data** table) relative to which the distance to the beginning of the described area will be given. When editing this value, the user must select a node from the list of already existing ones. The description will be assigned to the selected node, i.e. the user can freely change its position in the **Data** table without having to update the position of an additional point each time.
- **Distance** - horizontal distance from a node from the **Node** column to the additional point. This value can be negative if it refers to a node existing behind a point (looking in the direction of the profile drawing).

Below is an excerpt from the profile with additional ordinates points plotted.

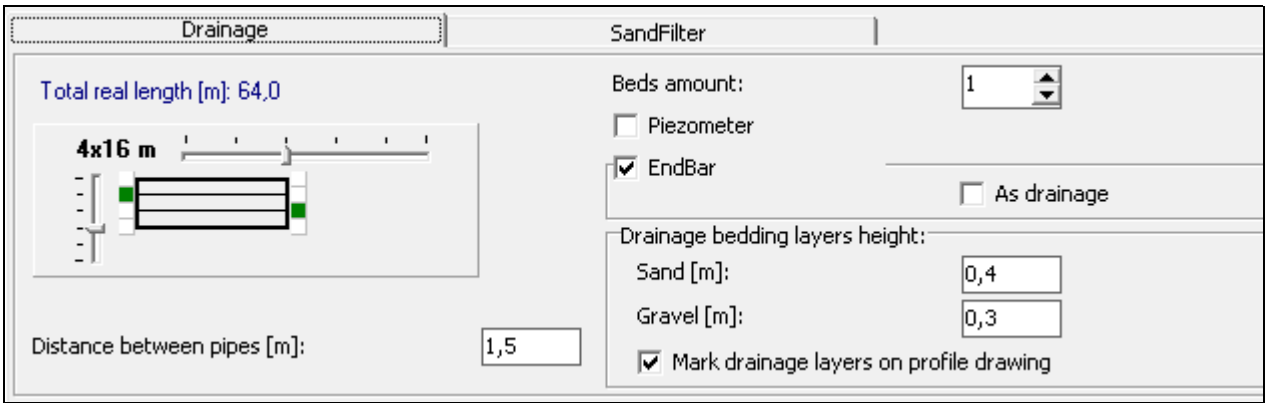

210
209
208
207
206
Point no 2
Additional point
Datum level 205,00 m AOD
Ground level [m AOD]
210,00
208,50
210,50
Proj. ground level [m AOD]
209,00
209,80
208,40
Pipe invert level [m AOD]
208,00
207,90
Depth [m]
2,00
2,60

Sewage treament plant

By default, in the main program window, the **Sewage treatment plant** tab is disabled. To enable it, select the **Show Sewage Treatment tab** checkbox in the **Settings** window - menu: (Tools > Settings > Other).

The program allows you to choose between the two systems of sewage treatment:

Drainage



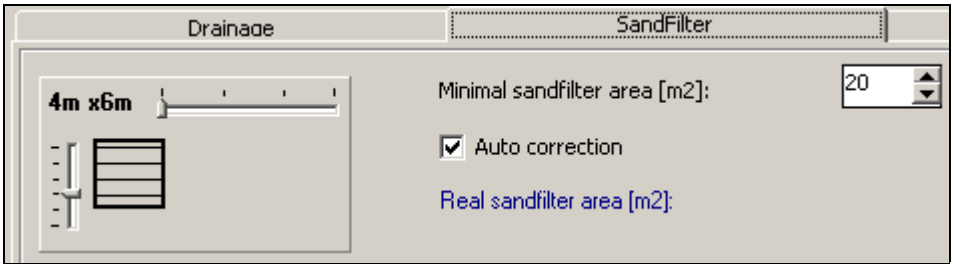
If you checked the **Piezometer** checkbox and in the **Data** table will be inserted ending manhole (with **Kind** parameter as "closing") then on the profile drawing appears a piezometer.

Checking **End bar** option and the amount given in boxes in **Drainage bedding layers height** group has an impact on the calculated volume of excavation.

Selecting the **As drainage** option at the end and the quantities given in the fields **Drainage bedding layers height** affects the determined volume of excavations.

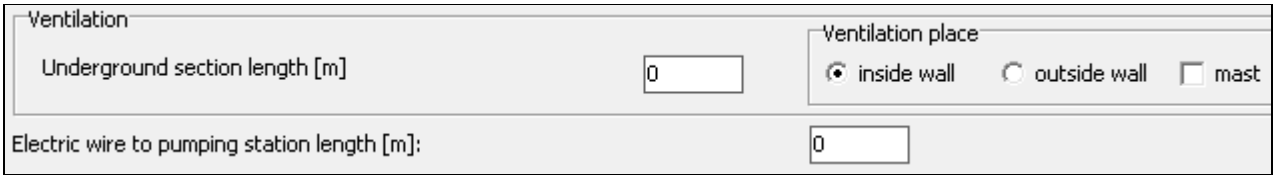
If drainage bedding layers are to be marked on the profile, the option **Mark drainage layers on profile drawing** should be selected.

Vertical sand filter

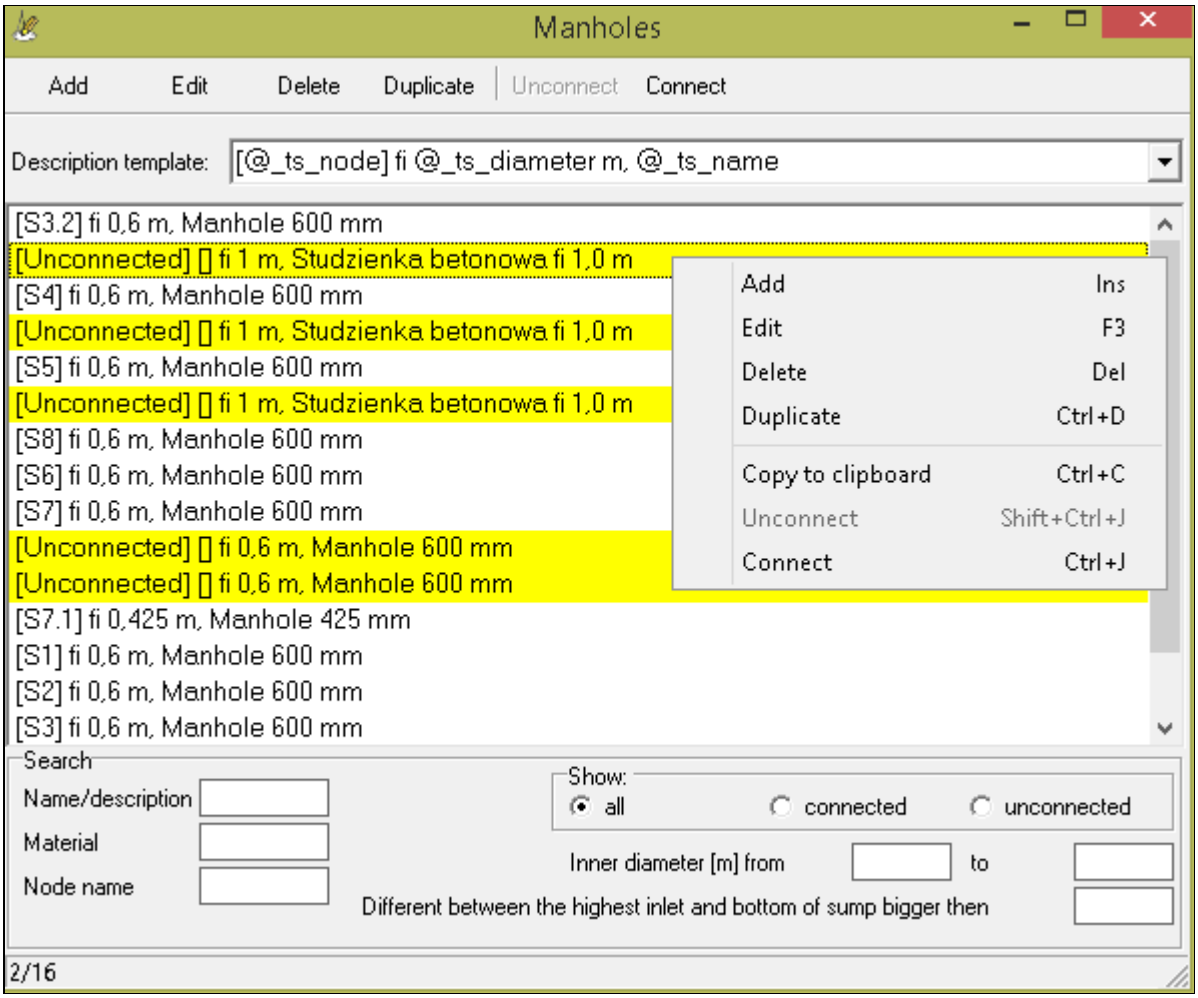


Checking the **Auto correction** box to maintain a constant total surface of the filter. When font color is red it is mean dimensions of the filter are not recommended.

In addition, you can specify the length of the underground section of the "high ventilation" pipe. It will be used to calculate statistics: the volume of excavation and the length PVC110 pipe.



Manholes



Methods of activation:

icon: menu: (Tools > Manholes)

The **Manholes** window lists all the manholes in the project (regardless of the profile in which they appear).

In the **Description template** field, you can use variables to define the format of the displayed data. To edit the parameters of a selected manhole, double-click (or press **F3**) on the choosen list entry.

The manholes marked in yellow are not currently assigned to any node in the project. They can be assigned to a node (**Connect** button), deleted (**Delete** key on keyboard) or left for later use.

Clicking the **Duplicate** button will create a copy of the selected manhole, unassigned to any node, which can be assigned to another node.

The fields in the **Search** group allow you to search for manholes that meet the conditions you specify. You can search based on:

- name or description,
- the material of which the manhole is made,
- name of node where the manhole occurs,
- manhole diameter,
- difference between manhole bottom elevation and the elevation of the bottom of the uppermost inlet,
- status: manhole connected to the node or not.

Right-clicking on the manhole list brings up a menu with an option to copy the list to the clipboard. The copied list can be pasted from the clipboard into any text editor, e.g. by pressing **Ctrl** + **V** .

Project summary

The program provides a statistical data of created profile. They are:

1. Excavation volume,
2. Sidefill volume (height of the sidefill layer is set in **Settings** window on **Cross-section** tab),
3. Bedding volume (height of the bedding layer is set in **Settings** window on **Cross-section** tab),
4. Max. depth,
5. Min. depth,
6. Max. gradient,
7. Min. gradient,
8. Total length,
9. Section quantity,
10. The longest section,
11. Turf area,
12. Trench side walls area,
13. Volume of ground to exchange.

Statistics		Materials
Excavation volume	[m3]	50.04
incl.: sidefill	[m3]	5.74
bedding	[m3]	1.46
Depth maximal	[m]	1.15
Depth minimal	[m]	0.36
Gradient maximal	[%]	2.00
Gradient minimal	[%]	1.00
Total length	[m]	40.40
Sections amount	[-]	5.00
Longest section	[m]	16.00
Turf area	[m2]	40.12
Side walls area	[m2]	120.94

Double-clicking in the value of items: 4, 5, 6, 7, 10 (or pressing **Go to node**) to close the window **Statistics** and transferring to the **Data** table, to the section, which displays the value in question.

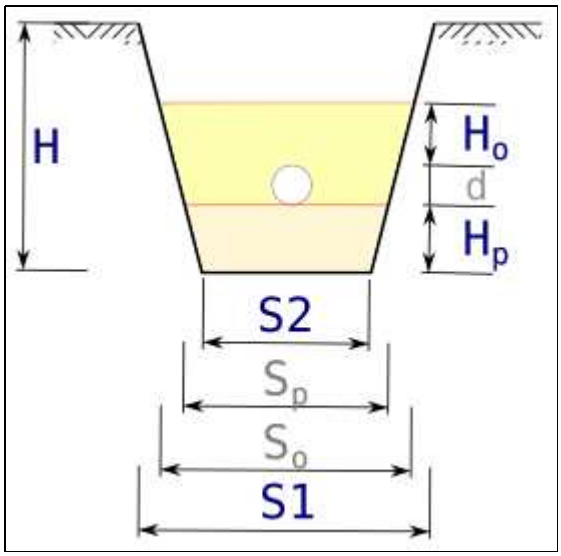
From the selectable list on the top of the window you can select the profile for which you want to view statistics. Selecting **all marked profile**, you can view a summary of the profiles contained in the project (only those marked on the sheets in the **Data** table by **[+]** next to the name). This option is only available for projects involving more than one profile.

Statistic - algorithms

There are algorithms used in some formulas in Drafter's statistic module.

Volume:

Excavation volume is calculate as sum of volumes:



- sections:

  V1 = H x (S1+S2)/2 x L [m3]

  where:

  V1 - one section volume,
  H - averaged depth of pipe bottom increased by bedding layer height,
  L - section length.

- tanks that exists in profile, like a: septic tanks, biofilters, separators etc.:

  V2 = H x (S + Rx2) x (L + Rx2) [m3]

  where:

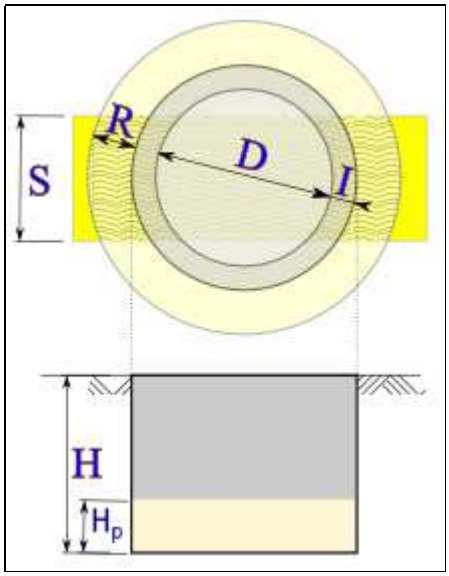
  V2 - one tank volume,
  H - averaged depth tank bottom increased by bedding layer height,
  S - tank width,
  L - tank length,
  R - width of additional hollow (menu: Tools > Settings > Cross-section).

- manholes:

  V3 = H x (D + Gx2 + Rx2) [m3]

  where:

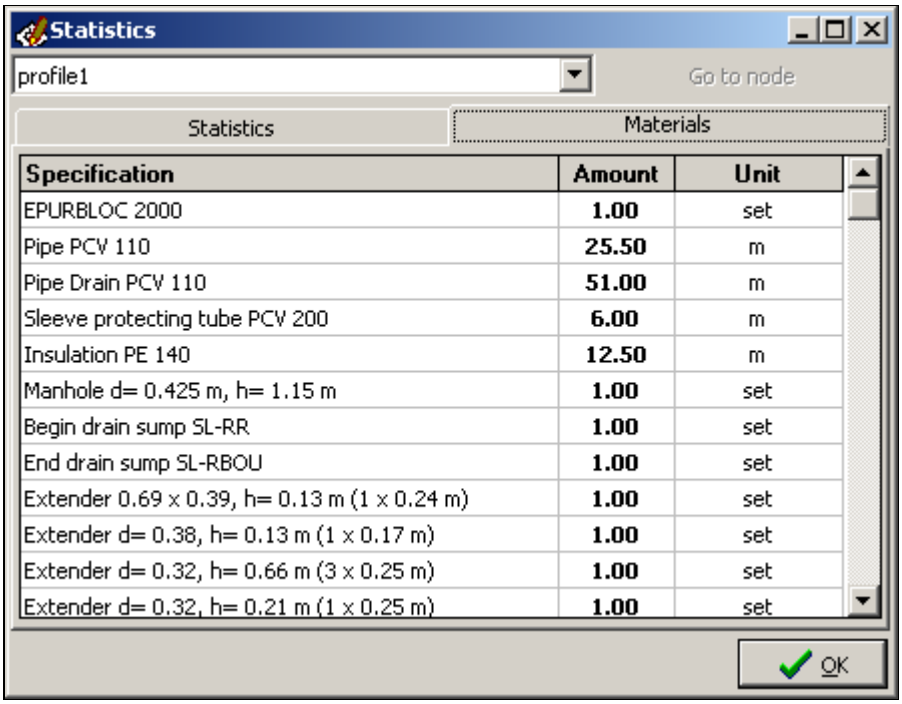
  V3 - one sump volume,
  H - averaged depth sump bottom increased by bedding layer height,
  D - sump diameter,
  G - sump wall thickness (menu: Tools > Settings > Cross-section),
  R - width of additional hollow (menu: Tools > Settings > Cross-section).



List of materials

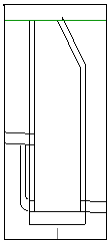
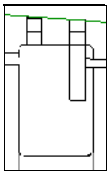
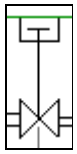
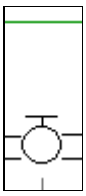
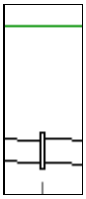
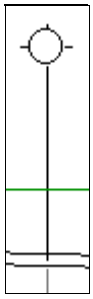
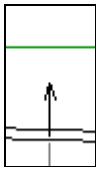
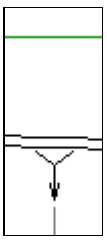
The program automatically creates a specification of materials for a single profile, and the whole project (if the project contains more than one profile). Materials specification can be printed using the included templates (see the **Templates** chapter) or by creating your own.

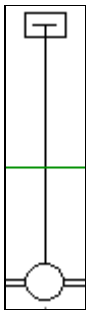
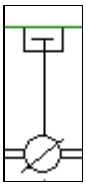
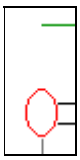
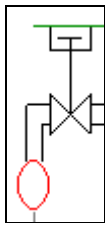
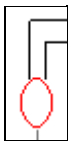
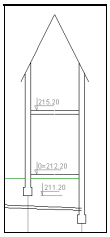
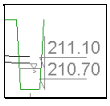
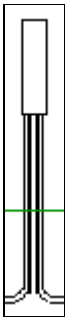
If in the different profiles of the same project, in identically named nodes, there are objects with the same parameters (eg, manholess of equal diameter and height) is in the statement of material for the entire project is not the program will increase their number (to be taken into account only the first occurrence of the object).

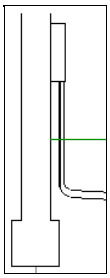
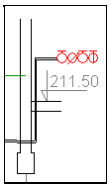
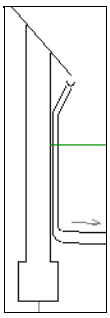
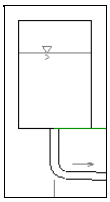
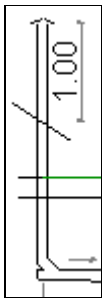


Specification	Amount	Unit
EPURBLOC 2000	**1.00**	set
Pipe PCV 110	**25.50**	m
Pipe Drain PCV 110	**51.00**	m
Sleeve protecting tube PCV 200	**6.00**	m
Insulation PE 140	**12.50**	m
Manhole d= 0.425 m, h= 1.15 m	**1.00**	set
Begin drain sump SL-RR	**1.00**	set
End drain sump SL-RBOU	**1.00**	set
Extender 0.69 x 0.39, h= 0.13 m (1 x 0.24 m)	**1.00**	set
Extender d= 0.38, h= 0.13 m (1 x 0.17 m)	**1.00**	set
Extender d= 0.32, h= 0.66 m (3 x 0.25 m)	**1.00**	set
Extender d= 0.32, h= 0.21 m (1 x 0.25 m)	**1.00**	set

Available objects

Object	Example	Description	Parameters F3
		clears the cell contents	-
manhole		Inserts the manhole selected in the **Objects** window	Description in Manholes section.
tank		inserts a "tank" object into the profile, e.g. a septic tank, sewage treatment plant, grease or hydrocarbon separator, liquid gas tank, etc. The selection must be made more specific by selecting a particular device in the **Objects** window. The program automatically inserts the appropriate tank length into the **Data** table.	Description in a Tanks section.
damper		inserts a damper symbol.	-
valve		inserts a valve symbol.	-
transition		inserts a transition symbol between sections of different diameters and/or pipe materials.	-
street water pump		inserts the symbol for a street water pump.	-
deaerator		inserts the symbol for the deareating device.	-
drain valve		inserts a symbol for drain valve.	-

Object	Example	Description	Parameters F3
ground hydrant		inserts symbol for ground hydrant.	- diameter (default value: 80 mm) - whether on a branch (default value: no)
underground hydrant		inserts symbol for underground hydrant.	- diameter (default value: 80 mm) - whether on a branch (default value: no)
Pipe tee (T-connection)		inserts a branch symbol.	- network diameter (default value: pipeline diameter) - ordinate of the network axis (default value: pipeline ordinate)
clamp saddle with a valve		inserts a pipeline symbol (circle) and a gate valve symbol next to it.	- network diameter (default value: pipeline diameter) - ordinate of the network axis (default value: ordinate of the pipeline) - whether the gate valve on the right side (default value: no) - height of the vertical section of the clamp saddle (default value: 0 m)
pipe junction		inserts a circle on the designed pipeline.	- network diameter (default value: pipeline diameter) - ordinate of the network axis (default value: pipeline ordinate)
building		inserts the outline of building foundation with ceilings and roof (depending on settings).	Description in Building section
ditch/slope		inserts, depending on user settings, a ditch or a slope (road).	Description in Ditch section
gas box		inserts a gas box.	-

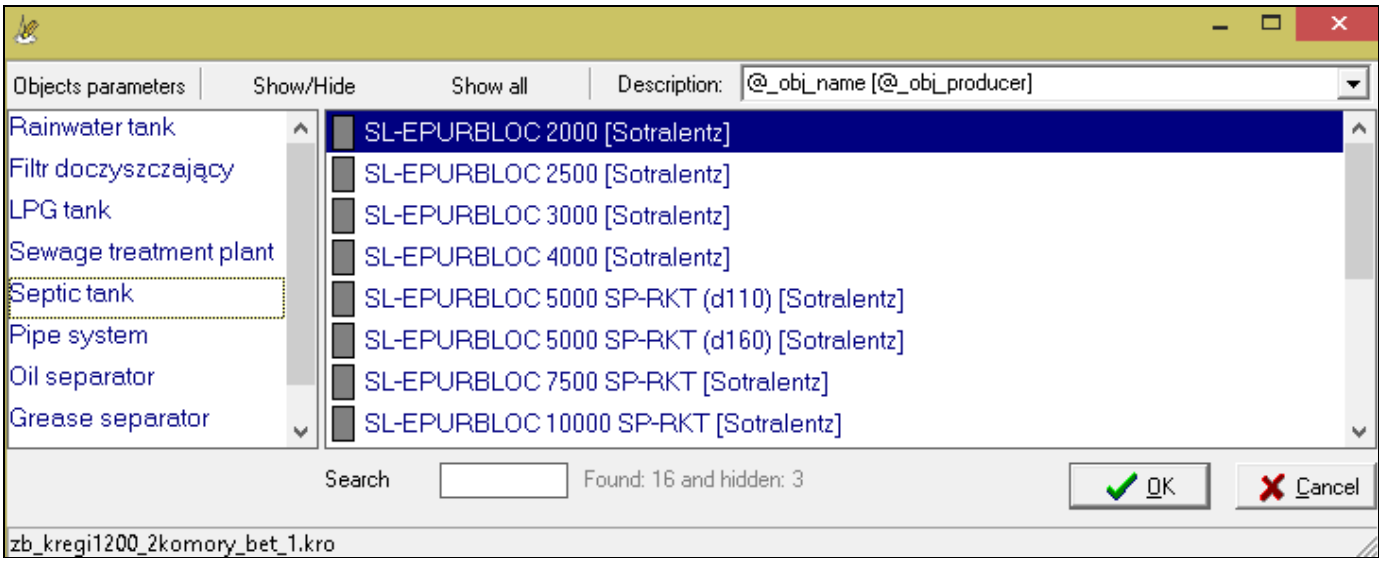
Object	Example	Description	Parameters F3
gas box on the wall		inserts a gas box on the wall of building.	Description in Building section
water meter set		inserts the symbols of the fittings that make up the water meter set into the profile. They are placed behind the wall outline (inside the building).	Description in Building section
gutter		inserts the outline of the building's foundation into the profile, along with a vertical section of gutter run alongside it, connecting to the designed pipeline.	Description in Building section
reservoir		inserts a symbolically marked above-ground water reservoir into the profile. This object is intended for use especially in profiles from which data will be exported for further analysis in Epanet.	-
stack		inserts a waste/vent stack pipe into the profile with connection to horizontal branch of building main drain.	Description in Stack section
inspection / water meter		This object type was left in for compatibility with older versions of Drafter. Instead, it is recommended to use the "manhole" object, which has much greater configuration capabilities.	-

How to navigate the drop-down list Objects using the keyboard.

After hovering (with the keyboard cursor) in the **Data** table over the "Object" column, press the **Enter** key, and then using the **cursor up** and **cursor down** keys, select the object category, e.g.: "other..." and press the **Enter** key. After that, the drop-down list will disappear and the object selection should be refined in the **Objects** window that appears. The first item in the category list is an empty item. It allows you to remove an already existing object from a node.

Inserting objects to **Data** table is also possible from main menu by choosing **Insert object**.

Decription of Objects window.



Buttons definitions:

Object parameters - display detailed parameters of chosen object,

Show/hide - press to hide rarely used objects or show earlier hidden (after click **Show all** button),

Show all - add to list also hidden objects (marked as red),

Description - In the text field Description you can specify filter pattern using variables:

@_obj_name - object's name,

@_obj_description - object's description,

@_obj_producer - object's producer name,

@_obj_idx - catalog index,

@_obj_date - date,

@_obj_material - material.

The most used description's patterns is possible to change by choosing last item: **customize list...**.

For most objects inserted into the **Data** table, you can modify their characteristic parameters by invoking the Parameters dialog box using the **F3** key or the menu: **Node > Object Parameters**. For most categories, you can add objects to the program database yourself. Once added once, an object is available for quick use in projects. The method for adding objects is described in the **Object Parameters** section.

Parameters of objects

In the **Object parameters** window user can set some parameters for chosen object's kind. There are different parameters for each of kinds. Except that some parameters are common for all objects:

Kind - Kind/category of object - read only value,

Name - Object's name,

Description - Description,

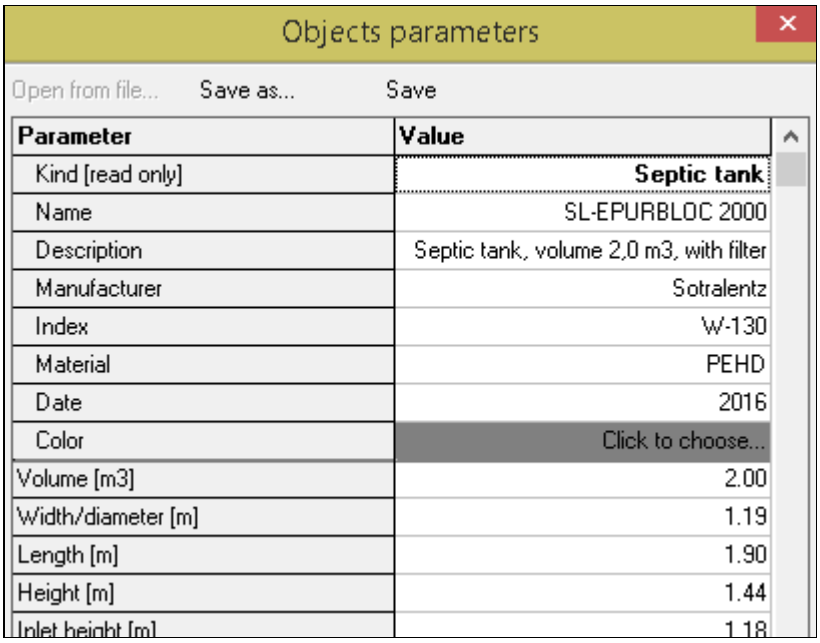
Producer - Producer,

Index - Catalog index (not applicable to pipe systems),

Material - Kind of predominant plastic,

Date - Date,

Color - Object's color (click in **Value** column to choose). Chosen color is shown in **Objects** window next to its name. It's allow for easy find objects on the list.



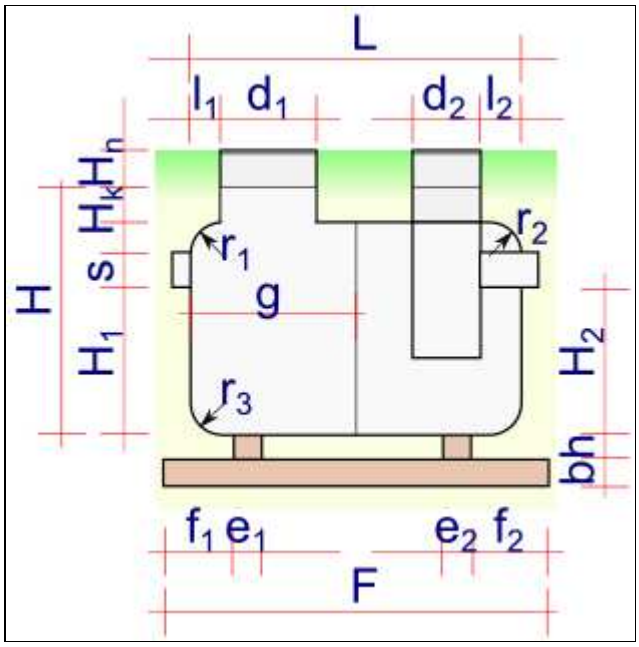
Drafter has got included some of predefinied objects. Files with them are KRO extensions and are placed in "objects" directory. User can create own object's defintions (tanks and sumps) based on already exists. To do it open **Objects** window (menu: Tools > Objects). Next choose and double click an object with the most similar parameters. In **Objects parameters** window input new values and click **Save as...** button. After that file with definition of new object will be save with chosen by user name.

A new object can also be created based on an object already inserted into a project. To do this, go to edit its parameters by pressing the **F3** key on your keyboard. It is important that currently selected row in **Data** table is the row with inserted object, on the basis of which new object will be created. Saving the entered data is similar to the above.

Opening **Objects parameters** window from **Objects** window is allowed to change definition of existing objects by click **Save** button. However, keep in mind that some of the files that are delivered with the program are overwritten with newer versions when the program is updated.

Custom objects - tanks

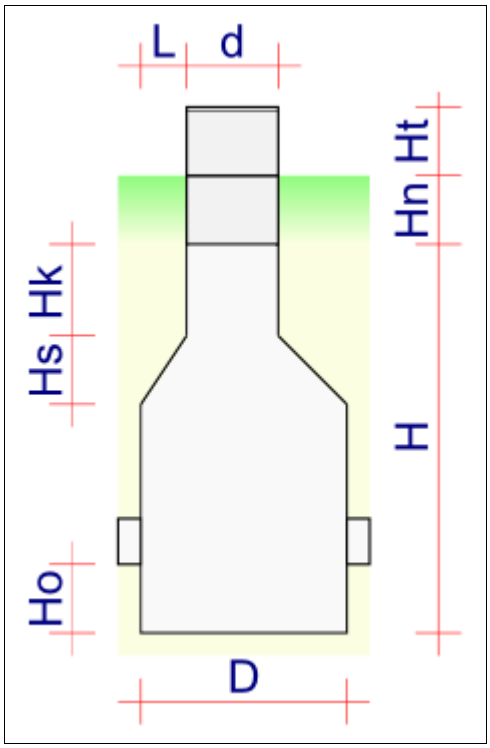
For objects in the shape of the tank ie septic tanks, sewage treatment plants, grease separators, filters and tanks used for storage of liquid gas can be determined:



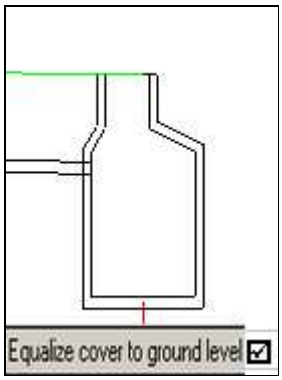
- **Volume [m3]**,
- **Width/diameter [m]** - if the tank is "projected" circle then the length should enter 0 (zero),
- **Length - L [m]**, - if value isn't set tank's width will be its diameter, that means will be rounded in top view,
- **Height - H [m]**,
- **Inlet's bottom height - H1 [m]**,
- **Outlet's bottom height - H2 [m]**,
- **Inlet/outlet diameter - s [mm]**,
- **Cover 1/2: width/diameter [m]** - if cover shape is circle, then length value must be zero,
- **Cover 1/2: length - d1/d2 [m]**,
- **Cover 1: dist. from left l1 [m]**,
- **Cover 1/2: manhole height - Hk [m]**,
- **Cover 1/2: Extension shaft height - Hn [m]** - if value is set to 0 (zero) then program choose one extension shaft,
- **Cover 1/2: Extension shaft max. count [pcs]** - maximal, accepted by producer, extension shaft quantity. If value isn't set by user then Drafter will choose appropriate quantity to reach ground level,
- **Cover 2: distance from left side - l2 [m]**,
- **The radius of rounding upper corners - r1/r2 [m]** - notation "x/y" is also accepted, where "x" means radius of the left one, and "y" - the right corner,
- **The radius of rounding bottom corners - r3 [m]**,
- **With filter** - checking this option determine to draw filter before tank's outlet
- **Base: width/diameter [m]** - if value isn't set Drafter assumes value equal to tank's width
- **Base: length - F [m]** - if value isn't set Drafter assumes value equal to its diameter
- **Base: height - b [m]** - base is drawing if its height is above zero.
- **Pillar 1: distance from left - f1 [m]**
- **Pillar 1: length - e1 [m]**
- **Pillar 2: distance from right - f2 [m]**
- **Pillar 2: length - e2 [m]**
- **Pillars: height - h [m]**
- **Wall thickness [mm]** - wall thickness of the tank
- **Partition 1/2: distance from the left - g [m]** - distance from the left tank edge to the vertical inner partition
- **Partition 1/2: height - H-Hk [m]** - height of partition in meters. Value 0 (zero) will cause drawing the partition for the whole tank height. Height different from zero will cause drawing a partial partition. In case of positive value - starting from the tank bottom or negative - starting from its top.

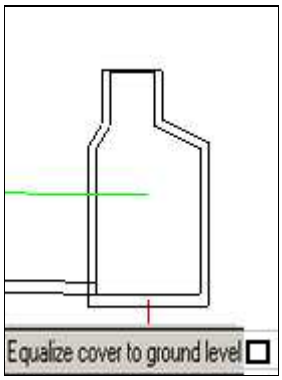
Custom objects - manholes

User can modify the following sump's parameters:

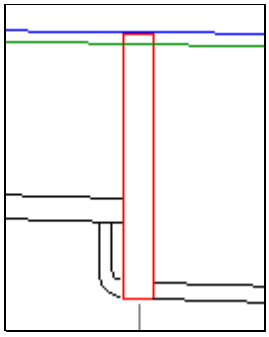


- **Height - H [m]**,
  If value is set then sump will has got a specified height regardless projected AOD level. In other words: sump's cover may lay above or under projected ground level. Default value is 0,00 (zero) - then height is automatically adjusted so that the sump's top was at the projected ground level.
- **Height from bottom to inlet bottom - Ho [m]**,
  Constans height, that user can set from sump's bottom to the bottom of the lowest inlet/outlet. This parameter can be used eg for sediment sump, pump stations etc. Default value is 0,00 (zero) - it's means sump's bottom is equated to inlet/outlet bottom.
- **Height above projected AOD level - Ht [m]**,
  Constans height, that user can set from sump's top to the projected groun/AOD level. This parameter can be used eg for sumps protruding above the ground. Default value is 0,00 (zero) - it's means sump's top is equated to projected ground level.
- **Bottom ordinate [m AOD]**,
  Setting ordinate of sump bottom allow to pin its bottom to entered ordinate. Then ordinate of bottom inlets/outlets haven't got influence on sump's bottom position. Default value is 0,00 (zero) - it's means sump's bottom ordinate is automtically adjusted to inlets/outlets level.
- **Inner diameter - D [m]**,
  Value of the inner diameter of sump. If value is 0,00 then diameter is set to 0,2 m.
- **Cover diamter - d [m]**,
  Diameter of cover of sump. It's may have got a diameter other than the sump. Default value is 0,00 (zero) - it's means sump's cover's diameter is equal to sump's diameter.
- **Distance from side of sump - L [m]**,
  Distance from cover's left side to sump's left side. Default value is 0,00 (zero) - it's means cover's left edge is adjusted sump's left edge.
- **Manhole height - Hk [m]**,
  Height of sump's components where their diameter is equal to sump's diameter. Default value is 0,00 (zero) - it's means count of components is none.
- **Cone height - Hs [m]**,
  Height of cone adapter between component with sump diameter and component with cover diameter Default value is 0,00 (zero) - it's means adapter is flat.
- **Extension shaft height - Hn [m]**,
  Height of the component (with constans height) that allow to extend sump to ground level. Default value is 0,00 (zero) - it's means application choose one extension shaft.
- **Extension shaft max. count [pcs]** - maximal extension shaft quantity. If value isn't set by user then Drafter will choose appropriate quantity to reach ground level (if sump cover is below ground level). If sump producer doesn't provide extension shafts then choose **none** option or type -1.
- **Wall thickness [mm]**,
  Thickness of the sump wall. Thickness is drawing in the cross-section drawing and in the profile drawing if thickness is greater than 50 mm. Default value is 0,00 (zero) - .
- **Sump kind**,
  Kind of sump can be chosen from the list, appeared after going to edit mode. User can select from:
  - standard - default,
  - watermeter - drawing with equipment,
  - distributing - for drainage,
  - closing drainage pipes,
  - aerating vertical filter,
  - pump station
- **Align cover to ground level**,
  If option is checked then sump drawing is begin from its cover, that is placed on ground level. If option is unchecked then begining draw is sump bottom, that is equated to inlet's bottom level.

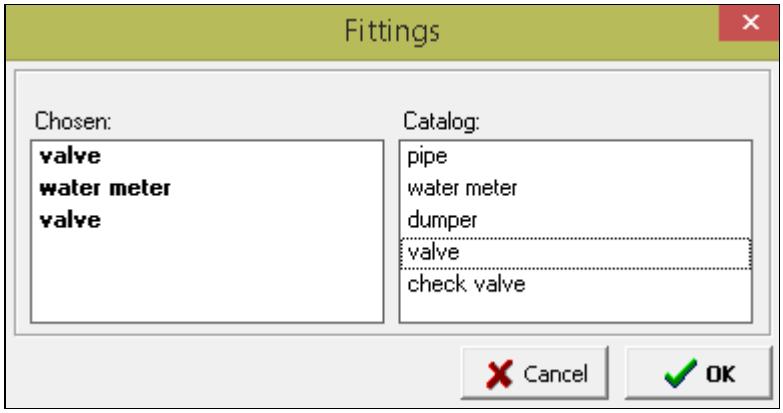




- **Without bottom**,
  Checking **Without bottom** option causes drawing sump without bottom, eg: infiltration sump.
- **Backdrop**,
  Draw backdrop pipe next to manhole while checkbox is checked and when distance from bottom to invert level is min. 50 cm.

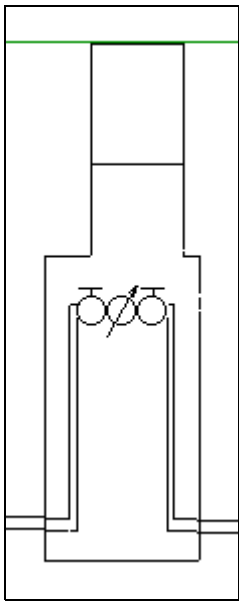


- **Fittings**,



In the **Fittings** window, you can select which fittings will be placed inside the well.

- **Fitting Elevation**,
  height, expressed in meters, by which the fitting in the manhole will be raised in relation to the pipe ordinate specified in the Data table. Below is an excerpt from a profile with a manhole with raised armature.



- **Additional inlet [m AOD/mm/degrees]**,
  Parameters of the additional inflow, i.e. the one, which is not automatically determined on the basis of other profiles being part of the design, and which is to be marked on the manhole drawing. The data should be given in the notation **O/D/A**, where O stands for the ordinate, D for the diameter of the inlet, and A for the horizontal angle of the inlet, e.g: "211,00/110/90". The inlet will be marked if the **Mark manhole inlets...** option is enabled. in the **Settings** window of the menu: Tools > Settings > Drawing > Longitudinal section.

Additionally in sump parameters window are showing parameters automatically calculated (read only):

- [auto] **Inlets** - list of inlets/outlets of sump, also from other profiles in current project.
- [auto] **Height [m]** - sump height

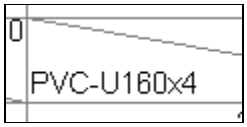
Custom objects - pipe systems

Adding pipe system to project allows on (if diameter in **Data** table given by user exists in pipe system):

- automatically assigning to pipe additional parameters, among other: material, wall thickness, class etc.
- automatically upgrade material's name and pipe description in **Data** table (coloring by pipe system color), which are assigned to pipe system,

t	Pipe	Material	Diameter	Chainage
	-	-	mm	m
0	D	PVC-U	110.00	0.00
0	D	PVC-U	160.00	20.00
0	D	PVC-U	200.00	43.00
0	D	PVC-U	200.00	61.00

- place on longitudinal section drawing additional inormations about pipe, eg. wall thickness,

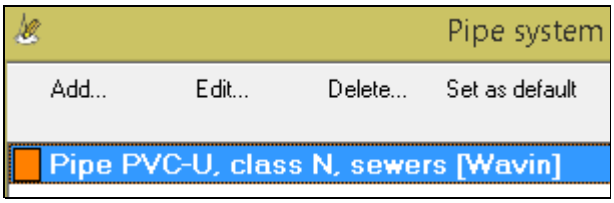


- "trading" pipes split in material list (mainly sewers). Splitting is allow to turn off on **Other** tab of **Settings** window.

Material	Quantity	Unit	Node	Index	Producer
Pipe PVC-U 110	**20.00**	m			
Pipe PVC-U, Wavin, 160x4 mm, l=6 m	**3.00**	szt.		3062923446	Wavin
Pipe PVC-U, Wavin, 160x4 mm, l=3 m	**1.00**	szt.		3062923443	Wavin
Pipe PVC-U, Wavin, 160x4 mm, l=2 m	**1.00**	szt.		3062923442	Wavin
Pipe PVC-U, Wavin, 200x4.9 mm, l=6 m	**3.00**	szt.		3064923862	Wavin

Adding pipe system to the project is not obligatory.

To add pipe system to the project click on  button on **Data** tab or choose in menu: (Tools > Pipe system).



In **Pipe system** window user can use pipe system attached to Drafter (due to changes in producers' assortment is not guaranteed, that pipe systems attached to Drafter are actual) or add own. The easiest way to add own is modifying exists one and save it with new file name.

One from chosen pipe system may be mark as default. It's mean, if in the new node user type pipe diameter, that exists in the default pipe system, this pipe parameters will be joined with default pipe system parameters. After change default pipe system all pipe data are automatically updated.

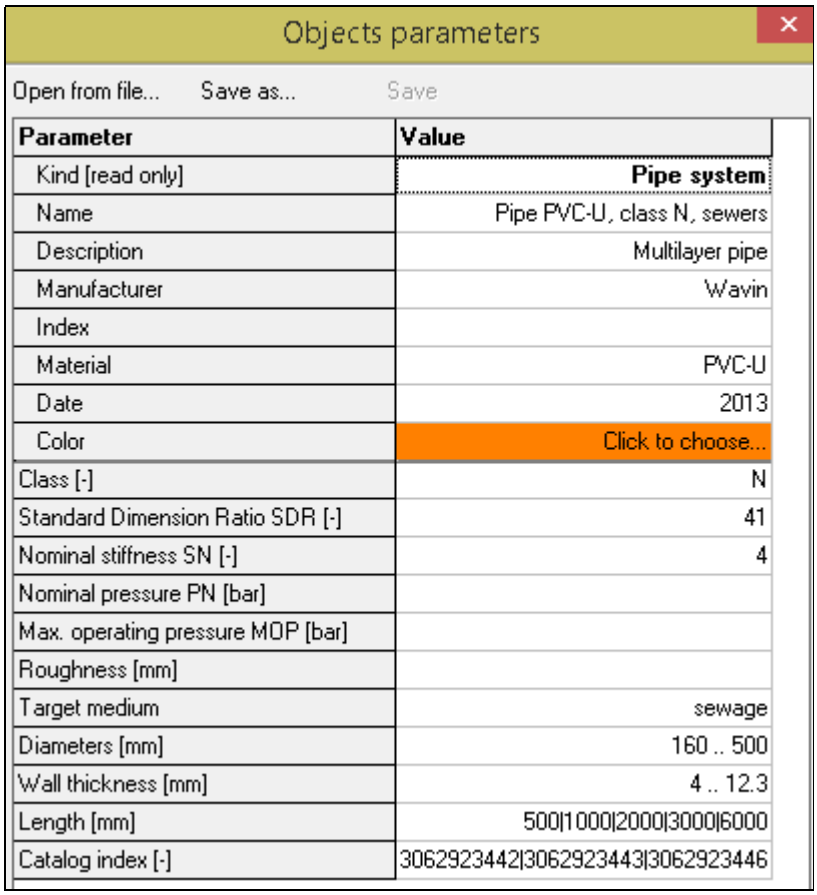
If diameter from node exists in more than one added pipe systems indicate proper pipe system in **Pipe** column in the **Datatable**.

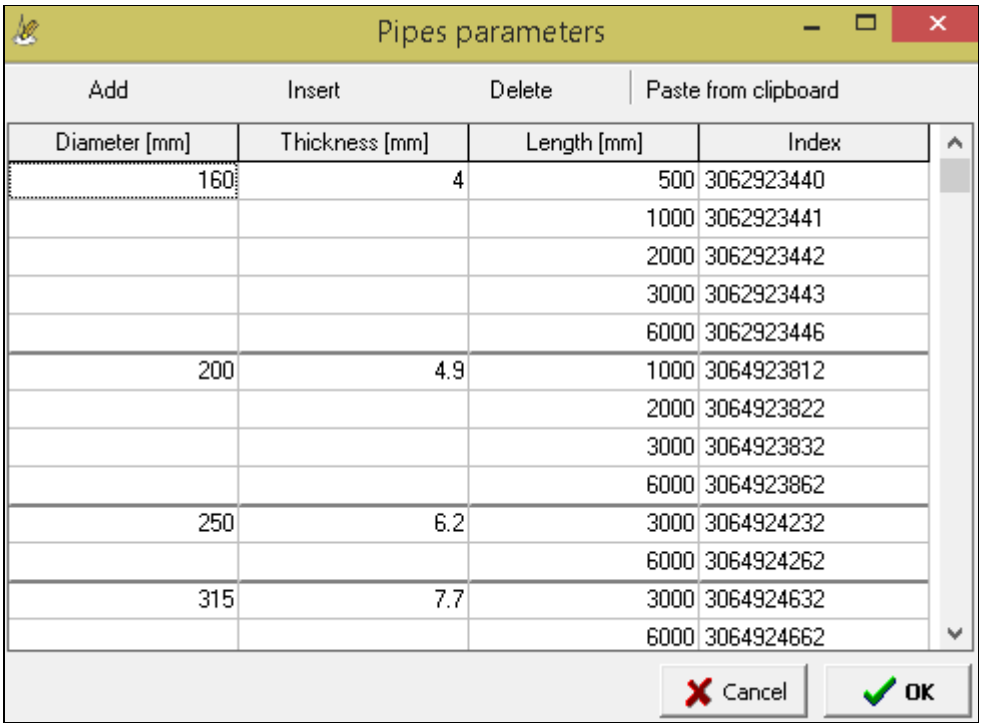
For each pipe system is allow to set:

- class,
- SDR (Standard dimension ratio),
- SN (Nominal stiffness),
- PN (Nominal pressure),
- MOP (Max. operating pressure),
- roughness,
- target medium, it's mean medium which pipe is for,

- diameters,
- walls thickness,
- trading lenght (for pipes sell in rolls set zero),
- catalog indexes,

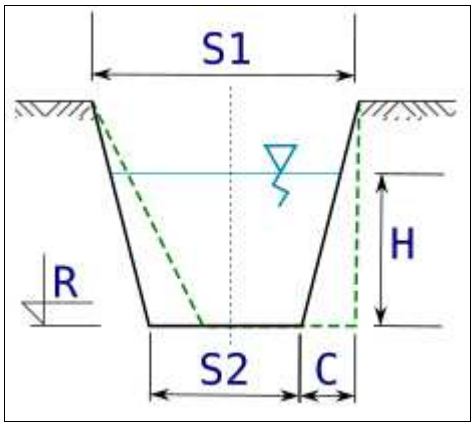
Four last items are modifying in **Pipes parameters** window. Both of whole table and each columns may be prepared eg. spreadsheet and paste to Drafter.



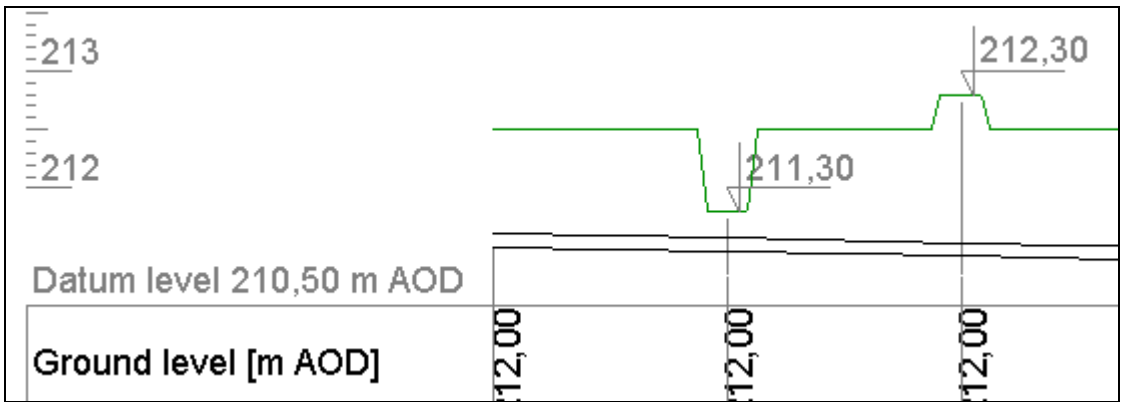


Object parameters - ditch

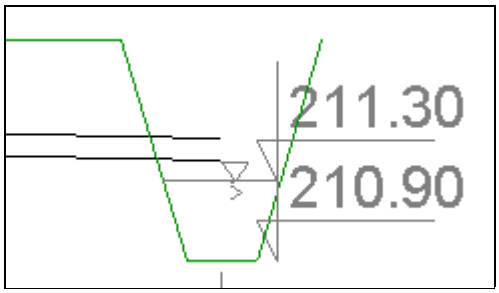
The "ditch" object allows to insert a cross-section through a ditch or slope (depending on the user settings) in the longitudinal profile drawing. The user can change the following parameters of the object:



- **Bottom ordinate - R [m AOD]** - this value can be defined as lower or higher than the ground level at a given node. A smaller value results in a ditch (on the figure below, left), while a larger value results in an escarpment (on the figure below, right). Using the version with an escarpment and setting its width appropriately, it is possible to obtain e.g. a cross-section of a road under which the designed pipeline passes. By default, the value of the trench bottom ordinate is assumed 0.5 m lower than the value of the pipe bottom ordinate at the given node.

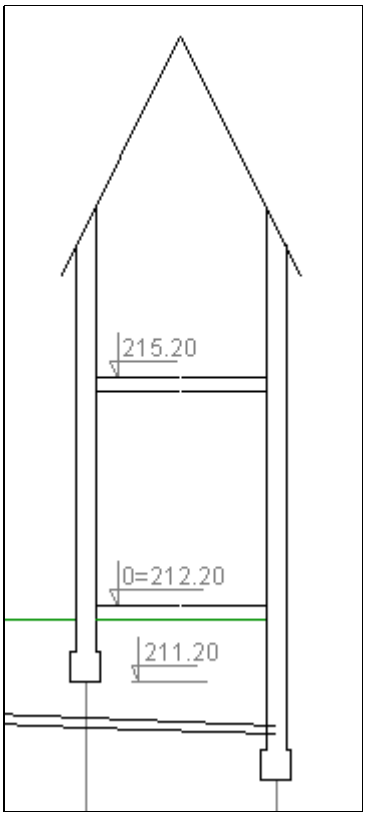


- **Width - top - S1 [m]** - width of the trench at ground level. The default value is 0.50 m.
- **Bottom Width - S2 [m]** - the width of the bottom of the ditch (or top of the slope). The default value is 0.35 m.
- **Bottom horizontal correction - C [m]** - horizontal bottom shift allows to obtain asymmetric slope inclination. The default value is 0.00 - which means that the slopes are inclined to the bottom at the same angle.
- **Water table level - H [m]** - if necessary, you can plot the water level on the ditch cross-section (example in the following figure), expressing it in meters as the vertical distance from the ditch bottom. By default, this level is not marked.

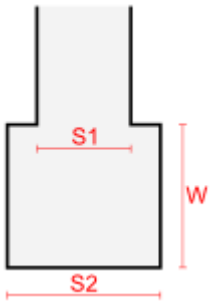


Object parameters - building

The "building" object allows to mark a passage through a wall on a longitudinal profile drawing. The user can define the following parameters of the object:



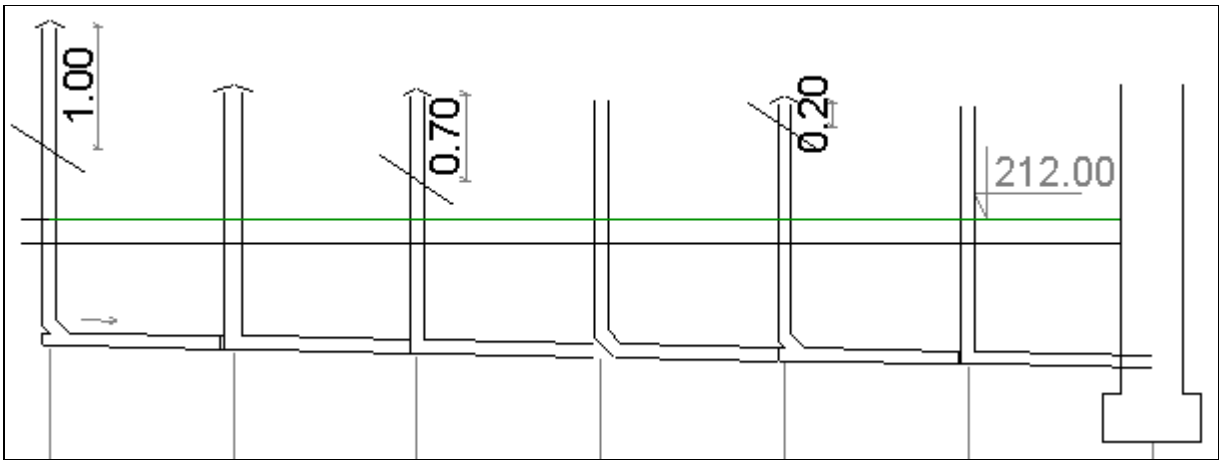
- **Footing bottom depth [m]** - this is the vertical distance between the ordinates of the existing terrain (or designed terrain, if the **Count depth in relation to the projected ground level** option is selected in the **Configuration** window) and the bottom of the footing. Default value is 0.6 m below the ordinate of the pipe bottom in a given node.
- **Floor ordinate(s) [m AOD]** - One or more (for multi-storey buildings) ordinates separated by semicolon ";" can be given. If the floor ordinate is not given, it will not be included in the drawing. The exception is when the floor length is given - then the top of the floor is equal to the ground level. To avoid plotting the ordinate on the drawing, precede the ordinate with "x". The ordinate which is to be marked as so called zero of the building should be preceded by zero.
- **Ceiling length [m]** - A negative length value places the floor drawing on the left side of the wall, while a positive value places it on the right. Default value is -1.5 m.
- **Ceiling thickness [m]** - Default value is 0.2 m.
- **Wall extra height [m]** - Additional height of the wall above the highest ceiling. Default value is 0.1 m.
- **Roof** - When checked, the roof outline is plotted on the drawing.
- **Roof pitch angle [°]** - Determine angle of roof slope inclination to horizontal. For the roof to be drawn, the **Roof** option must be checked. The default value is 45°.
- **Depth of the bottom of footing count to "0"** - Selecting this option causes calculation of footing bottom depth as a difference between "zero" of the building ordinate (it must be marked) and value given as **Footing bottom depth**.
- **Wall thickness - S1 [m]** - Definition of the wall thickness. Default value is 0.5 m.
- **Footing width - S2 [m]** - Defines the width of the footing. Default value is 0.8 m.
- **Footing height - W [m]** - Defines the height of the footing. Default value is 0.4 m.
- **Angle between pipe and wall [°]** - Specify the horizontal angle between the pipe entering the building and the wall. The default value is 90° when the pipe enters perpendicular the wall plane.



It is possible to draw the outline of the whole building (as in the first picture). To do that, a building object with the same parameters should be inserted into two adjacent nodes, except for the negative value of the floor length in the node on the right.

The above parameters can also be defined for other objects using the building outline, i.e.: gutter (additionally you can specify its diameter), water meter set (additionally you can specify the diameter of the casing pipe between the building wall and fittings - the pipe will not be drawn if you do not specify its diameter), gasbox on the wall.

Object parameters - stack



The "stack" object makes it possible to mark the stack and its horizontal branch on the longitudinal profile drawing. It can be especially useful during designing of sub-floor sewage systems. The user can define the following parameters of the object:

- **Diameter [mm]** - The default value is 110 mm.
- **Height [m]** - Distance from top of pipe/horizontal branch to top of stack.
- **Height above roof [m]** - The height to the top of the stack to the stack's passage through the roof. This height is not added to the **Height** parameter, but is part of it. If this height is specified then it will be dimensioned on the drawing. Default value is 0.5 m.
- **Roof** - Selecting this option symbolically marks the roof slope on the drawing.
- **Vent cowl** - Selecting this option causes adding to the drawing symbolic vent cowl on top of the stack.
- **Bottom of stack 2x45°** - Selecting this option causes drawing of the connection to the horizontal with two 45° elbows. If this option is not selected, the connection will be made with one 90° bend.
- **Bottom of stack with cleanout** - When this option is selected, the connection to the horizontal branch is drawn with a test tee.

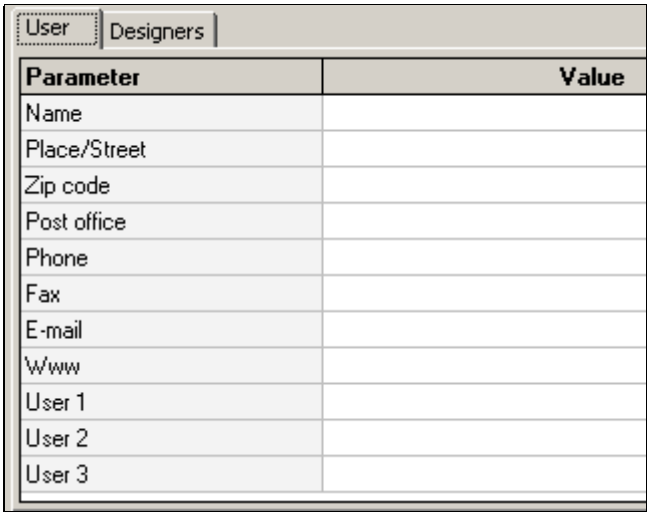
Depending on the settings made, elbows and/or T-pieces and a vent will be added to the list of materials.

If the above parameters are not specified, the program will assume the values given as default.

User data

Menu: (Tools > Settings > User)

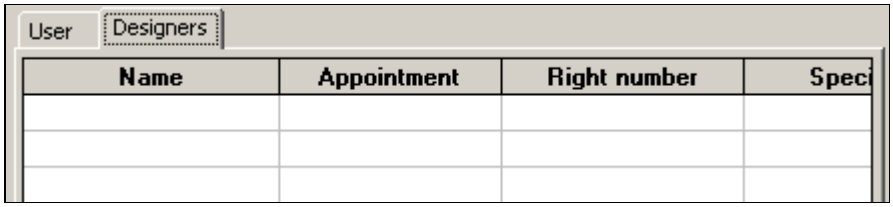
On the **User** tab you can set data describing the company - the program user. These data can be used in projects created using the variables in: **Infotable** and **Documents templates**.



Employees data

Menu: (Tools > Settings > Workers)

On the **Workers** tab, enter the following personal data of employees when they are developing technical specifications for the company:



- Name
- Appointment
- Right number
- Speciality

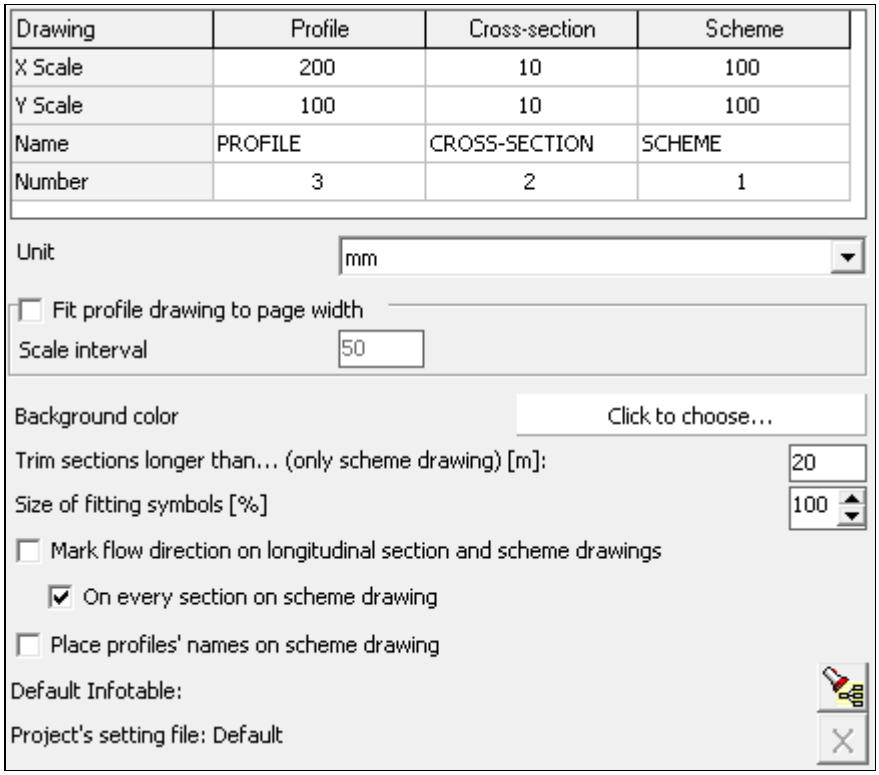
Inserted persons can then select the window **Project parameters** on the **Designers** tab.

Drawing parameters

Menu: (Tools > Settings > Drawing)

In the **Settings** window on the **Drawing** tab is possible to set for drawings (longitudinal section, schema, cross-section):

- scale, independent for X and Y axis,
- name,
- number.



Name and number of the drawing may be used as variables in associated documents and in the **Infotable** (eg by using the variable **@_drawing_name**, you can use one infotable for all types of drawings).

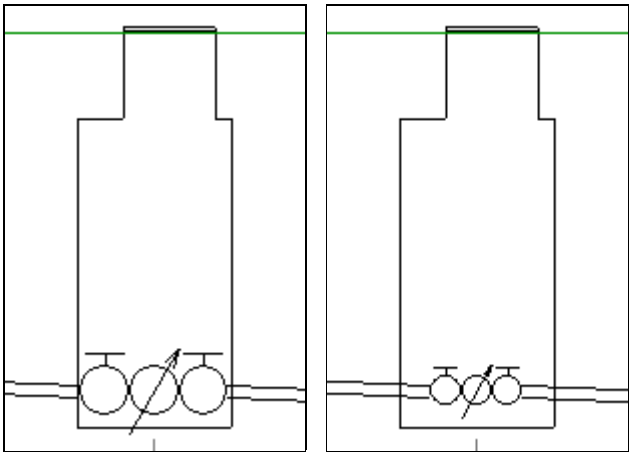
Use the **Unit** drop-down list to select the smallest unit of the generated drawing. The default unit is [mm].

If you check **Fit profile drawing to page width** checkbox, the Drafter automatically adjusts the horizontal ("X") scale, so that the generated lungitudinal cross-section drawing fit the currently selected paper format (eg A4 format). Additionally, the user can impose a minimum stroke in the selection of scale by entering the value in the text box **Scale interval**. For example: when you are given a jump equal to "50" is a program that will analyze the following scale, starting at 1:1, increasing further to 50 (1:50, 1:100, 1:150, etc.) until it finds such at which the total drawing will fit on the currently selected (in the print settings) page size. This option works only for the lungitudinal cross-section drawing.

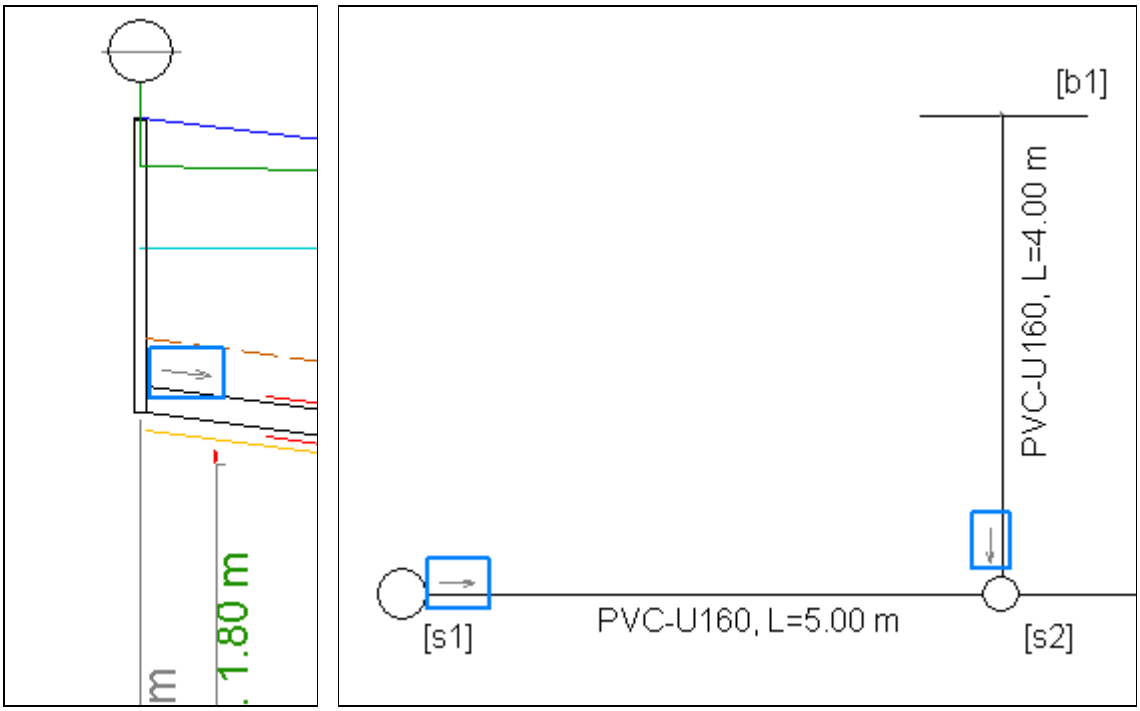
By clicking the **Background Color** box, you can change the background color of the generated drawing on the **Preview** tab.

On the network schema drawing program can automatically cut sections. Will be shortened sections longer than that given in the field: **Trim sections longer than ... [m]**.

By changing the **Size of fitting symbols** you can adjust the size of the symbols so that they are not too large, but still maintain legibility. This option affects the drawings: profile and schema. In the diagrams below: on the left the symbol size is 80% and on the right 50%.



If you select the option **Mark flow direction...** on the longitudinal section and the schema drawing, arrows will be placed behind the first node in accordance with the flow direction selected in the **Panel**. The following examples show the flow direction markers (in blue borders) on the profile (left) and schema drawings.



Selecting the **Place profiles' names on scheme drawing** option causes the profile names to be applied to the schema drawing. This option can be particularly useful for multi-profile projects.

By clicking **Default Infotable**, you can choose a **Infotable** template that will be used by default in newly created projects.

For the changes have been saved even after you close the program, press the button **Save**.

Layers settings

Menu: (Tools > Settings > Layers)

On the **Layers** tab user can manage layers, which can be attributed to the elements of generated drawing (lines and fonts).

For each layer, is possible to specify its name, color, line type and line thickness.

No	Name	Color	Line type	Line thick.	Locked
1	0	1	———	1	☑
2	terrain_proj	3	- — — — — -	1	☐
3	pipe	7	———	3	☐

Line thickness can be freely determined only by a continuous line. For others, the line thickness can only take the value 1.

By checking the option in the **Locked** column you can lock the layer. This possibility can be useful in case of further processing of the generated drawing in CAD program. Elements placed on a locked layer cannot be edited/deleted until the layer is unlocked.

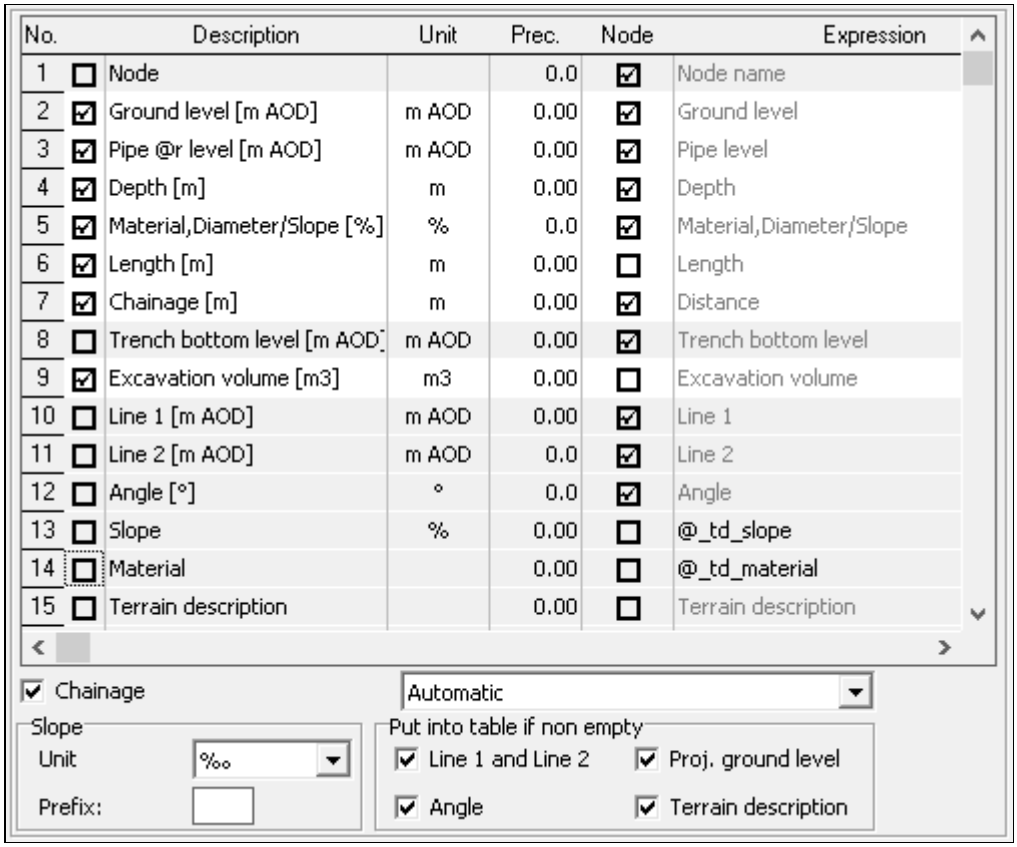
Shortcuts:

Insert - adding new layer,
Delete - deleting selected layer (layer "0" can't be deleted),
Cursor down on last item - adding new layer on the end of table,
Enter on **Color** column (or double click on **Color** column) - opens **Select color** window,
Enter on **Locked** column (or click on **Locked** column) - check/uncheck checkbox.

Table below longitudinal section

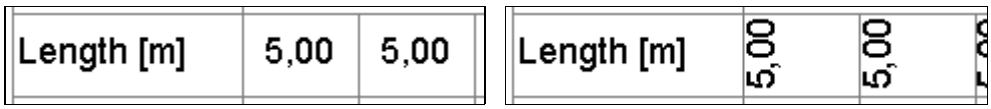
Menu: (Tools > Settings > Drawing > Table)

The user can modify the contents of the table inserted under the logintudinal cross-section drawing. List of data that can be placed in the table is stored in the other table in the **Settings** window - menu: (Tools > Settings > Drawing > Table). The table contains the following columns:



- **No.** - item number determines the order of the data. **The order can be changed by clicking (on a column No.) and dragging the row in another location,**
- Checkbox row - only the selected rows in the second column are placed in the table below the profile,
- **Description** - The title line that appears at the beginning of the table. In the **pipe level** row, is possible to use a variable **@r**, which at the time of generating the drawing will be replaced by the word "axis", "bottom" or "top" - depending on the profile settings. In rows applicable to flow is possible to use the **@flow_unit** variable, which will be converted to the currently selected **Flow unit**.
- **Unit** - A unit in which values are presented in a row,
- **Precision** - The precision with which it is presented by a number (number of decimal places),
- **Node** - a checkbox to be selected if the data presented is related to a node and not to the section between the nodes. From the appropriate selection depends, among others direction of the text in the row (vertical or horizontal),

  Example with the aforementioned option disabled (left) and with it enabled:



- **Expression** - Expression can only change in rows added by the user. A formula can consist of variables and values. For example, using the expression "@_td_ground_level+1" will result in the elevation of terrain value from **Data** table increased by 1. After adding a line based on this formula, you can get a line parallel to the line of site.
- **Height** - is determined as a percentage of the default row height of the table.
- **Connect** - when this option is checked, if the same value occurs in neighboring nodes, the description will appear only once. This option works only with section data (e.g., slope, material, scale), that is, the option in the **Node** column cannot be checked.

  Example with the aforementioned option enabled...

  ... and with the disabled:

Material,Diameter/Slope [%]	2,0 / PVC160	2,0 / PVC160	2,0 / PVC160	3,0 / PVC160	3,0 / PVC160

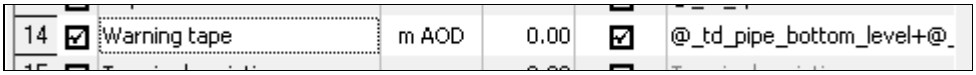
Example: Adding the row "Underground warning tape" to the table under the longitudinal cross-section.

Assumption: the tape is to be laid 30 cm above the top of the pipeline.

First, add a new row to the end of the table by pressing the cursor down on the keyboard (while on the lowest row of the table) or by right-clicking on the table and selecting **Add**.

Then, in the newly inserted row, fill data in columns: the first one after **No.** we mark to make the row appear under the profile. In the next ones we specify: the title - **Warning tape**, the unit - **m AOD**, the required rounding level. In the **Node** column, check the existing option, because the ordinates of the tape are to appear at the nodal points.

The most important column in this example is the **Expression**. In it, you should use the available variables so as to obtain an action that will result in the ordinate of the tape. According to the previous assumption, the ordinate of the tape at a given node should be 30 cm higher than the ordinate of the top of the pipe. So, the action will take the form [ordinate of the bottom of the pipe]+[diameter of the pipe]+0.3 m, which with the use of variables will look as follows: **@_td_pipe_bottom_level+@_td_diameter_m+0.3**. If it would be necessary to place the tape above the pipeline by another value than 30 cm (0.3 m), then you need to change 0.3 to the required value in the formula. The other columns can be left with automatically added default values.



If the "Warning tape" row is also to be used in projects created in the future then click the **Save as default** button.

If a line is to be drawn on the profile drawing based on the determined ordinates of the tape, it should be added in the **Lines** section.

By checking the box **Chainage** to get under the table line with those distances. The listbox **Chainage** allows you to specify the unit, which will be given full value of the distance. Choosing **Automatic** causes selection the largest unit for given profile, eg. when the total length is 321.50 m profile unit in which distances will be given hectometres.

If the data group in the table **Put into table when non empty** will be awarded an option **"Line 1" and "Line 2"** and/or **"Angle"** and/or **"Terrain description"** then will be placed into table under the profile data from the column "Line 1", "Line 2" and "Angle" and "Terrain description" (regardless of the settings in the appropriate positions in the table above), if the above data has been entered.

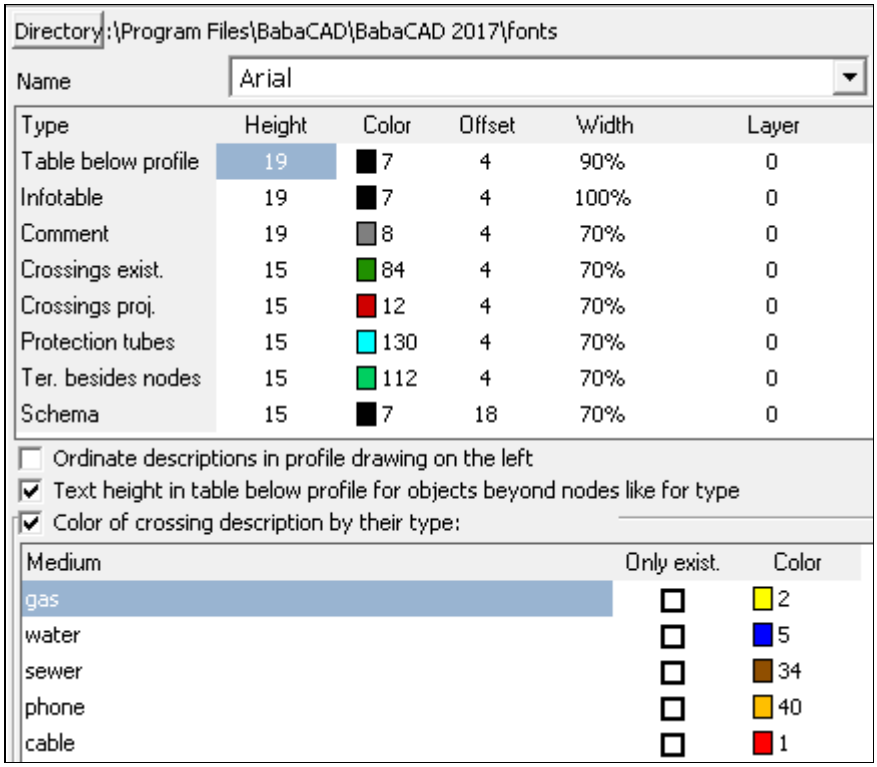
The **Slope > Unit** list allows you to choose how the slope value is presented in the table under the profile. Selecting the **Inverted** option from this list causes the inverse of the slope to be entered in the table (e.g.: **50** for a slope of **2%**). This type of notation is used in some countries. The text entered in the text field **Slope > Prefix** will be placed before the drop value.

For the changes made are saved when you quit, press the button **Save**.

Fonts settings

Menu: (Tools > Settings > Drawing > Fonts)

On the **Fonts** tab, you can specify the parameters of the fonts that are used in the generated drawing.



The requested typeface, choose from the dropdown list **Name**. It can be found on only those fonts installed on the system of which lend themselves to scaling. Moreover **Drafter** automatically added to the list of all files (not all fonts need to be) from the directory "Fonts" default CAD application. The names of these fonts are gray. If the font of this type are found, their use is stoned. If the program does not automatically find these fonts can be identified manually by pressing **Directory** button, the font directory application used to edit the generated profile.

Regardless for comments, the texts contained in the table below the profile and the imprint of the project can change the fonts, the following parameters:

- **height**
- **color**; Chosen from **Colors** window, opened by double clicking the left mouse button. There are 255 standard colors. Their subsequent numbers correspond to the numbers assigned to corresponding colors used in most CAD applications.
- **distance from vertical line** (**Space** column).
- **width percentage** (**Width** column). Described as a percentage (100% means that the width is not modified by changing the value - is changed character width).
- **layer**, to which the text will be assigned on the drawing (**Layer** column).

By default, in the table under the profile descriptions are placed on the right of vertical line (passing through the node). Option **Ordinate descriptions in profile drawing on the left** allows you to put names to the left of the vertical line.

If **Text height in table below [...]** checkbox is checked then for descriptions of elements eg. collision, protecting tubes etc. placed in the table below longitudinal cross-section will be used text height assigned to them instead of assigned to row titled **Table below profile**.

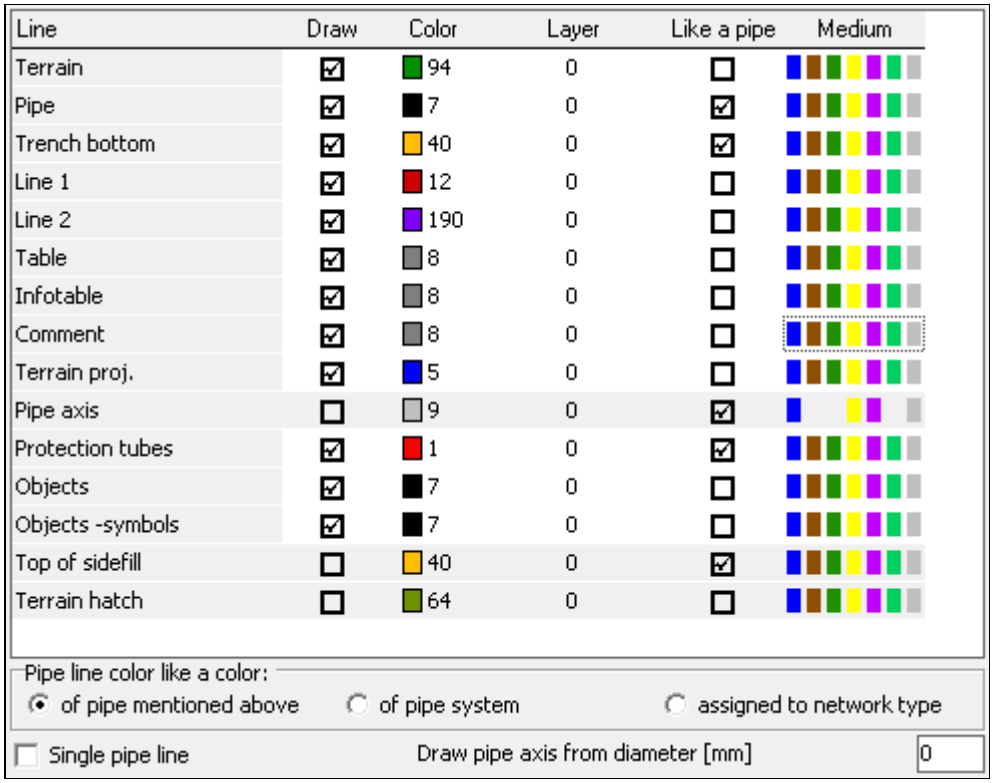
By checking the **Color of crossing description by their type** option, you can use the collision description color defined independently for different crossing types. To add a new crossing type to the table, press the **Insert** button on the keyboard. Each type of crossing type must be in a separate row of the table.

Lines settings

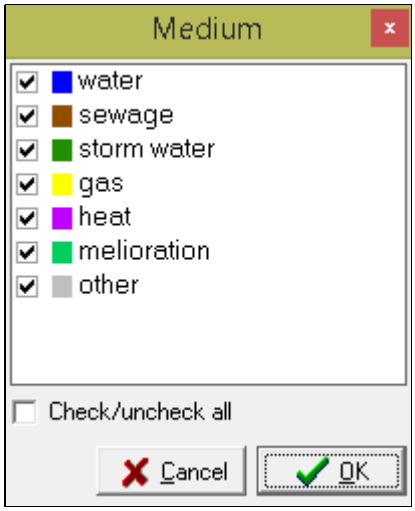
Menu: (Tools > Settings > Drawing > Lines)

On the **Lines** tab, you can set among other color and layer of the following lines created in the drawing:

- terrain,
- pipe,
- trench bottom,
- additional line - first,
- additional line - second,
- table below profile,
- **Infotable**,
- comments line,
- design terrain line,
- axis of the pipe,
- protecting sleeve tube,
- object - refers to manhole and tank type objects, i.e. septic tanks, treatment plants, separators, etc.,
- object marked with a symbol, i.e. gate valves, hydrants, etc,
- top of sidefill,
- terrain hatching.



By removing the check from the box in **Draw** column is possible to block its generation. Line the bottom of the trench is generated only when the amount of bedding on the **Cross-section** tab is greater than zero. Checking the option in the field **Like a pipe** causes lines to be drawn with offsets from some objects, such as manholes. If unchecked, lines are always drawn to nodes. In the Media column (by double-clicking with the mouse or pressing **Enter** on the keyboard), you can specify for which types of utilities, the line is to be drawn (e.g., the pipe axis line is practically not applicable to sewers).



If **Single pipe line** option is checked on the generated drawing pipe will draw by one line.

In the **Draw pipe axis from diameter [mm]** field, you can specify the minimum diameter of the pipe from which its axis will be drawn in the profile drawing (assuming that it was previously selected in the **Draw** to draw column at all).

The **Pipe line color like color** option group allows you to specify how to give color to the pipe line:

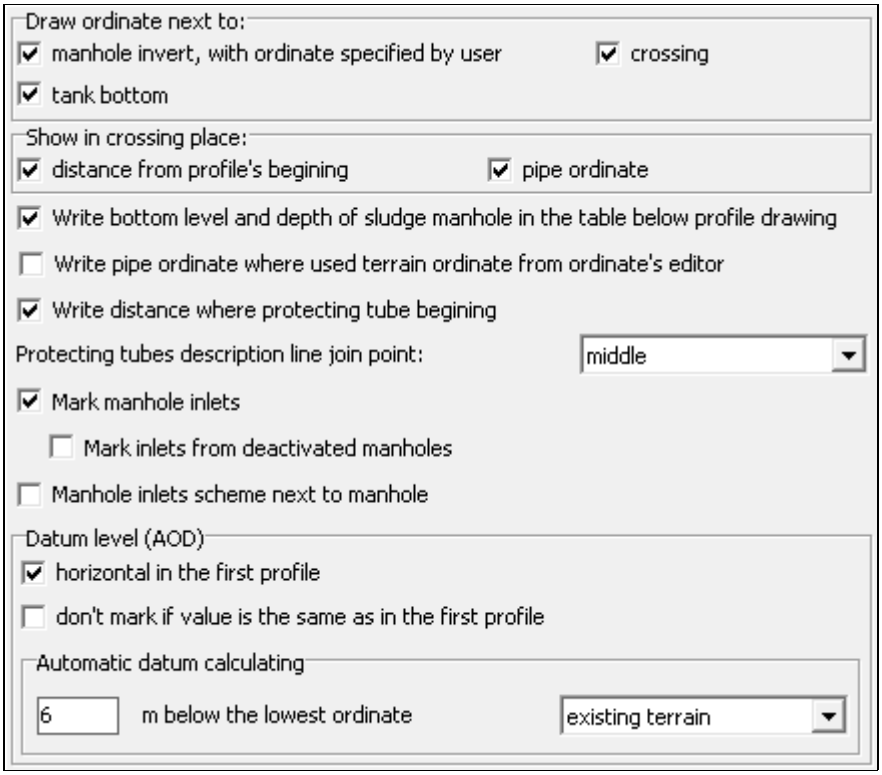
- **pipe from the above table** - the color as in the table for the pipe line will be assigned,
- **pipes in the pipe system** - the color as assigned to the default pipe system in the profile will be assigned. If the pipe system is not selected in the **Data** table then the color will be assigned as for the option **Pipe from the table above**,
- **assigned to the network type** - color will be adopted as assigned to the network type selected for the profile (as in the above screenshot of the **Medium** window).

In addition to the default line, you can add your own line. To do this you must first add a new item in Table with the expression, which will be determined by the ordinates of the new line. Then set the cursor to the last row of the table with the lines and press the key **cursor down** on your keyboard.

For the changes made are saved when you quit, press the button **Save**.

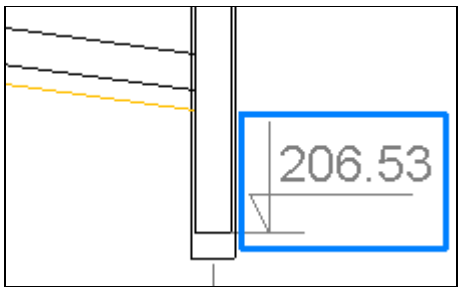
Additional drawing parameters

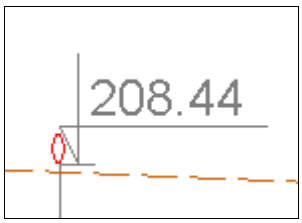
Menu: (Tools > Settings > Drawing > Longitudinal section)



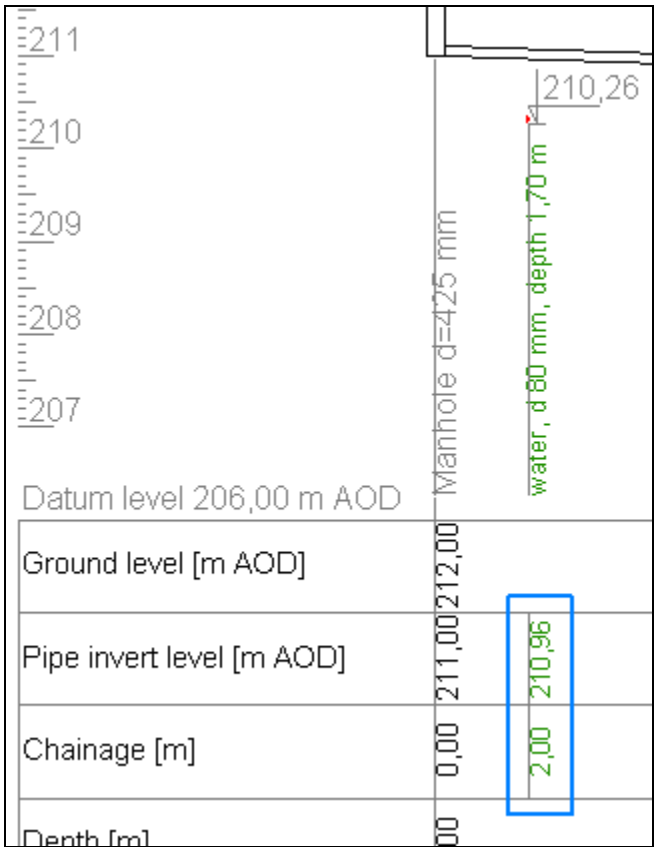
- Checking **Draw ordinate next to... manhole invert, with ordinate specified by user** option, will put on profile, next to the bottom of the manhole, the marker with ordinate value. This option applies only to those manholes for which the user has defined the ordinate.
  Similarly, selecting this option on crossings or tanks causes the ordinate of the crossing or of the tank, on the created profile will next to the its bottom location.

  Below are placed samples with markers next to manhole and crossing.

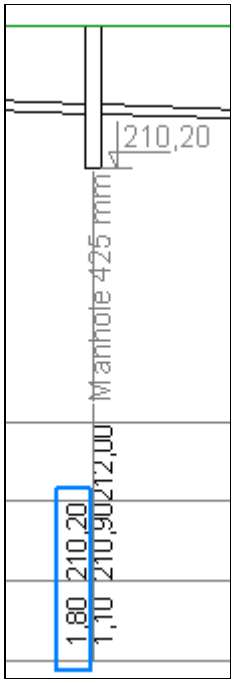




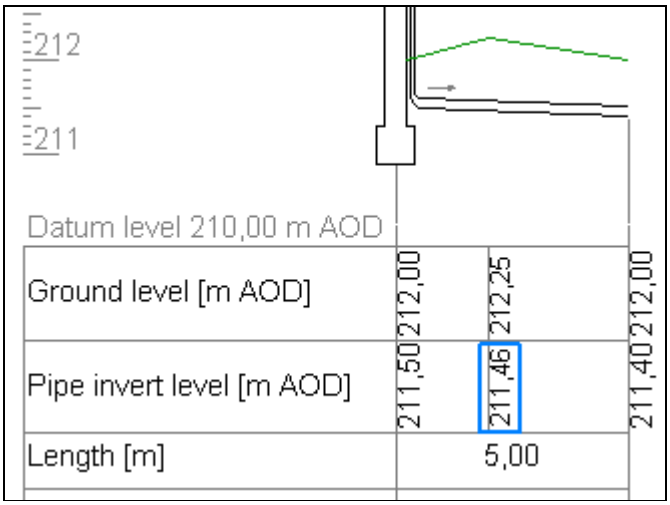
- Checking **Show in crossing place: distance from profile's begining** and/or **pipe ordinate** options causes displaying distance and/or ordinate of the proposed pipeline to the table under the profile, in crossing's location.



- By checking **Write bottom level and depth of sludge manhole...** checkbox, in the table below profile additionaly, next to pipe ordinate and depth, appears bottom level and depth of the sludge manhole (if they are different of pipe ordinate and depth).

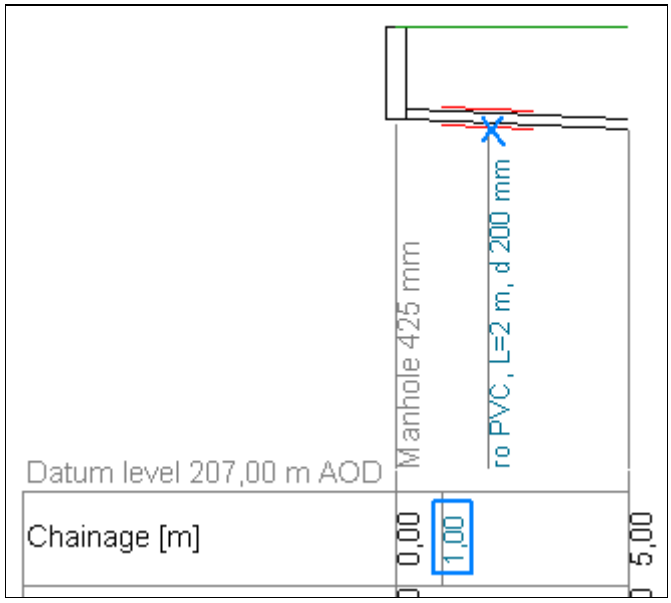


- Selecting the option **Write pipe ordinate where used terrain ordinate from ordinate's editor** causes the pipe ordinate (in the following figure in the blue border), to be entered in the table under the profile, in the place where the user



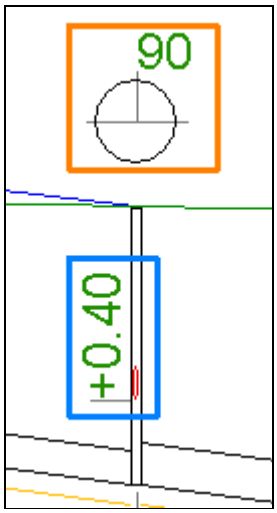
has inserted an additional terrain ordinate.

- Selecting **Write distance where protecting tube begining** option place of the beginning of the protecting pipe will enter the distance in the table below the profile.
- Option **Protecting tube description line join point** allows you to specify the location from which will be derived vertical profile description in the drawing. The choices are: a beginning, middle and end of the casing pipe. In the example below,



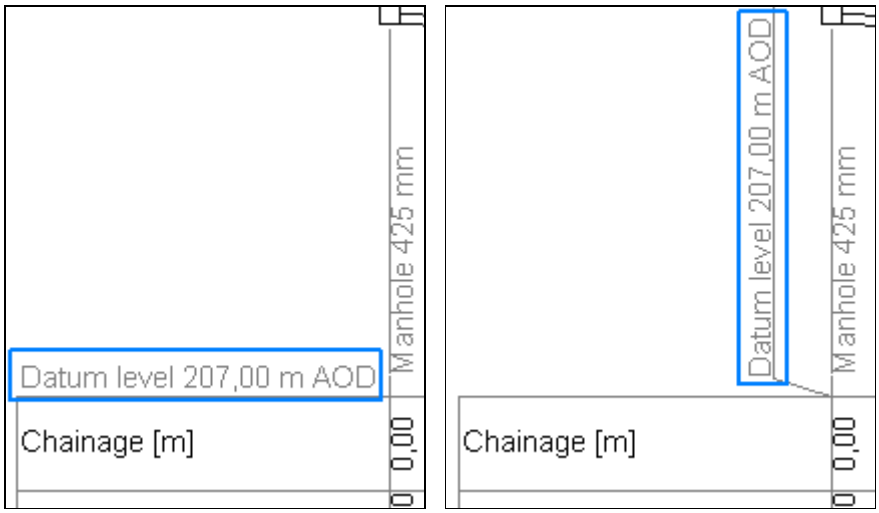
"center" selected and marked with a blue cross.

- Selecting **Mark manhole inlets...** puts on manhole inlets ordinates (from other profiles/tabs located in the same project). Inlets are marked only for projects containing more than one profile.
- The **Mark inlets from deactivated manholes** option works only when the **Mark manhole inlets...** option described above is checked. Selecting it marks inlets on the manholes that also come from profiles that have been disabled from the **Data** tab.
- Checking **Manhole inlets scheme next to manhole** checkbox causes placement of inlets scheme above manhole. This option also affects the placement of the connection diagram on the drawing of the cross section through the manhole. In the example below, marked with an orange border.



- To force the inclusion on the **first** profile string "Datum ground level..." horizontally check **Horizontal datum in first profile.**

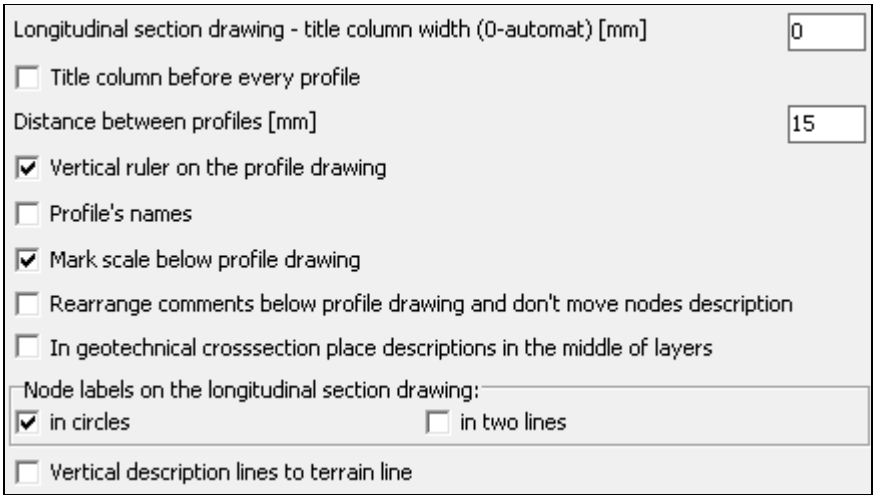
  The example on the left with the option selected:



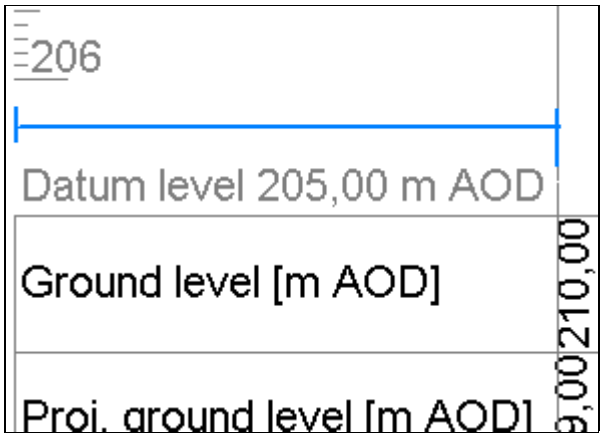
- Checking the option **Don't mark if value is the same as in the first profile** allows to hide value of the datum level on profiles if this value is the same as in the first profile. This option is relevant only in projects with more than one profile.
- Using the options found in the group **Datum level (AOD) > Automatic datum calculating**, you can determine how the comparative terrain level is determined. It can be determined as the smallest value of one of the ordinates: the existing terrain level, the designed terrain level or the bottom of the pipe, decreased by the value specified by the user.

Additional drawing parameters 2

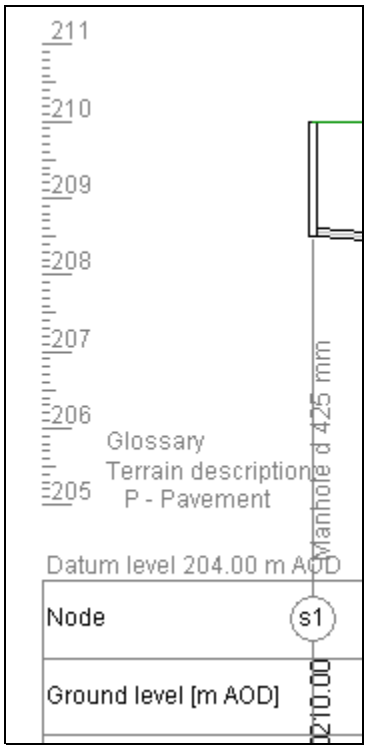
Menu: (Tools > Settings > Drawing > Longitudinal section 2)



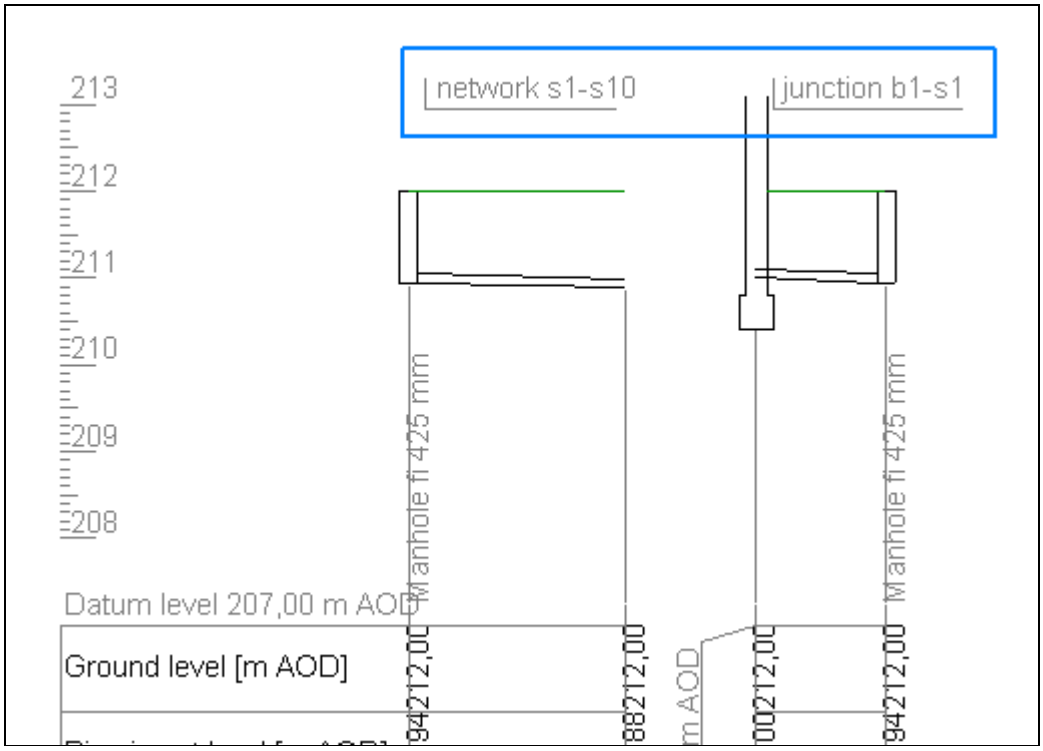
- The edit field **Title column width...** is used to specify the width of the title column in the table under the profile. If a value of 0 (zero) is entered, the width is automatically determined as narrow as possible based on the length of the titles. The width must be expressed in print millimetres. In the example, the column width is indicated by a blue dimension line.



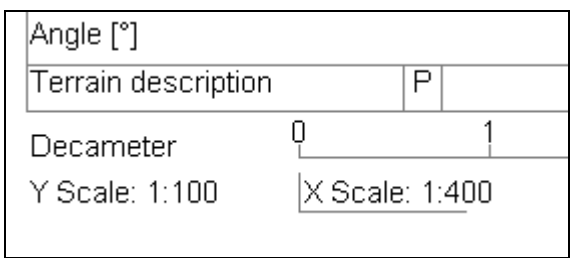
- If the option **Title column before every profile** is selected, a column with the titles of the rows is placed under the profile drawing before each profile. If the option is unchecked, the title column is placed only before the first profile. This option is only applicable to projects with more than 1 profile.
- Edit box **Distance between profiles [mm]** is used to determine the interval between each picture profiles (if the project contains more than one profile). The distance should be expressed in millimeters of print.
- Checking **Vertical ruler on the profile drawing** checkbox place ruler on the left side of the drawing. It's makes easy measure objects ordinate which it hasn't directly specified. If project includes more than one profile ruler is placed only when profile datum level is different from previous.



- When the **Profiles names** option is selected, the profile names (those displayed on the tabs under the Data table) will be inserted at the top of each profile drawing. In the example below, the inserted profile names are surrounded by a blue border.



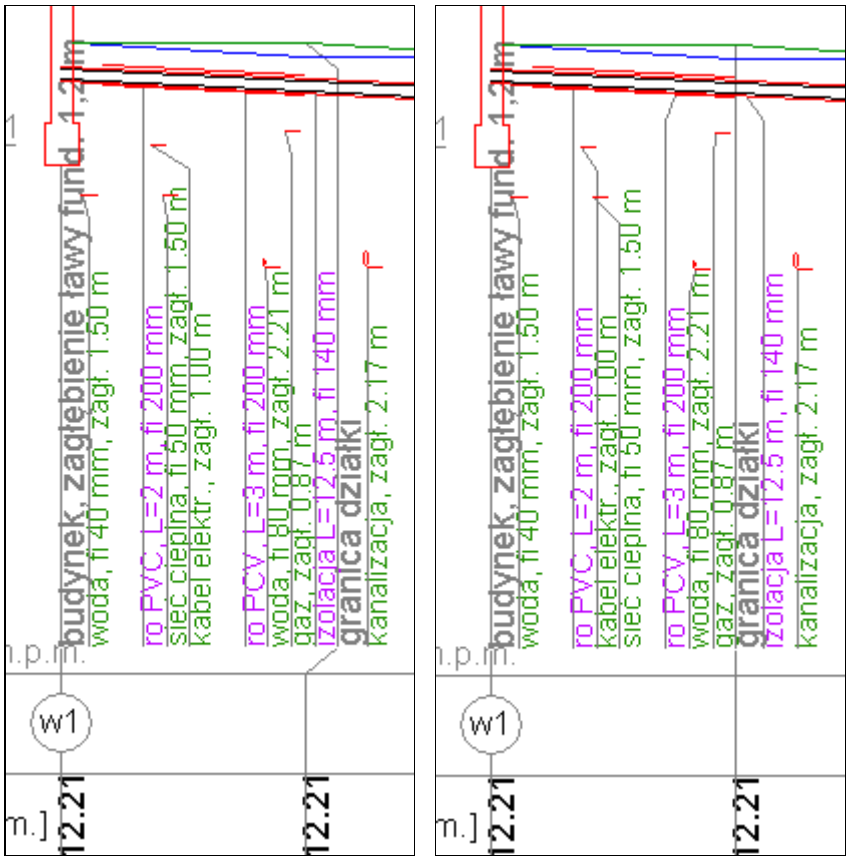
- By checking **Mark scale on the profile drawing** is possible to place information about scales values below table on the profile drawing. Information is also placed, regardless checkbox state, if project contains profiles with different horizontal scales values.



- The option **Rearange comments...** allows one of two algorithms to be selected, responsible for the way in which

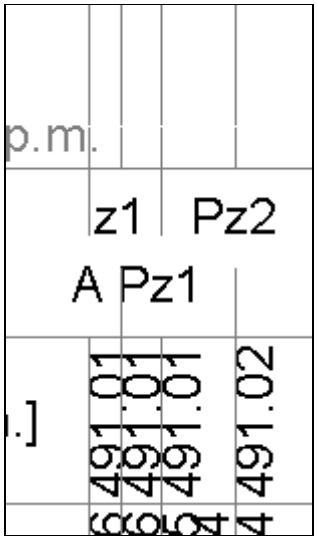
overlapping: comments, collision descriptions, etc., are spread over the profile drawing. When enabled, it allows node descriptions to be left in place, if possible.

Sample with unchecked (on the left) and checked:



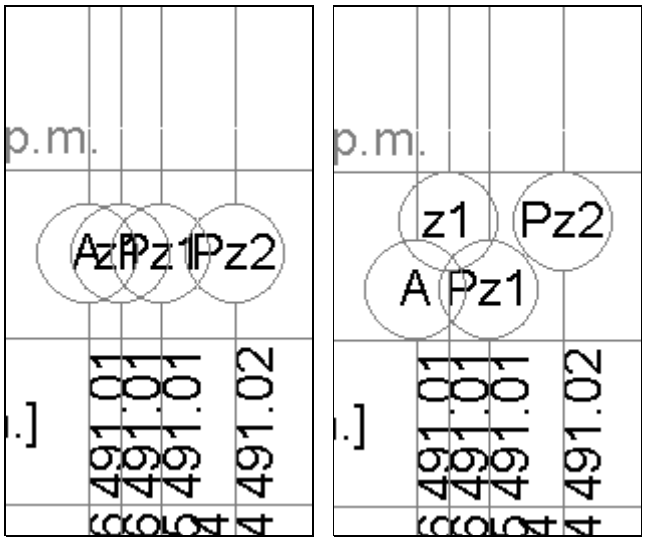
- An option **In geotechnical...** determines form and place of descriptions on cross-section.
- Group of settings **Node labels...** has an influence on placing node labels on the longitudinal section drawing.

When **in circles** checkbox is unchecking nodes are placed on drawing without surrounding circles. Unchecking may helps when node labels contains more than 3 characters and they goes out beyond circle.



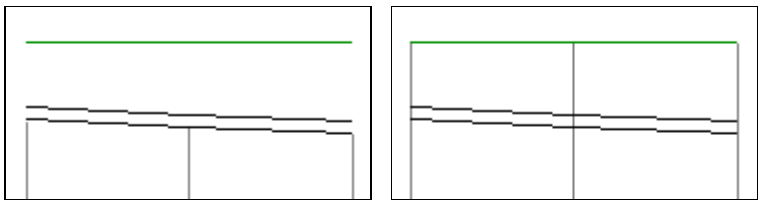
In case small distances between neighbour nodes may be useful to check **in two lines** checkbox. It allow to placing nodes in two lines. It decrease probability of overlapping labels.

Sample with unchecked (on the left) and checked option:



- If the option **Vertical description lines to terrain line** is selected, the vertical line of description/commentary above the pipe is extended to the site line. With the option off, the vertical lines are only under the pipe.

  Sample with unchecked (on the left) and checked option:



Data table - settings

Menu: (Tools > Settings > Data Table)

For each column of the **Data** table, you can specify:

- custom name,
- precision of the displayed numbers (for calculations are used full unrounded numbers),
- width.

The white background shows the columns displayed directly in the **Data** table. While on the colored one - the columns displayed in the **Panel**.

In the drop-down list **Slope unit**, you can select the unit in which the slope will be entered in the **Data** table.

In the **Flow Unit** drop-down lists, you can select independently, the constituent units of volume and time, forming the flow unit in which the flow will be entered in the **Data** table.

No	Name	Custom name	Unit	Prec.	Width
1	Exist. ground level	Exist. ground level	m AOD	0.00	75
2	Pipe level	Pipe level	m AOD	0.00	75
3	Depth	Depth	m	0.00	62
4	Length	Length	m	0.00	50
5	Slope	Slope	%	0.0	50
6	Pipe	Pipe	-	0	30
7	Material	Material	-	0	58
8	Diameter	Diameter	mm	0	58
9	Chainage	Chainage	m	0.00	58
10	Object	Object	-	0	105
11	Comment	Comment	-	0	205
12	Node	Node	-	0	39
13	Proj. ground level	Proj. ground level	m AOD	0.00	75
14	Angle	Angle	°	0.000	75
15	Line 1	Line 1	m AOD	0.00	75
16	Line 2	Line 2	m AOD	0.00	75
17	Crossings	Crossings	pcs.	0	75

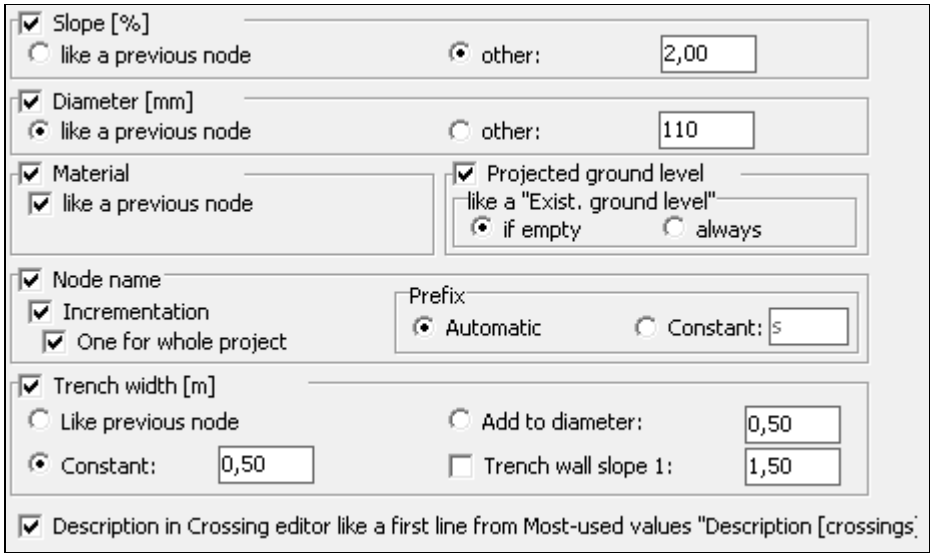
Slope unit: %

Flow unit: m3 / h

Data table - autofilling

Menu: (Tools > Settings > Data Table > Autofilling)

In order to speed up profiles data entry, the program allows to select values that will be automatically inserted into the table. These values can be modified in the **Settings** window on **Autofilling** tab. Values, which may be automatically filled are:

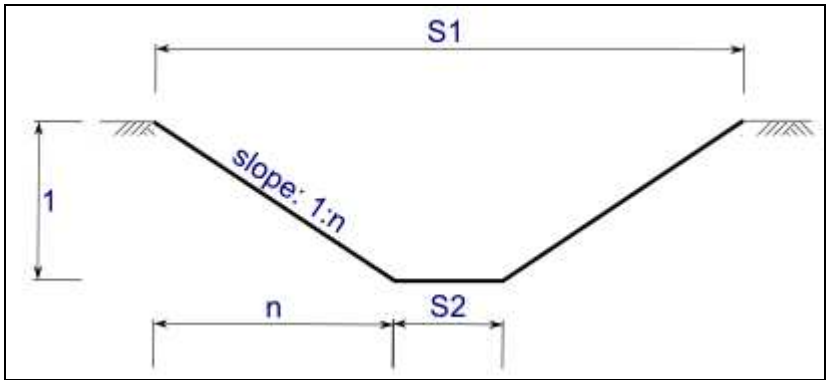


- **slope**,
- **diameter**,
- **material**,
- **node name**

  Node prefix can be created automatically (by the name of the object in a given node) or on the basis of the prefix specified by the user (allowable number of characters in the prefix is 3).
  Checking the **Incrementation** checkbox will automatically add numbers of nodes. Node numbering can be separate for each profile or common for the entire project, in which case the option **One for the entire project** should be selected.

- **trench width**

  In addition to standard options, the width can be calculated by adding the current value of the diameter specified in the **Add to diameter:**. To get the trench with slope walls, choose to **Trench wall slope 1:** and specify the size of the slope.



Checking **like a previous node** causes the insertion to the next, the new node value given in the previous node.

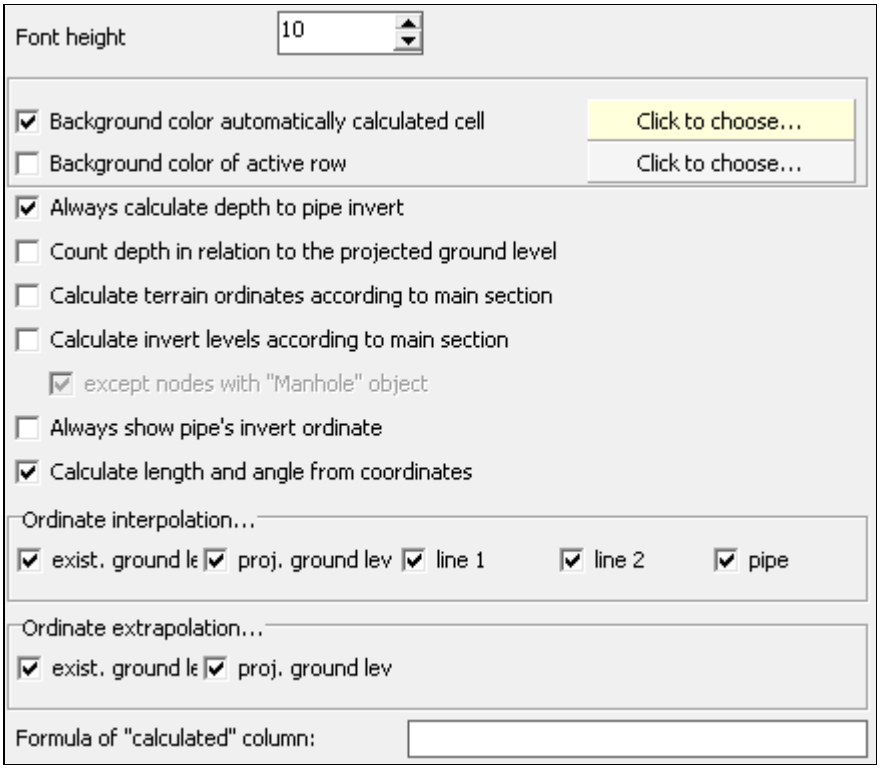
An unchecked checkbox next to a value results in the value not being considered during data entry.

Selecting the **Description in Crossing editor like a first line from Most-used values [...]** option causes, when a new collision is added, the description located in the first line of the Most-used values **Description [crossings]** to be inserted.

For the changes made are saved when you quit, press the button **Save**.

Data table - settings

Menu: (Tools > Settings > Data Table > Table parameters)



- By increasing the **Font height** value in the **Data** table, among other things, the readability of the data entered into it can be improved.
- By selecting the **Background color automatically calculated cell**, you can have the automatically calculated fields in the **Data** table 'highlighted'. The colour of the 'highlight' can be selected by clicking on the button in the currently selected color. Setting of color of active row is possible too.
- The pipe's depth in the **Data** table is calculated to the bottom, axis or top of the pipe (depending on the **Pipe level as** option selected in **Panel** on the **Data** tab). By checking the **Always calculate depth to pipe invert** field, the pipe depth is calculated always to the bottom of the pipe.
- When the option **Count depth in relation to the projected ground level**, the depth value is calculated as the difference between the ordinate of the design ground and the ordinate of the pipe. Otherwise the depth is calculated as the difference between the ordinate of the existing ground and the ordinate of the pipe.
- If the **Calculate terrain ordinates according to main section** option is selected, in a multi-profile project in which the main profile has been chosen, the ordinates of the existing ground and projected ground in connections diverging from the main profile will be automatically updated. This happens every time the user changes the ordinate of the ground in the main profile, even if it results from interpolation. Connections are identified by their node names.
- If the **Calculate invert levels according to main section** option is selected, in a multi-profile project in which the main profile has been selected, the pipe ordinates of the connections diverging from the main profile will be updated automatically. This happens every time the user changes the ordinate of a pipe in the main profile. The connections are identified by their node names. The ordinates are recalculated when in the Panel:
  - on all connections the **Pipe level** must be selected as **Value entered manually**,
  - both the main profile and the connections must have the same Pipe **Network type** set,
  - the connections must have the correct **Flow direction** set

  Manholes can be excluded from automatic updating by selecting the option **except nodes with "Manhole" object**. This may be necessary in the case of cascading sewer connections, for example.
- When the **Always show pipe invert ordinate** option is selected, the bottom ordinate is displayed in the Data table in the Pipe ordinate column regardless of the selection in the **Pipe level as** option.
- With **Calculate length and angle from coordinates** option is checked, the fields in the **Data** table, **Length** and **Angle**, will be calculated from the node coordinates entered (only if two neighbouring nodes have different coordinates).
- Selecting the option from the group **Ordinate interpolation...** causes automatic determination of ordinate (if it has not been entered manually into the table): existing ground level, designed ground level, line 1, line 2 and pipe (if **Pipe level** is set in the **Panel** as manually entered value).

  A prerequisite for automatic interpolation is the presence of a manually entered ordinate at any node **before** and **after** the node to be interpolated.

  In addition, any number of ordinates can be interpolated simultaneously between nodes with already determined ordinates. To do this, press the Ctrl key on the keyboard and keep the Ctrl key pressed while holding the left mouse button down to select the nodes to be interpolated together with edge nodes with already defined ordinates. Then right-click on the selected area and select **Interpolate between nodes**.

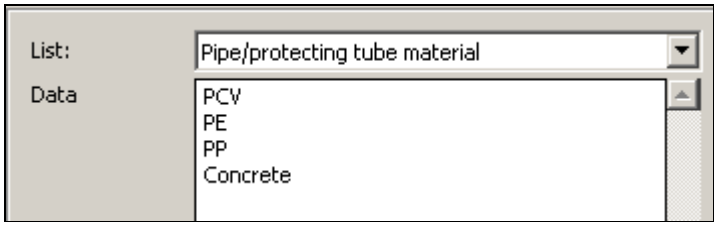
- Selecting the option from the **Ordinate extrapolation...** group. causes automatic determination of ordinate (if it has not been entered manually into the table): of the existing terrain and of the designed terrain.

A prerequisite for automatic extrapolation is the presence of a manually entered ordinate at any node. For e.g. flat terrain, it is sufficient to enter the ordinate manually in only one node - the others will be filled in automatically.
For better orientation, nodes where the ordinates are interpolated/extrapolated are coloured differently from those where the value has been manually entered.

The Formula edit box of the **Formula of "calculated" column** allows you to define an expression using variables (starting @_td_) and numeric values. From the expression, its value is calculated, which is displayed in the calculation column of the **Data table** (**Calculated** column). E.g.: If you enter @_td_bottom_level as a formula, in the Data table the ordinate of the bottom of a manhole in a given node (if a manhole exists there) will appears in the **Calculated** column.

Configuration of most used values

Menu: (Tools > Settings > Most-used values)



In order to speed up data entry to the **Data** table, to Crossing editor and to Sleeve protecting tubes editor program allows the user to define values that will appear on the lists of hints. These values can be modified in the window **Settings > Most-used values**. You have the following data list:

- **Material** - appears on lists in **Data** table and in the **Sleeve protecting tubes editor**;
- **Comment** - used in **Data** table in **Comment** column;
- **Medium** - used in **Crossing editor**;

In addition to the name of the medium, its default depth expressed in metres can be defined, e.g.: "water|1.8". It is important to separate the name from the depth with the "|" character.
The prerequisite for collision completion to work is that the option **Crossing's depth is constant** is selected in the **"Crossings editor"** group, menu: (Tools > Settings > Other)

- **Description - crossings** - used in **Crossing editor**;
- **Description - sleeve protecting tubes** - used in **Sleeve protecting tubes editor**;
- **Node description - schema** - used on schema drawing to describe the nodes;
- **Section description - schema** - used on schema drawing to describe the sections;
- **Pipe type - sleeve protecting tubes** - used in **Sleeve protecting tubes editor**;
- **Calculation type - calculation** - used in **Calculation configuration**;
- **Terrain descrition - terrain descriptions** - used in **Terrain descriptions** window;
- **Manhole node description - scheme** - used in the schematic diagram to describe nodes containing manholes
- **Object description - objects** - used in the Objects window

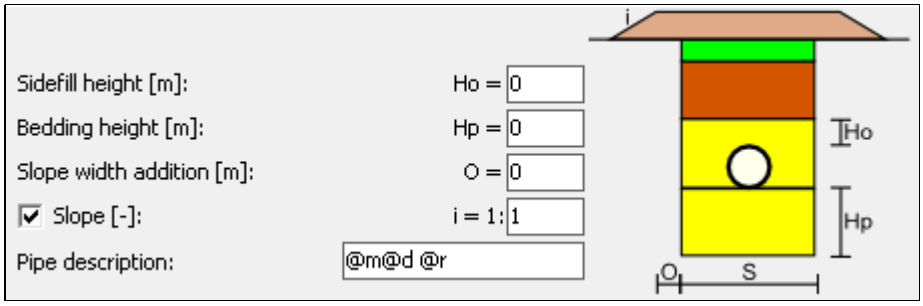
The selection of an appropriate list of data is done via drop-down **List**.

In the case of most lists is the preferred use of the available variables. This allows you to use the same description for many cases.

To make permanent the new value to your data, add it and press **Save**.

Cross section - settings

Menu: (Tools > Settings > Cross-section)

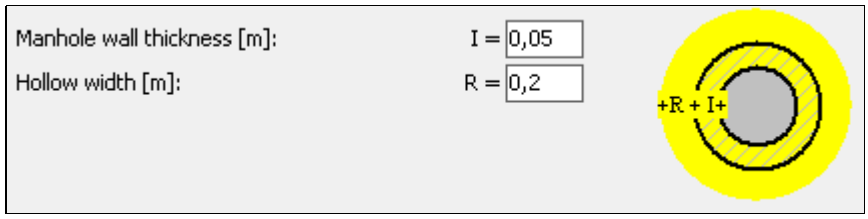


Changing the value **Sidefill height** and **Bedding height** affect the determination of approximate volume of excavation, sidefil and bedding in the **Statistics** window. In addition, these parameters are taken into account when generating the cross-sectional drawings.

Text box **Pipe description** is used to describe the pipe placed on the cross-section. It is advisable to use the description of the following variables:

- **@m** - pipe material,
- **@d** - pipe diamter [mm],
- **@o** - sleeve protecting tube diameter [mm],
- **@r** - pipe ordinate [m AOD],
- **@z** - pipe bottom depth [m]

For example, write "@m@d @z m" will be converted automatically in the drawing for the description of "PCV160 1.60 m".

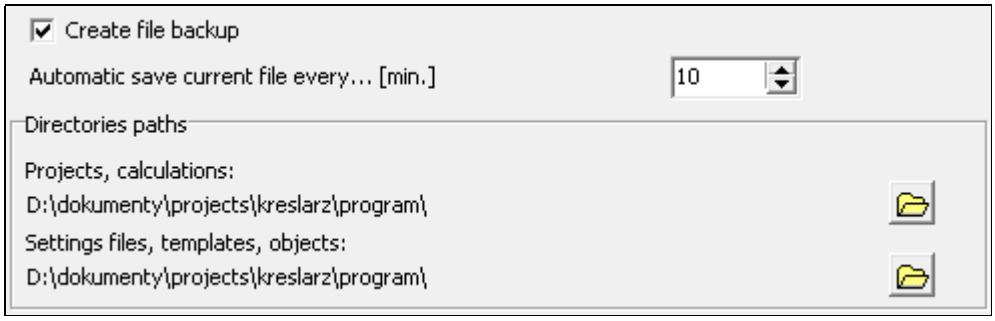


Enter the manhole wall thickness (applies only to the **Inspection** object) in the **Sump wall thickness** text box. This will be used when generating the manhole cross-section. For a manhole object, the wall thickness can be defined independently for each manhole in the **Object parameters** window.

The value given as **Hollow width** indicates how much wider than the diameter of the manhole the excavation underneath the manhole must be in order to be able to place the manhole. It will be used, among other things, to determine the excavation volume. The volume resulting from the specified hollow width will be included in the statistics as sidefill.

Safety settings

Menu: (Tools > Settings > Safety)



To improve the security of data, stored in a project file, you can select **Create file backup**. Selection makes when you open the file, create a copy of it. A copy is saved in the same directory as the opened file. Name the backup file is created with the name of the file to open and an additional extension .bak.

A backup is created if it does not yet exist or if the existing copy of the file being opened is older than 1 day. To recover data from a backup file, remove the ".bak" extension from the file name. The file extension will then become ".kre", i.e. the file will be opened with Drafter.

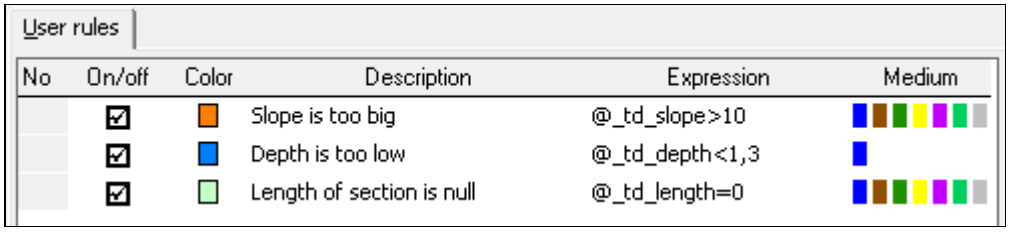
In addition, it is possible to activate the function of automatically saving an open file. To do this, enter a time, expressed in minutes, after which the program should save the data. Drafter saves the project to a file named "autosave.~kr", located in the directory with the Drafter configuration files. The location of this directory can be checked in the **Directory paths** box. **It should be noted that the autosave feature does not relieve you from manually saving the file**. It provides a safeguard against the loss of recently entered data in the event of a possible sudden spontaneous shutdown of the program (e.g. an unplanned power cut). If such an event occurs, after restarting Drafter, a message will be displayed indicating the existence of an unsaved file and the possibility of recovering it.

Specifying a time equal to zero minutes disables the autosave function.

Warnings settings

Menu: (Tools > Settings > Warnings)

Any profile made in the Drafter can be checked using self-created rules (several examples are built into the program). Rules are created using variables, usually from the **Data table** group, starting with @_td_. Each rule can be assigned to specific types of networks (e.g. sewer only).



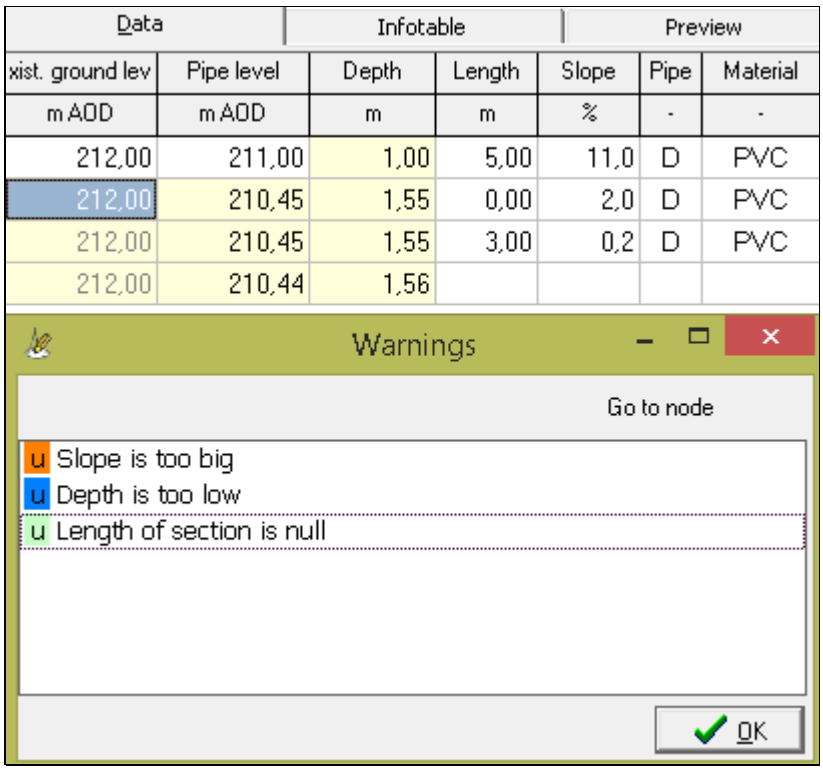
For each rule, you can specify:

- **Colour** - this will be visible in the warning list when the condition specified in the rule formula is met,
- **Description** - this will be visible in the warning list when the condition given in the rule formula is met.

  Additional variables can be used in the Description field:

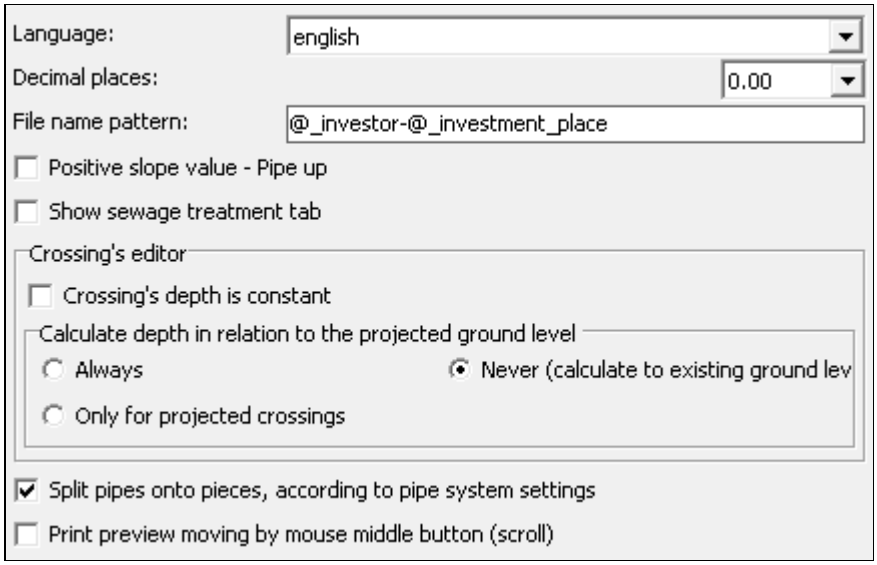
  - **@s** - the name of the profile in which the condition has been met,
  - **@n** - the name of the node where the condition has been met,
  - **@vmin** - the smaller of the values compared in the formula,
  - **@vmax** - the bigger of the values compared in the formula,
  - **@vl** - the value of the left-hand side of the equation in the formula,
  - **@vr** - the value of the right-hand side of the equation in the formula.
- **Expression** - an equation whose truth will be checked for each node in the profile. The equation should use variables (starting @_td_) and numeric values (e.g. for the formula @_td_slope<1 all nodes where the slope is less than 1 will be returned).
- **Media** - select the media to be affected by the rule (to call up the selection window, double-click with the mouse or press the **Enter** key on the keyboard.

The project can be checked by selecting **Tools > Check with rules** from the menu. If the condition of any of the formulas is met then a warning window will be displayed. Clicking the **Go to node** button will highlight the node in the **Data** table where the formula has been met.



Other settings

Menu: (Tools > Settings > Other)



- In the **J?zyk/Language** drop-down list, you can select the language in which all texts in the Drafter will be presented.
- From the selection list **Decimal places**, you can specify to how many decimal places the numbers displayed in the program's tables will be rounded (e.g. Editors: collisions, casing pipes, etc.). Only numbers displayed and not used in calculations are rounded. This option does not apply to the **Data** table, as the precision can be selected independently for each of its columns.
- In the **File name pattern** textbox, you can define, using variables, the template according to which the program will suggest a file name when trying to save a newly created project.
- The unchecked **Positive slope value - pipe up** checkbox means that if the slope is given as a positive value, the programme will calculate the ordinate of the bottom of the next node which is smaller than the ordinate of the current node. The opposite interpretation can be obtained by checking the aforementioned checkbox.
- If the **Sewage treatment plant** tab in main window is not used, it can be hidden by unchecking the **Show sewage treatment tab** checkbox.
- If the **Crossing's depth is constant** option is selected in the **Crossings editor** group, the depth of the entered collision will not change, but the collision ordinate will be recalculated. Similarly, if the option is left unchecked, the depth of the collision will be recalculated and the collision ordinate will not be changed.
- In addition, in the option group **Calculate depth in relation to the projected ground level**, the calculation of the collision depth can be specified. The depth can be calculated in relation to the ordinate of the proj. ground level:
  - always
  - never - then it will always be calculated in relation to the existing ground ordinate
  - only for collisions marked as designed in the **Crossings Editor** - then the depth of collisions marked as existing will be calculated to the ordinate of the existing ground.
- By selecting the option **Divide pipes into commercial lengths...** in the list of materials, you can obtain specific pipe quantities, divided into lengths of commercial lengths. This option can be particularly useful for sewer profiles. The division of pipes into commercial lengths is only possible for pipes assigned to a previously added pipe type series.

  Example of a statement with the aforementioned option enabled....

Material	Quantity	Unit	Node	Index	Producer
Pipe PVC-U 110	**20.00**	m			
Pipe PVC-U, Wavin, 160x4 mm, l=6 m	**3.00**	szt.		3062923446	Wavin
Pipe PVC-U, Wavin, 160x4 mm, l=3 m	**1.00**	szt.		3062923443	Wavin
Pipe PVC-U, Wavin, 160x4 mm, l=2 m	**1.00**	szt.		3062923442	Wavin
Pipe PVC-U, Wavin, 200x4.9 mm, l=6 m	**3.00**	szt.		3064923862	Wavin

  ...and with the disabled

Material	Quantity	Unit	Node	Index	Producer
Pipe PVC-U 110	**20.00**	m			
Pipe PVC-U 160	**23.00**	m			
Pipe PVC-U 200	**18.00**	m			

- If you select the option **Print preview moving by mouse middle button (scroll)**, you can change the way the drawing on the **Preview** tab is moved from the right mouse button to the middle mouse button.

Infotable editor - features

Infotable editor is located in the main window on the **Infotable** tab. With it you can create a table, which will include basic data such as describing the development, designer, etc. It can be virtually any shape and include additional data.

Inserting text boxes in the editor:

- **Text box** - Insert a text box.
- **Title list** - Placed in the editor drop-down list the titles of the drawings. If you type the title that is not listed, then it will be stored permanently after going to the **Preview** tab.

Navigation: Use the arrows buttons 

to change **Location** or **Size** of box in the editor where the cursor is currently positioned. After pressing **x10** button is increased ten-fold jump during the operation using the arrow keys.
To drag textbox within editor hold **Ctrl** key on the keyboard.

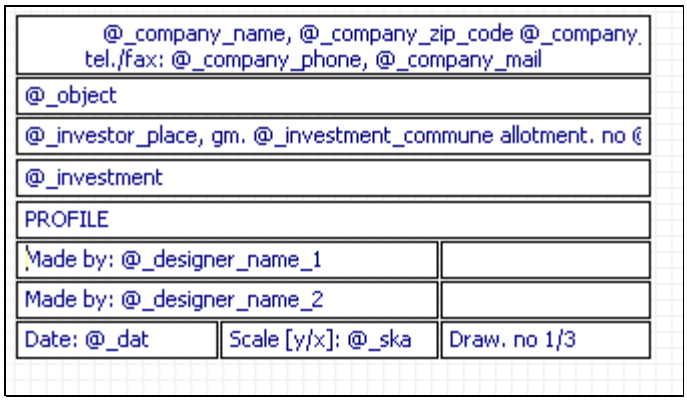
Deleting text boxes: Pressing **Delete text box** button will delete the text box in which the cursor is currently positioned.

If there is no need to put the **Infotable** in the drawing, press the button.

Infotable editor - working with templates

Formed metric can be saved as a template by pressing the button. The saved template can be used in new projects after pressing button and selecting from a list file with the appropriate template. You can create unlimited number of templates.

To make the template the default for each new project, it must be indicated as the default in the **Settings** window (Tools > Settings > Drawing).



You can enter anywhere in the template variables listed in **Variables** chapter or one from the following ones:

- **@_ska** - in the generated drawing is converted to the current scale. (e.g.: the template entry "Scale [y/x]: @_ska" in the generated drawing will look as follows: "Scale [y/x]: 1:100/100").
- **@_dat** - in the generated drawing is converted into the current date. The format of the inserted date depends on the Windows settings.

The picture shows a sample of **Infotable** template. It draws on all available types of items, and some variables.

Print preview

To view the generated drawing, click on the **Preview** tab in main window of the Drafter. The Drafter can generate the following drawings:

- longitudinal cross-section,
- diagram, i.e. for example the network with the connections as seen as on a map,
- cross-section through manholes, without having to define it in the **Cross-section Editor**,
- cross-section defined in the **Cross-section editor** at any point in the pipeline (also through manholes and tanks).

The following describes functions of the buttons on the tab:

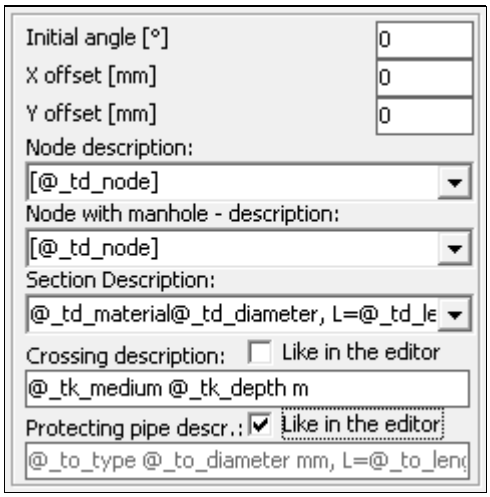
- zoom in, (also key + on numeric keyboard)
- zoom out, (also key - on numeric keyboard)
- original size,
- previous view.

User can hide/show some groups of elements of the drawing by pressing below mentioned buttons:

- crossings,
- sleeve protecting tubes,
- vertical cross-sections,
- parcel boundaries,
- pagination.

Marking of collisions and plot boundaries in the schematic drawing is in a simplified version, i.e. it does not take into account the angle projected on the horizontal plane between the projected pipeline and the collision or the plot boundary (in the schematic it is always a right angle: 90 degrees). Objects located in the hidden part of the "shortened" pipeline are not shown in the schematic drawing (when using the option: **Trim sections longer than...**).

Additionaly after **Panel** activation (by pressing **F4** key) following options are available to adapt the scheme drawing:



- Determining the initial angle of the scheme.
- X Offset - Horizontal offset from the position imposed automatically by the program.
- Y Offset - Vertical offset from the position imposed automatically by the program.
- Node description - pattern of node descriptions.
- Description of a node with a manhole - template according to which descriptions of nodes where the **Manhole** object is located will be applied.
- Section description - pattern of section descriptions.
- Crossing description - a template according to which crossings descriptions will be applied to the profile and scheme drawings.
- Sleeve protecting pipe description - template according to which the sleeve protecting pipe descriptions will be applied on the profile and scheme drawings.

By unchecking the **Like in the editor** checkbox, a template for the description of the crossing and protecting pipes can be imposed (independently). With this option selected, the descriptions from the respective editors will be used.

In the node and section descriptions, the variables (located in the **Data table** section) should be used. On the other hand, in the crossing and protecting pipe descriptions, the variables placed in the crossings and protecting pipe sections respectively. Inserting **\P** in the body of the description of a node and in the description of a node with a manhole moves the fragment of text behind the **\P** to a new line.

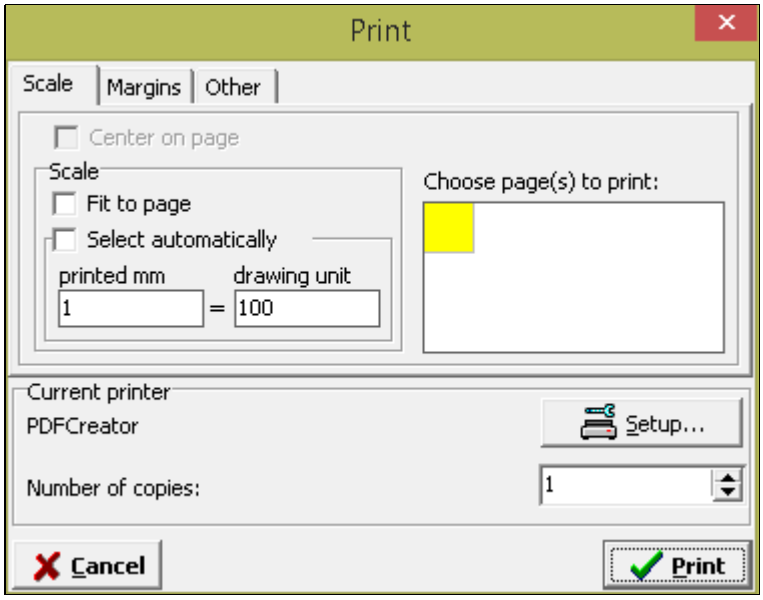
By clicking **Open preview in new window** can move the preview to the new window. This allows you to "live" observe the impact of changes on the created drawing. This option can significantly improve the ease of working with computers designed for simultaneous display on two monitors. Not recommended for use in a separate preview window for older computers.

Hints:

- double clicking the left mouse button anywhere in the preview is the place to move to the center of the preview,
- circle with the mouse any viewing area, with the left button, causes its enlargement,
- moving the mouse while pressing the right button to move the generated drawing,
- mouse wheel rotation, depending on its direction, causing zooming in or out of the picture,
- drawing can be moved using the cursor keys,
- hold **Ctrl** key while pressing the cursor keys result in faster movement.

Printing

If the automatically generated profile does not require further modifications in the CAD application you can print it by clicking the button with the printer or by choosing in menu **Project > Print ...**.



In the window that appears on the tab **Scale** need to decide on the scale of the print giving the ratio of output to the amount of millimeters adopted drawing entities. Optimal choice can be made automatically by selecting **Select automatically**.

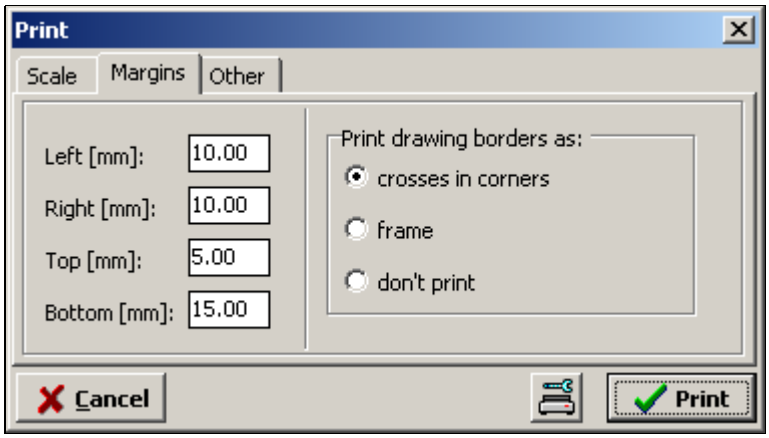
If the printed image has to fit on one page, select **Fit to page**. Then the program selects the same scale so that the drawing is best suited to your page. If **Fit to page** checkbox is checked you can center the drawing on the printed page by selecting **Center on page**.

On the right side of the window you can select pages to be printed. Each grid corresponds to one printed page for the currently selected paper size. By clicking on individual boxes or mark them for printing. Will be printed only in yellow boxes marked. The program shall appoint the necessary number of sheets of paper (squares) for the profile. The user may try to change the number of pages by changing the margins settings or choose a different paper size by pressing the **Printer Setup** (next to **Print**).

In addition, in the **Number of copies** field you can specify how many copies the printout will be made. The default value is 1.

Margins settings

In **Print** window on the **Margins** tab, you can change the range of printing on paper, for example, typing in the text box **Left [mm]** value of "11.0" get unprinted area with a width of 11.0 mm, starting from the left edge of the paper.
If the user enters a value less than that provided for a particular type of printer from the manufacturer, then the program will impose a minimum allowable value of the modified margin.

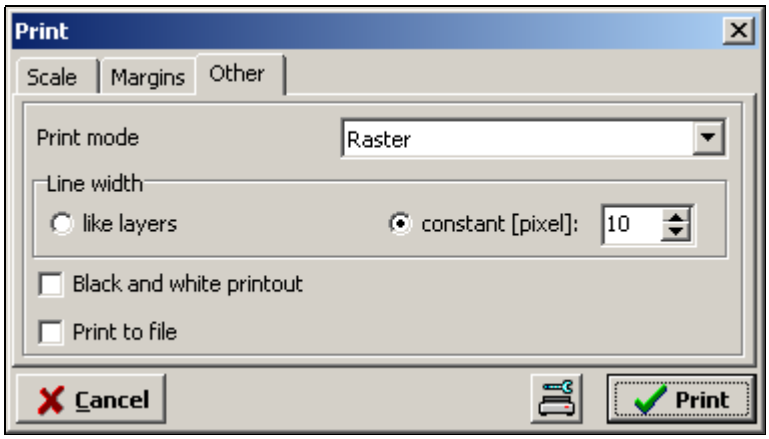


If the output profile will take more than one piece of paper worth uncheck the box **Print drawing boundaries as > crosses in corners**. This reprinting of small crosses marks in the corners of the drawing, which may be useful for gluing the individual pieces of the profile.

To obtain a printout drawing a border, choose Print **drawing boundaries as > frame**.

Other settings

In **Print** window on the **Other** tab, you can choose how to create a print between vector and raster. Raster method is a bit slower and should be selected only when the print using the vector method is incorrect (e.g. out of the text or lines).

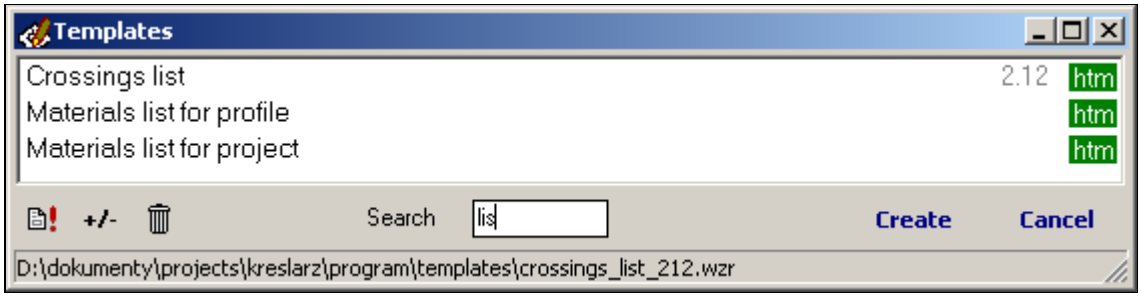


If the raster method is chosen it is possible to obtain a print black and white. To do this you need to select **Black and white printout**. This option allows for a clearer print. This is evident especially in printers with low resolution or poorly printing grayscale.

When you select **Print to file** option after pressing **Print** in the Drafter directory is created file with the drawing (without sending the print to the printer.) In the vector method to create the print, the file is **print.emf**, while the raster file **print.bmp**.

Working with templates

Menu: (Tools > Templates)



Content of generated documents is drawn from a template placed in a subdirectory "templates". To edit the template press **Template edit** button placed in **Templates** window. Template file has WZR extension. Its content can also be modified in any text editor (e.g.: Notepad).

Warning! Modification of the template involves the appearance of change generated in the future, documents that are based on the template. It is advisable to save the modified template with a different name to the original, as template files are overwritten during program updates.

Lesser-used templates can be hidden by pressing the **+/-** button. Hidden templates, you can see by pressing the button with the image of the trash. Restoring the hidden template is held by analogy as hidding.

With the program among others templates are being delivered: specification of **Data** table, materials specification, profile statistics. Example prints using them can be viewed on the website in the Screenshots section.

You can create your own document template that uses customer data (e.g. request to the office, technical description, imprint to the envelope, any protocols, etc.). To do this, create a file of any name and WZR extension (eg test.wzr) and open it in a text editor (e.g. Notepad). Template must include a heading and actual content of the document. When you create a new template must be placed in a "templates" subdirectory. It is best to start learning how to work with variables by reviewing the template files supplied with the Drafter in a text editor (WZR files in the "templates" directory).

Template header sample:

```
<HEADER>
<TITLE>Profile statistics</TITLE>
<FONT_SIZE>12</FONT_SIZE>
<FONT_NAME>Tahoma</FONT_NAME>
<TYPE>htm<TYPE>
</HEADER>
```

Where:

TITLE: The title of the template, which appears in the drop-down list of templates
FONT_SIZE: The font size for a given template
FONT_NAME: Font name for the template
TYPE: Document type (**htm** - allows text formatting, placement of tables, but it is difficult for edit; **txt** - have fewer opportunities generated document formatting, but readily undergo editing).

To put the data in the created document, you must use the variables. They can be inserted anywhere in a document containing plain, non-variable text.

Variables

There is a list of variables you can use in Drafter. Especially, in work with templates.

User data
@_company_name - company name
@_company_place - place/street
@_company_zip_code - zip code
@_company_post - post office
@_company_phone - phone number
@_company_fax - fax
@_company_mail - e-mail
@_company_www - www address
@_company_user1 - user data
@_company_user2 - user data
@_company_user3 - user data

Project data
@_investor - investor
@_investor_place - address: city / street
@_investor_zip_code - address: postal code
@_investor_post - address: post office
@_object - object
@_investment - investment
@_investment_place - city / street
@_investment_commune - community
@_investment_zip_code - postal code
@_investment_post - post office
@_investment_allotment_no - allotment number
@_project_type - trade
@_project_oder_no - order number
@_project_date - date
@_user1 - user data
@_user2 - user data
@_user3 - user data

Calculation data
@_k_investor - investor
@_k_investor_place - address: city / street
@_k_investor_zip_code - address: postal code
@_k_investor_post - address: post office
@_k_object - object
@_k_investment - investment
@_k_investment_place - city / street
@_k_investment_zip_code - postal code
@_k_investment_post - post office
@_k_investment_commune - community
@_k_investment_allotment_no - allotment number
@_k_project_type - trade
@_k_project_oder_no - order number
@_k_project_date - date
@_k_user1 - user data
@_k_user2 - user data
@_k_user3 - user data

Adding at the end of the variable ' (apostrophe) character its value is retrieved from the data contained in the calculation.
Conversely, if you select **Save calculation parameters with KRE file** as the value of the variable will be inserted into the similar, the data retrieved from the project.

People taking part in the creation of documentation (max. 9)
@_designer_name_1 .. @_designer_name_9 - names
@_designer_offic1 .. @_designer_offic9 - job titles
@_designer_right1 .. @_designer_right9 - right number
@_designer_speci1 .. @_designer_speci9 - specialty

Current profile statistics
@_stat_volume_excavation - volume of excavation
@_stat_volume_sidefill - volume of sidefill
@_stat_volume_embankment - volume of enbankment
@_stat_volume_bedding - volume of bedding
@_stat_volume_to_exchange - volume to exchange
@_stat_area - turf area
@_stat_walls_area - side walls area
@_stat_depth_max - max. depth
@_stat_depth_min - min. depth
@_stat_slope_max - max. slope
@_stat_slope_min - min. slope
@_stat_stat_length - profile length
@_stat_sections_quantity - number of sections in the profile
@_stat_section_max - length of the longest section in the profile

Total profiles statistics
@_stat_avolume_excavation - volume of excavation
@_stat_avolume_sidefill - volume of sidefill
@_stat_avolume_embankment - volume of enbankment
@_stat_avolume_bedding - volume of bedding
@_stat_avolume_to_exchange - volume to exchange
@_stat_aarea - turf area
@_stat_awalls_area - side walls area
@_stat_adepth_max - max. depth
@_stat_adepth_min - min. depth
@_stat_aslope_max - max. slope
@_stat_aslope_min - min. slope
@_stat_alength - profile length
@_stat_asections_quantity - number of sections in the profile
@_stat_asection_max - length of the longest section in the profile
@_stat_tank_excavation_volume - volume of excavation for the septic tanks occurring in the profile
@_stat_joins_amount - joins amount to existing pipelines
@_stat_wall_quantity - amount of break through the wall of the pipeline route
@_stat_yield_surplus_disposal_quantity - quantity of exports of surplus excavated material
@_stat_pumpstation_quantity - number of pumping stations in the profile
@_stat_acrossings_count - crossings count
@_pumpstation_exists - if the profile is pumping a variable returns the value 1 (otherwise 0)

Excavation cross-section
@_trench_bedding_height - bedding layer height
@_trench_sidefill_height - sidefill layer height

Water treatment plant
@_o_filter_width - vertical sandfilter width
@_o_filter_length - vertical sandfilter length
@_o_filter_area - vertical sandfilter area
@_o_drain_trench_quantity - drainage trench quantity
@_o_drain_trench_length - drainage trench length
@_o_drain_field_quantity - drainage fields quantity
@_o_drain_length - drainage total length
@_o_gravel_volume - amount of gravel below drainage [m3]
@_o_sand_volume - amount of sand below drainage [m3]
@_o_geotextile_length - geotextile length
@_o_septic_tanks_volume - septic tanks volume
@_o_septic_tank - septic tank type
@_o_separator - separator type
@_o_vent_length - underground vetilation length
@_o_pumpstation_distance - electric wire to pumping station length
@_ventilation_wall_inside - if vertical ventilation pipe is on inner side of the wall then variable value is set to 1 (otherwise is set to 0)
@_ventilation_wall_outside - if vertical ventilation pipe is on outer side of the wall then variable value is set to 1 (otherwise is set to 0)
@_ventilation_mast - if vertical ventilation pipe is on mast then variable value is set to 1 (otherwise is set to 0)

Material specfication for current profile
<_MAT> - beginning of the line with the following variables
@_tm_name - material name
@_tm_quantity - quantity
@_tm_unit - unit
@_tm_node - node name (only for manholes)
@_tm_cat_no - catalog index
@_tm_producer - producer's name

Material specfication for all profiles
<_AMAT> - beginning of the line with the following variables
@_tm_aname - material name
@_tm_aquantity - quantity
@_tm_aunit - unit
@_tm_anode - node name (only for manholes)
@_tm_acat_no - catalog index
@_tm_aproducer - producer's name

Data table
<_DAN> - beginning of the line with the following variables
@_td_ground_level - ground level (optionally: @_td_ground_level_l or @_td_ground_level_r for ordinate from the left side or right side of the node)
@_td_max_ground_level - max. ground level in node
@_td_ground_level_proj - projected ground level in node
@_td_pipe_level - invert level (bottom or axis) depending on the settings
@_td_pipe_bottom_level - invert level regardless of the settings
@_td_pipe_level_mode - depending on the settings replaced with the word "bottom" or "axis"
@_td_line1 - ordinate of additional line
@_td_line2 - ordinate of additional line 2
@_td_bottom_level - manhole bottom ordinate in given node
@_td_bottom_levels - ordinates all the pipes reaching the node are separated by a _ character
@_td_depth - pipe depth
@_td_flow_from_section_as_node - flow from the node (variable only to describe the node)
@_td_flow_from_section_as_sec - flow from the section (variable only to describe the section)
@_td_summary_flow_as_node - summary flow in the node (variable only to describe the node)
@_td_summary_flow_as_sec - summary flow in the section (variable only to describe the section)
@_td_slope_unit - slope unit
@_td_slope - slope
@_td_slope_terrain - slope of terrain line between neighbouring nodes
@_td_slope_terrain_proj - slope of projected terrain line between neighbouring nodes
@_td_material - material
@_td_diameter - diameter [mm]
@_td_diameter_m - diameter [m]
@_td_pipe_wall_thickness - pipe's wall thickness [mm] (available when diameter exists in pipe system added to project)
@_td_pipe_wall_class - pipe's class
@_td_pipe_wall_sdr - pipe's sdr parameter
@_td_pipe_wall_sn - pipe's sn parameter
@_td_pipe_pn - pipe nominal pressure
@_td_length - section length
@_td_chainage - chainage
@_td_object - object
@_td_comment - comment
@_td_node - node
@_td_angle - refraction angle in the node
@_td_coord_x - X (horizontal) coordinate of the node
@_td_coord_y - Y (vertical) coordinate of the node
@_td_width - trench width
@_td_embankment - embankment volume
@_td_volume_to_exchange - excavation volume to exchange

Crossings
<_KOL> - beginning of the line with the following variables
@_tk_medium - medium name
@_tk_diameter - crossing diameter
@_tk_pipe_bottom_level - crossing bottom ordinate

@_tk_comment - crossing description
@_tk_node - chosen node
@_tk_distance_from_node - distance from the crossing of the chosen node
@_tk_distance_from_begin - distance from the crossing to begin
@_tk_h - vertical distance from the pipe to crossing
@_tk_depth - crossing depth
@_tk_color - color of the medium type assigned to crossing
@_tk_is_to_remove|'True'|'False' - text set by user (between '', instead True and False) if crosssing is marked as "to remove"
@_tk_is_projected|'True'|'False' - text set by user (between '', instead True and False) if crosssing is marked as projected

Sleeve protecting tubes
<_OSL> - beginning of the line with the following variables
@_to_diameter - sleeve protecting tube diameter
@_to_material - sleeve protecting tube material
@_to_length - length
@_to_comment - description
@_to_node - chosen node
@_to_node_distance - distance from the sleeve protecting tube beginning of the chosen node
@_to_type - pipe type (from „Kind" column of editor)

Manholes
<_SUMP> - beginning of the line with the following variables
@_ts_node - name of the node with the manhole
@_ts_producer - producer's name
@_ts_kind - manhole type
@_ts_name - manhole name
@_ts_description - description
@_ts_diameter - diameter of the manhole [m]
@_ts_cover_diameter - cover diameter [m]
@_ts_wall_thickness - wall thickness [mm]
@_ts_height - height [m]
@_ts_cover_height - cover height [m]
@_ts_cone_height - cone height [m]
@_ts_extension_height - extension height [m]
@_ts_extension_count - extension count [szt.]
@_ts_inlets - inlets' ordinates separated by coma [m AOD]
@_ts_bottom_ordinate - bottom ordinate [m AOD]

Other
@_t_no - sequence number in the tables
@_data - current date
@_compare_level - compare level
@_profile_name - profile name
@_crosssection_name - cross-section name
@_drawing_name - name of the generated figure (profile or cross-section)
@_drawing_number - number of the generated drawing
@_ska - scale of drawing described as „1: y/x"
@_scale_x - horizontal scale drawing (X)
@_scale_y - vertical scale drawing (Y)

Calculations
@_ko_calc_kind - calculation type chosen in "Calculation parameters" window

<_CALCULATION> - beginning of the line with the following variables
<_ELEMENT> - beginning of the line with the following variables
<_POSITION> - beginning of the line with the following variables
<_MATERIAL> - beginning of the line with the following variables

Summaries
@_ko_[option]w_netto - net worth with mark-ups
@_ko_[option]w_vat - value of the vat tax
@_ko_[option]w_brutto - gr value with mark-ups
@_ko_[option]w_nettor - net worth of the labor (without mark-ups)
@_ko_[option]w_nettom - net worth of the materials (without mark-ups)
@_ko_[option]w_nettos - net worth of the equipment (without mark-ups)

@_ko_[option]w_skpr - value of the indirect costs of labor
@_ko_[option]w_skzm - value of the indirect costs of materials
@_ko_[option]w_skps - value of the indirect costs of equipment
@_ko_[option]w_skp - value of the indirect costs
@_ko_[option]w_szkpr - value of profits from the indirect costs of labor
@_ko_[option]w_szkzm - value of profits from the indirect costs of materials
@_ko_[option]w_szkps - value of profits from the indirect costs of equipment
@_ko_[option]w_szr - value of profits from the labor
@_ko_[option]w_szm - value of profits from the materials
@_ko_[option]w_szs - value of profits from the equipment
@_ko_[option]w_sz - value of profits
@_ko_[option]w_net_narzr - net worth of the labor (with mark-ups)
@_ko_[option]w_net_narzm - net worth of the materials (with mark-ups)
@_ko_[option]w_net_narzs - net worth of the equipment (with mark-ups)

@_ko_[option]w_brutto_narzr - gr worth of the labor (with mark-ups)
@_ko_[option]w_brutto_narzm - gr worth of the materials (with mark-ups)
@_ko_[option]w_brutto_narzs - gr worth of the equipment (with mark-ups)

[option] can take values:
calc_ - values refer to the calculation
element_ - values refer to the calculation elements
position_ - values refer to the calculation positions

e.g.: variable @_ko_element_w_netto will be converted into calculation element worth netto with mark-ups

Mark-ups
@_[option]n_vat - VAT rate
@_[option]n_kpr - value of the indirect costs of labor [%]
@_[option]n_kzm - value of the indirect costs of materials [%]
@_[option]n_kps - value of the indirect costs of equipment [%]
@_[option]n_zr - value of profits from the labor [%]
@_[option]n_zm - value of profits from the materials [%]
@_[option]n_zs - value of profits from the equipment [%]
@_[option]n_zkpr - value of profits from the indirect costs of labor [%]
@_[option]n_zkzm - value of profits from the indirect costs of materials [%]
@_[option]n_zkps - value of profits from the indirect costs of equipment [%]

[option] can take values:
calc_ - values refer to the calculation
element_ - values refer to the calculation elements

@_ko_catalogs - KNR list used in calculation
@_ko_element_table_no - element number
@_ko_element_table_description - elementu name

@_ko_calc_no - calculation number
@_ko_calc_no_all - absolute calculation number
@_ko_calc_n_jobrate - calculation job rate
@_ko_calc_kind - calculation type
@_ko_element_no - element number (numbering starts from the beginning of each calculation)
@_ko_element_no_all - absolute element number (numbered from the beginning of a calculation)
@_ko_element_description - element name
@_ko_element_quantity - multiplicity of the element
@_ko_position_description - name of the calculation position
@_ko_position_catalog_name - KNR name
@_ko_position_catalog - KNR name and number
@_ko_position_specification_link - Refer to the chapter in the technical specifications, which are described in works included in the item
@_ko_position_quantity - The number of calculation units of an item
@_ko_position_unit - position unit as number (e.g. 20)
@_ko_position_unit3 - position unit as three digit number with leading zeros (e.g. 020)
@_ko_position_unit_txt - position unit expressed as text (e.g. pc)
@_ko_position_par1 - additional parameter of calculation position
@_ko_position_w_netto_1 - the net unit value of position

@_ko_position_no - position number (numbering starts from the beginning of each element)
@_ko_position_no_all - absolute position number (numbered from the beginning of a calculation)

RMS specification
@_rms_material_no_[option] - sequence number of component
@_rms_material_number_[option] - component index
@_rms_material_description_[option] - name
@_rms_material_unit_[option] - unit
@_rms_material_quantity_[option] - quantity
@_rms_material_netto_[option] - net price (without mark-ups)
@_rms_material_w_netto_[option] - net worth (without mark-ups)

[option] can take values:
r - for labor specification,
m - for material specification,
s - for equipment specification,

Lines that begin with "/ /" are not included in the resulting document