instalSOFT

Net-san

**Calculation of outdoor water supply and sewerage networks
complete with graphic editing
of network plan and profiles**

Version 4

© Instalsoft

Rev. 2012-01-12

Foreword

THE PRODUCER OF THIS SOFTWARE APPLICATIONS SHALL NOT BE LIABLE FOR LOSS OF PROFITS OR INCOME, LOSS OF DATA, DAMAGE OF EQUIPMENT OR ANY OTHER LOSS ARISING OUT OF THE USE OF THIS SOFTWARE.

The software application presented in this manual is protected by Copyright. The User is therefore obliged not to distribute or duplicate this application.

The software Copyright owner is InstalSoft s.c.. Legal purchasers of the software are entitled to obtain explanations regarding its use.

The authors shall not be liable for the consequences of wrong installation of the software, abuse or calculation results.

Contact with the authors:

InstalSoft s.c.
http://www.instalsoft.com
E-mail: info@instalsoft.com

Contents:

1. INTRODUCTION

1.1. Purpose

Net-san is intended for designers of outdoor sewerage and water supply networks. It enables plotting horizontal plans (schematic drawings of network) and profiles of water supply and sewerage systems, and drawing schematics of nodes in the water supply network. The data entered into the network diagram form basis for hydraulic calculations, which include sizing pipe diameters in a water supply (radial) network or sewerage (gravitational or pressurized) network and determination of the required pressure of the water source. Hydraulic calculations are integrated with an algorithm of determining optimum vertical layout of the network (avoiding interference with other utilities and appurtenances).

It is also possible to plot profiles not associated with the network plan. In this case no calculations are made.

With Net-san you can also plot heating and gas supply networks, although some of the components will have to be drawn manually with the use of graphical components. Calculations for this type of systems cannot be made.

The program is handled in a manner similar to that of other InstalSoft programs based on the graphical editor. The screen layout, the manner of placing components and appending their data, the keyboard shortcuts are nearly identical.

Net-san features extensive capabilities of drawing configuration and of placing additional appurtenances. It runs on PC computers under MS Windows 98, Me, NT, 2000 and XP.

1.2. Scope of calculations

Calculations made by Net-san include sizing pipe diameters in gravitational or pressurized sewerage network and in water supply (radial) network, and automatic determination of the vertical layout of these networks, where defined clearance must be maintained with respect to other utilities and appurtenances. The minimum pressure of the source is determined for a water supply network to ensure required pressures for all terminal units.

Water demand balance calculations are not made for terminal units and the sewage flow rates are not determined for network nodes – these data must be specified by the user for terminal nodes of the network.

A bill of materials is automatically drawn up for a network calculated by the program. This bill of materials includes pipelines and manholes, provided that the latter are selected from a catalogue of a defined manufacturer. Pipeline fittings are included in the bill of materials, if they are selected from a catalogue of a defined manufacturer.

1.3. Arrangement of the Manual

The sections of this Manual are arranged in a manner that enables a beginner to start working with the software right away and to let every user get acquainted with its advanced functions.

Section 2 describes shortly how to work with the software. The information given there is sufficient to start working with the program and to create projects with the use of its basic functions. The arrangement of information follows a typical scheme of creating a project.

Subsequent sections of the Manual describe all functions of the program arranged in a more encyclopaedic way. Information provided there is, to some extent, a repetition of what is included in section 2, but it is much more detailed.

Further parts of the Manual (section 3 and following sections) is also provided on the installation disk of the software in the form of easy-to-browse HTML files and a PDF file, which enables printout of selected sections. Microsoft® Internet Explorer version 4 or higher is recommended to look through the Manual in the HTML format. To view or print the Manual in the PDF format, Adobe® Acrobat® Reader or Adobe® eBook Reader software is required.

1.4. Symbols used

The following symbols and abbreviation are used in the manual:

Paragraphs marked with "♦" indicate lists of actions required for performing a function. For instance:

♦ In order to place a component in the project:
   1. Click the button on the toolbar representing the component.
   2. Move the mouse pointer to the workspace. The mouse pointer takes the shape of [...].

Paragraphs marked with an exclamation mark and printed in bold and italics contain particularly important information. For instance:

! *Double mouse click in the realtime zoom mode or "grabber hand" mode, toggles these modes on and off. This enables convenient and fast viewing of the project.*

The following notation:

» command "File / Save project" (**Ctrl+S**, "Program" →) «

indicates the selection of the command "Save project" from the "File" menu. The combination of **Ctrl** and **S** keys is the shortcut to this command – this means pressing the **CTRL** key, keeping it depressed and pressing the **S** key. This command can be called from the "Program" toolbar by pressing the button.

1.5. Terminology and designations

The following terms are used in this manual, as well as in the program itself:

- **Project**: A network or several networks included in one data file is referred to as a project, which forms one whole and is calculated as a whole. Each project consists of at least one worksheet.

- **Worksheet**: Part of a project that is edited independently of other parts and fills the entire area of the drawing board. Worksheets are edited separately, however they are saved to one project file. There are two types of worksheets: 'Maps' and 'Profiles'.

- **Map**: Worksheet that constitutes a schematic representation (plan) of the network. It may be superimposed over a base map, ensuring thereby good agreement with the actual layout of the network in the terrain.

- **Profile**: Worksheet that represents a longitudinal layout of the network. It can be automatically generated on the basis of a diagram in the 'Map' worksheet or created by the user.

- **Component**: Components are the basic and indivisible parts of the network in the program. Examples of components:
  - node,
  - pipe-run,
  - profile section,
  - appurtenance or fitting component.

- **Pipe-run**: A section of network that carries a defined flux of medium (water, sewage) invariable along the entire length of that section.

- **Node**: The starting or terminal point of a pipe-run. In practice it is a point where pipe-runs connect or it is a source or a terminal unit (demand node).

- **Profile section**: Stretch of profile between one manhole and other (or between one point of network inflection and other). The slope of sewer or water pipe in a single profile section is constant.

- **Node section**: Stretch of water pipe of constant diameter along its entire length, or stretch of water pipe between one fitting component placed on the drawing and other (e.g. valve, expansion joint).

- **Cover depth**: Thickness of soil layer over a sewer or pipeline measured from ground level to the upper surface of sewer or pipeline.

- **Depth**: Thickness of soil layer over a sewer or pipeline measured from ground level to sewer bottom (in sewerage network) or to pipeline axis (in water supply network).

- **Appurtenances and fittings**: An appurtenance or a fitting is a component that cannot stand alone, it can only be a part of another component (e.g. pipe-run). Fittings are placed on sections of the network under design (e.g. gate valves), whereas

appurtenances are components that exist along the route of the network and are placed directly on the profile or map (e.g. other utility lines that run across the network under design).

- **Data**: Each component in the project has data assigned to it, either determined automatically from the drawing (e.g. pipe-run length), adopted as default values or requiring specifying by the user (e.g. required water flow in demand node).

- **Profile** fragment: A series of sections in a profile together with associated fittings and appurtenances.

- **Module**: To facilitate copying of repeatable fragments of project, such fragments may be saved to clipboard or toolbar. The saved project fragment is called a 'module'. Node fragments may also be saved.

- **Group**: A set of components defined by the user. Each group has the following characteristics:
    1. None of its components may not be situated outside group boundaries
    2. When a group is moved, all its components are moved

When a component of a group is moved outside group boundaries, the group range is extended automatically

1.6. File name extensions

The following file name extensions have been adopted:

*.NTS - files with network and profile data,
*.~NS - files with data of previous versions,
*.HLP - graphical catalogues (additional information on products included in catalogues),
*.CHM - help files,
*.INI,
*.SST - files with program configuration,
*.KTR - files with pipe, pipe fitting and manhole catalogues,
*.PRF - files with designs created with Gredi-profile 2.0,
*.ISM - files with base drawings created with "Scan Combine".

2. SHORT DESCRIPTION OF THE STAGES OF CREATING A PROJECT

2.1. Introduction and stages of creating a project

In this section the basic stages of entering data, calculating a network and of generating profiles are presented. The subsequent subsections provide a short description of each of theses stages, whereas the detailed description of the many functions and mechanisms of the program is given in the following sections of the Manual. Further in this Manual you will also find detailed descriptions of all available components and their data.

There are two basic manners of working with the program:
1. The user designs a sewerage or water supply network on a plan (map), calculates the network, and then the program automatically generates profiles for selected routes.
2. The user manually creates a profile (or profiles) using the available components. This mode of work is similar to that applied in previous version of the program (Gredi-profile 2.0). In this case network calculations cannot be made.

Both manners of work are described below and in subsequent sections of the Manual.

2.2. Creating a new project

A new project is created by means of the "File / New project" command or by

clicking on the "New project" button ![button] – first button on the "Program" tab. You can also use the "New project" button in the welcome window, which is displayed after launching the program. The "Project options" window is displayed. By default the program assumes that the network will be drawn on a location plan and the options box is adapted to a 'Map' type worksheet. If you intend to draw profiles only, close the options box, switch to 'Profile' type worksheet and open the options box again (Options / General data). Proper appending of data (in particular 'Reference level' in the 'Profile' type worksheet) enables correct completion of the second stage of project – creation of a profile.

Review the other tabs in the "Project options" window and modify as required the default values that were entered automatically.

Project options can be changed at a later stage of design work. To do this select "Options / General data" (**F7**) or "Options / Info" (**Shift+F7**) – each of these commands displays the project options window in the appropriate tab. Later you can move freely from one tab to another.

2.3. Project involving network plan and profile

♦ To create a project that includes a plan and a profile of the network:
1. Create a new project and specify its general data

2. (option) Paste together fragments of map scanned on a scanner to make a large map – see Appendix C
3. (option) Load the terrain map (drawing) on the "Base" tab of the "Map" worksheet
4. Draw the diagram of the water supply and/or sewerage network and, if necessary, of other systems (not to be calculated by the program) that may interfere with the network under design
5. Append numerical data of the network nodes and pipe-runs
6. Save the created project to a disk file
7. Make the calculations
8. Create a new "Profile" type worksheet or switch to an existing one, define the profile route, specify the reference level and create the profile in this worksheet as defined
9. If necessary, repeat the above steps to create subsequent profile worksheets
10. Print the project as required (network plan, profiles, calculation results tables, bill of materials).

2.3.1. General data

General data of a project include many values, which can be modified during or after editing, but before making calculations. In practice, before proceeding, the following should be specified:

– catalogues used – click on the 'Catalogues" button
– default pipe types for individual types of networks included in the project
  minimum and maximum values of cover depths and minimum and maximum gradients
– meteorological data for determining stormwater flow,
– data for determining required values of water flow at abstraction points and flow of sanitary sewage.

2.3.2. Inserting terrain map as base

Drawing the network plan on a terrain map enables automatic determination of the lengths of all pipe-runs.

♦ To insert a terrain map as base saved in DWG/DXF file format:
1. Select "Import base from DWG/DXF file" in the "File" main menu or select "Import base from DWG/DXF file" in the top toolbar in the "Program" tab.
2. In the window that opens find on the disk and select the file to be imported and click "Open".
3. Find on the disk and select the font file or click "Cancel".
4. Select the unit of measure for the drawing in the import window.
5. Click "Next" in the top part of the window. In the list of layers of the imported file displayed on the left-hand side deselect those that are not required in the base drawing and would make the drawing less clear.
6. Click "Next". The drawing is entered into the "Base" layer of the project. As the structure of the file is not altered during import, it is possible to change in the graphical editor the visibility of the individual layers of the DWG/DXF file in the data table in the box "Displayed layers".

♦   To insert a terrain map as base saved with the "Combine Maps" application or in bmp format:
   1. Select the "Base" tab in the set of tabs indicating project levels in the lower right part of the screen
   2. Select the "Base" tab in the upper set of tabs that enable access to tools and functions, and then the "Map" component
   3. Insert the component, represented by a large square, into the worksheet intended for the network map (by default it should be the first worksheet in a new project)
   4. In the data table of the new component select the file with the scanned map and load that file, then specify scale of the map, if necessary.

♦   To insert a terrain map as base saved in popular bitmap format:
   1. Select the "Base" tab in the set of tabs indicating project levels in the lower right part of the screen
   2. Select the "Graphics" tab in the upper set of tabs that enable access to tools and functions, and then the "Drawing" component
   3. Insert the component, represented by a small square, into the worksheet intended for the network map (by default it should be the first worksheet in a new project)
   4. In the data table of the new component select the file with the scanned map saved in one of the formats: bmp, wmf, jpg, emf, and load that file into the drawings gallery window and specify size of the drawing in that gallery

   To facilitate drawing the network diagram, it may be convenient to set the map display colour to light grey or other colour that will not obscure the diagram.

   You will find more on map editing in section 5.1.1

2.3.3.   Drawing network plan

   To start drawing the diagram (plan) of the network, switch to the "Network" tab. There are no strict rules on the sequence of drawing and placing components.

   If you draw the network diagram on map of terrain, the following sequence is recommended:

   Mark buildings:

   You can place rectangles or polygons in places where buildings and other objects (e.g. storm water drainage basin) are situated and covered by the water supply and/or sewerage network project. To make the drawing more clear you can fill in the polygons with a colour and select the style, colour and thickness of line. This has no effect on calculations.

   Water supply network:

1. Place the terminal nodes of the water supply network at points indicated by the map:

- water supply network receptacles, meter boxes , ,

- external hydrants

- building supply points

- network supply point

2. Plan the water supply network route

3. Insert junctions at planned connection points of pipe-runs

4. Connect all junctions by means of water supply network pipe-runs .

You can find more information on water supply network components in section 5.3.

Sewerage network:

1. Place the terminal nodes of the sewerage network at points indicated by the map:

- sewage drainage from building

- other wastewater sources
- manhole at point of connection to existing network or at inlet to treatment plant

- manhole at point of connection to existing network or at inlet to treatment plant , measuring drain in pressure sewerage at point of connection to existing network or at inlet to treatment plant, or outlet to open reservoir .

-

2. Plan the sewerage network route, wastewater pump stations , and intermediate junctions.
3. Insert junctions at planned connection and direction change points of pipe-runs and at other points where manholes must be used , or junctions .

4. Connect all junctions by means of sewerage network pipe-runs or pressure network pipe-runs .
.

8

You can find more information on sewerage network components in section 5.2.

As the same symbol may be used to indicate both terminal nodes and flow-through junctions, you must specify the function of the node immediately after placing it by selecting the appropriate text (receptor, flow / combine streams, source) in the 'Function' field of the data table.

After placing all components it is a good idea to check all connections (**Shift+F2**). If any unconnected components are found in the network, then a list of messages is displayed indicating missing connections or wrong connection directions.
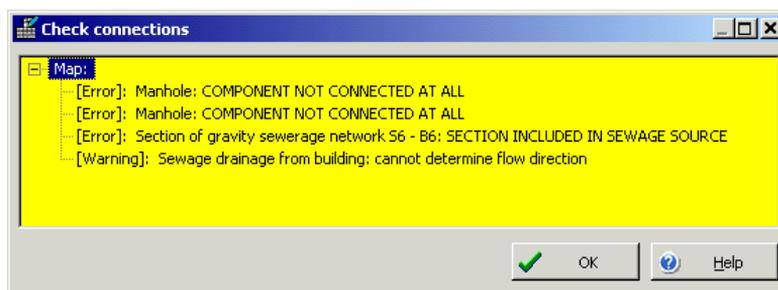
When this list is displayed, you cannot edit the diagram. After closing the error diagnostics list you can look up the same messages in the basic diagnostics list, which you can open and close by pressing the **F8** function key.

Other networks:

In the case of other networks that are not subject of calculations, it is sufficient to draw them only in the vicinity of expected points of interference with the networks to be calculated. The following symbols are used for this purpose:

– heating network:

– gas supply network:

– other networks:

You can find more information on components of other networks in sections 5.2.3 – 5.2.5.

2.3.4. Support for drawing network plan

The "Drawing Assistant" tool helps in precise planning of network. Using this tool enables drawing or inserting components at defined distances from and/or angles to a component in the DWG/DXF base drawing or other network component. These angles are particularly important when connecting network sections to bottom of manhole. Incorrect angle will make proper sizing of manhole invert impossible.

1. To use the Drawing Assistant, select "View" / "Show/hide drawing assistant".
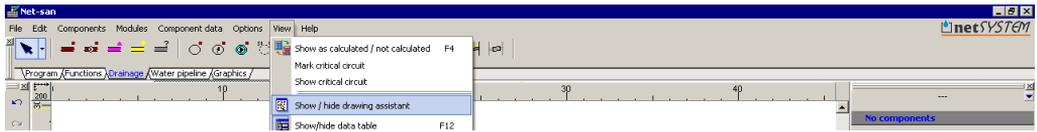
2. The Drawing Assistant window is displayed.
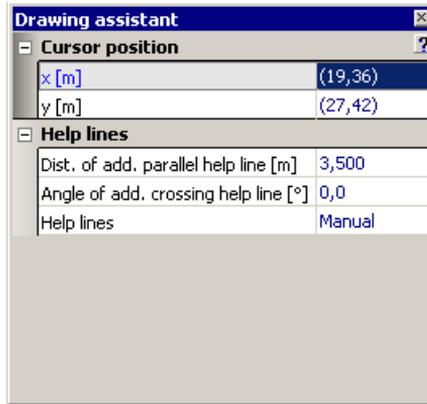
3. When drawing, the Drawing Assistant window displays information on current position of cursor, Δx and Δy increment from starting point of network section, length of current section, and angle of intersection with horizontal axis.
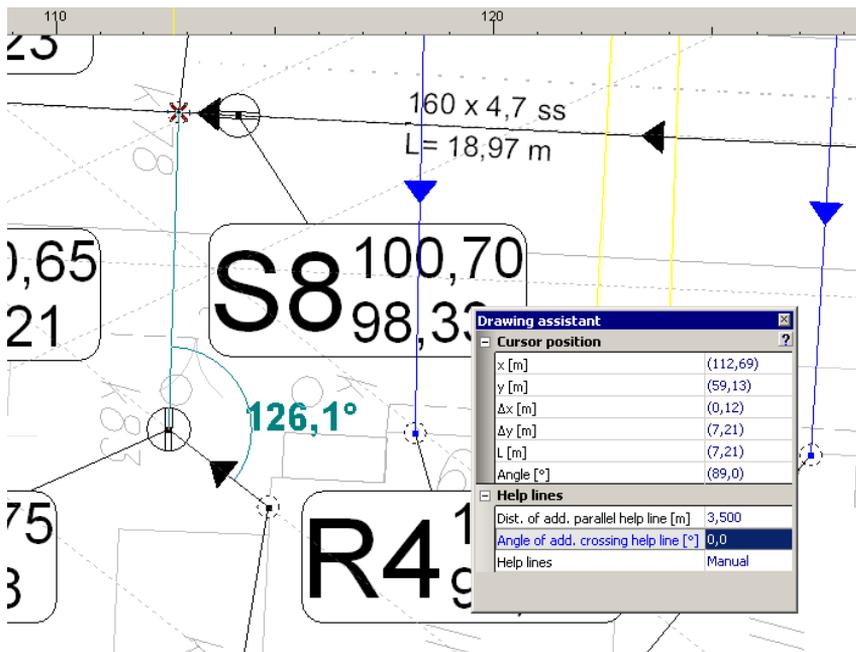
In the "Cursor position" window you can specify the length and connection angle of subsequent section. To do this enter the length of the subsequent network section in the "L [m]" field, and the angle of the subsequent section in relation to preceding section in the field "Angle [°]". Specifying the values in these fields enables precise determination of the parameters of a given network section.

10

4. After specifying the value of the angle and/or distance of additional lines and positioning the cursor on any component of the DWG/DXF base or on a network component, help lines are displayed to facilitate precise drawing.

5. For example, to draw a section at an angle of 126° to the preceding section, enter the value of angle in the field "Angle of add. crossing help line [°]" in the Drawing Assistant window. Broken help lines inclined at the specified angle are displayed during drawing. When you want to draw a section at an angle of e.g. 90° to another section, after entering the value of the angle and after starting to draw, position the cursor over the section to which the new section is to be perpendicular. Help lines, at an angle of 90° exact, are displayed.
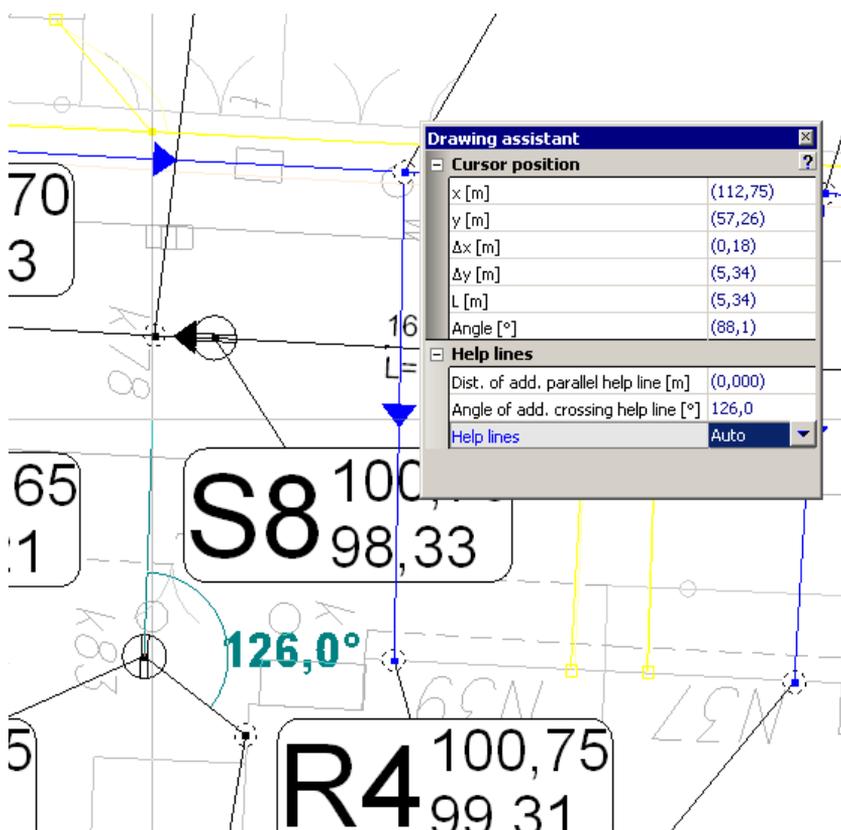
The Drawing Assistant also facilitates drawing a network parallel to another, existing network. To do this:

1. Enter the distance between the networks in the field "Dist. of add. parallel help line [m]".
2. Select the type of network to be drawn.
3. Place the cursor over the existing network (without clicking). Help lines are displayed at the specified distance.
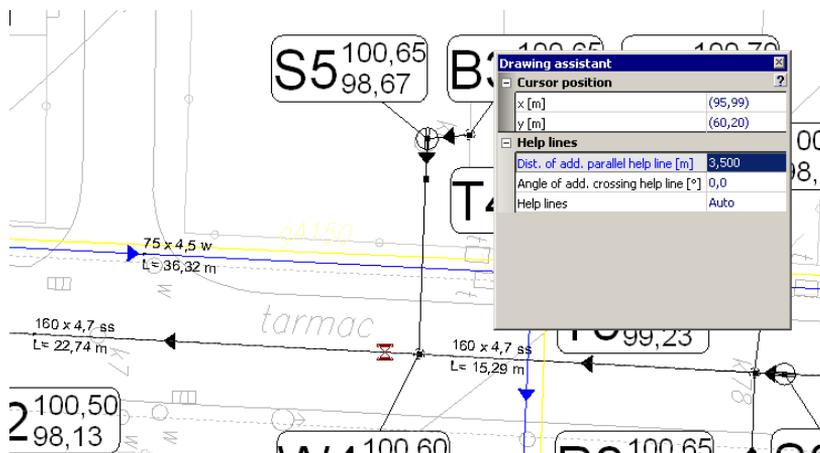
2.3.5. Appending component data

When the diagram of the network under design is ready and correct, you should then enter data of the network components into the data table. To open (and close) the data table press the **F12** function key. The data table displayed applies to the component that is marked currently – to mark a component position the mouse pointer over it and press the left mouse button. It is possible (and useful) to have a number of components of the same type marked and to enter the same data applied to all those components at once. To mark a number of components click on each of them while holding the Shift key depressed. It is also possible to mark all components of specified type within an area or in the entire project – these functions will be described further in this manual.

! ***The most important data that need to be specified for individual network component types include:***

- for terminal nodes of water supply network: designation (symbol identification), water outflow at point (water demand), required pressure, ground elevation;
- for terminal nodes of sewerage network: designation (symbol identification), inflow at point (sewage flow), ground elevation;
- for intermediate nodes of water supply network: designation, ground elevation;
- for intermediate nodes of sewerage network: designation, ground elevation, inflow at point (direct wastewater inflow from surrounding area into manhole is permitted);
- in the case of pressure network, select pumping station from catalogue or define pump specifications if pumping station is not from catalogue,
- in gravity network, specify manhole type (if not specified, default values are adopted from calculation options),
- for water supply network sections: pipe type (if not specified, the default type from general data is adopted);
- for sewerage network sections: pipe type (if not specified, the default type from general data is adopted), minimum and maximum gradient, maximum fill ratio.

Many fields in the data table are filled in with default values, which might, or might not be proper for the given project. Therefore the values displayed in the table should always be checked.

2.3.6. Calculations

To invoke calculations click on the calculator button [image] on the upper "Program" tab (**F10**). Calculations are then checked and data diagnostics are run. Diagnostic messages are displayed in a separate box.
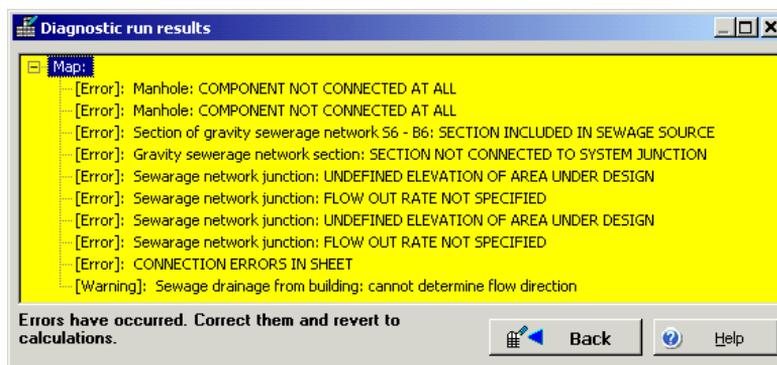
If errors are found, calculations will not be possible – you have to go back to editing and correct the errors. If no errors are found, a "Next" button appears (even if warnings or hints were displayed). Upon clicking on this button, a set of tabs is displayed for presenting calculation options.

The first tab presents minimum spacing (in vertical plane) to be maintained between the calculated networks and between them and other networks during automatic determination of network route.

The second tab presents restrictions applied when sizing sewerage network line diameters: minimum diameters and velocity limits for pipe-runs.

The third tab presents restrictions applied when sizing water supply network line diameters: minimum diameters and velocity limits for pipe-runs.

Moving to the last tab (by clicking on it or on the "Next" button) starts the calculations proper. Upon completing calculations, sub-tabs appear with tables of results. The tables show parameters of network sections and nodes. They include sized diameters of network sections, types of manholes, hydraulic parameters affecting the sizing of diameters and pressures in the nodes of water supply network. A separate sub-tab displays the bill of materials.

You can find more information on data diagnostics and calculations in section 6.

2.3.7. Creating a network profile

Upon successful completion of calculations it is possible to automatically create a profile of each of the calculated networks. To do this go back to the graphical editor, open a new "Profile" type worksheet ("File / Worksheets...") and invoke editing. Now press the **F7** key and select the "Profile definitions" tab. In the upper part of this tab there is a table where you can define one or more profiles by selecting the sheet from which data is to be taken (this must be a "Map" type sheet), and by indicating the profile route – from start node, through intermediate node to terminal node. Map of the

network around the indicated nodes is displayed in the lower part of the tab. Upon defining the first profile, you can define subsequent profiles, or generate the profile drawing by clicking on the "Create profile as defined" button [icon] in the upper "Program" tab.

After pressing the **F7** key again, you can select the "General data on profile" tab in the project options, where you can change the reference level, profile scale, manhole label description, etc.

You can find more information on components making up sewerage and water supply profiles 5.4 and 5.5.

2.4. Designing a network profile only

♦   To make a profile drawing only:
1. Create a new project and delete the "Map" sheet within it, and create a new "Profile" type sheet
2. Construct the profile or node diagram using the components available in the program and the network route drawn on the site plan
3. Save the created profile to a disk file
4. Print out the project.

2.4.1. General data

The most important general data of a profile project include the reference level and horizontal and vertical scales.

2.4.2. Creating the profile

To start creating a profile, select the "P – Sewerage" or "P – Water supply" tab in the toolbar, depending on the type of profile you want to create.

Creating a profile consists of the following stages:
1. Placing the start of the profile (table with description of profile table fields together with the first manhole / starting point of network)
2. Placing the other (middle) profile sections and, if necessary, drawing house drains or connection lines with the use of the so-called contracted sewerage profile starts
3. Inserting other network components: roadways, existing pipelines, knickpoints, beddings, backfills, and fittings (in a water supply profile).

2.4.3. Placing the start of a profile

To start creating a profile click on the first button, which represents profile start in the toolbar on the "Sewerage" or "Water supply" tab ([icon] or [icon]). The component insert mode is enabled. The symbol of the component being placed appears at the mouse pointer, and the outline of the component is displayed in the workspace:

14
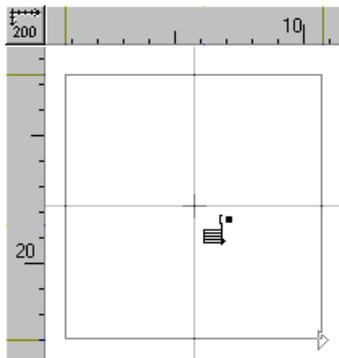
To place the selected component click once anywhere within the drawing. The program switches back to the select mode.

The crosshairs cursor, with a vertical and a horizontal line, facilitates accurate positioning of the cursor in relation to the reference system (vertical and horizontal ruler) and to other components in the drawing. The vertical and horizontal rulers are not the only indicator for placing components. We recommend placing components in such manner that the bottom of the drawing table coincides with level 0 on the vertical ruler.

Upon placing the start of the profile, data of the component placed appear in the data table. They should be appended before placing subsequent components. If the data table does not show up, click on the component placed and then press the **F12** key.

All fields are described in detail in section 5.

By default the cursor is positioned in the first field of the data table: "Designation". Enter the symbol (designation) of the component in the network.

! ***If the distance between manholes is small and the designations obstruct each other, one of them can be moved down or up. To move a designation down enter > before the symbol (e.g. >S0), to move it up enter < before the symbol (e.g. <S0). To delete a designation, enter ? in the field of the data table.***

The second field of the table is provided for specifying the start of the current fragment of profile.

In the next field ("At section end") you can select a component to be placed at the end of the profile section. To display available options, click on the field. A button with an arrow is displayed to the right of the field. Click on the button to display the list of options. You can pick your selection from this list (inlet to building, crossing, manhole, pumping station, outlet to open reservoir, etc. – depending on the profile type) by clicking on the appropriate component name. Depending on your choice, further table fields are displayed, which require filling in.

15

Elevation of area under design and elevation of existing area, appended in the following two fields of the data table, should be taken from the site map. The program enables transferring the elevation of the existing area into the area under design and vice versa. To transfer the elevation of the existing area into the area under design, enter "i" or "*" in the "Area des. el." field. To transfer the elevation of the area under design into the existing area, enter "p" or "*" in the "Area exist. el." field.

The following field for specifying the elevation of duct bottom or of pipeline axis is named "Duct bottom end" or "Pipeline axis end", respectively. "End" indicates the elevation of the end – right-hand side of the profile section just placed. These elevations ensue from the design and they must be specified. Instead of entering the elevation, you can specify the gradient (percentage) or burial depth in relation to area under design or to existing area (see the yellow hint bubble in the field of the data table.

Information entered in the "Additional caption" field appears on the leader line of the component being placed. You can enter the information directly into the field of the table or make use of an additional box that is displayed when you click on the button with an arrow. Take notice of the "Write above line only" check box, as by default part of the caption (depending on its length) is placed above the line, and the remaining part under the line. When the caption is too long, to break it into two lines press **Enter** where desired.

Take notice of the last field in the table ("Cascade") of a sewerage profile. When the distance between the elevation of the duct bottom inlet and the elevation of the manhole bottom is more than 0.5 m, then a cascade is automatically drawn. That is why "Auto" appears by default in the "Cascade" field. To eliminate the cascade click on the button with an arrow to the right of the field and select "No".

2.4.4. Placing the middle sections of a profile

To start creating middle sections of the profile click on the third button in the toolbar on the "Sewerage" or "Water supply" tab (or) and place the components in the same manner as the start of sewerage profile. If the AUTO mode is enabled, the section will be connected automatically when the mouse pointer is positioned near the connection point – the connection point is marked with a cross. Accurate adjusting of the position of the component is not required.
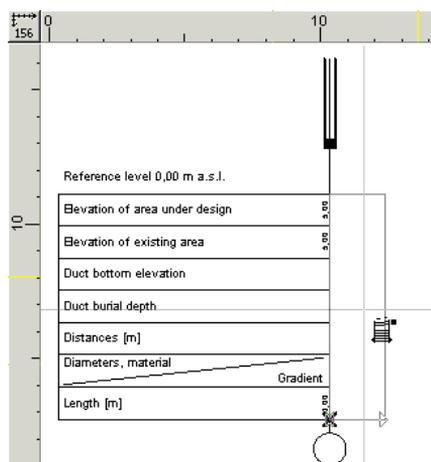
Subsequent sections of the profiles are connected together. Connection of components is marked with a purple square at the point of contact of the two components. Components that are not connected are marked with green triangles:

Connected sections of a profile	Sections of a profile not connected

Upon placing the middle component, the data table needs to be filled in. Most of the fields of the data table is the same as for profile start. Only the new fields are described below.

The fields "Duct diameter" or "Pipeline diameter", "Diameter designation", "Material" ensue from the design and do not require clarification.

After placing the component, "Auto" appears by default in the "Duct bottom start" / "Pipeline axis start" field. This means that the data of the start of duct bottom / pipeline axis of the middle component are equal to those of the end of duct bottom / pipeline axis of the preceding component. You can leave the "auto" setting or enter the data manually.

Placing a contracted profile start enables drawing of house drain profiles that belong to the sewerage network (in the case of sewerage profile) and water supply connection lines in the case of water pipeline profile. This component does not include any verbal description of the start component.

When creating a project, the profile display may be changed: you can move it, zoom it in or out. This is done by means of buttons in the "Functions" toolbar on the left side of the screen.

2.5. Placing fittings and appurtenances

At any moment of the project creation process you can insert other supplementary components:

SEWERAGE PROFILE :

 – "Perpendicular pipe" – existing or planned

 – "Double pipe" - existing or planned, e.g. heating network

 – "Knickpoint"

 – "Roadway"

 – "Bedding"

 – "Backfill" – e.g. thermal insulation layer

 – "Protection tube"

WATER PIPELINE PROFILE:

The same components as for sewerage profile are available, and in addition:

 – "Gate valve".

♦   To insert one of the components above:
1. Mark the profile section where the component is to be placed by clicking on that section
2. Click on the button representing the required component
3. Insert the component by positioning the mouse pointer at the proper point of the profile section and clicking on it.

Additional fields describing the data of the inserted component appear in the table of the marked profile section. These fields should be filled in. The fields have buttons with a "+" or "–" symbol. By clicking on these buttons you can expand or collapse these fields. After expanding the field of a component additional fields appear with data of this component. These fields should be filled in – in particular the elevation and location of the inserted component may be altered.

2.6. Creating junctions drawings on water pipeline profile

The program enables drawing diagrams of junctions (nodes) on water supply network together with the specifications of individual components.

♦ To start drawing junctions, switch to the "Junctions" tab on the toolbar.

Every type of junction component, e.g. line (pipe), reducing adapter, gate valve, hydrant, is represented by a button on the "Junctions" tab. The type of component represented by each button is displayed in a yellow hint bubble, when the mouse pointer is positioned over the component. To insert a component, click on the appropriate button. The component insert mode is then enabled.

When designing a junction, first draw the layout of pipes, e.g. straight pipe, tee. To do this click on the "Junction section" button and then click within the drawing area to mark the starting point of the junction. Click once more to mark the end point. The standard select mode is enabled again. To make a branch, e.g. a tee, click on the "Junction section" button again or use the F3 key – "Insert last item".

♦ To insert a selected component, e.g. a gate valve, on the lines drawn:
1. Click on the button representing the selected component
2. Position the selected component on the drawn lines of the junction at a selected point and click the mouse button
3. Append the data of the component in the data table.

The data of the inserted components can be changed in the data table at any time. To display the data of a component in the data table, mark the component by positioning the mouse pointer and clicking on the base of the component (e.g. crossing of junction section line with valve – the valve is marked). Now the data in the table may be changed.

2.7. Inserting component labels

The next step is the placing of labels with data describing the diameters of pipes and specifications of individual components.

♦ To place the diameter of a given pipe in the drawing:
1. Click on the given line section (the component is highlighted and the data of the component appear in the data table)

2. Click on the appropriate button representing the diameter
3. Click on the line (pipe) to insert the label.

The diameter selected in the table is now displayed in the drawing. In addition to the diameter, the material of the pipe can also be displayed.
The junction diagram may also include captions or numbers consistent with the specifications table or with the bill of materials.

♦ To insert a component label:
1. Click on the selected component within the junction diagram
2. Click on the appropriate button representing the reference mark – component label
3. Position the mouse pointer on the component and press the mouse button to mark the starting point of the reference line

4. Move the mouse pointer to the point where you want to place the end of the reference line and press the left mouse button again
5. Enter the name or number of the component in the data table of the component label. As you fill in the table, the caption is automatically displayed on the drawing.

You can select the font and the thickness of the reference line. To change font select the "Font" field. A button with three dots appears to the right of the field. Click on this button to display a table with font types and styles and make your selection.

To indicate the orientation of the reference text select the appropriate option from the list (horizontal, vertical upwards, vertical downwards). To select the reference line type click on the button with three dots to the right of the "Line type" field. A table with line types is displayed where you can select the appropriate style, colour and thickness of line.

2.8. Saving the project to a disk file

Each project should be saved to disk to enable its opening and modification or printing in future. The project is saved to one file on the disk. When the project is saved for the first time, you'll be prompted to enter a name for the file.

File names should be clear - file name length may be up to 255 characters and it may include national alphabet letters and spaces. An example of file name: "Sewerage, Oxford St., network ver. 1".

♦ To save a project to a disk:

1. Select "File / Save project" (**Ctrl+S**,)
2. You are prompted for the file name
3. Click OK.

2.9. Printing the project

To print the created profile select the "Print" tab in the lower right corner of the screen. Fields are displayed in the data table that enable setting the printing options.

Auxiliary lines indicating sheet edges – paper cutting lines are displayed on the drawing.

These lines may be adjusted by dragging them with the mouse.

To change printer settings (e.g. print quality, paper size) select the "Settings" compound field in the table. The button with dots on the right side of the field opens a window for specifying printer settings, which depend on the printer. After entering appropriate data and accepting the print preview, you can proceed with the printing.

You can specify other printing parameters in the data table: margins, overlaps, etc.

To start printing select "File / Print" (**Ctrl+P**,). You can specify the range of pages to be printed (page numbers are shown in the print preview) and number of copies. Printing is executed when you click OK.

You can find more information on print options and printer settings in section 7.

3. PROJECTS AND FILES

3.1. The concept and structure of project file

The basic structure on which Net-san operates is the project, saved to a single file on disk. Project files have an ".nts" extension. The project file may in practice cover a network of any size and is usually divided into sheets, which include a map (network plan) and one or more profiles. The sheets facilitate management of individual parts of the project. A single sheet cannot include both a map and a profile. The project file may include only one sheet with the map of the network (always loaded by the user as data for making further calculations and for the profile) and optionally several profile type sheets. In addition to the network to be calculated (water supply or sewerage), the project may also include data on other networks that already exist and that may interfere with the planned network.

Operations on sheets, such as creating a new sheet, renaming, deleting are executed by selecting "File / Worksheets ...".

3.2. Create a new project

To create a new project launch the graphical editor and click on the "New project" button in the welcome window or, if the program has already been launched, select the

"File / New project" command or click on the first button in the "Program" tab (first tab in the upper left part of the screen).

Welcome window:
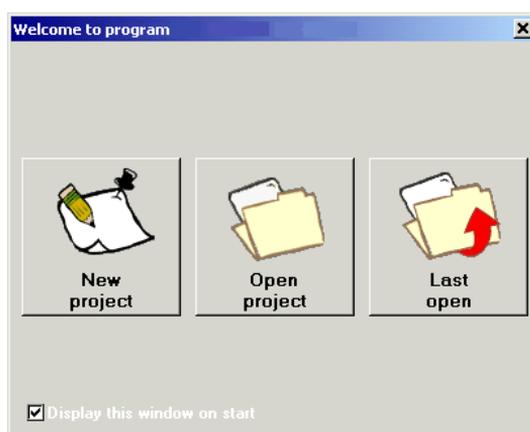
3.3. Project options and general data

After selecting the "New project" command a new file is created (initially without a name) and a project configuration window is opened. This window has several tabs

shown in its upper part: "Info", "General data". etc. By default the second tab is activated.

If you would like to amend project options at a later stage you should select "Options / Project options / General data" (**F7**) or "Options / Project options / Project Info" (**Shift + F7**) – the project options window is displayed with the appropriate tab activated. You can then switch between the various tabs, e.g. "Edit".

! ***The data shown on each tab apply to the current sheet only – you can therefore specify different data for each sheet of the project. The set of tabs displayed also depends on the sheet type.***

3.3.1. Info tab

The "info" tab has in its upper part information fields (read-only) concerning the project file, such as file name and version, number of worksheets, date of creation and modification, etc. In the lower part of the tab there are three sub-tabs for entering data (description, street, town, telephone, etc.) concerning the edited project as well as the project developer and designer.

3.3.2. Map data tab

The "Map" tab is displayed only for a "Map" type sheet. It includes a.o. the following data:

- "Default pipe type ..." – you can specify default types of pipes for individual types of network sections (you can specify other pipe types for subsequent sections in the data table)
- "Catalogues" – a "Catalogues" window is displayed, where you can associate several or all available catalogues with the project. If no catalogue is associated with the project file, it will not be possible to select default type of pipes
- "Component appearance" – a "Component appearance" window is displayed, where you can configure graphically individual components present in the network.

3.3.3. General data on profile tab

The "General data on profile" tab is displayed only for "Profile" type sheets. It includes the following data:
- Reference level for current sheet
- Profile table field height
- Horizontal scale of profile
- Vertical scale of profile
- Field for changing the elevations of profile sections by means of the mouse. When this box is checked, ⬆ symbols are displayed which enable dragging the elevation up or down by means of the mouse
- Field for indicating manner of specifying the location of appurtenances in the data table (from profile start or from profile section start)
- Field for specifying the manner of automatic description of manholes on profiles created according to definition

–   Default material name, to be assigned to all new profile sections – only in open version of the program and only when the profile is created directly (not on the basis of network diagram)

–   "Component appearance" – a "Component appearance" window is displayed, where you can configure graphically individual components present in the profile.

When you position the mouse pointer over a field, after a while a hint bubble is displayed describing the field.

3.3.4. Profile definitions tab

The "Profile definitions" is displayed only for "Profile" type sheets. It is used to define routes which form basis for drawing profiles. This applies to profiles that are drawn automatically.

A table of routes is displayed in the upper part of window. In the lower part of window you can see the "Map" type sheet from which the route is selected.

The columns of the table have the following use:

–   "From dwg. sh." – specify the sheet with the network plan (components of junction and network section type) used to create profile

–   Specify profile type

–   "Start des." – designation of the starting node of the route. A button with arrow on the right side of the field enables selection from list

–   "Int. des." – designation of the intermediate node. Used only when the route is ambiguous

–   "End des." – designation of the end node of the route

–   "Start type" – type of profile start. "Normal" indicates that a component with a table describing individual fields of the profile will be inserted, "Compacted" indicates that a simplified profile start will be inserted.

In the lower part you can specify the spacing (in centimetres) between individual routes in the drawing.

When the profile definition is changed, the profile is not redrawn automatically. To

redraw the profile, select "File / Create profile as defined" (**Shift+F11**,).

3.3.5. Edit tab

The "Edit" tab contains current data on display of sheet and edit parameters:

–   "Decimal place accuracy" – indicate the accuracy of displaying elevations, distances and gradients (number of decimal places)

–   Field for changing the elevations of profile sections by means of the mouse. When this box is checked, symbols are displayed which enable dragging the elevation up or down by means of the mouse.

–   "Express gradient in" – select a unit for specifying gradients

–   "Horizontal and vertical grid" and "Snap to grid" - specify parameters of grid to which the components will be aligned. Each component that is inserted or moved is snapped to the grid

- "Connecting distance with AUTO disabled" – distance at which network sections or profile sections will be connected to each other when the AUTO mode is not on
- "Find connection in AUTO mode" – distance at which a connection will be looked for in the AUTO mode for basic components and modules
- "Additional angle in ORTO" – the ORTO mode supports, in addition to horizontal and vertical orientations, an additional angle, which may be specified here. Its value is stated in degrees as an inclination from horizontal plane
- "Workspace size" – indicate the point of origin and the size of the drawing board
- "Always show page layout" – toggle on / off display of page size of the printout and paper cutting lines. By default the page layout is shown only in the "Print" layer
- "Duct elevation shown as" – the designer may choose whether the "Duct bottom elevation" is defined as the elevation of the inner or of the outer wall of pipe.
- "Start of default designations of sewerage system junctions" – there is a function for automatic designation of network junctions according to a defined formula (e.g. S1, S2, …), which saves time spent on entering data. Junctions are numbered starting from sewage receptor.
- "Start of default designations of water pipeline junctions" – there is a function for automatic designation of network junctions according to a defined formula (e.g. W1, W2, …), which saves time spent on entering data. Junctions are numbered starting from water supply source.
- "Automatic junction designation" – program automatically assigns designations to junctions according to a set pattern.

3.4. Catalogue selection

Catalogues for Net-san constitute files prepared by the software producer with data on the components which make up the installation. The user cannot create his or her own catalogues, neither can he or she modify existing catalogues. Moreover, various versions of the software (e.g. complete version and version dedicated to pipe manufacturer) have different levels of authority to read individual catalogues.

Each project has a defined set of catalogues, which are used during system editing and calculations. Of course it is possible to use all available catalogues in a project. Then, however, you will sometimes have to go through long lists of available types. Calculations cannot be done without at least declaring a default pipe catalogue. Usually every project also requires a catalogue of miscellaneous fittings (without specifying manufacturer) with generally available components.

♦ To be able to specify catalogues to be used in the project select "Options / Catalogues" ( ) or click on the "Catalogues" button in the general data window.

A window with two lists is displayed: the list on the left side contains all available catalogues, whereas the list on the right shows catalogues selected for the project. Tabs at the top of the window are used to switch between different catalogues used in the project.

Catalogues can be moved from one list to the other. For this purpose arrow buttons located between the lists are used. The single arrow button moves marked catalogues only, while the double arrow button moves all catalogues.

A 📖 symbol can appear in both lists beside a catalogue name. It indicates that the catalogue includes graphical information. This information can be displayed by pressing **F1** and **Shift + F1** keys when selecting a catalogue or a component from that catalogue.

The list of catalogues used in the project can also include the following symbols: "Z", "U!" and "U". They have the following meanings:

- **Z** – catalogue always loaded from disc (read into every project),
- **U!** – catalogue already used in current sheet (at least in one component),
- **U** - catalogue used in other sheet of the project.

! Catalogues marked with "U!" or "U" cannot be removed from the list.

3.5. Saving a project to disk and loading a project from disk

Each project is saved to a disk in one file with a ".nts" extension. A new project must be always saved with a name specified by the user. To do this select "Save project" or "Save as" – in case of a new project both commands act in the same way – you'll be prompted to enter a name for the project (file). You can use long file names – it is advisable to select names that help identify the contents of the files. The file name length may be up to 255 characters. If you move to calculations in the new project for the first time (before having saved the project to the disk), you'll be prompted for the project name. Then data diagnostics will be performed and the calculation options tab will be activated.

The following project save and load operations are possible:

♦ To save a project to disk:

Select "File / Save project" (**Ctrl+S**, 💾)
If the file has no name yet, then the above command is equivalent to "Save as ..." – see below.

♦ To save a project under another name (the file will be copied under the new name):

1. Select "File / Save as ..."
2. If necessary change the default folder where you want to save the file. The new folder should be indicated in the "Save in:" field in the upper part of the window
3. Type in the new name of the file in the "File name" field
4. Click the "Save" button.

♦ To load a project from disk:

1. Select "File / Open project" (📂).If the project that is currently open has been changed, a dialog box will be displayed asking if you want to save the changes.

27

2. Select file from the list. If the file you are looking for does not show up on the list, check whether the proper folder in the "Search in:" field has been selected. To facilitate identification of projects, fields are displayed on the right hand side with information on the project indicated in the list. The upper field shows information on the project and designer (specified in General data / Info), and the lower field lists sections contained in the project.
3. Click the "Open" button.

The last items in the "File /" menu are the names of projects that were edited recently. After selecting one of these items, the indicated file is opened. File names may be displayed with or without full access path (see section 8.3.1).

Files are automatically saved in the background at specified time intervals. You can specify that time interval in the program settings (see section 8.3.1). It is not the original opened file that is saved, these are files the name of which begins with "Autosave file ...". These files are deleted when the original file is saved.

3.6. Worksheets in file

Projects are divided into worksheets. Each worksheet is formed by a separate drawing sheet, represented by a separate tab in the lower left part of the screen. To go to a selected worksheet click on the corresponding tab. At any given moment only one sheet is displayed on the screen. Worksheets facilitate managing the entire project.

You can make operations on worksheets, such as renaming, copying or other, using the command "File / Worksheets..." ( ),which opens a window for handling worksheets in the project:
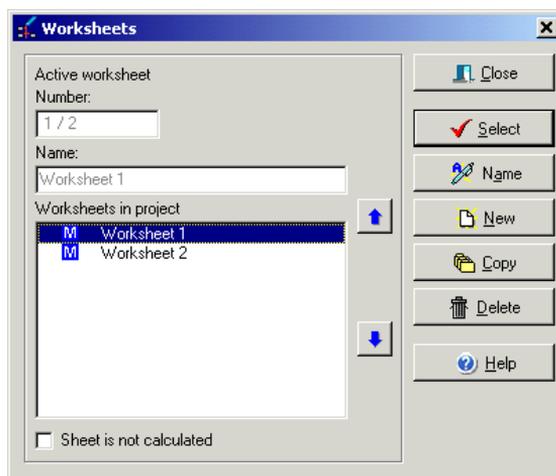
The central part of the window displays a list of worksheets in the project. All information displayed above and operations carried out apply to the worksheet marked on the list.

Information on the indicated worksheet is displayed in the upper part. The number of the current worksheet and total number of worksheets in the file is displayed in the "Number" field. Field "Worksheet is not calculated" is used to indicate that the marked worksheet will not be calculated and will not be accounted for in the bill of materials. This way you can append to the project drawings or diagrams (made using the program or imported) or keep in one file several variants of the project, of which only the current one is calculated.

Buttons on the right side of the window are used to perform various operations on the current worksheet: select it for editing, renaming, copying or deleting. You can also create a new worksheet of specified type. To close the window, click the "Close" button.

♦   To add a new worksheet:
1. Open the window for handling worksheets ("File / Worksheets...").
2. Click the "New" button.
3. Select drawing sheet type
4. If the project has not been saved before, you will be prompted to save the project to the disk.
5. The new worksheet is added and it becomes the current one.

♦   To copy a worksheet:
1. Open the window for handling worksheets ("File / Worksheets...").
2. Click the worksheet you want to copy.
3. Click the "Copy" button.
4. The worksheet is copied.

♦   To rename a worksheet:
1. Open the window for handling worksheets ("File / Worksheets...").
2. Click the worksheet you want to rename.
3. Click the "Name" button.
4. Type in the new name of the worksheet.

♦ To delete a worksheet:
   1. Open the window for handling worksheets ("File / worksheets...").
   2. Click the worksheet you want to delete.
   3. Click the "Delete" button.
   4. Confirm the action.

! *YOU CANNOT UNDO WORKSHEET DELETION.*

♦ To change worksheet order:
   1. Select worksheet.
   2. Move the selected worksheet up or down using the arrow buttons.
      The order of worksheets is altered and saved.

4. GRAPHICAL EDITOR FUNDAMENTALS

4.1. Introduction

This section provides basic information on the graphical editor. In the following subsections you'll find descriptions of the components of screen, operating modes, rules of editing drawing components and other. The arrangement of the subsections is of a rather encyclopedic nature.

The following sections, which pertain to base and installation editing, describe the use of the editor in editing of the individual project components.

4.2. Screen components

An example of a screen is shown below:
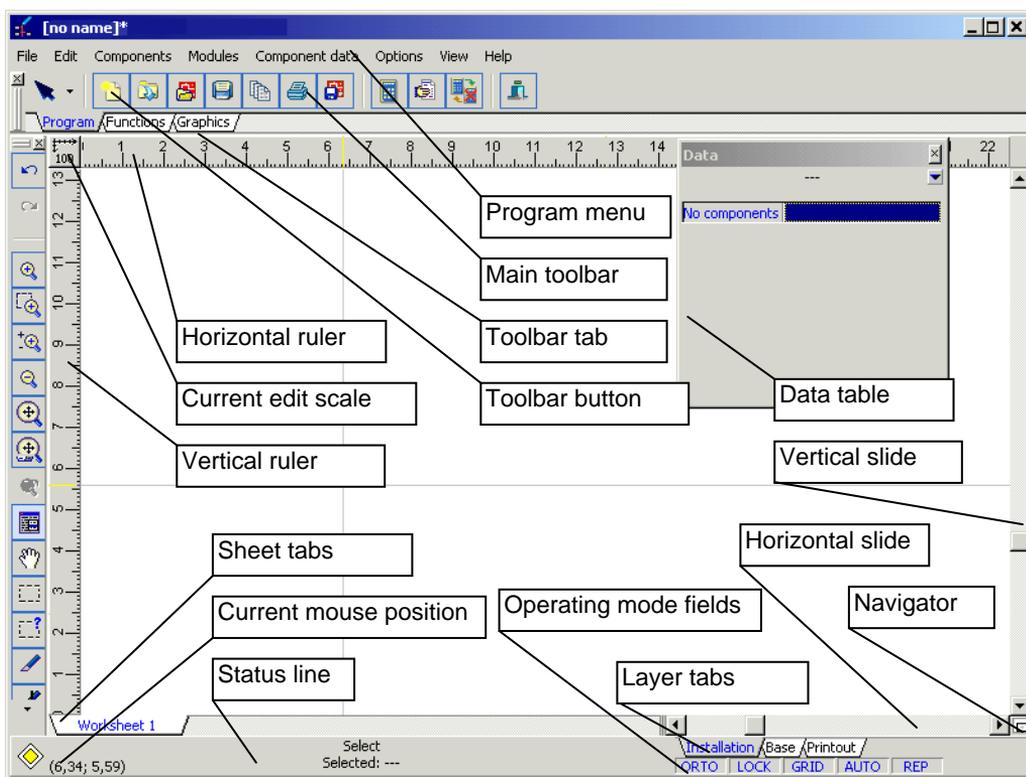
The entire screen is occupied by the main window of the program. Auxiliary windows may be pasted into the main window and they can function as its part (like the data table shown in the figure above) or they can form separate windows displayed against the background of the main window. The auxiliary windows comprise among other the error list, data sets list and the toolbars. You can toggle the windows on and off, just like the additional toolbars (the "Functions" toolbar on the left side of the screen in the example shown above). Screen configuration is described in section 8.3.2.

The main components of the screen (main window) include:

1. Title:

On the left side of the title bar of the main window is the name of the program and name of the current file that is being edited. An "*" (asterisk) beside the file name indicates that the file has been changed since the last save operation (in the figure above). On the right side of the title bar are the standard system buttons.

2. Program menu:

Under the title bar is the program menu. When you click one of the menu items, the commands of that menu are displayed. If you find in this manual a passage like » "File / Save as..." «, it indicates that the "File" menu should be selected on the menu bar shown above and then the "Save as..." command should be selected from the drop-down menu.

3. Main toolbar and toolbar tabs:

The main toolbar has several tabs. When you click a selected tab the contents of the toolbar changes and new buttons show up. Clicking a button calls the function assigned to that button or enables the insertion mode of the given component. If you position the mouse pointer on a button (without clicking), the name of the function or component assigned to that button is displayed in a hint bubble.

The button on the left side of the main toolbar is used to switch to the select mode (when other operating mode was enabled, component insertion mode for instance). This button is displayed no matter which toolbar tab is selected.

Beside the button is a pull-down menu, which enables moving between worksheet layers and switching on and off operating modes.

4. Workspace with horizontal and vertical rulers:
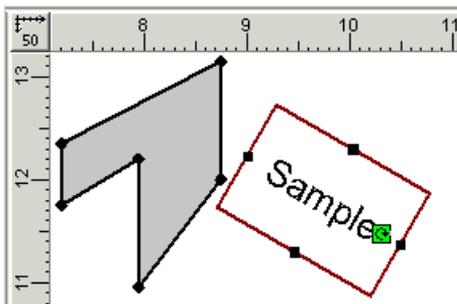
A fragment of workspace with a fragment of a drawing plan is shown above. The workspace is bounded from above and left side by rulers: horizontal and vertical. The rulers let you control position of components you place in the drawing. The element that indicates the position of the mouse pointer in relation to the reference scale (rulers) is a vertical and a horizontal line that forms a kind of a target positioner. The button in the upper left corner between the rulers is used to change the viewing scale and displays the current scale (1:50 in the figure above).

5. Vertical and horizontal slider:
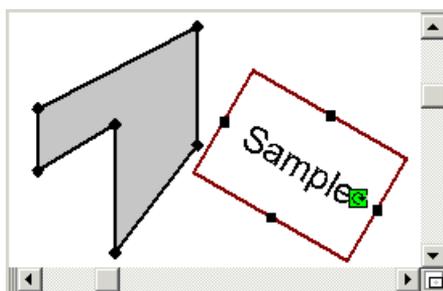
On the borders of the workspace opposite the rulers there are sliders, vertical and horizontal. The sliders are used to move the area of the drawing displayed on the screen. There are two ways of doing that. You can either drag the slider along the scrollbar which causes the drawing to scroll on the screen, or you can click the arrows on the scrollbars to move the displayed part of the drawing in small steps or you can click elsewhere on the scrollbar to move the drawing in larger steps. To move quickly within a large project, you can also use the navigator in the lower right-hand part of the screen (see section 4.4).

6. Project worksheet tabs:

In the lower left part of workspace, next to the horizontal ruler, there are tabs that represent worksheets in the project. If not all worksheet tabs fit into the screen, buttons with arrows are displayed, which enable scrolling the tabs. To enable editing a worksheet, click the relevant tab. The activated tab has a white background. Clicking the right mouse button on a tab opens the worksheets handling window – see section 3.6).

7. Status bar:
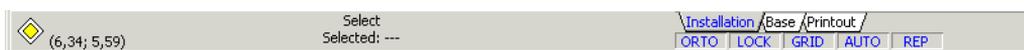
The status bar is at the bottom of the screen. The status bar displays current information about the status of the program (status icon on the left side) and position of the mouse pointer on the drawing area (numbers in brackets). Text messages are displayed in the middle with information on current operations.

Project layer tabs or layer check box and operating mode fields are on the right side of the status bar.

Project layer tabs or layer check box enable switching between different layers of the project (see section 4.5):

In the program settings (see section) you can switch between the tabs and the check box.

The operating mode fields are used to enable / disable operating modes (see section 4.6). Blue colour indicates an enabled mode, grey indicates a disabled mode. To toggle a mode on / off click the appropriate field (in the figure below the ORTO and GRID modes are enabled):

To facilitate switching between the different modes, convenient key combinations are assigned (see section 4.6).

4.3. Moving around the project – viewing and scale

A project can be created and edited in any scale. With large projects, the scale required to display the entire project on the screen will not allow precise editing or component selection. For that reason the program is equipped with many advanced functions which enable zooming in and out and panning fragments of the installation.

The current scale of the installation is shown in the upper left corner of the drawing area, between the vertical and horizontal rulers: . Upon clicking this button, a menu appears where you can select the editing scale:
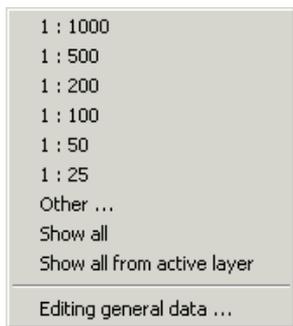
Selecting the viewing scale from the list is not very convenient. It is easier to use the following buttons in the "Functions" tab to zoom in or zoom out:

⊕ – Zoom in (magnify) in steps (**Ctrl+"+"**). The magnification step is configurable (see section 8.3.1),

⊕ – Zoom in selected area (**Ctrl+"*"**). Upon clicking this button you mark the area to be zoomed in. The eventual viewing scale depends on the size of the area marked: if the marked area is large, the viewing scale is increased only slightly, if the marked area is small, it will be displayed in large scale.

⊕ – Realtime zoom in/out – stepless zooming (**Ctrl+"/"**). The area around the point indicated by the mouse pointer is magnified or contracted – the advantage is that you don't have to look for the location you want to edit. Realtime zoom is assigned by default to mouse roll.

⊖ – Zoom out (reduce) in steps (**Ctrl+"-"**).

⊕ – View entire project (**F5**): the viewing scale and area are changed so that the entire project is displayed.

⊕ – View entire active layer (**Shift+F5**): the viewing scale and area are changed so that all components of the active layer are displayed. For instance only the installation can be magnified in this way, no matter components set on the different layers.

⊗ – Revert to recent magnification and location – restores the scale used and location displayed before the last zoom or move operation.

♦ To use the realtime zoom function:
   1. Switch the realtime zoom mode on.
   2. Position the mouse pointer on the part of the drawing you want to zoom in or out.
   3. Press and hold the left mouse button.
   4. While keeping the left mouse button depressed move the mouse upwards to zoom in on the fragment or downwards to zoom out the fragment.

Stepless changing of the viewing scale (realtime zoom) provides a convenient way of observing individual fragments of the project in association with the "View entire project" function. After having edited a fragment of the drawing you press the **F5** key and use the realtime zoom to magnify another area of the drawing you want to edit.

The functions described above are used to change the viewing scale. A supplement to that is a group of functions that enable changing the part of the drawing that is displayed without changing the scale.

You can move the viewed part of the project on the screen by using the vertical and horizontal scroll bars or by using the "grabber hand". You can switch the "grabber hand" on with the button on the "Functions" tab.

♦ To move the viewed fragment of the project with the "grabber hand":
   1. Click the "grabber hand" button.
   2. Position the pointer within project area.
   3. Press and hold the left mouse button ("grabber hand" pointer appears).
   4. While keeping the left mouse button depressed move the viewed fragment in the desired direction.

! ***Double clicking the mouse when in realtime zoom mode or "grabber hand" mode, toggles these modes on and off. This enables convenient and fast viewing of the project***

The graphical editor supports mice with scroll wheels that operate in the MS IntelliMouse™ standard or with an enhanced scroll wheel driver (usually dual wheel mice). The wheels may be used to move the viewed project on the screen (function of the "grabber hand") or to zoom in or out stepless ("realtime zoom" function). The operating modes of the wheels are configurable (see section 8.3.3).
You can use the following keys when working with a wheel mouse to modify the functions of the wheels:
   – no key used – realtime zoom in and out
   – **Alt** – move the viewed project using default functions of the mouse wheels
   – **Shift+Alt** – reduces the scrolling speed (more precise operation) using default functions of the mouse wheels
   – **Ctrl+Alt** – move the viewed project with mouse wheel functions swapped - useful with single-wheeled mice.

4.4. Moving around the project - navigator

Navigator is a useful tool that helps moving from one place of the project to another. The button that activates the navigator is in the lower right-hand part of the screen:
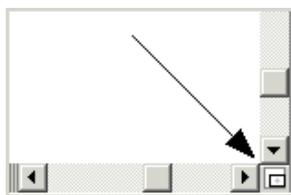
When you click and hold the left mouse button on the navigator button a preview of the entire current worksheet is displayed with the current view open for editing shown as a negative. When you move the mouse while holding the left mouse button,

the viewing window is moved accordingly. This facilitates viewing the entire worksheet at selected scale.

4.5. Project layers

A project consists of layers. Layers may consist of installation (network) layers, base (map, building plan) layers, etc. You can think of the project layers as of transparent films superimposed one over the other. At any given moment you have access to only one layer, while the other is displayed, but cannot be edited. Layers are represented in the status line by a tab or by a check list:

The last tab or the last item in the check list ("Print") differs from the others in that it is not a drawing layer and is used for configuring the printout of the drawing – defining page size, paper cutting lines, etc. (see section 7.2). Each worksheet includes the "Print" layer.

The "Map" type sheet consists of network and base map layers, whereas "Profile" type sheets have only one profile layer.

4.6. Operating modes of the editor: AUTO, LOCK, GRID, ORTO, REP

Special operating modes of the graphical editor facilitate some editing operations. You can toggle the modes on / off by double clicking the field representing the mode in the status bar:

ORTO | LOCK | GRID | AUTO | REP

Blue colour indicates an enabled mode, grey indicates a disabled mode.

To facilitate switching between the different modes, convenient combinations of keys are assigned situated one near the other and near the **Alt** key: **Alt+Z**, **Alt+X**, **Alt+C**, **Alt+V, Alt+B**. The letter keys used in these combinations do not indicate the first letters of mode names. These keys are arranged in the same order as the modes on the bar shown above.

4.6.1. ORTO mode - Inserting horizontal and vertical components

The graphical editor facilitates placing horizontal and vertical components when the orthogonal (ORTO in short) mode is enabled. You can also use this mode when drawing sections or polylines.

♦ To enable or disable the ORTO mode:
  – Click once on the ORTO box in the lower right corner of the screen
  or
  – Press the **Alt+Z** keys.

The ORTO mode also supports drawing at an angle specified on the "Edit" tab under "File / General data" (**F7**). This angle is specified as an inclination from horizontal line in degrees.

When the ORTO mode is enabled it is possible to place a horizontal, vertical or a slanting section inclined at the specified angle. This way the parallel arrangement is easily maintained without the need to operate the mouse precisely.

4.6.2. LOCK mode – Locking components

Locking (immobilising) components is applied to protect them against unintended displacement when pointing to them to append their data.

There are two methods in the graphical editor of locking and protecting components against movement:
1. global – switching on the LOCK mode – all components are locked
2. local – enables locking selected components only.

Global locking of all components

Using the global method is recommended, as it enables faster locking and unlocking of components.

Global lock of all components is indicated in the LOCK field in the lower right-hand corner of the screen (see section 4.2). Blue letters in that field indicate that the LOCK mode is activated. Grey letters mean that the LOCK mode is disabled.

The LOCK mode does not protect components against deletion. When the LOCK mode is enabled it is possible to mark and delete components – it is only the moving of components that is disabled.

♦ To globally lock or unlock all components in a worksheet:
  – click once on the LOCK field in the lower right-hand corner of the screen,
or
  – press the **Scroll Lock** key or **Alt+X** keys.

As global lock of components is assigned to the **Scroll Lock** key, which has a LED linked to it on the keyboard, you have an additional indicator of the lock mode.

When saving a project to a disk file, the current setting of the LOCK mode is also saved. You can also specify how the LOCK mode is to be set on opening of a saved file.

♦ To specify how the LOCK mode is to be set on opening of a file:
  1. Select "Options / Settings"
  2. Select the "Files" tab
  3. Set the "Lock on open file" field

  There are three options available:
  – No change – opening of a file does not change the current LOCK mode setting

–   Same as last edit – the same LOCK mode is set as during last save of the file
–   Lock – movement of components is always locked after opening of a file.

Lock selected components

Local locking enables locking selected components in the drawing. In addition it enables locking vertical movements while allowing horizontal movements.

In the standard program setting there are no buttons on the toolbar representing local locking/unlocking. Under such setting this function is only accessible from the menu. However, the user can append relevant buttons to the toolbar. The procedure is described in section 8.3.3.

♦   To lock / unlock selected components:
   1. Mark the component(s)
   2. To:
      a) completely lock a component, select "Components / Lock component"
      (**Ctrl+B**,),
      b) lock component elevation (vertical movements), select "Components / Lock
      component vertical" (),
   3. To unlock components, select "Components / Unlock component" (**Ctrl+D**,),
   4. Components are now locked / unlocked; this is indicated by a padlock symbol on the component (complete lock) or padlock symbol with vertical arrows (lock vertical).

! If a component is locked locally, it cannot be moved even if the LOCK mode is disabled. You have to unlock the component to be able to move it.

4.6.3.   GRID mode – Drawing grid

The drawing grid is designed to match components to each other when editing a project.

When the grid is switched on, then during inserting or moving the components, they are all snapped to the grid. This facilitates placing components on the same position or arranging them evenly at grid spacing intervals.

! Switching the GRID mode on does not cause the grid to be displayed in the workspace, but snaps to the grid the components that are inserted.

The standard grid spacing is 20 cm, in absolute scale, which means that with a typical scale of 1:100, the components will be positioned in the drawing with an accuracy of 2 mm.

The GRID mode has its field in the lower right-hand corner of the screen (see section 4.2). Blue letters in that field indicate that the GRID mode is activated. Grey letters mean that the GRID mode is disabled.

♦   To enable or disable the GRID mode:
   –   click once on the GRID field in the lower right-hand corner the screen,

or
  –   press the **Alt+C** keys.

    Grid parameters may be changed as required.

♦   To change grid parameters:
    1. Select "Options / General data" (**F7**)
    2. Select "Edit" tab
    3. Specify values for "Horizontal grid" and "Vertical grid"

4.6.4. AUTO mode – Automatic component connecting

    In the AUTO mode the editor automatically searches for and proposes a connection point for the component that is being inserted.

    The AUTO mode has its field in the lower right-hand corner of the screen (see section 4.2). Blue letters in that field indicate that the AUTO mode is activated. Grey letters mean that the AUTO mode is disabled.

    The AUTO mode is indicated when inserting components by diagonal cross marks, which indicate proposed point of placing the component. The component is moved in a manner enabling its connection to the existing structure.

*!   **When placing components, you can switch the AUTO mode on/off using the SHIFT key:**
    If the AUTO mode is globally disabled, and you need that mode to insert a new module, press and hold the SHIFT key while placing the module.
    If the AUTO mode is globally enabled, but it introduces some undesirable effects of pipe-run connecting when inserting a new module, press and hold the SHIFT key while placing the module.
    This could be formulated as follows:
    ~~AUTO~~ **+ Shift => AUTO**
    AUTO + Shift => ~~AUTO~~*

4.6.5. REP mode – Repetitive insertion of components

    Repetitive insertion mode is useful when editing both the plan of the building and the installation. The REP mode is designed to facilitate inserting of components of the same type.

    Blue REP letters in the field indicate REP mode activated. Grey letters mean that the REP mode is disabled.
    When this mode is activated, components of the selected type are inserted until the right mouse button is pressed (after inserting a component, the mode of inserting that component is still activated). When REP mode is disabled, after inserting a component the mode of selecting components is activated.

    The REP mode applies to structural, installation and fitting components (walls, windows, doors, radiator, heating floor, manifold, etc.).

♦   To enable or disable the REP mode:

  –   Click once on the REP field in the upper right corner of the screen,
or
  –   Press the **Alt+B** keys.

4.7. Undo and redo

The program features a multi-level undo and redo system. You can undo and redo almost any operation performed on the project.

♦   To undo the last operation select "Edit / Undo" (**Ctrl+Z**,).

If the last undo operation was unnecessary, you can restore the previous operation by performing the "redo" operation.

♦   To restore the undone operation select "Edit / Redo" (**Ctrl+Y**,).

Undo and redo has a multilevel structure. You should, however, remember that after undoing an operation and then performing other operations on the project (changing data, placing a component), it is not possible to redo the undone operation. This is because the status of the project has changed and the operation to be redone may, for instance, apply to non-existing components.

The number of possible undo operations depends on program settings and specifications of the computer. The need to change the settings may result from the fact that the undo and redo functions require saving of the project in memory after each change.

♦   To change the settings of "Undo" and "Redo":
  1. Select "Options / Settings"
  2. Select "General" tab
  3. Specify the required value in the "Undo options" field.

Depending on hardware configurations, you can impose some restrictions on the number of operations stored in the memory:
  –   no restrictions
  –   restriction on the number of operations stored
  –   restriction on the size of memory occupied.

The default setting is maximum 8MB of memory. In most of the usual cases this is sufficient for much more than 20 undo steps. When you position the mouse pointer on the field with the number of megabytes (MB) allocated to the last project status, a hint appears with the memory size currently occupied for undo and redo operations.

There is an additional function "Undo and Redo saves screen position". When this option is checked, then on undoing or redoing the displayed part of the drawing and the scale of the drawing are changed.

4.8. Inserting components and operations performed on components

4.8.1. Inserting components

The basic operation performed when using the graphical editor is the placing of components in the drawing. Each type of component is represented by a button on the toolbar in the "Components" tab. When you position the mouse pointer on a button, a hint is displayed with the description of the component.

♦ To insert a component in the project:
   1. Click the toolbar button representing the required component type.
   2. Move the mouse pointer to the drawing area. The pointer takes on the shape that indicates the kind of component to be placed. Contour of the component and connection points of components are drawn and position of the component is indicated on the vertical and horizontal rulers. You can observe the position of the component on the status line.
   3. Click the left mouse button on the position where you want to place the component. The component will be inserted and the basic mode of selecting and moving components will be restored.

♦ To place a segment of an installation or a polyline in the project:
   1. Follow steps 1 and 2 above
   2. Click the left mouse button at the starting point of the polyline
   3. Move the mouse pointer to the point where the end or deflection point of the polyline is to be located
   4. In case of a deflection point, click the left mouse button and change the direction of drawing the line. Repeat this step as many times as necessary
   5. Upon reaching the end point click the left mouse button, and then the right button. The polyline is completed.

When placing components, you can use the AUTO mode, which finds possible connection points for the new components, and the ORTO mode, which facilitates placing of horizontal and vertical components. The use of these modes depends on the type of component and is described in detail in sections on construction and installation editing.

User-defined modules are placed in a similar manner.

4.8.2. Marking single components

To be able to perform any operation on a component, such as move, delete, data change, etc., you must mark that component.

♦ To mark an existing component:
   1. Enable the mode of selecting (marking) components, if it is not enabled already. The component selection mode is the default mode. The black arrow button on

the toolbar should be depressed, and the mouse pointer must have the form of a cross.

2. Position the pointer on the component and click the left mouse button.

After marking the component it is highlighted, for instance by making its contour thicker or by changing its colour, and a message is displayed on the status bar informing that the component has been selected. If the data table is displayed, the data fields of the selected component appear.

! ***Every component upon inserting it is automatically marked. Inserting another component cancels that marking.***

! ***When marking a polyline clicking on one of its segments marks that segment only. To mark entire length of a polyline double click any fragment of it.***

You can mark a component which is obscured by another component placed on top of it.

♦ To mark a component obscured by another:

1. Click the component which is "on top"; this component becomes marked.
2. Press and hold the **Ctrl** key.
3. While keeping the **Ctrl** key depressed, click the component which is underneath (within the area which is shared by the two components).
4. Release the **Ctrl** key.
5. The component "on top" will be unmarked and the component "underneath" will be marked.

♦ To unmark a component click a point within the workspace where there is no component. The status line should then display "Selected: ---". You can also use the

"Unmark all components" function ( ).

4.8.3. Marking multiple components

Most of the operations can be done not only on single components, but on groups of components. To perform an operation on multiple components, you have to mark them.

! ***Marking multiple components and editing them simultaneously can substantially speed up your work. Marking multiple components is of special significance when appending data (see section 4.9.3).***

There are several methods of marking multiple components in the graphical editor. The choice of the method depends on the components that are to be marked and on their relative arrangement. Available methods of marking groups of components are described below.

♦ To unmark multiple components proceed as in the case of a single component. Click a point within the workspace where there is no component. The status line should then display "Selected: ---". You can also use the "Unmark all components" function ( ).

4.8.4. Marking multiple components with the use of the Shift key

♦ To mark several components:
   1. Enable the mode of selecting (marking) components, if it is not enabled already. The component selection mode is the default mode. The black arrow button on the toolbar should be depressed, and the mouse pointer must have the form of a cross.
   2. Position the pointer on one of the components and click the left mouse button to mark it.
   3. Press and hold the **Shift** key.
   4. While keeping the **Shift** key depressed, click the second and following components. These components will be marked without unmarking the previous ones.
   5. If you have marked a wrong component, click it once more (while still pressing the **Shift** key) and it will be unmarked.
   6. Release the **Shift** key.
   7. All the indicated components will be marked. This is indicated by changed colour or by thicker outline. The status line displays the number of marked components.

To unmark multiple components click a point within the workspace where there is no component. The status line should then display "Selected: ---". You can also use the "Unmark all components" function ( ).

! *You can combine the functions of the Shift and Ctrl keys (marking obscured components; function described in the previous item) when marking components.*

4.8.5. Marking multiple components within a specified area

It is possible to mark all components located within a specified area. You can also restrict marking to components of a specified type. The latter feature is particularly helpful when appending data.

♦ To mark all components within a specified area:

   1. Select "Edit / Select components in rectangle" ("Functions" –>). This command is also available in the pop-up menu displayed upon clicking the right mouse button with the mouse pointer positioned within the workspace.
   2. Position the mouse pointer in one corner of the area with the components to be marked and press the left mouse button.
   3. While keeping the mouse button depressed, mark the required area. The area is represented by a pink rectangle.

4. The components within the selected area will be marked when you release the mouse button.

*! **The following rules apply to component marking within selected area:**
– if you select the area from left to right, then only those components that are entirely enclosed within the selected area will be marked. The selected area is in this case drawn with a solid line
*– if you select the area from right to left, then all components entirely and partly enclosed within the selected area will be marked. The selected area is in this case drawn with a dashed line.**

It is also possible to mark components of a specified type only (e.g. sections, etc.) This is very helpful in appending data.

♦   To mark all components of a specified type within selected area:

1. Select "Edit / Select in rectangle components of type ..." ("Functions" –>)..
2. Select from the pull-down menu the required type of components to be marked.
3. Position the mouse pointer in one corner of the area with the components to be marked and press the left mouse button.
4. While keeping the mouse button depressed, mark the area. The area is represented by a pink rectangle.
5. The components within the selected area will be marked when you release the mouse button.

*! **You can configure the menu of component types to be selected (marked).***

4.8.6. Marking multiple components of specified type in the entire sheet

If you want to mark all components in the project or all components of a specified type, there are functions available similar to marking components within a specified area. These functions, however, apply to the entire sheet:

–   "Edit / Select all components" (**Ctrl+A**),

and

–   "Edit / Select all components of type ...", which display pop-up menus to select component type to be marked.

4.8.7. Moving component

Nearly every component inserted into the project can be moved freely and placed in any location. The exceptions are components that are placed on other components, such as valves. Components of this type cannot be moved at all or they can be moved only within a specified range. A valve, for instance, can only be moved within its system section.

♦   To move a component:

1. Make sure that the mode of selecting components is enabled (the button with a black arrow on the toolbar is depressed).

2. Position the mouse pointer on the component and click and hold the left mouse button (the component is marked).
3. With the left mouse button depressed drag the component to a new position.
4. Release the mouse button (the component remains marked).

! If more than one component is marked, dragging any of them causes all the marked components to be moved.

If the GRID mode (drawing grid) is enabled when components are moved, then the components are moved in accordance with the specified horizontal and vertical grid spacing.

It is also possible to move one or more selected components using cursor control keys, which is one of the ways to effect a vertical only or horizontal only movement. Pressing the **Alt+<cursor control key>** key combination moves the component by a distance equal to the minimum ruler interval, pressing the **Alt+Shift+< cursor control key >** key combination moves the component by a distance equal to 0.1 of the minimum ruler interval. In the GRID mode small movements cannot be effected. The described manner of moving components is convenient for precise positioning of components.

4.8.8. Changing the dimensions of components

Some components have fixed size, which cannot be changed. Other components may be modified freely by means of the mouse. Such components are, for instance, graphical components: rectangles, ellipses, etc.

All components, the size of which may be modified, have handles (black squares) on their outlines. These handles enable expanding and contracting of the component. For instance, a rectangle may be expanded and contracted in any direction:
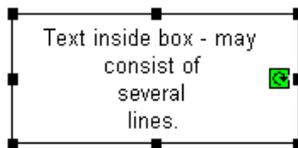
♦ To change the size of a component:
1. Make sure that the mode of selecting components is enabled (the black arrow button on the toolbar should be depressed).
2. Position the pointer on the first component and click the left mouse button to mark the component
3. Position the mouse pointer on one of the handles (squares). The pointer changes shape to suggest the manner of resizing.
4. Press and hold the left mouse button.
5. While keeping the left mouse button depressed, expand or contract the component.
6. Release the left mouse button.
7. The component takes on the modified shape.

4.8.9. Rotating a component

Some components can be rotated around its centre. Such components include, for instance, graphical components: rectangles, ellipses, etc.

All components that can be rotated have an icon , which can be grabbed and dragged to rotate the component.
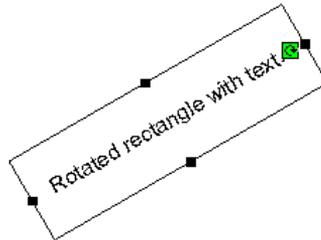
♦ To rotate a component:
1. Make sure that the mode of selecting components is (the black arrow button on the toolbar should be depressed)
1. Position the pointer on the component and click the left mouse button to mark it
2. Position the mouse pointer on the rotation icon . The pointer changes shape to suggest that the component may be rotated
3. Press and hold the left mouse button
4. While keeping the left mouse button depressed, rotate the component
5. Release the left mouse button.

! ***Pressing the Shift key when rotating a component limits the rotating capability to several (typical) angles only: 0°, 45°, 90°.***

4.8.10. Deleting components

The grahical editor allows simple deletion of components. If you want to delete a group of components, you can use the functions of marking groups described above and delete those components simultaneously.

♦ To delete a component from a project:
1. Mark the components you want to delete. You can use any of the methods described in sections 4.8.3 to 4.8.6

2. Select "Edit / Delete" (**Del**,).

4.8.11. Repeated inserting of an identical component

By default, after placing a component or module, the basic operating mode of marking and moving is restored. Often, however, you will want to insert several components of the same type. In these cases it is convenient to use the "Insert last item" command, which is assigned to the F3 function key.

♦ To place another component or module of the same type after inserting the first one, select "Edit / Insert last item" (**F3**,).

When inserting several identical components in sequence it is also possible to use the REP mode – repetitive insertion of components. When this mode is activated, components of the selected type are inserted until the right mouse button is pressed (after inserting a component, the mode of inserting that component is still activated). When REP mode is disabled, after inserting a component the mode of selecting components is activated.

The REP mode applies to structural, installation and fitting components (walls, windows, doors, radiator, heating floor, manifold, etc.).

♦ To enable or disable the REP mode:

– Click once on the REP field in the upper right corner of the screen,

or

Press the **Alt+B** keys.

4.9.   Appending component data

4.9.1.   Data appending fundamentals

Most of the components on which the program operates have defined data or parameters. The data table is used to edit these data.

♦ To switch the data table on / off select "View / Show / Hide data table" (**F12,**)

The left-hand column of the data table shows the names of the fields, while the right-hand column displays the values. Fields assigned to fittings additionally include symbols indicating whether these data fields are displayed (⊟) or hidden (⊞). In the program settings you can indicate whether the fields assigned to fittings are to be displayed by default or not, meaning that groups in the data table can be represented as closed or open (see section 8.3.1).

! *Some of the fields have hints assigned to them, which appear on the screen when you position the mouse pointer on the field (without pressing the mouse keys).*

The contents of the data table varies, and the fields that appear in the table depend on the component marked. If several components of the same type have been selected, the table contains fields common to all marked components. If components of different types have been marked, the data table will not contain any fields.

! *Marking many components of the same type and editing them simultaneously may speed up data appending substantially (see section 4.9.5).*

48

! ***To append data you have to mark (select) components. To avoid unintended displacement of components, it is advisable to lock them (LOCK mode) before starting data appending (see section 4.6.2).***

4.9.2. Data table field types and editing

The data table may consist of fields of various types. The field type determines the manner of editing it. Many data and values are determined automatically on the basis of the drawing or other data. These values are shown enclosed in brackets and the user can overwrite them.

! ***To restore the default value (assigned by the program) enter "?" in the given field and press the "Enter" key.***

The data table can include the following types of fields (examples are provided for graphical components):
1. Text fields
2. Numerical fields
3. Selection fields (multiple option fields, with an arrow next to the field name)
4. Compound fields (name followed by dots ...)
5. Information fields

! ***Data tables may also include fields of mixed nature.***

1. Text fields

In the text field you can enter text. In some cases the number of characters that can be entered is limited. To confirm the new contents of the text field you have just entered, press the **Enter** key.

An example of a text field is "Text" for a rectangle.

2. Numerical fields

Numerical fields enable you to enter numerical values. The value that is entered into a numerical field is usually checked automatically to see whether it is within specified range. If an out-of-range value is entered, then an audible warning signal is issued and the lowest or highest value within the permissible range is adopted, depending on the entered value being too low or too high. To confirm the new value in the numerical field press the **Enter** key.

In some numerical fields it is possible to enter special values, represented by letters or other characters. Special values are always explained in field hints. An example of a numerical field is "Height [mm]" for a rectangle.

3. Selection fields

In the case of some fields the range of values that can be entered is limited (e.g. "Arrow type" indicating arrowhead type at the starting point and at the end of a polyline). After clicking on a field of this type an additional button appears on the right side of the field which enables displaying a list of available values. Clicking on this button expands the list:
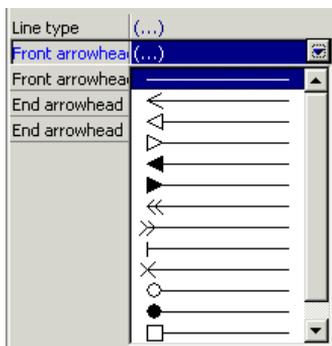
! ***When entering data, instead of using the mouse to display the list of values, you can press the Ctrl + Enter key combination. Ctrl + Enter is also used to close the list.***

To complete editing the field, after indicating the proper option, you can click on the arrow on the right side of the field or click on another field in the data table. Double clicking on the selected option also closes the list.

! ***Double clicking on a selection field selects the following option in the list.***

The lists that are displayed in the data table may be single-level or multilevel lists. In the multilevel lists sub-options are represented by yellow folders. To expand or contract a group, double click on it or position the mouse pointer on it and press the **Enter** key.

<u>4. Compound fields</u>

Some fields require that more than one value is specified. In this case compound fields are used. When you click on a compound field, a button with an arrow or three dots (ellipsis) appears on the right side of the field, similar to the one in selection fields. Clicking on this button opens an additional box with various fields, depending on the needs. An example of a compound field is the "Font" field.

You can close the window after specifying the values in the fields. To open or close a compound field, as was the case with selection fields, you can use the **Ctrl+Enter** keys. Editing can also be completed by clicking on the arrow on the right side of the field or by clicking on another field in the data table.

<u>5. Information fields</u>

Some fields serve only the purpose of informing and the user cannot modify them. After clicking on a field of this type a narrow coloured outline appears around it.

4.9.3. Repeated entering of the same data

Often many components in a project have the same data. If this is the case, to speed up editing, you can apply one of the methods, which facilitate repeated entering of the same data:

–   Repeat the last value
–   Enter data simultaneously for multiple components
–   Sets of typical values

These methods are described in the subsections that follow.

4.9.4. Repeating the last value

This is the simplest method of duplicating component data. After specifying data for one component and marking the following component you don't have to enter the data once more, just make use of the repeat feature.

♦   To enter data by repeating last value:
  1. Mark one component.
  2. Enter data.
  3. Mark next component – the same field should be indicated in the table.
  4. Select "Component data / Repeat last value" (**F2**).

! Last values are stored for 10 data fields.

If no last value is stored for the field selected, this is indicated by an audible warning signal and an appropriate message in the status bar.

4.9.5. Simultaneous entering of data for multiple components

This is the second fast method of entering data for multiple components. After marking several components, fields common to all these components are displayed in the data table. The values in the fields that are identical for all marked components are displayed in the table; other fields are empty.

If you enter a value in any of these fields, that value becomes valid for all marked components, no matter whether the values for individual components before entering the new value were identical or different. The fields that are not edited will not change their contents.

♦   To change a value for multiple components simultaneously:
  1. Mark the components. You can use all the techniques described in section 4.8.3 to 4.8.6.
  2. Enter data in appropriate fields in the data table.

4.9.6. Data sets and data sets gallery

In this method selected component data are stored in the form of a set and then the set is pasted into other components. The stored data sets form the contents of the data sets list, which can be used during further editing.

♦   To save a data set:
  1. Mark the component and append its data (if not appended before)
  2. Click the right mouse button with the pointer positioned within the data table or click the scroll button of the pop-up menu in the data table:

51

3. Select "Save as data set" in the pop-up menu
4. A list of fields to be saved in the set appears (initially the list is empty)
5. Using the arrow buttons, move the fields to be saved in the data set. To do this select the field or several fields using the Shift or Ctrl key and click the appropriate button. A single arrow moves only the marked fields, double arrow moves all fields
6. Click "OK" to save the data set or "Cancel" to abandon

An example of a data set creation window is shown below:
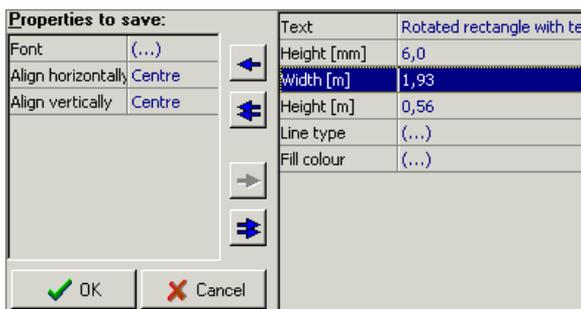
The created data sets are displayed in a special window of the data sets list.

♦ To switch a data sets list on or off select "View / Show/Hide data sets list" (**F9**,)

The list presents the stored data sets in the following form:
<field_value> (<field_name>); <field_value> (<field_name >); [...]

for example:
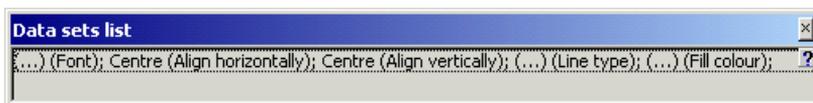
! ***When you position the mouse pointer on a data set, a hint is displayed showing which fields are stored in the set.***

Data sets may be used for individual components or multiple marked components.

♦ To paste a data set into individual components one by one:
1. Switch on the data sets list (if it is not displayed yet)
2. Click on the selected data set within the data sets list. The shape of the mouse pointer changes into an arrow with a data table attached
3. Click the left mouse button on all the components which are to take on the field values of the selected data set
4. Click the right mouse button to complete the "pasting" of data sets

♦ To paste a data set into many components at once:
1. Switch on the data sets list (if it is not displayed yet).

2. Mark the components which are to take on the data from the data set. You can use all the techniques described in sections 4.8.3 to 4.8.6.
3. Double click on the data set in the data sets list.
4. The data set is "pasted" into all marked components.

Data sets gallery

You can process the saved data sets: delete them, change their descriptions, etc. The data sets gallery is designed for this purpose.

♦ To switch a data sets gallery on select "Component data / Data sets gallery..."

The data sets gallery window is presented below:
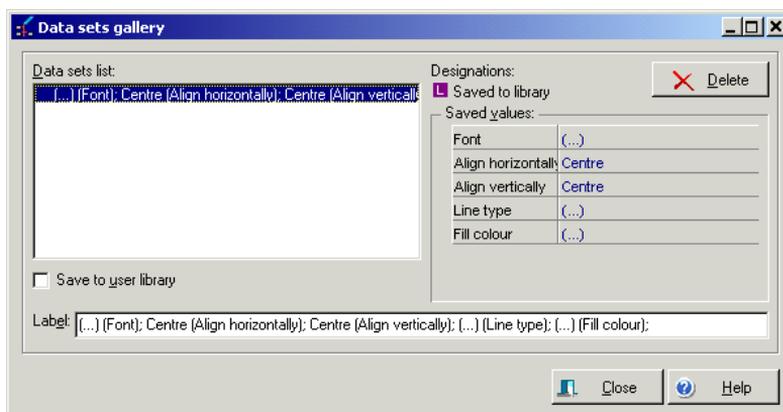
The main part of the window is the list of data sets. Data sets are presented in the same manner as in the data sets list, the only difference being letter "**L**", which may appear beside some of the data sets. This symbol indicates that the data set will be saved to a user library and not to the project file. It will then be possible to use that data set in all your projects. Data sets that are not marked with the "**L**" symbol are saved to the project file.

When you click on a data set, its contents are shown on the right side of the window. A checkbox under the list of sets is used to specify whether the selected data set is to be saved to a user library or not. In the "Label" field you can change the description of the selected data set. The "Delete" button enables you to remove the data set from the list.

4.9.7. *Moving between components using the keyboard*

The mouse is a convenient tool for editing drawings. However, when you want to edit component data, you need the keyboard. To avoid repeated shifting of your hand from the keyboard to the mouse and back, you can use the keyboard to move from one component to another.

The graphical editor features two modes of moving between components by means of the keyboard:

– moving to the nearest component in specified direction
– moving to the next connected component in specified direction.

Moving to the nearest component in specified direction:

This mode is most convenient for moving between components of the same type within a project.

♦ To move to the nearest component by means of the keyboard:
1. Mark one of the components. Its data is shown in the data table. To move about between the data in the table, use the cursor control keys:
– Press **Ctrl+<cursor control key>** to move in the direction indicated by the cursor control key to the nearest component of the same type (e.g. from a middle section of a profile to the next or preceding middle section)
– Press **Ctrl+Alt+<cursor control key>** to move in the direction indicated by the cursor control key to the nearest component of other type (e.g. from a middle section of a profile to the starting point of the profile).

<cursor control key> indicates one of the keys in the cursor control keypad.

! The movement is made to the component that is nearest in geometrical terms. Sometimes this may not be what the user expects.

After shifting to another component of the same type, its data are shown in the data table, which you can alter. The current field in the data table is not changed. This enables fast entering of data in one field. You can also use the function of repeating the last value (see section 4.9.4).

If the value is the same for all components, it is better to use one of the methods of marking multiple components (see section 4.8.3).

Moving to the next connected component in specified direction:

This mode is most convenient for moving between segments of a system within a project.

♦ To move to the next connected component in specified direction:
1. Mark one of the components. Its data is shown in the data table. To move about between the data in the table, use the cursor control keys
2. Make sure that the **Num Lock** mode of the keyboard is enabled – relevant LED is on
3. Press **Ctrl+< key of the numeric keypad >**. The direction of movement is determined by the position of the key in relation to key 5, and accordingly: key 8 – move to the component connected "at the top" of the selected component, key 9 - component connected "at the top and to the right" of the selected component, key 6 – "to the right", key 3 – "down, to the right", etc. The diagram below illustrates this:

ã 7	8 á	9 ä
ß 4	5	6 à
å 1	2 â	3 æ

! ***The keyboard can only be used to move from one component to another to append their data. To move or enter new components you have to use the mouse.***

4.10. Hardware and fittings

4.10.1. Basic information

Besides the main components, which make up the structure of the installation in Net-san, there are many additional components, such as valves or gates, protecting tubes, etc. They all have been given a collective name of "hardware and fittings". Their characteristic and common feature is that they are placed on other, existing components in the project.

! ***Hardware or fitting is a component that cannot stand alone in the project, it must be assigned to other component.***
! ***Graphical components, such as pipe-run labels, have similar properties. All operations that can be executed on fitting components can also be done on pipe-run labels.***

Description of the various hardware and fitting components is given in the following sections of the manual.

4.10.2. Inserting single fitting components

Fitting components are placed in a similar manner to ordinary components. The difference is that a fitting component must be inserted to an existing component, which we named basic component (for instance pipe-runs).

♦ To insert a fitting component:
1. Make sure that the basic component, onto which fittings will be added, already exists in the project.
2. Click the button representing the required fitting component
3. As you move the mouse within the project, the outline of the selected component is displayed.
4. Click the basic component into which you want to insert the fitting. The fitting will be inserted only if the fitting component is logically compatible with the basic component.
5. Depending on the set mode of inserting fittings, the component-marking mode is enabled or insertion of another identical fitting is proposed. In the latter case, if

you want to abandon inserting fittings, press the right mouse button.
Configuration of the mode of inserting fittings is described in section 8.3.1.

! ***Always click on the component (fragment of the system under design), onto which you want to install the fitting.***

4.10.3. Deleting fitting components

To delete a fitting or hardware component you first have to select (mark) it. Therefore, when selecting fitting or hardware components for deletion, always observe messages in the status line to be sure not to select and delete the basic component on which the fitting or hardware is placed.

Remember, that when you delete a basic component that includes fittings, those fittings will also be deleted. You can always undo a delete operation (see section 4.7).

4.10.4. Inserting multiple fittings

The graphical editor enables fast insertion of many identical fittings into existing basic components in a project.

♦ To insert fittings into many basic components fast:
1. Mark the basic components into which you want to insert the fittings. You can use all the techniques described in sections 4.8.3 to 4.8.6.
2. Double click the toolbar button representing the required fitting component. Fittings are inserted into all marked components, provided that the component type is logically compatible with the basic components.

4.10.5. Appending fitting component data

As mentioned, fittings are not independent components. They must always be assigned to other components. Fitting component data are also appended in the data table. When you click on the fitting or the corresponding basic component, the data table displays data of the main component at the top, and the data of the fitting beneath:

These data always begin with the name of the selected hardware or fitting component followed by fields of the selected fitting. These fields are marked with "•".
First field of each fitting is marked with a white symbol ⊞ or ⊟. The ⊞ symbol indicates that the remaining fields of that component are contracted (hidden). The ⊟

Rotate dwg.:

Complex field. Provide rotating pipe-run label. Used mainly when configuring many pipe-run labels.

A configured label placed on a pipe-run may be saved to the toolbar provided for user-defined modules.

♦ To save a pipe-run label and its settings, mark that label on the drawing and select "Modules / Append to toolbar" in the menu. The modules gallery window appears where you can add your label to a new component. After clicking OK., the new pipe-run label becomes available on the user-defined modules tab.

4.12. Splitting and inserting additional profile section into an existing fragment

To enable moving an entire fragment of profile (several sections simultaneously), all connections between profile sections must be stored in memory.

When constructing a profile functions are required that enable disconnecting a component on its right side only or on its left side only. The program is provided with such functions.

Disconnecting on one side only is used most often to insert an additional profile section into an existing fragment of profile.

♦ To insert a new section of profile into an existing fragment of profile:
1. Click on the profile section to the right of the point where you want to insert the new section of the profile
2. The section is marked
3. Select "Components / Disconnect on left" (**Ctrl+L**,)
4. Click and drag the disconnected profile fragment to the right
5. Insert the new profile section into the free space
6. Move the disconnected fragment back so that it is reconnected to the inserted section
7. The above steps may also be executed in a similar manner after disconnecting a section on the right side and moving the profile fragment to the left.

! To avoid instant reconnecting of a disconnected component, it must be moved away.

Sometimes you have to disconnect a component completely (on its right and left sides).

♦ To disconnect a component completely:
1. Click on the component
2. Select "Components / Disconnect" (**Shift+Ctrl+R**,)
3. The component is disconnected completely.

4.13. Connection points

Both during drawing network layout on a plan and when creating network profiles in the AUTO mode, when you move the mouse pointer near the connection point of subsequent section, the program will try to connect that section – the connection point is indicated by a cross.

When the **Fehler! Unbekannter Name für Dokument-Eigenschaft.** mode is enabled the program also helps in connecting network sections. "Connecting distance with AUTO disabled" and "Find connection in **Fehler! Unbekannter Name für Dokument-Eigenschaft.** mode" may be configured in the "General data" window in the "Edit" tab.

Connection of components is marked with a purple square at the point of contact of the two components. Components that are not connected are marked with green triangles:

Connected sections of a profile	Sections of a profile not connected
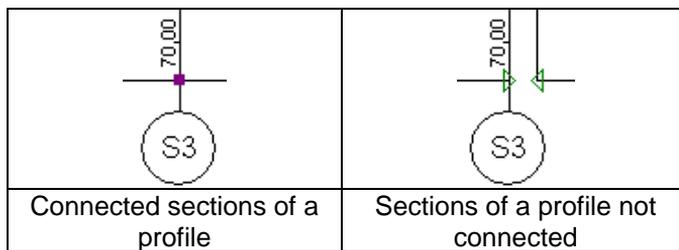
A black solid square appears between connected components on the network map (plan). If the component is not connected the square is empty (when checking connections (Shift+F2) unconnected components are displayed in a list):

Components on map connected	Components on map not connected
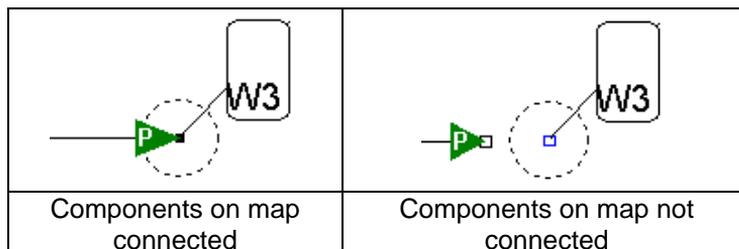
If the profile is generated automatically from network layout (map) created by the user, profile sections are marked with two partly superimposed squares – blue and green. When changes are made in the layout, then the marking of sections of profile also changes (grey and green squares). You must then repeat the "Create profile as defined" command.

Profile sections associated with map	Profile sections not associated with map
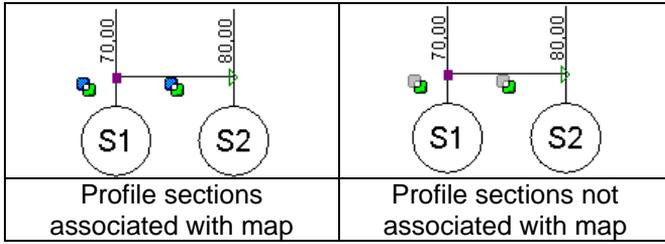
5. COMPONENTS FORMING THE PROJECT

This section describes the basic components that are used to build network plans and profiles in Net-san. Every component has a button representing it on the toolbar and used to insert the component and has a detailed description of the component data fields.

5.1. Components of base map for network plan

5.1.1. Map

Map is an optional component placed on the "Base map" layer. It helps drawing the layout of the network according to its routes.

! *Objects shown on the map are not identified by the program – it just forms a uniform background for the drawing.*

Button on the toolbar ("Base map" tab):

Map data:

Map file
File in a proprietary format (.ism) created with "Combine Maps" application or a bitmap file (.bmp) with the map of the area.

Resolution
Numerical field. Resolution [DPI], at which the map was scanned. Usual values include 100, 150 and 300.

Scale
Numerical field Scale of map specified by user. The values in "Resolution" and "Scale" fields are used to scale the map properly.

Edit colour
Selection field. Colour in which the map is displayed during editing. A light colour is recommended to avoid obscuring network components displayed against the background of the map. (none) indicates original colours of the map.

Print colour
Selection field. Colour in which the map is printed on a colour printer or a plotter. (none) indicates original colours of the map – for some printers this is the only proper setting.

5.2. Components of sewerage network plan

5.2.1. *Section of gravity sewerage network*

Button on the toolbar ("M – Sewerage" tab):

Section data:

Designation
Text field. Designation (symbol) of section of the network.

Component status
Selection field. Select current status of component from list: existing, under design, calculated or not calculated, to be removed or to be moved.

Location
Selection field. Select location of the component in the network: connection line / house drain, side branch, main network.

Section length [m]
Information field. Distance between start and end of network section. Section lengths are taken automatically from network diagram.

Pipe type
Selection field. A list of available catalogues is displayed, from which pipe type may be selected for one or for all sections. If the field is empty, then default pipe type is sized from catalogues specified in the General data of project (before calculations the following message is displayed: "undefined pipe type - default type applied").

Connection cat. and type
Selection field. Catalogue and, optionally, type of pipe fittings used when automatically making connections. The catalogue specified may be the one assigned to the pipe type in section.

Component appearance
Selection field. Select type of sewerage component and its appearance in the drawing. In addition you can configure the appearance of other components of the network.

Duct bottom start [m]
Numerical field. Elevation of duct bottom at the start of section. Value in brackets indicates elevation determined by the program. Entering "?" indicates reverting to automatic setting. This value must be specified for existing sections.

Duct bottom end [m]
Numerical field. Elevation of duct bottom at the end of section. Value in brackets indicates elevation determined by the program. Entering "?" indicates reverting to automatic setting.

Min. gradient [‰]
Numerical field. You can specify the minimum gradient for sized ducts. If the field is empty, then default value is adopted.

Max. gradient [‰]
Numerical field. You can specify maximum gradient for the ducts sized. If this is not specified, then default value is adopted.

Min. cover depth [m]
Numerical field. Minimum cover depth of ducts in relation to ground surface level. If the field is empty, then default value is adopted.

Max. cover depth [m]
Numerical field. Maximum cover depth of ducts in relation to ground surface level. If the field is empty, then default value is adopted.

Max. fill ratio [%]
Numerical field. Specifies percentage of maximum filling of duct under design with medium. If the field is empty, then default value is adopted.

Label
Compound field. Description of section appurtenance or pipe-run. You can select the status of appurtenance (under design or existing). Type of designation may be selected for pipe-run label to be positioned over, under or beside description line.

Self-defecation
Selection field. Fehler! Unbekannter Name für Dokument-Eigenschaft. - default selection defined for all sections of network in calculation options, **YES** - adopt the self-defecation criterion for the selected section, **NO** - disregard the self-defecation criterion for the selected section,

Connection to manhole
Selection field. Precise control over the manner of connecting a duct to manhole: Fehler! Unbekannter Name für Dokument-Eigenschaft. - optimum solution is adopted depending on available types of invert, land configuration and other specifications of the network.
To manhole bottom - connects duct to manhole bottom if the selected manhole has an invert with appropriate angles and number of connections.
Above bottom, no backdrop pipe - forces a cascade to be created when selected section is connected to manhole.
Above bottom, with backdrop pipe - forces a cascade to be created when selected section is connected to manhole, additionally an external backdrop pipe will appear on the profile.

5.2.2. Section of pressure sewerage network

Button on the toolbar ("M – Sewerage" tab):

Data similar to that of gravity sewerage network, except for "Max. fill ratio".

5.2.3. Section of heating network

Indicates the route of heating network, which is not calculated by the program, but may interfere with the network under design.

Button on the toolbar:

Section data:

Designation
Text field. Designation (symbol) of section of the network.

Component status
Selection field. Select current status of component from list: existing – not calculated, to be removed or to be moved.

Section length [m]
Information field. Distance between start and end of network section. Section lengths are taken automatically from network diagram.

Pipeline diam. [mm]
Numerical field. Diameter of pipeline. Specified by user.

Component appearance
Selection field. Select type of sewerage component and its appearance in the drawing. In addition you can configure the appearance of other components of the network.

Axis el. start [m]
Numerical field. Elevation of pipe axis at the start of section. This value must be specified for existing sections.

Axis el. end [m]
Numerical field. Elevation of pipe axis at the end of section. This value must be specified for existing sections.

Wall thickness [mm]
Numerical field. Thickness of pipeline wall. User-specified data.

Material
Text field. Material of the pipeline.

5.2.4. Section of gas supply network

Indicates the route of gas supply network, which is not calculated by the program, but may interfere with the network under design.

Button on the toolbar:

Data similar to that of heating network section.

5.2.5. Section of other network

Indicates the route of other network, which is not calculated by the program, but may interfere with the network under design.

Button on the toolbar:

Data similar to that of heating network section.

5.2.6. Manhole

Button on the toolbar:

Manhole data:

Designation
Text field. Designation (symbol) of manhole in the network..

Component status
Selection field. Select current status of component from list: under design, existing – calculated, not calculated, to be removed.

Function
Selection field. Select current function of component from list. Select:
– sewage source – first manhole in the calculated network. No network section bringing in sewage should be connected to this manhole
– flow / combine streams – middle manhole in the calculated network. This manhole should be connected with network sections bringing in sewage and one section taking sewage away
– sewage receptor – terminal manhole in the calculated network. This manhole should only be connected with network sections bringing in sewage.

Manhole type
Selection field. Type of manhole selected from catalogue or defined without catalogue.

Manhole top

Selection field. List of manhole tops – available only to manholes from catalogue.

Sediment trap height [m]

Numerical field. Height of sediment trap – in manholes with sediment trap.

Inflow at point [dm3/s]

Numerical field. Additional stream of medium flowing into given point (outside of network pipeline).

Area des. el. [m]

Numerical field. Elevation of area under design. Enter 'X' or '*' to adopt elevation same as that of existing ground surface level. This value is used in calculations – it must be defined.

Area exist. el. [m]

Numerical field. Elevation of existing area. Enter 'D' or '*' to adopt elevation same as that of planned ground surface level.

Manhole diam. [mm]

Numerical field – if manhole type was selected from catalogue. Diameter of manhole – you can specify the size of component under design. If rectangular manhole was selected, this field is disabled. In the component appearance window (see section 8.2.1) you can specify the minimum component diameter in drawing. If manhole type was selected from catalogue, this field is only for information purposes.

Dimension 1 [mm]

Numerical field. Dimension of one of manhole sides. Enabled for rectangular manholes only.

Dimension 2 [mm]

Numerical field. Dimension of second side of manhole. Enabled for rectangular manholes only.

Manhole bottom el. [m]

Numerical field. Elevation of the bottom of manhole. Value in brackets indicates elevation determined by the program. Entering '?' causes the manhole bottom to be positioned at the level of the lowermost connection.

Additional description

Text field. Additional caption displayed on the profile above the table. To edit this caption, open a special window by clicking on the button on the right side. You can also enter the text from the keyboard without opening the window. Additional options:

– ! – preceding the description with an '!' indicates that the complete description will be located above the line indicating the end of the profile section. Otherwise only the first line of the description is located above the line, the remaining lines of description displayed below the line

– | – indicates end of line and start of new line. To enter this character, press Shift+\
– @ – entering this character makes a diameter symbol ø (phi) appear in the description.

Cascade
Selection field. Determines the manner of drawing cascade on a profile:
auto (default) - cascade will be drawn when elevation difference between duct bottom and manhole bottom is more than 0.5 m
– no - cascade not drawn at all
– yes - cascade drawn always.

Component appearance
Selection field. Manhole description style. In addition you can configure the appearance of other components of the network (see section 8.2.1).

Runoff coefficient Y
Numerical field. Ratio of precipitation amount falling on given surface to amount flowing away or infiltrating the soil.

Catchment area [m^2]
Numerical field.

Stormwater retention
Selection field. After selecting **YES**, two additional fields appear which enable specifying:
– maximum value of outflow from retention reservoir in dm3/s,
– minimum retention capacity. The required height of reservoir (which is in fact the elevation of the of the reservoir) is calculated based on the value entered and on diameter or lengths of the sides of the reservoir.

Manhole build-up
Selection field. You can choose between automatic and manual selection of invert type.

5.2.7. *Sewage pumping station*

Button on the toolbar:

Pumping station data:

Designation
Text field. Designation (symbol) of component of the network.

Component status
Selection field. Select current status of component from list: under design, existing – calculated, not calculated, to be removed.

Pumping station type
Selection field. Type of pumping station selected from catalogue or defined without catalogue.

Outlet cover depth [m]
Numerical field. Cover depth of outlet of pumping station in relation to ground surface level. Editable in case of component not taken from catalogue. If component was selected from catalogue, this field is only for information purposes.

Delivery head [m]
Numerical field. Delivery head of pump. Editable in case of component not taken from catalogue.

Dead depth [m]
Numerical field. Minimum submergence (flooding depth) of pump at which the pump may start operation. Editable in case of component not taken from catalogue. If component was selected from catalogue, this field is only for information purposes.

Av. pump operation time [min]
Numerical field. Average time of pump operation within 1 hour with time accounted for cooling of pumping station.

Del.head / Pump characteristics
Compound field. Delivery head of pump at design point or pump performance characteristic.

Other data same as for manhole.

5.2.8. Outlet to open reservoir

Button on the toolbar:

Open reservoir data:

Open res. el. [m]
Numerical field. Elevation of inlet to open reservoir. This value must be specified.

Other data same as for manhole.

El. outflow to open res. [m]
Numerical field. Elevation of inlet to open reservoir. Must be specified.

5.2.9. Sewage drainage from building

Button on the toolbar:

Drainage data:

Designation
Text field. Designation (symbol) of component of the network.

Component status
Selection field. Select current status of component from list: under design, existing – calculated, not calculated, to be removed.

Inflow at point [dm3/s]
Numerical field. Flux of medium discharged from building.

Area des. el. [m]
Numerical field. Elevation of area under design. Enter 'X' or '*' to adopt elevation same as that of existing ground surface level. This value is used in calculations – it must be defined.

Area exist. el. [m]
Numerical field. Elevation of existing area. Enter 'D' or '*' to adopt elevation same as that of planned ground surface level.

Bldg. inlet el. [m]
Numerical field. Elevation of inlet into building. This value must be specified by user.

Component appearance
Selection field. Building drainage outlet description style. In addition you can configure the appearance of other components of the network (see section 8.2.1).

Drain type in bldg.
Selection field. Select from displayed list type of wastewater drainage from building: sanitary system (drain pipe crossing the wall or under foundation), drain pipe with cleanout, drain pipe without cleanout.

Wall th. [mm]
Numerical field. Thickness of building wall at basement level.

Found. depth [m]
Numerical field. Depth of building foundation in relation to basement level.

Found. height [m]
Numerical field. Height of strip foundation.

Basement level [m]
Numerical field. Elevation of basement level in building.

Ground floor elevation [m]
　　Numerical field. Elevation of ground floor in building.

Y
　　Numerical field. Runoff coefficient.

Catchment area [m²]
　　Numerical field.

Qd [dm³/s]
　　Numerical field. Additional flow of medium reaching given point (outside of network ducts).

No. of inhabitants
　　Numerical field. Mean number of residents in building.

　　Infrastructure components (such as: roadway, pipeline, knickpoint, bedding, etc.) and additional caption on profile are placed in the project on both the map diagram and the profile. They are all described in detail in section 5.6.

5.3.　Components of water supply network plan

5.3.1.　*Section of water-pipe network*

Button on the toolbar:

Section data:

Designation
　　Text field. Designation (symbol) of section of the network.

Component status
　　Selection field. Select current status of component from list: existing, under design, calculated or not calculated, to be removed or to be moved.

Location
　　Selection field. Select location of the component in the network: connection line / house drain, side branch, main network.

Section length [m]
　　Information field. Distance between start and end of profile section. Section lengths are taken automatically from network diagram.

Pipe type
　　Selection field. A list of available catalogues is displayed, from which pipe type may be selected for one or for all sections. If the field is empty, then default pipe type is sized from catalogues specified in the General data of project (before

calculations the following message is displayed: "undefined pipe type - default type applied").

Component appearance
Selection field. Style of water pipeline network description. In addition you can configure the appearance of other components of the network (see section 8.2.1).

Min. cover depth [m]
Numerical field. Minimum burial depth of pipelines in relation to ground surface level. If the field is empty, then default value is adopted.

Max. cover depth [m]
Numerical field. Maximum burial depth of pipelines in relation to ground surface level. If the field is empty, then default value is adopted.

Network type
Selection field. You can create network of appropriately configured appearance and assign to selected pipe-runs.

Max. gradient [‰]
Numerical field. Defines maximum value of gradient. If this is not specified by user, then default value is adopted.

User 1
Text field.

User 2
Text field.

Bedding
Text field. When this field is filled in, then appropriate text appears in the profile in the Bedding field. This field is not displayed by default, it must be activated in the Configuration of table fields in the Project Options.

Bedding layer thickness [m]
Numerical field.

5.3.2. Section of heating network

See item 5.2.3

5.3.3. Section of gas supply network

See item 5.2.4

5.3.4. Section of other network

See item 5.2.5

5.3.5. *Water-pipe network junction*

Button on the toolbar:

Junction data:

Designation
Text field. Designation (symbol) of section of the network.

Component status
Selection field. Select current status of component from list: existing, under design, calculated or not calculated, to be removed or to be moved.

Function
Selection field. Select current function of component from list: water receipt, flow / combine streams, water source. Depending on the choice, additional boxes are displayed for entering data.

Outflow at point [dm3/s]
Numerical field. Flux of medium discharged at given point (outside of network pipeline). Appears only in case of "water receipt" type junctions.

Required pressure [kPa]
Numerical field. Required pressure at medium reception point. This value must be specified by the user. Appears only in case of "water receipt" type junctions.

Pressure at source [kPa]
Numerical field. Required pressure at source. May be specified by the user or calculated by the program (size). Applies only to junction of 'water source' function.

Area des. el. [m]
Numerical field. Elevation of area under design. Enter 'X' or '*' to adopt elevation same as that of existing ground surface level.

Area exist. el. [m]
Numerical field. Elevation of existing area. Enter 'D' or '*' to adopt elevation same as that of planned ground surface level. This value must be defined.

Pipeline axis el. [m]
Numerical field. Elevation of pipeline axis. Value in brackets indicates elevation determined by the program. Entering '?' indicates that elevation is to be calculated. In case of junction of a 'water source' function, the value must be specified.

Additional description
Text field. Additional caption displayed on the profile above the table. To edit this caption, open a special window by clicking on the button on the right side. You can also enter the text from the keyboard without opening the window. Additional options:

– ! – preceding the description with an '!' indicates that the complete description will be located above the line indicating the end of the profile section. Otherwise only the first line of the description is located above the line, the remaining lines of description displayed below the line
– | – indicates end of line and start of new line. To enter this character, press Shift+\
– @ – entering this character makes a diameter symbol ø (phi) appear in the description.

Component appearance
Selection field. Junction description style. In addition you can configure the appearance of other components of the network (see section 8.2.1).

Straight through pipe/duct: z
Numerical field. Local loss coefficient; enables taking into account of local pressure drop across a pipe fitting.

Branch: z
Numerical field. Local loss coefficient; enables taking into account of local pressure drop across a pipe fitting.

Crossing type
Selection field. Selection of one of the four available options has an effect on the manner of presenting a junction on a profile.

5.3.6. Hydrant

Button on the toolbar:

Hydrant data:

Outflow at point [dm3/s]
Numerical field. Flux of medium discharged at given point.

Required pressure [kPa]
Numerical field. Required pressure at medium reception point. This value must be specified by the user.

Hydrant type
Selection field. Select type of hydrant: underground or surface.

z
Numerical field.

Other data as for water pipeline junction.

5.3.7. *Water-pipe inlet in building*

Button on the toolbar:

Inlet data:

Elev. of inlet into build. [m]
Numerical field. Elevation of pipeline inlet in the building. The value may be specified by the user or determined by the program during network route optimisation.

Outflow at point [dm3/s]
Numerical field. Flux of medium discharged at given point.

Required pressure [kPa]
Numerical field. Required pressure at medium reception point. This value must be specified by the user.

Bldg. inlet el. [m]
Numerical field. Elevation of inlet into building. This value must be specified by user.

Component appearance
Selection field. Building inlet description style. In addition you can configure the appearance of other components of the network (see section 8.2.1).

Inlet type in bldg.
Selection field. Select from displayed list type of pipeline inlet into building: water meter (screw conn.), water meter (flange conn.), no water meter.

Wall th. [mm]
Numerical field. Thickness of building wall at basement level.

Found. depth [m]
Numerical field. Depth of building foundation in relation to basement level.

Basement level [m]
Numerical field. Elevation of basement level in building.

Ground floor elevation [m]
Numerical field. Elevation of ground floor in building.

Water meter above basement lev. [m]
Numerical field. Elevation of water meter above basement level in building.

Other data as for water pipeline junction.

5.3.8. Water meter chamber

Button on the toolbar:

Chamber data:

Manhole diam. [mm]
Numerical field. Diameter of chamber. Enter the diameter of the component under design.

Manhole bottom el. [m]
Numerical field. Elevation of the bottom of chamber. Value in brackets indicates elevation determined by the program. Entering '?' causes the chamber bottom to be positioned at the level of the lowermost connection.

Other data same as for water pipeline junction, except for "Junction type", "Function" and "Pipeline axis el.".

Infrastructure components (such as: roadway, pipeline, knickpoint, bedding, etc.), fittings and additional caption on profile are placed in the project on both the map diagram and the profile. They are all described in detail in section 5.6.

5.4. Components of sewerage profile

Components described in this section are used in principle only when creating a profile of sewerage network not associated with network map. Profile generated automatically by the program includes its basic components. Nevertheless, when you know the properties of the various components, you will better understand the possibilities of automatic profile generation and, to some extent, you will be able to modify profiles generated automatically.

5.4.1. Start of sewerage profile

The start of sewerage profile describes the table with profile description and first manhole.

Button on the toolbar:

Sewerage profile start data:

Designation
Text field. Designation of first manhole of the profile or other component. You can select from list a designation already entered. If the entered or selected designation already exists, then data of profile section end will be copied. Sometimes, when sections are short, manhole designations are superimposed. In the "Designation" field you can move the captions up or down. To lower the caption

- enter '>' before the text (symbol), to raise the caption - enter '<'. You can enter the designation and move it down at the same time. Examples:
– S1 – assign designation "S1"
– > – move the existing caption down,
– >S1 – assign designation "S1" and move it down.

Start X [m]

Numerical field. Location of first manhole determined by start of profile. 0 adopted by default.

At section end

Selection field. Specify component to be placed at end of profile section. You can select from list: sewage / water drainage from building, manhole, pumping station, outlet to open reservoir.

Area des. el. [m]

Numerical field. Elevation of area under design at end of profile section. Enter 'X' or '*' to adopt elevation same as that of existing ground surface level.

Area exist. el. [m]

Numerical field. Elevation of existing area at end of profile section. Enter 'D' or '*' to adopt elevation same as that of planned ground surface level.

Additional description

Text field. Additional caption displayed on the profile above the table. To edit this caption, open a special window by clicking on the button on the right side. You can also enter the text from the keyboard without opening the window. Additional options:
– ! – preceding the description with an '!' indicates that the complete description will be located above the line indicating the end of the profile section. Otherwise only the first line of the description is located above the line, the remaining lines of description displayed below the line
– | – indicates end of line and start of new line. To enter this character, press Shift+\
– @ – entering this character makes a diameter symbol ø (phi) appear in the description.

Other data depend on the component selected:

For sewage / water drainage from building:

Drain type in bldg.

Selection field. Select from displayed list type of wastewater drainage from building: sanitary system (drain pipe crossing the wall or under foundation), drain pipe with cleanout, drain pipe without cleanout.

Wall th. [mm]

Numerical field. Thickness of building wall at basement level.

Found. depth [m]

Numerical field. Depth of building foundation in relation to basement level.

Basement level [m]
   Numerical field. Elevation of basement level in building. Required to enable drawing of wall on profile.

Ground floor elevation [m]
   Numerical field. Elevation of ground floor in building.

   For a manhole:

Manhole type
   Selection field. Type of manhole selected from catalogue or defined without catalogue.

Manhole top
   Selection field. List of manhole tops – available only to manholes from catalogue.

Manhole diam. [mm]
   Numerical field Diameter of manhole. Specify the size of component under design.

Manhole bottom el. [m]
   Numerical field. Elevation of the bottom of manhole. Value in brackets indicates elevation determined by the program. Entering '?' causes the manhole bottom to be positioned at the level of the lowermost connection.

Side inlets
   Compound field. You can specify data of manhole side inlets. Button to the right opens a table for editing elevations and diameters of side inlets.

Cascade
   Selection field. Determines the manner of drawing cascade on a profile:
   auto (default) - cascade will be drawn when elevation difference between duct bottom and manhole bottom is more than 0.5 m
   – no - cascade not drawn at all
   – yes - cascade drawn always.

5.4.2. Contracted sewerage profile start

A contracted sewerage profile start defines the start of profile without a table with descriptions and the first manhole. This component is usually used to draw house drains.

Button on the toolbar:

Data same as for sewerage profile start.

5.4.3. Sewerage profile section (middle)

A sewerage profile section defines the middle fragment of sewerage profile.

Button on the toolbar:

Data of middle section of sewerage profile:

Designation
Text field. Designation of first manhole of the profile or other component. You can select from list a designation already entered. If the entered or selected designation already exists, then data of profile section end will be copied. Sometimes, when sections are short, manhole designations are superimposed. In the "Designation" field you can move the captions up or down. To lower the caption - enter '>' before the text (symbol), to raise the caption - enter '<'. You can enter the designation and move it down at the same time. Examples:
– S1 – assign designation "S1"
– > – move the existing caption down,
– >S1 – assign designation " S1" and move it down.

Component appearance
Selection field. Select sewerage type and assigned appearance from list. In addition you can configure the appearance of other network components.

At section end
Selection field. Specify component to be placed at end of profile section. You can select from list: sewage / water drainage from building, manhole, pumping station, outlet to open reservoir.

Section length [m]
Numerical field. Distance between manholes at the start and at the end of profile section. Enter the distance to calculate the position of the end of profile section. Enter a value preceded by "!" to specify the position of section end - the distance will be calculated by the program.

Area des. el. [m]
Numerical field. Elevation of area under design at end of section. Enter 'X' or '*' to adopt elevation same as that of existing ground surface level.

Area exist. el. [m]
Numerical field. Elevation of existing area at end of section. Enter 'D' or '*' to adopt elevation same as that of planned ground surface level.

Duct bottom start [m]
Numerical field. Elevation of duct bottom at the start of section. The default setting is (auto), which means that the elevation of the end of preceding profile section is adopted. You can enter "?" to revert to "auto" setting or enter the elevation directly. You can also precede the value entered with one of the following characters:

– "p" or "i" – indicating that the value entered is burial depth in relation to area under design or existing area, respectively
– "<" – indicating that the value entered is gradient (%)
– ">" – indicating that the value entered is gradient (%) in the opposite direction.

Duct bottom end [m]

Numerical field. Elevation of duct bottom at the end of section. Enter elevation directly or precede the value entered with:
– "D" or "X" – indicating that the value entered is burial depth in relation to area under design or existing area, respectively
– "<" – indicating that the value entered is gradient (%)
– ">" – indicating that the value entered is gradient (%) in the opposite direction.

Pipe type

Selection field. Material, which the section is made of. A list of available catalogues is displayed, from which pipe type may be selected for the profile section.

Duct diam. [mm]

Numerical field. Diameter of duct.

Diam. design.

Selection field. Manner of designating diameter and name of material displayed in "Diameters, material / Gradients". The following options are available:
– (none) - diameter not displayed. Only material name displayed
– Phi - diameter displayed as ø, followed by material name, e.g. "ø 160 PVC"
– Dn - diameter designated as Dn, followed by material name, e.g. "Dn 150 steel"
– (material name) - diameter follows material name without any designation, e.g. "PVC 400".

Gradient with next

Selection field. If you enter "Yes" here, the gradient will be calculated and drawn with the next profile section included:

first and second sections have the field "Gradient with next" set to "Yes", the third is set to "No"	All sections have their field "Gradient with next" set to "No"

Additional description
Text field. Additional caption displayed on the profile above the table. To edit this caption, open a special window by clicking on the button on the right side. You can also enter the text from the keyboard without opening the window. Additional options:

- ! – preceding the description with an '!' indicates that the complete description will be located above the line indicating the end of the profile section. Otherwise only the first line of the description is located above the line, the remaining lines of description displayed below the line
- | – indicates end of line and start of new line. To enter this character, press Shift+\
- @ – entering this character makes a diameter symbol ø (phi) appear in the description.

Component status
Selection field. Current component status can be selected from the list: existing, under design, to be removed, to be moved.

Duct bottom end [m]
Numerical field. Elevation of duct bottom at end of profile section. This elevation should be entered directly or preceded by:

- "p" or "x" – indicates that the value entered is the depth of the duct in relation to ground level, under design or existing, respectively,
- "<" – indicates that the value entered is the gradient expressed in percent,
- ">" – indicates that the value entered is the gradient in opposite direction expressed in percent.

Backdrop pipe
Selection field. Determines the manner of drawing a backdrop pipe on a profile.

- „auto (default)" – indicates that the program will draw a backdrop pipe when the elevation difference between the duct bottom and manhole bottom exceeds 0.5 m,
- "no" – backdrop pipe will not be drawn,
- "yes" – backdrop pipe will always be drawn.

Manhole status
Selection field. Current status can be selected from the list: existing, under design, to be removed, to be moved.

5.5. Components of water pipeline profile

Components described in this section are used in principle only when creating a profile of water supply network not associated with network map. Profile generated automatically by the program includes its basic components. Nevertheless, when you know the properties of the various components, you will better understand the possibilities of automatic profile generation and, to some extent, you will be able to modify profiles generated automatically.

5.5.1. *Start of water pipeline profile*

The start of water pipeline profile describes the table with profile description and first component of the profile.

Button on the toolbar:

Water pipeline profile start data:

Designation
Text field. Designation of the first component of the water pipeline profile. You can select from list a designation already entered. If the entered or selected designation already exists, then data of profile section end will be copied. Sometimes, when sections are short, designations are superimposed. In the "Designation" field you can move the captions up or down. To lower the caption - enter '>' before the text (symbol), to raise the caption - enter '<'. You can enter the designation and move it down at the same time. Examples:
– W1 – assign designation "W1"
– > – move the existing caption down,
– >W1 – assign designation "W1" and move it down.

Start X [m]
Numerical field. Location of first component of the profile. 0 adopted by default.

Component status
Selection field. Current component status can be selected from the list: existing, under design, to be removed, to be moved.

Angle of route direction change [º]
Numerical field. With this field filled in, when the function "Project Options" / "Component appearance" / "Profile components" / "Draw angles of route direction changes" is activated, designations of route inflections and angles appear above the plotted profiles.

At section end
Selection field. Specify component and its drawing to be placed at end of profile section. You can select from list: inlet to bldg. (building to which the pipeline is connected), crossing, water meter chamber, hydrant.

Area des. el. [m]
Numerical field. Elevation of area under design at end of section. Enter 'X' or '*' to adopt elevation same as that of existing ground surface level.

Area exist. el. [m]
Numerical field. Elevation of existing area at end of section. Enter 'D' or '*' to adopt elevation same as that of planned ground surface level.

Pipeline axis end [m]
Numerical field. Elevation of pipeline axis at end of profile section. Enter elevation directly or precede the value entered with:

- "p" or "i" – indicating that the value entered is burial depth in relation to area under design or existing area, respectively
- "<" – indicating that the value entered is gradient (%)
- ">" – indicating that the value entered is gradient (%) in the opposite direction.

Additional description

Text field. Additional caption displayed on the profile above the table. To edit this caption, open a special window by clicking on the button on the right side. You can also enter the text from the keyboard without opening the window. Additional options:

- ! – preceding the description with an '!' indicates that the complete description will be located above the line indicating the end of the profile section. Otherwise only the first line of the description is located above the line, the remaining lines of description displayed below the line
- | – indicates end of line and start of new line. To enter this character, press Shift+\
- @ – entering this character makes a diameter symbol ø (phi) appear in the description.

Branches …

Numerical field.

Other data depend on the drawing selected:

For inlet into building:

Inlet type in bldg.

Selection field. Select from displayed list type of pipeline inlet into building: water meter (screw conn.), water meter (flange conn.), no water meter.

Wall th. [mm]

Numerical field. Thickness of building wall at basement level.

Found. depth [m]

Numerical field. Depth of building foundation in relation to basement level.

Basement level [m]

Numerical field. Elevation of basement level in building. Required to enable drawing of wall on profile.

Ground floor elevation [m]

Numerical field. Elevation of ground floor in building.

Found. height [m]

Numerical field. Height of strip foundation.

Water meter above basement lev. [m]

Numerical field. Elevation of water meter above level of basement in building.

For crossing:

Crossing type
Selection field. Select from displayed list type of crossing at section end: direct, tee/crosspiece, crosspiece integrated with gate valves, angle valve, saddle fitting.

For hydrant:

Hydrant type
Selection field. Select type of hydrant: underground or surface.

5.5.2. *Contracted water pipeline profile start*

A contracted water pipeline profile start defines the start of profile without a table with descriptions and the first component of the profile. This component is usually used to draw building connection lines.

Button on the toolbar:

Data same as for water pipeline profile start.

5.5.3. *Water pipeline profile section (middle)*

A water pipeline profile section defines the middle fragment of sewerage profile.

Button on the toolbar:

Data of middle section of water pipeline profile:

Designation
Text field. Designation of component of the water pipeline profile. You can select from list a designation already entered. If the entered or selected designation already exists, then data of profile section end will be copied. Sometimes, when sections are short, designations are superimposed. In the "Designation" field you can move the captions up or down. To lower the caption - enter '>' before the text (symbol), to raise the caption - enter '<'. You can enter the designation and move it down at the same time. Examples:
– W1 – assign designation "W1"
– > – move the existing caption down,
– >W1 – assign designation "W1" and move it down.

Component status
Selection field. Current component status can be selected from the list: existing, under design, to be removed, to be moved.

Angle of route direction change [º]
Numerical field.

83

Type of network
Selection field. Network type and drawing appearance assigned to it can be selected from list.

Component appearance
Selection field. Style of profile section description. In addition you can configure the appearance of other components of the network (see section 8.2.1).

At section end
Selection field. Specify component and its drawing to be placed at end of profile section. You can select from list: inlet to bldg. (building to which the pipeline is connected), crossing, water meter chamber, hydrant.

Section length [m]
Numerical field. Distance between start and at end of profile section. Enter the distance to calculate the position of the end of profile section. Enter a value preceded by "!" to specify the position of section end - the distance will be calculated by the program.

Area des. el. [m]
Numerical field. Elevation of area under design at end of section. Enter 'X' or '*' to adopt elevation same as that of existing ground surface level.

Area exist. el. [m]
Numerical field. Elevation of existing area at end of section. Enter 'D' or '*' to adopt elevation same as that of planned ground surface level.

Pipeline axis start [m]
Numerical field. Elevation of pipeline axis at the start of profile section. The default setting is (auto), which means that the elevation of the end of preceding section is adopted. You can enter "?" to revert to "auto" setting or enter the elevation directly. You can also precede the value entered with one of the following characters:
–   "p" or "i" – indicating that the value entered is burial depth in relation to area under design or existing area, respectively
–   "<" – indicating that the value entered is gradient (%)
–   ">" – indicating that the value entered is gradient (%) in the opposite direction.

Pipeline axis end [m]
Numerical field. Elevation of pipeline axis at end of profile section. Enter elevation directly or precede the value entered with:
–   "p" or "i" – indicating that the value entered is burial depth in relation to area under design or existing area, respectively
–   "<" – indicating that the value entered is gradient (%)
–   ">" – indicating that the value entered is gradient (%) in the opposite direction.

Pipe type
Selection field. Material, which the section is made of. A list of available catalogues is displayed, from which pipe type may be selected for the profile section.

Pipeline diam. [mm]
    Numerical field. Diameter of pipeline.

Wall thickness [mm]
    Numerical field. Thickness of pipeline wall. User specified data.

Branches…
    Numerical field.

Diam. design.
    Selection field. Manner of designating diameter and name of material displayed in "Diameters, material / Gradients". The following options are available:
–   (none) - diameter not displayed. Only material name displayed
–   Phi - diameter displayed as ø, followed by material name, e.g. "ø 160 PVC"
–   Dn - diameter designated as Dn, followed by material name, e.g. "Dn 150 steel"
–   (material name) - diameter follows material name without any designation, e.g. "PVC 400".

Gradient with next
    Selection field. If you enter "Yes" here, the gradient will be calculated and drawn with the next profile section included:

6,00	10,20	12,40		6,00	10,20	12,40

first and second sections have the field "Gradient with next" set to "Yes", the third is set to "No"	All sections have their field "Gradient with next" set to "No"

Additional description
    Text field. Additional caption displayed on the profile above the table. To edit this caption, open a special window by clicking on the button on the right side. You can also enter the text from the keyboard without opening the window. Additional options:
–   ! – preceding the description with an '!' indicates that the complete description will be located above the line indicating the end of the profile section. Otherwise only the first line of the description is located above the line, the remaining lines of description displayed below the line
– | – indicates end of line and start of new line. To enter this character, press Shift+\
–   @ – entering this character makes a diameter symbol ø (phi) appear in the description.

Other data depend on the component selected and its drawing at the end of profile section, and they are described for the start of water pipeline profile.

Bedding
  Text field.

Bedding layer thickness [m]
  Numerical field.

Backfill
  Text field.

Top backfill thickness [m]
  Numerical field.

User 1
  Text field.

User 2
  Text field.

Reference level [m]
  Numerical field. Reference level for profile section.

Elevation [m]
  Numerical field. Elevation of conduit. Program determines elevation automatically from drawing. The value can be overridden by user.

5.6. Appurtenances in profile sections

Additional components that modify the appearance of profile or constitute appurtenances on the profile route can be placed on sections of network profiles, both sewerage as well as water supply.

The following components can be placed on sewerage and water pipeline profiles.

5.6.1. Additional caption on profile

Button on the toolbar:

Data of additional caption:

Caption
  Text field. Additional caption on profile. You can enter text which will be displayed in the indicated area on the profile. You can also specify the orientation of the text: upwards or downwards.

5.6.2. *Perpendicular pipe and double perpendicular pipe*

Button on the toolbar: or

Perpendicular pipe data:

Line
    Selection field. Specify data of the line (pipe) by entering letters directly into the data table or selecting them from a window opened by clicking on the button on the right side:
–   1 / 2 – single or double line,
–   X – existing line, to be calculated or not calculated
–   D – line under design,
–   L – existing line to be removed
–   M – existing line to be moved
–   O – designate distance (position) in profile table
–   E – designate elevation in profile table.

Diameter [mm]
    Numerical field. Diameter of line. Specified by user.

Type of network
    Selection field. Type of appurtenance (conduit) entered on profile drawing. Select from available list or configure in "Component appearance" window.

Description
    Text field. Description of line. Generated automatically – may be modified by user. To recall automatic settings enter "?". You can also specify the orientation of the text: upwards or downwards.

Position [m]
    Numerical field. Position of line in relation to profile start. Position of component is determined automatically from the drawing. You can specify the distance from start of current section by entering a value preceded by '+'.

Elevation [m]
    Numerical field. Elevation of line. Elevation is determined automatically from the drawing. You can override it by entering other value.

In square duct
    Selection field.

5.6.3. *Knickpoint*

Button on the toolbar:

Knickpoint data:

Knickpoint
Selection field. Specify knickpoint data by entering letters directly into the data table or selecting them from a window opened by clicking on the button on the right side:
- X – existing knickpoint
- D – knickpoint under design
- O – designate distance (position) in profile table
- E – designate elevation in profile table.

Position [m]
Numerical field. Position of knickpoint in relation to profile start. Position of component is determined automatically from the drawing. You can specify the distance from start of current section by entering a value preceded by '+'.

Elevation [m]
Numerical field. Elevation of knickpoint. Elevation is determined automatically from the drawing. You can override it by entering other value.

Description
Text field. Description of knickpoint. You can enter text which will be displayed in drawing. You can also specify the orientation of the text: upwards or downwards.

5.6.4. Roadway

Button on the toolbar:

Roadway data:

Roadway
Selection field. Specify roadway data by entering letters directly into the data table or selecting them from a window opened by clicking on the button on the right side:
- X – existing roadway
- D – roadway under design
- O – designate distance (position) in profile table
- E – designate elevation in profile table.

Position [m]
Numerical field. Position of roadway in relation to profile start. Position of component is determined automatically from the drawing. You can specify the distance from start of current section by entering a value preceded by '+'.

Elevation [m]
Numerical field. Elevation of roadway. Elevation is determined automatically from the drawing. You can override it by entering other value.

Width [m]
Numerical field. Width of roadway. 15 metres is the default value. You can override it by entering other value.

Depth at kerb [m]
Numerical field. Depression of roadway edge in relation to kerb (kerb upstand). 0.5 metre is the default value. You can override it by entering other value.

Depth of roadway axis [m]
Numerical field. Depression of roadway axis in relation to kerbs. 0.0 metre is the default value. You can override it by entering other value.

Description
Text field. Description of roadway. You can enter text which will be displayed in drawing. You can also specify the orientation of the text: upwards or downwards.

5.6.5. Bedding and backfill

Button on the toolbar: or

Bedding / backfill data:

Length [m]
Numerical field. Distance along which the bedding or backfill is arranged. 10.0 metres is the default value. You can override it by entering other value.

Width [m]
Numerical field. Width of bedding or backfill. 0.2 metre is the default value. You can override it by entering other value.

Thickness [m]
Numerical field. Thickness of bedding or backfill. 0.2 metre is the default value. You can override it by entering other value.

5.6.6. Protection tube

Button on the toolbar:

Protection tube data:

Length [m]
Numerical field. Length of protection tube. 5.0 metres is the default value. You can override it by entering other value.

Diameter [mm]
Numerical field. Diameter of protection tube. 200 mm is the default value. You can override it by entering other value.

In water pipeline profiles you can also place fittings:

5.6.7. Gate valve

Button on the toolbar:

Gate valve data:

Appurtenances
Selection field. Specify appurtenance (gate valve) data by entering letters directly into the data table or selecting them from a window opened by clicking on the button on the right side:
- X – existing appurtenance
- D – appurtenance under design
- O – designate distance (position) in profile table
- E – designate elevation in profile table.

Type
Selection field. Type of gate valve selected from catalogue or defined without catalogue

Position [m]
Numerical field. Position of appurtenance (gate valve) in relation to profile start. Position of component is determined automatically from the drawing. You can specify the distance from start of current section by entering a value preceded by '+'. Elevation of appurtenance is determined from its position.

After placing the component, the data table is extended with data of that component.

5.7. Components of drawings of water supply network junctions

5.7.1. Junction section

Component used to designate section of water pipeline junction.

Button on the toolbar:

Junction section data:

Diameter [mm]
Numerical field. Diameter of water pipeline section. Specified by user.

Material
Text field. Material of the junction section. material name entered by user.

Diam. design.
Selection field. Manner of designating diameter and name of material displayed after selecting "Diameter and material label". The following options are available:
–   (none) - diameter not displayed. Only material name displayed
–   Phi - diameter displayed as ø, followed by material name, e.g. "ø 160 PVC"
–   Dn - diameter designated as Dn, followed by material name, e.g. "Dn 150 steel"
–   (material name) - diameter follows material name without any designation, e.g. "PVC 400".

Line type
Compound field. Type of line representing the junction section. You can change the style, colour and thickness of the line.

Line th.
Numerical field. Thickness of line. You can change the thickness of the line with which the junction section is drawn.

5.7.2. Junction label

Component used to describe sections of water pipeline junctions or fittings on the water pipeline.

Button on the toolbar:

Junction label data:

Text
Text field. Text displayed as description of component, specified by user.

Font
Compound field. Font used in the component label. You can select the type, style, size, colour and additional effects of font.

Height [mm]
Numerical field. Height of font in the drawing. Irrespective of font size chosen, you can freely adjust its size in the drawing without having to open the "Font" options.

Line type
Compound field. Type of line representing the junction section. You can change the style, colour and thickness of the line.

5.7.3. Appurtenances in junction sections

To make the drawings of water pipeline sections complete, appurtenances should be appended:

 – valve

 – expansion joint

 – angle valve

 – reducing adapter

 – flange connection

 – underground hydrant

 – surface hydrant

Diameter and material label is an additional component represented by the button: .

Connection type (flange / no flange) can be specified for valves, expansion joints and reducing adapters.

6. CALCULATIONS AND RESULTS

6.1. Invoking data diagnostic and calculations

After completing editing diagram of installation and appending data, you can proceed to calculations.

♦ To invoke calculations click on the 🖩 button on the "Program" toolbar or select "File / Calculations..." (**F10**).

Data diagnostics are run and if there are no errors in data, proper calculations can be initiated by clicking the "Next" button (you can also use **Ctrl+Enter** keys). A set of tabs is displayed for specifying calculation options. The last tab "Results" contains tables of calculation results. Actually the calculations are finally done when this tab is activated.

To disable calculations after displaying list of diagnostic messages, click the "Back" button or press the **Esc** key.

6.2. Data diagnostics

6.2.1. Types and syntax of messages

There are several windows and lists that display messages regarding the various stages of editing and calculating the heating system. All of them are uniform with respect to syntax and the manner of finding components to which these messages pertain.

The syntax of the message is as follows:

[Message_type] Component_to_which_message_refers Message_text

There are following types of messages:

Errors
These are the most important messages, they are displayed in capital letters. All errors that occur in the course of creating a project must be rectified; otherwise it will not be possible to obtain all results for the project.

Warnings
These messages are of lesser importance than errors. They are displayed in small letters and they do not block further calculations or display of results. Warnings are usually associated with data or results that are presumably incorrect and should be checked.

Hints
The purpose of these messages is only to remind you of or draw your attention to some data or results. Like warnings, hints do not block further calculations or display of results and are also displayed in small letters.

Connections

These messages are warnings, which may appear after calculations are finished. They apply to automatic sizing of connections in the system and describe cases, where no connection or junction has been sized.

You can display information on the errors in the windows that show messages. To do this click on the error and press the **F1** key, or press the right mouse button and from the pop-up menu select "More info on message".

A more detailed description of diagnostic messages is given in an appendix at the back of this manual.

6.2.2. Checking connections

Before invoking calculations and the thorough data diagnostics associated with that, it is advisable to check the connections. This consists in making sure that there are no unconnected components. If there are connection errors in the system, they will be indicated in the diagnostics results window displayed after invoking calculations, so it is more convenient to make the necessary corrections before invoking calculations.

♦ To start connection checking select "Component data / Check connections" (**Shift+F2**).

A window is opened with a list of messages on connection errors found in the system:
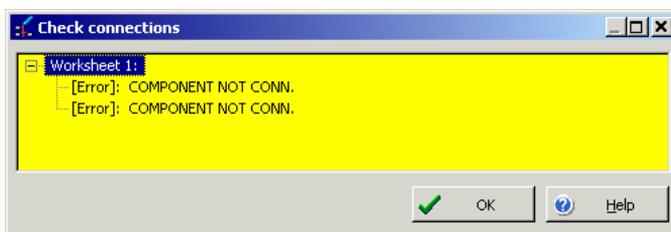
The structure of this window, types of messages that appear there and the manner of finding components associated with the errors are identical to that of the diagnostics window and are described below. If the system is completely correct, the window is empty.

Data editing is not possible when the connection diagnostics window is displayed – you have to close it by clicking on the "OK" button. Information on erors is not lost – it will be displayed in the error list of the graphical editor.

6.2.3. Data diagnostics

Always after invoking calculations a data diagnostics run is made and the diagnostics results window is displayed. The diagnostics window does not appear if there are no messages to display – then the first calculations tab is automatically selected.
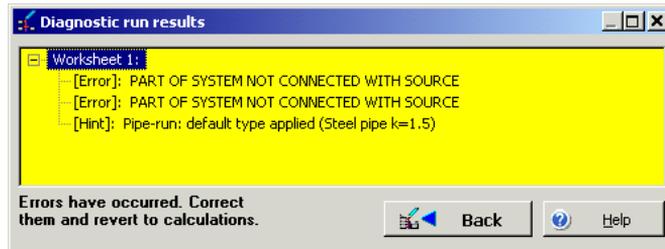
The window displays messages on incorrect or suspected data found during the diagnostic run. If connection errors are found, appropriate messages are displayed and other data are not checked. These errors must be corrected and calculations should be invoked again.

The types and syntax of messages and the manner of finding components associated with these messages is described in the others subsections.

After analysing the diagnostics messages you can go back to editing and correcting data (press the "Back" button) or proceed to calculations (press the "Next" button). If errors occur, you cannot carry on with the calculations: an appropriate message is displayed in the lower left corner of the window and the "Next" button is not available.

If you go back to editing all messages that were displayed in the diagnostics window are transferred to the error list (see section 7.3). You can then easily find and rectify all irregularities in the data.

6.2.4. Finding the component or field to which a message applies

When browsing the message list in the graphical editor the mouse pointer positioned on the message takes on the form of a "grabber hand". When you click on the message, the component or field to which the message applies is marked and highlighted in yellow.

! *Apart from being highlighted in yellow, the component to which the message applies is marked (selected) and its data appear in the data table (if it is displayed).*

This way you can easily find the cause of errors or data associated with the warnings or hints.

You can also obtain more information on the message. Mark the selected message and press the **F1** key or press the right mouse button to display the pop-up menu and select the appropriate item in the menu.

In the pop-up menu of the message list that appears after pressing the right mouse button there is also a function for marking all components where the given error has occurred. This function facilitates correction of the same type of error, which may be rectified in a group of components simultaneously – data of all marked components is displayed in the data table where they can be modified at once (see section 4.9.5).

Calculations and results

An example of an error that can be corrected in this way is the "pipe type not specified" error.

The list of messages in a project which consists of many worksheets may have a tree structure (when messages apply to several worksheets):
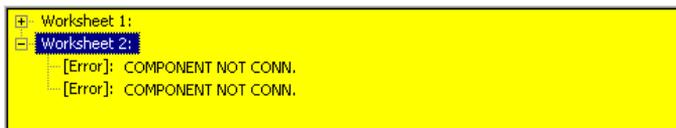
In the example above the list of messages for the "Worksheet 1" worksheet is collapsed – this is indicated by the plus symbol in the square to the left of worksheet name. The "Worksheet 2" worksheet has a minus and messages for this worksheet are displayed.

♦ To expand / collapse the list of messages that apply to the given worksheet, click on the square to the left of worksheet name or double click on the worksheet name in the error list.

When messages are marked, worksheets are automatically changed so that the component to which the marked message applies is shown.

6.3. Calculation options

When you click on the "Next" button in the diagnostic messages window (which is possible only if no errors were found), the window is closed and a set of tabs with calculation options appears – values that have substantial effect on calculations and results, which in principle form an extension of the general data of the project, but for the user's convenience have been moved out of the graphical editor to enable fast execution of several calculation variants. During the first attempt to run the calculations of a new project the values displayed in these fields are taken from the AUTOREAD file (unless the project is a modification of another project). During subsequent calculations these values are taken from the project file. Some options associated with pipe catalogues (e.g. maximum velocities) are taken for the first calculations from the pipe catalogue.

Fast calculations

It is possible to run the so-called "fast calculations". In this case calculations are performed without going through all the calculation options tabs (default values or values specified before are adopted).

! *Running complete calculations with calculation options specified is recommended in the first stage of the project.*

This function is useful when you want to refresh results displayed in the graphical editor (without viewing results in tables), or when you make slight corrections in the

project and want it to be recalculated. A message on the course of calculations is displayed and then the results in the graphical editor are refreshed.

6.4. Calculation options

6.4.1. Common options

Calculation options that are common to all networks are minimum distances in vertical plane at which the networks under design may pass each other or networks not calculated by the program and placed on the plan (gas supply, heating, power networks, etc.). These values are taken into account when determining vertical routes (profiles) of the designed networks – even if the profile drawing is not generated. The distances between networks (gas supply, heating, power, etc.) not calculated by the program are not controlled in any way.

6.4.2. Sewerage network calculation options

You can specify the following values for each pipe family used in the project:
– minimum sized diameter of network sections under design
– multiplier for maximum velocities, which are specified in pipe catalogues for each individual diameter – hence the need to use a multiplier instead of entering one value. However most pipe catalogues specify the same maximum velocities for all diameters. You can specify the multiplier depending on the location of the section within the network: house drain, branch, main line
– minimum cover depth in relation to ground level
– maximum cover depth in relation to ground level
– maximum fill ratio (percentage) of ducts under design (gravity sewerage network only)
– minimum gradient of sized ducts
– maximum specific flow resistance along length (pressure sewerage network only).

These pipe sizing options are separate for gravity sewerage and pressure sewerage networks. There is also differentiation between components depending on their location: house drain, branch or main line. You can specify separate sets of pipe sizing data for each fragment of network.

In addition, the tab also has fields for:
– resizing diameters
– retaining diameters specified by the user when sizing
– specifying the minimum angle at which sections of pressure sewerage network may be connected.

6.4.3. Water supply network calculation options

You can specify the following values for each pipe family used in the project:
– minimum sized diameter of network sections under design

- multiplier for maximum velocities, which are specified in pipe catalogues for each individual diameter – hence the need to use a multiplier instead of entering one value. However most pipe catalogues specify the same maximum velocities for all diameters. You can specify the multiplier depending on the location of the section within the network: house drain, branch, main line
- maximum specific flow resistance along length
- minimum cover depth in relation to ground level
- maximum cover depth in relation to ground level
- maximum gradient.

There is also differentiation between components depending on their location: house drain, branch or main line. You can specify separate sets of pipe sizing data for each fragment of network.

In addition, the tab also has fields for:
- resizing diameters
- retaining diameters specified by the user when sizing
specifying the minimum angle at which sections of water supply network may be connected.

6.5. Calculation results

Calculations are done when you move to the "Results" tab. If at least one error, warning or hint occurs, a window with an apprpriate message is displayed. Immediately after making calculations you can view all tables with results. To see the results in the drawings you have to click on the "Editor" button in the upper part of the screen. The other buttons at the top of the screen:
- "Print" – display the print preview and print all tables of results,
- "Cancel" – return to the graphical editor without completing calculations (e.g. if immediately after viewing calculation results of radiant heating we do not want to carry on with calculations),
- "Back" – go back to previous tab (calculation option).

6.6. Description of tables of results

All tables contain results for individual "Map" type worksheets (as data for calculations are derived only from these sheets). Each table, except for the bill of materials, begins with the name of the sheet, and then follow the results of calculations.

Tables of results for sections and junctions are also available after going back from calculations to the graphical editor.

Each line of the table has a button . After clicking this button the component to which the line in the table is assigned is indicated in the drawing.

6.6.1. Table of sewerage network sections

The table of sewerage network sections provides the following information:

Designation	symbols of starting and end junctions of section	
Elevation of start	elevation of section start	[m]
Elevation of end	elevation of section end	[m]
Flow rate	design flow rate of sewage in section	[dm^3/s]
Length	length of section	[m]
Gradient	geometric gradient of section	[‰]
Diameter	diameter and thickness of pipe wall in section	[mm]
Pipe type	pipe family used to construct the section	
Fill ratio	fill ratio of section at design flow rate	[%]
Velocity	sewage velocity at design flow rate	[m/s]
Flow rate 100%	flow rate at full fill ratio	[dm^3/s]
Velocity 100%	sewage velocity at full fill ratio	[m/s]
Cat. no.	catalogue number of pipe	
Rough.	absolute roughness of pipe	[mm]
Cov. depth start	cover depth of start of section	[m]
Cov. depth end	cover depth of end of section	[m]
No. collisions	number of collisions of the section with other networks	

If there are collisions between various systems in the network under design, then an auxiliary table of collisions is displayed below the section line. This table provides the following information:

Collision with network	name of network with which collision occurs	
Distance	distance between projection of collision point on section and start of section	[m]
Pass distance	the smallest distance at which networks pass each other	[m]

6.6.2. Table of sewerage network junctions

The table of sewerage network junctions provides the following information:

Designation	symbol of junction	
Inflow	design flow rate of sewage reaching network in the junction	[dm^3/s]
Elevation of area des.	elevation of area under design at junction point	[m]
Elevation of exist. area	elevation of existing area at junction point	[m]
Elevation manh.bot.	elevation of bottom of manhole in the junction	[m]
Manhole type	name of manhole type sized for the junction	

Junctions with manholes selected from catalogue will have an auxiliary table listing components of the manhole.

6.6.3. *Table of general results of water supply network*

The table of general results of water supply network provides general information on network size:

No. of sources	number of water sources
No. of receptors	number of receptors (consumers) in the network under design
No. of intermediate junctions	number of intermediate junctions
No. of network sections	number of all sections in the network

For each junction that is a source, the following is provided:

Flow rate at source	supply capacity of source	[dm^3/s]
Available pressure	water pressure, calculated or specified	[kPa]
Critical receptor	designation of critical receptor	
Route l. to crit. rec.	length of network route to critical receptor	[m]
Pressure drop along crit. route	water pressure drop along route to critical receptor under design conditions	[kPa]
Hydrostatic head	hydrostatic head resulting from elevation difference between source and critical receptor	[kPa]
Pressure upstream of crit. receptor	water pressure upstream of critical receptor	[kPa]

The values presented may be used to assess the impact of each item on the sized required pressure of water at source.

6.6.4. *Table of water supply network sections*

The table of water supply network sections provides the following information:

Designation	symbols of starting and end junctions of section	
Elevation of start	elevation of section start	[m]
Elevation of end	elevation of section end	[m]
Flow rate	design flow rate of water in section	[dm^3/s]
Length	length of section	[m]
Diameter	diameter and thickness of pipe wall in section	[mm]
Pipe type	pipe family used to construct the section	
Velocity	water velocity at design flow rate	[m/s]
R	pressure drop along section	[Pa/m]
Total loss	total pressure drop along section, including local resistance and fittings	[m. SW]
Cat. no.	catalogue number of pipe	
Rough.	absolute roughness of pipe	[mm]
Cov. depth start	cover depth of start of section	[m]
Cov. depth end	cover depth of end of section	[m]
No. collisions	number of collisions of the section with other networks	

If there are collisions between various systems in the network under design, then an auxiliary table of collisions is displayed below the section line. This table provides the following information:

Collision with network	name of network with which collision occurs	
Distance	distance between projection of collision point on section and start of section	[m]
Pass distance	the smallest distance at which networks pass each other	[m]

6.6.5. Table of water supply network junctions

The table of water supply network junctions provides the following information:

Designation	symbols of starting and end junctions of section	
Outflow	design flow rate of water taken out of the network in the junction	[dm^3/s]
Pressure at source	water pressure at the junction	[kPa]
Elevation of area des.	elevation of area under design at junction point	[m]
Elevation of exist. area	elevation of existing area at junction point	[m]
Elevation manh.bot.	elevation of bottom of manhole or pipeline	[m]

Junctions that only connect pipe-runs will have zero in the "Ouflow" column, whereas junctions that discharge into a reservoir, treatment plant or other network will have a negative value in this column.

6.6.6. Bill of materials

The bill of material tables are split into tables of individual component types located on separate tabs.

The "Pipes" tab

The list of pipes is structured with respect to catalogues and pipe families used in the project. List of pipes of one family includes the name (header) of the catalogue with that family and a table that provides the following information:

Product	full name of pipe type
Size	nominal or outer diameter of pipe
Catalogue code	catalogue number or code of the diameter provided by manufacturer
Quantity	length specified in units given below
Unit	unit of length

The "Manhole" tab

The list of manhole components includes the name (header) of the catalogue with manhole data and a table structured in the same manner as that of the pipe table:

Product	full name of manhole component
Size	extended designation or repeated name of component
Catalogue code	catalogue number or code of the component provided by manufacturer
Quantity	number of units given below
Unit	unit of measure

6.6.7. Finding components

When viewing tables of results you may want to find some components that meet specific criteria, e.g. pipe-runs of specified flow rate or water velocity. If the project is large, this may not be easy. You can then use the search function, which is available in the pop-up menu of the table of results displayed upon pressing the right mouse button (**Ctrl+F**).

Upon selecting "Search" in the menu, a window such as the one below is opened:
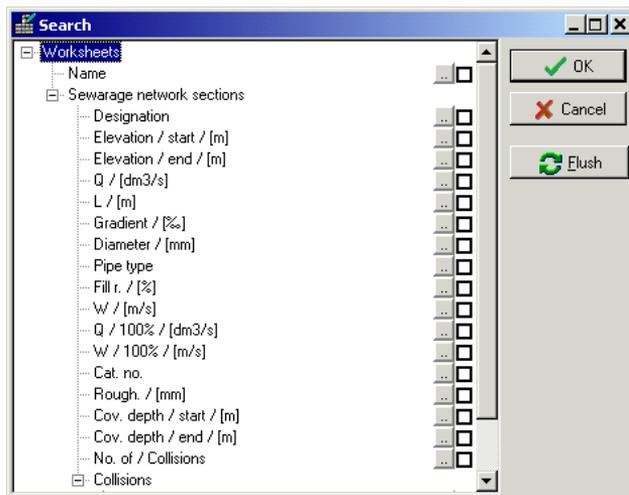
After marking the required item, clicking on on the right-hand side of the window opens a window for specifying details of selection criteria:
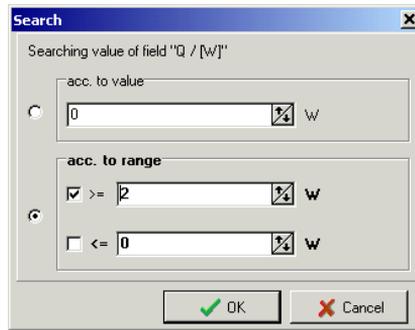
After confirming the data click the "OK" button in the "Search" window. As a result, all fields in the table of results that meet the search criterion will be highlighted. After defining and confirming another criterion, both the previously highlighted fields as well as those, which satisfy the new criterion will be marked. Thus the multi-parameter search is performed according to the principle of logical "or".

To start a new search with new criteria, click on the "Flush" button and then enter new search criteria as described above.

You can also use the fast find function, which is activated by pressing **Shift+Ctrl+F**. This option enables finding a component in a given column of results (e.g. with a specified symbol, catalogue code). Double click on any field in the appropriate column and press the keys mentioned – a search window is opened where you can enter the value you are looking for.
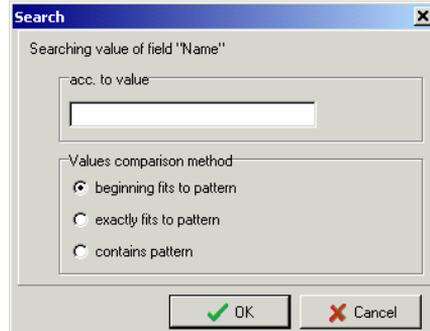
6.6.8. Configuring table appearance

Pop-up menu of the right mouse button also enables adjusting the appearance of tables ("Display configuration") to our needs. This is an example of a configuration:
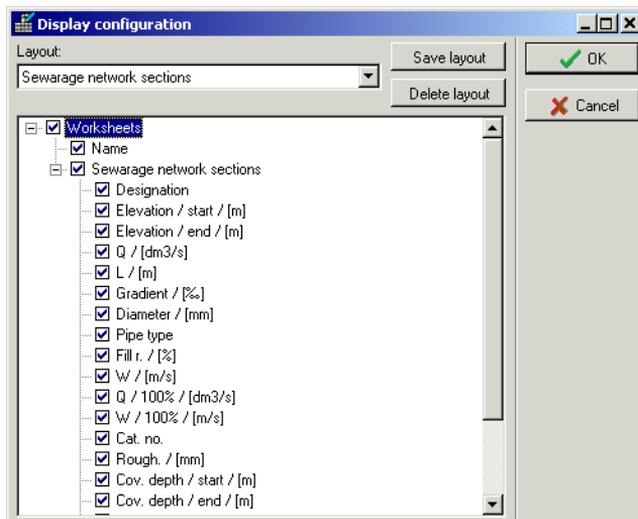
To display/hide particular columns click on the check box ☑ beside the column name. By default all boxes are checked and all columns are displayed.

If you often use the same settings in particular tabs, you can save them in a layout. To do this, mark the columns to be displayed on screen and click on the "Save layout" button. A window is displayed where you should enter the name of the new layout and click OK.

To delete a layout, mark it and click on the "Delete layout" button.

You can configure the appearance and search function in all secondary tabs, except "General results" and "Bill of materials". In the "General results" tab you can only configure the appearance of tables, while in the "Bill of materials" tab there is no possibility of configuring.

6.7. List of messages pertaining to calculation results

The list of all messages pertaining to the course of calculations – errors, warnings and hints, is displayed automatically under the tables of results. Clicking on a line in the list sets the table of results in such manner, that the project component to which the error is assigned to, is displayed. The active tab is also changed, if necessary.

The meaning of individual messages, and the way to avoid situations described in messages during subsequent calculations, is described in the appendix at the back of this manual.

6.8. Printing result tables and exporting to a spreadsheet

If calculations are complete, you may print results tables on a printer or export them to an MS Excel® spreadsheet.

6.8.1. Printing calculation results on a printer

♦ To print out the results:

1. Click the "Print" button. The print preview and print options window is displayed.
1. Determine the layout (page orientation, components to be included in the printout) and the style of the printout in the "Layout" and "Style" tabs on the right side of the window.
2. Set margins for the printout page on the "Print" tab.
3. Make sure that all the above settings are correct and in accordance with your intentions.
4. Determine general settings on the "Print" tab, such as printer type, range of pages to be printed, number of copies, page numbering, etc.
5. Click the "Print" button in the main configuration window to start printing.

Print preview and print options window

After clicking "Print" the print preview and print options window is displayed:
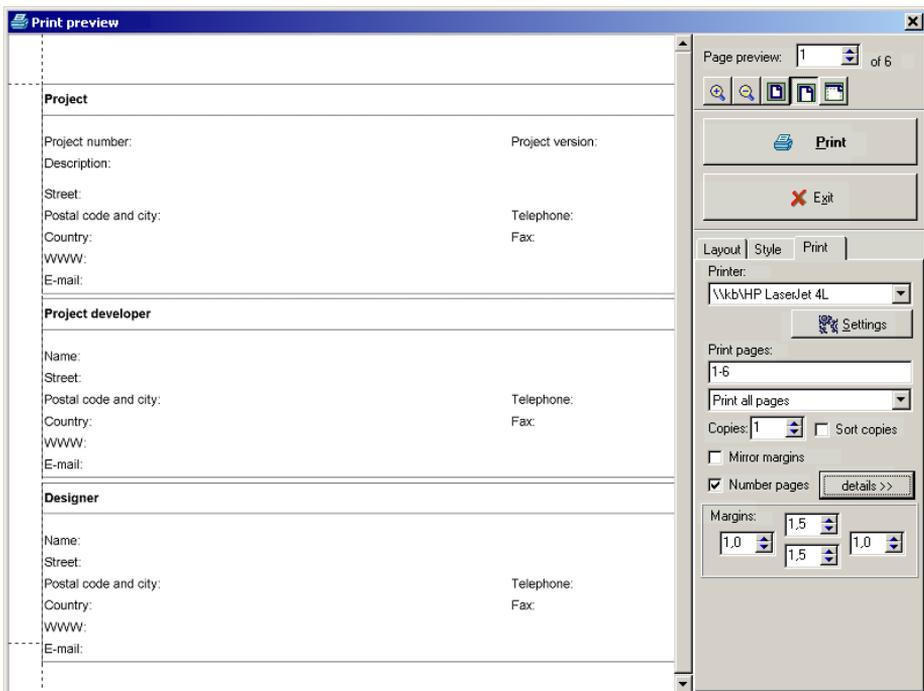
The print preview and print options window consists of two parts and enables printout configuration in a broad scope. The left side of the window is the print preview part. The displayed print preview reflects the current settings of print style and layout.

If only part of the page is displayed in the print preview, you can display other portions of the page by moving it using sliders. Alternatively, you can move the displayed part of the page with the "grabber hand" – when you position the mouse pointer in the print preview part of the window, it takes on the appearance of the "grabber hand" with which you can drag the displayed page of the printout.

The bar on the right side is used for setting print options. The bar has three tabs: "Layout", "Style" and "Print". The components of this bar are described below.

Change page displayed and preview scale

At the top of the print configuration tab there is a set of buttons and fields for changing the print preview scale:

The field "Preview pages ... of ..." enables previewing the next / previous page of printout and informs about the current total number of pages. The buttons below this field, from left to right, are used to:
- – increase preview scale (zoom in),
- – decrease preview scale (zoom out),
- – specify scale enabling preview of entire page,
- – specify scale to fit the width of the page, including margins, to the width of preview area,
- – specify scale to fit the width of the page, excluding margins, to the width of preview area.

Printing

The "Print" button below is used to start printing.

Using this button effects in printing of the results in a previously declared scope and print style.

Revert to calculations

The "Exit" button below is used to complete the print settings and go back to the display of the results tables and calculation options:

The lower part of the print settings bar includes three tabs: "Layout", "Style" and "Print".

6.8.2. General printout settings:

After opening the print preview and print settings window, the "Print" tab is selected, where you can change the general printout settings.

The "details" button opens and closes additional fields for configuring page numbering. These additional fields are displayed in the figure below:
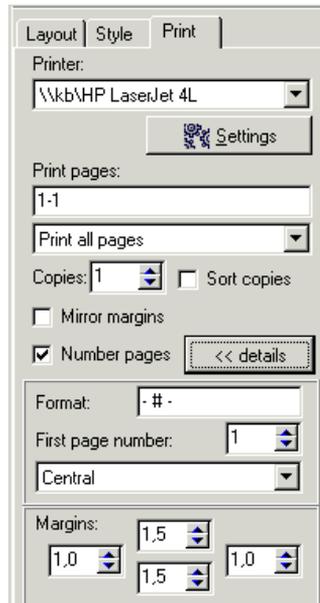
Change margins

The fields in the lower part of the "Print" tab are used to set the widths of margins: left, upper, right and lower, respectively:
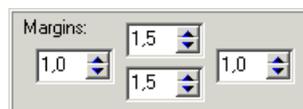
6.8.3. *Printout layouts – specify the range of results printed:*

In the "Layout" tab you can specify exactly which calculation results tables and fields you want to print:
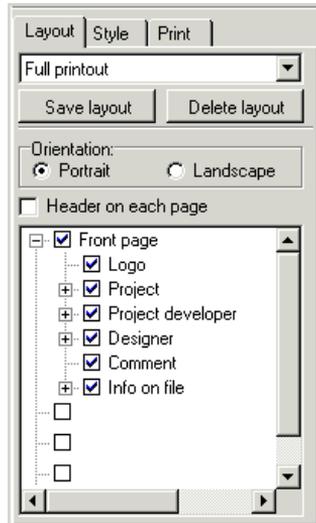
In the upper part of the tab is a field where you can select one of the predefined printout layouts. In the list in the lower part you can review or modify the selected layout. The changes made are introduced in real time in the left part of the screen.

After modifying any of the layouts you can save it under a new name to use not only for the current printout, but also in future. To do this click on the "Save layout" button and enter the name of the new layout. Existing layouts may be deleted by clicking on the "Delete layout" button. Standard layouts cannot be deleted.

In the middle of the tab is a field for changing the orientation of the printout (page).

! *The landscape orientation often produces good results when printing result tables. The rows of the table fit into the printed page. Another way of avoiding the splitting of table rows in the printout is to specify narrower margins, provided that the printer allows you to do that.*

6.8.4. Printout styles – specify colours, fonts, etc.:

In the "Style" tab you can specify the size and type of font used and the colour of background and letters, separately for individual types of fields in the result tables:
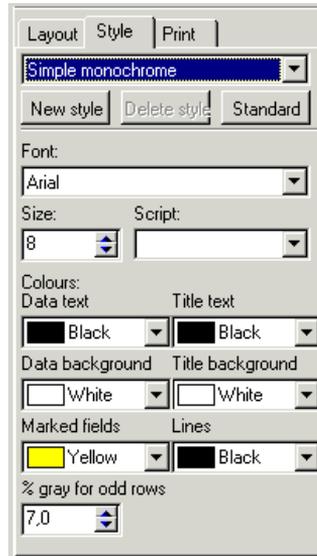
As was the case with printout layouts (see above), there is also a set of predefined printout styles, and new styles created by the user may also be saved. Predefined styles cannot be deleted.

To improve legibility of the results, every second line is grayish. It is possible to declare the grayness ratio for the odd lines in per cents.

6.8.5. Exporting calculation results to an MS Excel® spreadsheet

♦ To export calculation results to a spreadsheet:

1. Click on the arrow pointing down located to the right of the [🖨 Print ▼] button in the upper part of the calculations window. A menu appears with items "Print" and "Export to MS-Excel".
2. Select the second item – "Export to MS-Excel". A window is displayed where you can enter the name of the target file with an .xls extension.
3. Click on the "Save" button in this window. The file is saved to disk.

Check the created file by opening it in MS Excel®.

ization

7. RESULTS IN THE GRAPHICAL EDITOR

Calculation results are saved to the project file and most of them may be printed or displayed in hints in the graphical editor. The calcualtion error list shown under tables of results and tables of results are also available in the graphical editor.

The results which can be shown on the system diagram and printed, may be grouped in two categories:
- results of calculations of pipe-runs (system segments) – not displayed by default; a "Pipe-run label" component is required on each pipe-run the results of which are to be displayed
- results of calculations of receptors, fittings and possibly system junctions – displayed automatically; their configuration is required in the "Options/component appearance" menu. Empty sets of calculation results may be specified for some components by default.

Pipe-runs are not described automatically on drawings, as this could obscure other components in the drawing. The user can place labels on every pipe-run. It is also possible to place labels on a number of pipe-runs simultaneously. Labels can be placed when editing data (prior to calculations) or after completing calculations and reverting to the graphical editor.

♦ To place labels on multiple pipe-runs simultaneously:
- mark the appropriate pipe-runs using the functions for marking multiple components (see section: 4.8.3 – 4.8.6),
- double click one of the pipe-run labels in the "Graphics" tab
- labels are placed in the middle of pipe-run, and if there are many of them, it may be necessary to move some of them to make the drawing clearer.

In the "Profile" type worksheet with profiles created on the basis of results of calculations of network diagram, such results as the elevations of the start and end of sections and pipeline diameters are displayed automatically in the respective fields of the profiles.

7.1. Looking up component results in hint bubbles

Grafical editor enables displaying of component data and results for individual components in hint bubbles.

♦ To display a hint bubble with component data and results position the mouse pointer over the component and wait for a while.

If the system has been calculated, basic component data and some calculation results for that component are displayed. If the system has not been calculated yet, only the basic data are displayed.
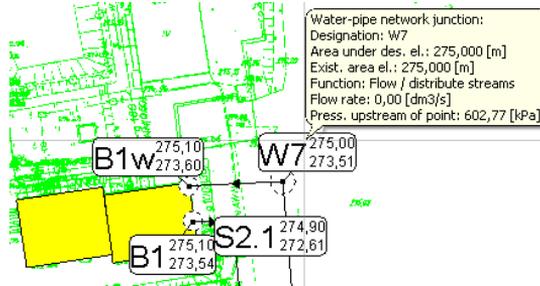
7.2. Printing, print options and printer settings

7.2.1. Printing and print options

After executing calculations, you can print the project on any printer installed in a Windows environment. Before printing you should set print options and establish paper cutting lines, if the project will not fit into one sheet.

♦   To print the project:
1. Select the "Print" layer tab in the lower right corner of the screen. The print preview mode is enabled.
2. Specify print parameters and options in the fields of the data table (the data table must be displayed).
3. Check positions of paper cutting lines; adjust them by dragging the lines with the mouse, if necessary.
4. Select "File / Print" (**Ctrl+P**, 🖨). The project is printed.

After selecting the „Print" tab, the print preview mode is enabled. The connection points, auxiliary points disappear, all lines take on the appropriate thickness. Print configuration fields appear in the data table:

Printer
The printer on which the project is to be printed.

Settings ...
Used to specify printer settings. The button with dots on the right side of the field opens a window for specifying printer settings.

Scale
Printout scale (in relation to real dimensions of the project).

Overlap [cm]
Width of overlap. Description of tabs and an example of page layout are given further in this section.

Orientation
You can change paper orientation from portrait to landscape and vice versa.

112

Left margin [cm], Right margin [cm], Top margin [cm], Bottom margin [cm]
Margin settings on the printout.

Paper length [mm]
Paper length. Paper dimension parallel to horizontal axis of project is meant as the paper length.

Paper width [mm]
Paper width. Paper dimension parallel to vertical axis of project is meant as the paper width.

Colour
Here you can switch colour printing on and off on colour printers (e.g. to spare colour ink or to make a quick test printout). If the Colour field is set to "No", then a monochrome printout will be made no matter what type of printer is used. The disabling of colour may also be useful when using monochrome printers –light colours are sometimes not printed at all on such printers.

Continuous paper
Cont. paper page length
Here you specify the settings for printing on continuous paper, (see section 7.2.3).

No. of pages
Indicates how many pages the project will take up with the current print options.

! NOTE: in some cases some of the pages may be empty.

While in print preview mode, purple and green lines are displayed on the screen together with a scissors symbol. These lines indicate paper cutting lines. Purple lines indicate paper cutting lines on the left, lower side, and the green lines indicate overlaps that enable pasting together sheets to form complete printout of a project. Overlap is a part of the printout that appears on two neighbouring pages. When pasting two pages together, one page should cover the other over the overlap width. Two short lines above and below the printout area indicate the place which the page should adjoin. One of the neighbouring pages comes on top (it may be any page of the two), the other comes underneath. You can inactivate the overlap in the data table by entering 0 (zero) in the "Overlap" field.

After positioning the mouse pointer on one of the purple lines, it takes on a shape that suggest the ability to move the paper cutting line. You can "grab" the line with the mouse and move it, so that, for instance, the paper is not cut in the middle of a room label. However, you can never set the page size larger than the printer page; you can only make it smaller.
You can revert to the default setting. To do this, select "File / Reset page layout". Also, changing the printer, scale, overlaps or margins resets the default page setting.

♦ To disable the print preview mode, click another tab in the lower right corner of the screen.

7.2.2. Printer settings

♦ To display the printer settings window:
    1. Select "File / Printer settings"
or
    2. Select the "Print" tab in the lower right corner of the screen and on the button on the right side of the "Settings ..." field

    The appearance of the window that is displayed depends on particular printer. You can always select page size and printout orientation.

7.2.3. Printing on continuous paper

    The graphical editor enables printing on continuous paper on printers that support such printing. This depends not only on the printer construction, but on the printer driver (software that controls printer operation).

! The "Continuous paper" field indicates that the continuous paper band is understood by the program (and printer driver) as a sequence of paper sheets joined together (e.g. a sequence of A4 size pages) and not as one long page. Interpretation of paper format by the program does not depend on the physical form of paper (a sequence of joined pages or continuous paper from a roll).

♦ To print on continuous papers first try to set a long page in the printer driver. Drivers of some printers enable specifying user paper size and long pages for this paper size (e.g. 5 m). To do this:

    1. Select the "Print" tab,
    2. Select the required printer in the first field of the data table,
    3. Click on the button with three dots on the right side of the second field,
    4. Printer settings window is displayed. Its appearance depends on printer type,
    5. Find the paper size field. Check if user paper setting is available,
    6. If long page setting is not available – see the procedure described below,
    7. Select user paper and try to set maximum paper length (or width – depends on printer driver). Enter a high value (e.g. 10 m); the driver will inform you of the maximum value possible,
    8. Set paper orientation to "Landscape",
    9. Ensure that "Continuous paper" field is set to "No",
    10. Print preview (printout bands) should be displayed on the screen,
    11. Execute the printing.

    If user paper setting is not available, try to print using other printer settings. To do this:

    1. Select the "Print" tab,
    2. Select the required printer in the first field of the data table,
    3. Click on the button with three dots on the right side of the second field,
    4. Printer settings window is displayed. Its appearance depends on printer type,
    5. Depending on the type of printer, try to:

- find that printer setting which says that continuous paper is installed in the printer. In HP DeskJet 1120 printer, for instance, such setting is named "Banner",
- set paper size that has "Fanfold" in its name. This paper size informs that internal printing margin should not be implemented (e.g. Epson 2150),
- if none of the above produces expected effects – set normal paper,

6. Before finishing printer settings note down the length and width in millimetres of the selected paper size,
7. If after closing the printer settings window the page layout is not appropriate (pages are very small, for instance), enter the noted down lengths of selected paper in the "Paper length" and "Paper width" fields of the data table,
8. Set paper orientation to "Landscape",
9. Set "Yes" in the "Continuous paper" field in the data table,
10. Enter page length expected by the Windows driver in the "Cont. paper page length" field. This value has to be determined experimentally. Usually it is equal to the TOTAL LENGTH (including margins) OF THE SELECTED PAPER TYPE – for instance, for an A4 page in landscape orientation this will be 297 mm. Sometimes you have to enter here the same value as in the "Paper length" field,
11. Print preview should be displayed on the screen. The vertical paper cutting lines are in this case lines of virtual cutting transferred to the printer driver and they cannot be moved. Horizontal paper cutting lines may be moved,
12. Execute the printing.

The procedure described above may not be applicable to some printers. If printing on continuous paper is not possible, this is usually because of the printer driver. You can try to find an updated version of the printer driver – for instance, at the printer manufacturer's Web site, by entering in the Web browser the address 'www.<manufacturer_name>.com', for instance www.hp.com or www.epson.com. Of course, there is no guarantee that an updated driver will solve the problem.

Printer settings can be saved to the AUTOREAD file, which is a template for creating new files. You'll find more on the AUTOREAD file in section 8.13.

7.3. Error list in graphical editor

The error list after reverting to the graphical editor, can be switched on / off select "View / Show/Hide error list" (**F8**). The contents of the error list depend on the last operation performed:

- if calculations of the current project have not been invoked before, the error list is empty,
- if the last operation performed was the checking of connections, the error list contains connection error messages (or is empty if no errors occur),
- if calculations were invoked and after diagnostic run editor was called again in order to correct data, the error list contains diagnostic messages,
- if calculations were made, the error list contains error, warning and hint messages created during the calculation process.

Information on types and syntax of messages in the error list is given in section 6.2.1, while section 6.2.4 provides hints on using the error list: description of its

structure, method of finding components associated with messages and of displaying additional information on selected messages.

Upon pressing the right mouse button on the error list, a pop-up menu is displayed with the following options:

- find component – find the component associated with the message,
- mark all components where this error has occurred,
- mark all components of the type ... where this error has occurred,
- mark all components where this error has occurred (entire file),
- additional information about the message,
- don't show this message again.

! *Initially all messages are displayed in bold font. Each message, after finding the component associated with it, is marked as "checked" and is then displayed in the error list in normal font. However, you can repeat the procedure of finding the component to which a "checked" message applies (menu opened by pressing the right mouse button). The font is changed as soon as the component is found, therefore an error indicated in plain font should not be treated as a rectified error.*

Information on the meaning of individual errors is given at the back of this manual.

8. ADVANCED FUNCTIONS

8.1. Sending and opening projects with the use of electronic mail

The program supports transfer of projects by electronic mail. This possibility, however, depends on your mail software. If you use Microsoft® software, you have to install Outlook® 97, 98 or 2000. In case of Outlook Express® the function is not enabled in the standard configuration.

♦ To send a project by e-mail using the built-in function:

1. Select "File / Send to / Send to new addressee ..." ()or "File / Send to / Send to addressee from address book ..." (),
2. If the project has not been saved to disk under a name, you'll be prompted to specify the name.
3. If you selected "File / Send to / Send to addressee from address book ...", a list of addressees is displayed.
4. A new mail message is created with data file attached. You have to specify the title, body of the message and list of addressees, if necessary.
5. Send the message.

If the mail sending support is not compatible with your mail software, you have to create the message and attach the file manually.

♦ To send a project by e-mail without using the built-in function:
1. Save the project to disk.
2. Launch your mail application.
3. Create a new message.
4. Fill in the list of addressees, title and body of message.
5. Select command for attaching an additional file (usually Insert / Attachment file)
6. A file selection window pops up. Select the saved project file – project files are saved to folder selected during program installation (by default the folder My documents \ <package_name> \ stores the projects and data files)
7. Send the message.

A project received by electronic mail should be saved in the program data folder. To do this:
1. Open the received message with the project attached in your mail application. An icon representing the attached project should be displayed in the body of the message or at the bottom of the window.
2. Click the right mouse button on the project icon. A pop-up menu is displayed.
3. Select "Save as ..." from the pop-up menu.
4. Navigate in the window that shows up to the folder with project data – by default the folder My documents \ <package_name> \ stores the projects and data files
5. Save the file.

After completing the operations listed above you can open the project using the usual command "Open project".

8.2. Configuring component appearance

Net-san enables configuring appearance of components of both "Map" type and "Profile" type sheets. To configure component appearance select "Options / Component appearance". The "Component appearance" window can also be opened in the "General data" window.

8.2.1. Configuring network component appearance

In a "Map" type sheet you can configure the appearance of various types of network sections, of network junction labels and general appearance settings during editing.

In the "Section of network" tab you can configure the font in section labels and label styles, style and thickness of lines in the different types of network.

When you click on an item in the table on the left side of the window, its properties are displayed on the right side. The fields have the following meanings:

- "Name on profile" – text displayed when the component of given type is a collision on the profile
- "Name in pipe-run label" – text displayed on map in pipe-run label
- "Existing network" and "Network under design" – configure style and colour of line for existing and planned components
- "Min. and max. line th. in dwg." – minimum and maximum thickness of line used to draw the network. The thickness of network sections is drawn in scale, but only within specified limits. If you want network thickness to be drawn in scale, enter different values here (e.g. 0.1 and 10, respectively); enter the same value in both fields to draw the network with a uniform line
- "Arrows" – specify size and colour of arrow for indicating direction of liquid flow in the network under design.

Buttons on the right side of the window enable copying, renaming and deleting network type.

On the "Network junctions" tab you can configure various styles of network junction labels. The program initially provides only one default type of junction label.

To open network junction label configuration window click on the item in the list and then on the "Configure" button. Here you can activate or disable the individual label fields and configure their appearance and mutual arrangement.

8.2.2. Configuring profile appearance

In a "Profile" type sheet you can configure the fonts used to draw the profile and the appearance of the individual components of the profile and general appearance settings during editing.

On the "Labels on profile" you can specify all options concerning the appearance of profile labels.

In the upper part of window you can specify fonts used to describe individual parts of the profile. When you position the mouse pointer on a button, a help bubble is displayed indicating the font represented by the button.

The following options are presented in the middle part of the window:
– "Move app. labels above max. elev." – the value entered here indicates how high will the appurtenance labels be positioned. The position of the point where the appurtenance label starts is calculated as max. elevation of ground plus the value entered here
– "App. labels downward by default" – by checking this box you make the captions in appurtenance labels point downwards (towards the profile table). This property can be specified individually for each separate label
– "Auto labels with upper case letters" – write automatic appurtenance labels using upper case letters
– "Designation circle diameter" – specify the diameter of circle inside which designations of profile sections and manholes are displayed
– "Draw designation circles" – toggle on/off designation circles. When the box is unchecked, profile section designations are displayed just below the table
– "Vertical text" – when this box is checked, designations are displayed as vertical captions. This may be useful with long texts (when this box is checked, disable designation circles)
– "Draw hectometer" – check this box to draw the hectometer
– "Distances 0 - 100 m" - check this box to have distances specified in the table with regard to the hectometer – all distances will be within 0 – 100 m range
– "Over profile" – labels placed above profile
– "Translation over profile" – how far the label is moved above the highest point of profile,
– "Specify distances in hectometres/decametres" – this field indicates whether distances in profile table are specified in hectometres or decametres,
– "Material name on profile" – indicates whether name is short or full,
– "Only vertical text in field "Distance"",
– "Reference level symbol",
– "Long name of reference level" – defines format of long name of reference level,
– "Short name of reference level" – defines format of short name of reference level.

A button in the lower part of the window enables specifying the sequence of fields in the profile table. Individual fields may also activated and disabled.

In the "Profile components" you can define the lines (thickness, colour and style) used to draw individual components of the profile, such as: area under design, existing area, existing and planned manholes, junction sections. In addition to the style and thickness of lines you can specify the following options:
– "Draw area under design" – toggle on/off the drawing of area under design
– "Draw existing area" – toggle on/off the drawing of existing area
– "Draw manholes as closed solid" – check this box to have manholes drawn as a closed solid (with no gap for duct):

as open solid	as closed solid
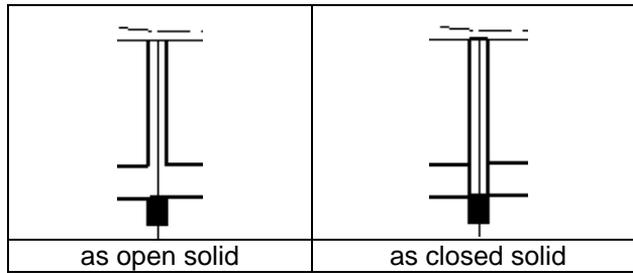
- – "Simplified manhole drawings" – draw manholes in simplified mode (outline of manhole)
- – "Non-standard hatching of landing slab" – use non-standard hatching of landing slab area. Some printer drivers do not feature this option – in such case uncheck this box.

On the "Network sections" tab you can configure the appearance of network sections on profile and colours of collisions. Configuration is done in a manner similar to that in the "Map" type sheet.

In the "Edit" tab you can configure the reference scale and drawing simplifications during editing.

Reference scale:	Simplification scale
500	750

Simplify (always)	Simplify acc. to scale
☐ No circles / arcs	☑ No circles / arcs
☐ No text	☑ No text
☐ No connections	☐ No connections
☐ No field graphics	☑ No field graphics
☐ No drawings and OLE	☐ No drawings and OLE
☑ No compound lines	☑ No compound lines
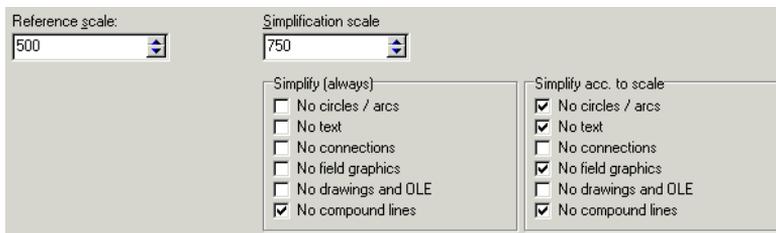
In the upper left corner is the "Reference scale" field. The value in this field determines the scale at which all font heights have the nominal value specified by the user. If printout scale is lower than reference scale (e.g. printout scale 1:100, reference scale 1:50), the fonts on printout will be of lower height than specified by the user. An opposite situation will occur if the printout scale is larger.

We recommend the reference scale for the "Map" type worksheet equal to the scale of the base map. In the profile sheet it is best to stay with the default setting (100).

8.2.3. Component label layout configuration window

The component label layout configuration window can be opened from the component appearance window. It is used for configuring the appearance of component labels in the project. You can toggle on and off the various fields of the label and configure their appearance and arrangement in relation to each other. The two buttons 🔍 🔍 are used to change the viewing scale of the component being configured.
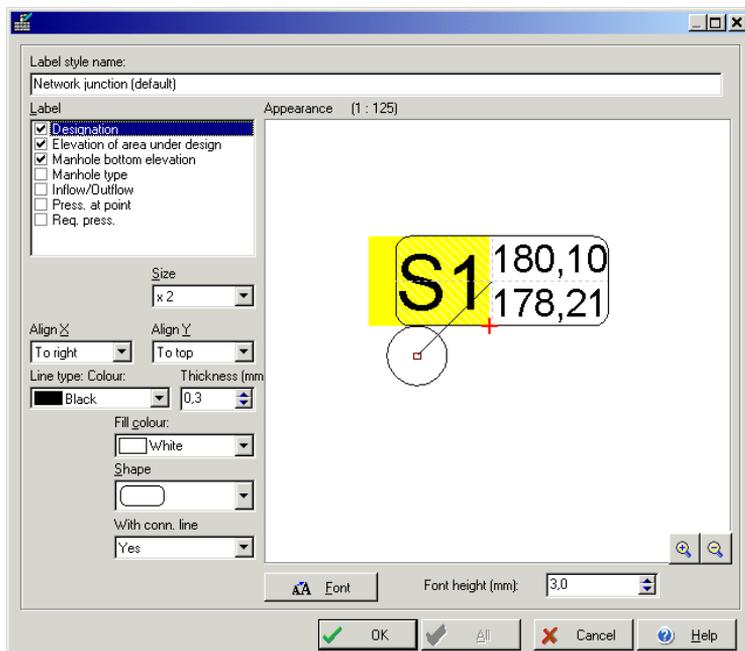
The following features may be specified for each component label:
- – label displayed or hidden (by marking or unmarking on the list in the upper left corner of the window)
- – label size
- – additional elements (e.g. frame, circle, etc.)
- – text justification in relation to position (horizontally and vertically)
- – colour of line used to draw the component
- – thickness of line used to draw the component
- – fill colour
- – text orientation
- – base point
- – component shape
- – line linking the component with its label displayed or not

- – buttons in the lower left corner of the component display area "Increase / decrease component height / length" enable changing the dimensions of the component and checking the manner of mutual displacement of labels when the dimensions are changed.

Moreover, the following can be specified for the entire label:
- – font type and size
- – font heigh – nominal value for the reference scale.

♦ To change the font (simultaneously for all label fields), click on the "Font" button and select a font. You can also specify font size in millimetres.

♦ To change the relative size of font for individual fields select a value in the "Size" field for each label.

The current field may be selected in the list displayed in the upper left corner of the window.

After selecting a field, a red cross and a green circle appear on the graphical symbol of the component. The red cross indicates the point of "attaching" the field, and the green circle determines the so-called base point.

The base point is of significance in case of components of variable height. It indicates the point to which the caption is to be attached when the component size is changed. In effect the red cross indicates the position of the caption in relation to the base point. For instance, if the base point is in the middle of the upper side, the label field will always be positioned at the same distance from the middle of the upper side. If the red cross coincides with the green circle, the caption will always be positioned in the middle of the upper side.

! ***Not all fields described above are available for each component.***

Confirm the settings in the component appearance window by clicking "OK" or cancel them by clicking "Cancel". For some labels an "All" button is available for simultaneous changing of appearance of all labels of a given type.

8.3.   Graphical editor configuration

Application configuration is meant as settings, which are not saved in the project (and therefore do not depend on any particular project), but are saved to a configuration file and are assigned to a particular user. This is the main difference between project options and User settings and remaining application configuration.

8.3.1.   User settings

♦   To display user settings window select "Options / Settings".

The "**Program settings**" window has six tabs:
  – "Designer data" – here you can append text data on the company, investor and designer
  – "Screen appearance" – here you can specify how part of the screen should look and the colours used in editing
  – "Files" – here you can specify what happens upon opening a file, automatic file saving frequency and saving options on exiting the application
  – AUTO, LOCK
  – "General" – here you can specify other settings
  – "Messages" – here you can specify which warnings and hints are not to be displayed upon completing calculations.

"**Screen appearance**" tab:
  – "Show" – toggles on and off rulers and status line
  – "Status line" – status line options
  – "Hint area shape" – enables specifying the shape of hint area which is displayed when you position the mouse pointer on an item on the screen

–   "Hints in data table" – when you position the mouse pointer on some of the fields in the data table, hints are displayed. Here you can switch off the display of these hints

–   "Hints for project components" - when you position the mouse pointer on a project component, hints are displayed with that component's data and, if available, calculation results. Here you can switch off the display of these hints

–   "Show group in data table as ..." – when a fitting component is placed on a component (e.g. a valve on a pipe-run), the data table includes both the pipe-run data and the valve data. The valve data may be displayed or hidden (see section 4.10.5). Here you can determine the manner of presenting data upon marking a component

–   "Auxiliary windows" – these include: data table, error list, data sets list, letter shortcut list, etc. Here you can force these windows to be displayed without titles. This enables more information to be displayed. Windows differ from one another in colour and contents, so mistakes are unlikely.

–   "Show layers as" – here you can determine how the project layers are to be displayed – as tabs or as check lists (see section 4.2).

–   "Selected component colour" – colour in which the marked components are drawn

–   "Workspace colour" – background colour during editing

–   "Crosshairs cursor" and " Crosshairs cursor colour" – crosshairs cursor consists of two perpendicular lines which are displayed in the workspace at the current position of the mouse pointer. This cursor facilitates positioning of components in relation to each other. Here you can switch this cursor on and specify its colour

–   "Zoom in/out increment" – here you can specify the percentage increment of zooming in and out (when using Zoom in and Zoom out commands)

–   "Animate zoom in/out" – when this box is checked, the magnified/shrunk area is displayed by means of animated rectangles during zoom in/out. This helps to see how the viewed area is changed.

"**Files**" tab:

–   "Position on open file" – project view displayed upon opening it

–   "Lock on open file" – LOCK mode (section 4.6.2), which locks components. This can be altered after opening the file:

1. "No change" – opening of a file does not change the current LOCK mode setting.
1. "Same as last edit" – the same LOCK mode is set as during last save of the file.
2. "Lock" – the LOCK mode is always enabled after opening of a file.

–   "Error list on open file" – here you can specify whether the error list is shown upon opening a file if errors occur

–   "Autosave" – data can be saved automatically at specified time intervals. Here you can toggle autosave on and off and specify the time interval

–   "Read AUTOREAD for new data" – toggles on and off the loading of options and initial project contents from the AUTOREAD file. You'll find more information in section 8.13

–   "Display welcome window on start-up" – here you can specify whether the welcome window (with three basic commands: New, Open, Open last) is to be displayed on program start

- "Save options" – here you can specify which settings are saved on exiting the application, this applies to: contents of toolbar, all settings in the "Program settings" window and screen layout (positions of such items as data table, error list, toolbars)
- "Save toolbar now" – enables immediate saving of the toolbar in its current state
- "Store names of last 'x' files" – the graphical editor stores the names of the files recently opened and displays them in the "File" menu to facilitate their reopening. Here you can specify how many names you want to store
- "Show recent file names with full path" - when this box is checked, the file names shown in the "File" menu are displayed with their full access paths. This is useful when you work with files saved to different disks.

"**General**" tab:
- "Message duration" – time duration of the display of minor messages in the status line (e.g. "cannot place component here")
- "Undo options" – options of the Undo and Redo commands. You'll find more information in section 4.7
- "Restore screen position on Undo and Redo" - when this box is checked, the program tries to restore screen position and zoom level on "Undo" and "Redo". If the box is not checked, screen position is not restored
- "Double click in data table" – operation executed when you double click on a selection field in the data table
- "Select values in check lists by" – select single or double click for confirming selections in lists for selection fields
- "Open group in check list by" – select single or double click for opening subgroups in lists for selection fields
- "Immediately scroll with the slider" – When this box is checked, the view of the project on the screen is moved as you move the slider on the horizontal or vertical scroll bar. When this box is not checked, the view of the project on the screen is refreshed only after you stop moving the slider
- "Display calculation results after changing project" – when this box is checked, calculation results are still displayed on the drawing even if data have been changed. Only the status icon changes to indicate that the project has been changed. When this box is not checked, calculation results disappear after each data change
- Print options:
- "Print on margins" – when this box is checked, the margins are available for printing. When this box is not checked, one letter in the text may be lost in the printout when page break is inserted in the middle of text
- ""Print page header vertically" – when this box is checked, page header (information on project name and page number) will be printed vertically on the left side of page instead of being printed horizontally in the upper left corner of page
- "Print page header on page 1 only" - when this box is checked, page header (information on project name and page number) will be printed on first page only
- "Print scissors – paper cutting lines" - when this box is checked, small scissors symbols will be printed to indicate location of page cutting
- "Print paper cutting lines" - when this box is checked, broken lines will be printed to indicate page cutting lines
- "Print numbers of adjacent pages" - when this box is checked, arrows and numbers of adjacent pages are printed on margin

- "Warn when trying to print without calculations" - when this box is checked, a warning will be displayed if you try to print a project which has not been calculated or which has been altered since last calculations have been done.

A button in the lower part of the window enables configuration of colour conversion table for exporting DWG/DXF files (to AutoCAD®). Each colour displayed by the graphical editor can have a number assigned, corresponding to the colour used in AutoCAD®. In addition, you can toggle on and off the use of Windows fonts in DWG/DXF files.

"**Messages**" tab:
– Here you can specify which of the warnings or hints are not to be displayed upon completing calculations. Message disabling applies to the entire editor, but it can be set at any time.
– A message can be disabled in the program settings or directly by pressing the right mouse button when the mouse pointer is over the message in the error list. Then select "Don't show this message again". The selected hint or warning will not be displayed the next time you start calculations.

8.3.2. Screen appearance configuration

Basic screen components are described in section 4.2. Apart from the main window there are auxiliary windows, which may be pasted into the main window and function as its part or as separate windows displayed over the main window.
Each auxiliary window that can be pasted into the main window (except for toolbars) has a set of special buttons in its upper right corner:

 is used for closing the auxiliary window.

 is used for moving the window on the screen. When moving the window its frame may be wide or narrow. The width of the frame depends on the location of the window. Wide frame indicates that the window will be independent, narrow frame indicates that the window will be pasted into the main window area. Narrow frame appears only when the auxiliary window is moved to the edge of the main window where it can be pasted.

When you position the mouse pointer on the button, a hint appears with information on the auxiliary window, and when you press the mouse button the entire help page is displayed.

Pasting toolbars into the main window is done the same way as pasting auxiliary windows. The only difference is that there are no special buttons on the toolbars. However, when you position the mouse pointer next to a button or between buttons of the toolbar, the pointer takes on the shape of a grabber hand which enables dragging the toolbar into another location on the screen.

The main (standard) toolbar is displayed in the graphical editor screen upon launching the program. It contains tabs, which enable switching between parts of the toolbar.

Apart from the main toolbar, you can also place other toolbars on the screen. These toolbars are copies of the main toolbar tabs.

♦ To place additional toolbars on the screen:
  1. Press **Shift+F9** or click the right mouse button on the main toolbar within an area with no button
  2. Mark the toolbars which you want to place separately on the screen
  3. Click on the "Close" button
  4. Move the new toolbars into appropriate location.

You can move the main toolbar around the screen just as any other toolbar. If, however, the main toolbar is pasted at the side of the main window, the tab names are not displayed. Instead of tab names, an empty field is displayed. Pressing the right mouse button when the pointer is within that area displays a menu that enables selection of any of the tabs.

Other screen appearance settings are defined in the program settings window on the "Screen appearance" tab (see section 8.3.1).

8.3.3. *Configuring toolbars, keyboard, mouse and pop-up menu*

The graphic editor enables you to configure the contents of toolbars, assign operations to individual keys and keyboard shortcuts, configure operations executed on pressing mouse buttons and define the contents of pop-up menu. You can execute these operations in the "Configure" window.

♦ To display the window for configuring toolbars, keyboard, mouse and pop-up menu select Options / Configure ...".

The window has three tabs. One of them is used for configuring toolbars, the second is for configuring the keyboard and the mouse, the third is for configuring the pop-up menu.

Toolbar configuration:
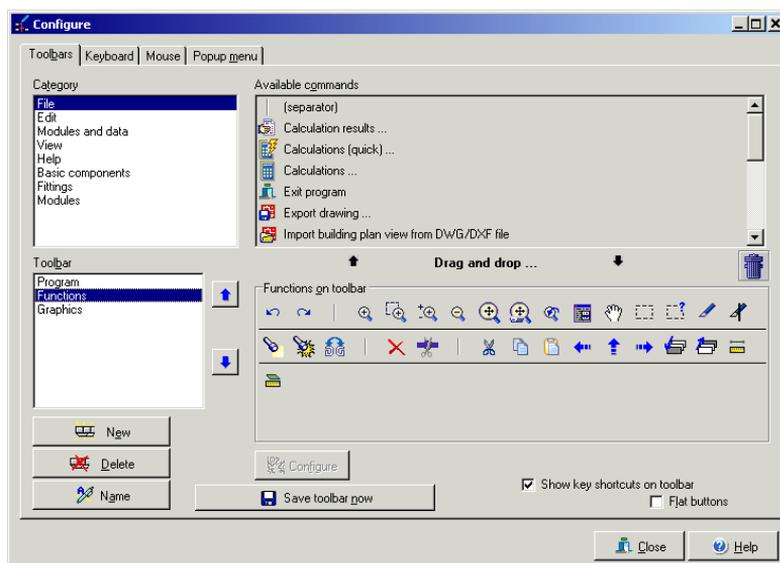
In this part of the window (i.e. the first tab) you can specify the contents of all toolbar tabs.

A list of operation categories is displayed in the upper left part of the window. When you select one of the items in the list, available operations in the form of buttons are displayed in the upper right part of the window or a list of modules appears. When you position the mouse pointer on a button, its function is displayed.

The lower left part of the window contains a list of available toolbars (tabs of the main toolbar) and buttons for creating a new tab, deleting and renaming an existing tab. Two buttons with blue arrows to the right of the toolbar list are used for changing the sequence of tabs displayed within the main toolbar.

The central right part of the window contains the set of buttons of the selected toolbar.

♦ You can move the buttons from the upper part of the window to the lower part and back by dragging and dropping:
   1. Position the mouse pointer on a button and click and hold the left mouse button
   2. While keeping the left mouse button depressed, move the pointer with the button to the desired part of the window
   3. Release the left mouse button.

You can delete buttons in a similar way by dragging them to the waste bin (also displayed in the window).

In the lower part of the window you can also change the appearance of buttons ("flat" or framed) and specify whether the hints for buttons are to display keyboard shortcuts.

♦   To create spacing between two neighbouring buttons, insert a "separator" between the buttons. The separator button is the first button in the upper part of the window.

Buttons that represent modules may be configured.

♦   To configure a button that represents a module:
1. Click the button, then click "Configure"
2. A window described in section 8.8.4 is displayed.

Keyboard and mouse configuration:
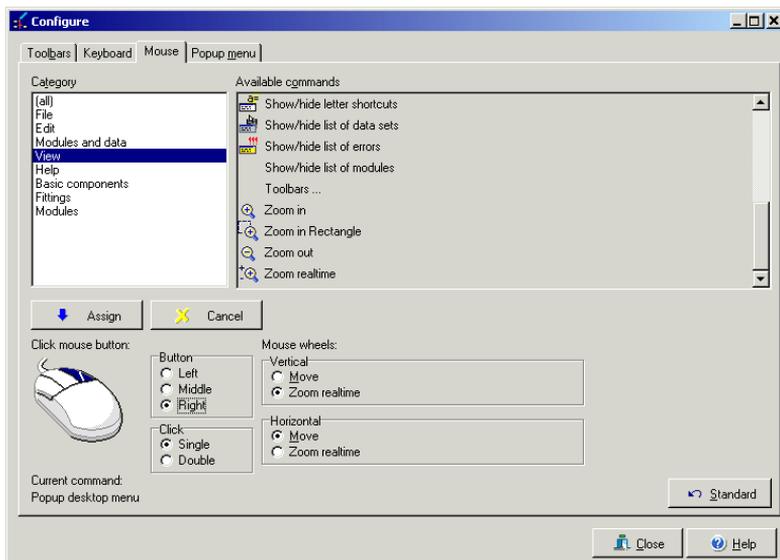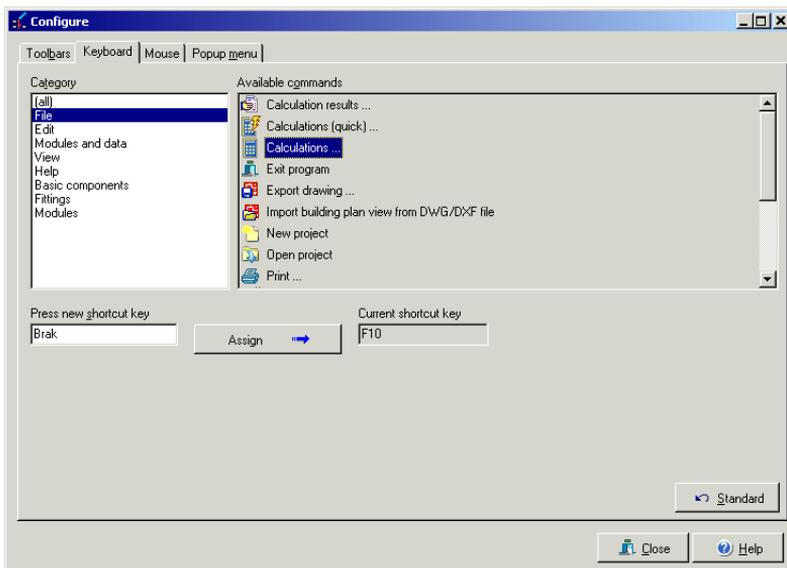
On this tabs you can assign operations to individual keys of the keyboard or to key combinations and assign operations to mouse buttons.

A list of operation categories is displayed in the upper left part of the window. When you select one of the items in the list, available operations of that category are displayed in the upper right part of the window.

♦   To assign an operation to a key or a combination of keys:
   1. Select a category of operations
   2. Select an operation from the list of operations – the key combination to which the operation is currently assigned is displayed in the "Current shortcut key" field. This field is empty if no keys are assigned to the operation
   3. Click on the "Press new shortcut key" field
   4. Press the key combination you want the operation to be assigned to. The combination is displayed and a warning is issued if that combination has already been assigned to an operation
   5. Click on the "Assign" button. The selected key or key combination is assigned to the indicated operation
   6. The "Default" button restores the default keyboard configuration.

♦   To assign an operation to a mouse button:
   1. Select a category of operations
   2. Select an operation from the list of operations
   3. Click on the appropriate "Button" and "Click" fields on the right side of the window. Information is displayed on the operation currently assigned to the mouse button
   4. Click on the assigning button
   5. By pressing the "Cancel" button you can cancel the assignment.

Fields for configuring mouse wheels are in the right part of the window. For a single wheel mouse, settings are those for a vertical wheel. For a dual wheel mouse, vertical and horizontal wheels may be configured separately. The use of mouse rolls and associated keys of the keyboard is described in section 4.3.

Pop-up menu configuration:
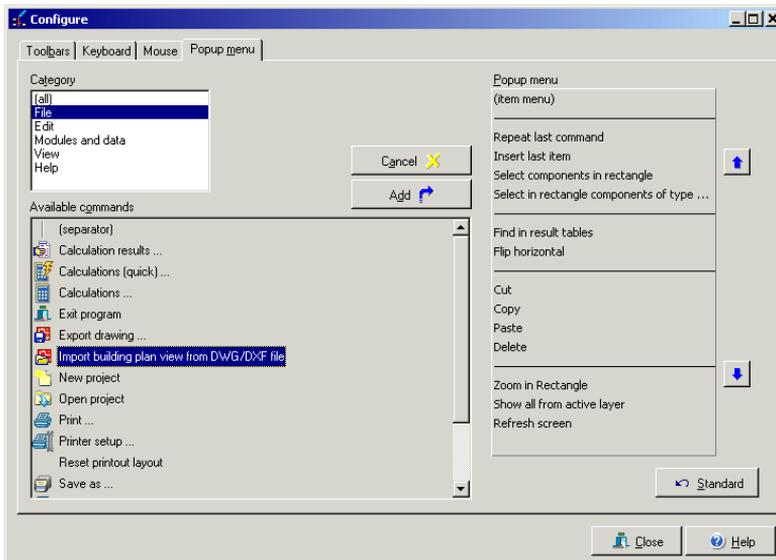
In this part of the window you can define the contents of pop-up menu, the invoking of which may be assigned, for instance, to the right mouse button.

A list of operation categories is displayed in the upper left part of the window. When you select one of the items in the list, available operations of that category are displayed in the lower left part of the window along with "(menu component)" and "(separator)" items, which enable inclusion of an operation that depends on the clicked component and of a separator – a line for separating menu lines, respectively.

A list of operations included in the pop-up menu is displayed on the right side of the window. You can change the sequence of functions by means of buttons with blue arrows on the right side of the list.

You can move operations from one list to the other by dragging and dropping or by using the "Add" and "Cancel" buttons.

The "Default" button restores the default pop-up menu.

8.3.4. Changing the sequence of data displayed in data table

Each time you select a component, its data are displayed in the data table. Sometimes not all of that data are necessary, sometimes a different sequence of the data would be more convenient. For this reason you can change the sequence of items in the data table and hide selected fields of the data table.

♦ To change the sequence of component data displayed in data table:
1. Select "Component data / Change data sequence / <component_type>",
2. A window is displayed for changing data sequence. Data fields are displayed in the sequence in which they are arranged in the data table; a symbol beside the field name indicates whether the field is displayed or not
3. Using the two arrow buttons on the right side you can move the fields into a different position (up or down). You can also do this by clicking on the field and dragging it

4. To disable the display of selected data in the data table, uncheck the box to the left of the data name.

8.4. Using drawings from other applications

8.4.1. Importing drawings; Drawings gallery

The graphical editor supports use of drawings from other applications. Such drawings are treated as "pure" graphics, which means that they cannot be part of the calculated system. Drawings are handled in the "Drawings gallery". In this window you can load drawings, delete them, re-scale them, modify their descriptions, etc.

♦ To activate the drawings gallery (window for handling drawings), select "Modules / Drawings gallery...".

The drawings gallery window is shown below:
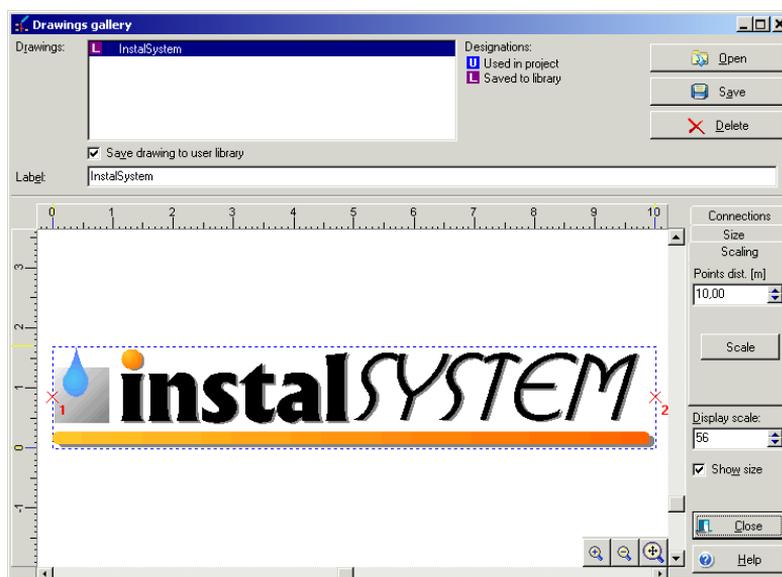
The upper part of the window includes a list of drawings in the project. Some items in the list may be preceded by symbols "L" or "U". They have the following meanings:

- "**L**" – the drawing is saved to user library; it is saved to project file only if it was used. Drawing without the "**L**" symbol is saved to project file only and is not available in other projects
- "**U**" – drawing used in the project.

After clicking the drawing title in the drawings list, the drawing is displayed in the lower part of the window. To zoom in or out the drawing use buttons with magnifying

glass , use the scroll bars to move the drawing horizontally and vertically.

Check the box ☑ under the drawings list to save the selected drawing to user library. You can change the label of the selected drawing in the "Label" field.

Buttons for loading the selected drawing from disk, saving the drawing to disk and deleting the drawing from the list are in the upper right part of the window. Drawings may be saved in several file formats.

! It is not possible to delete a drawing that is used in the project.

Tabs for configuring drawing data are on the right side. The "Scaling" tab is used for scaling the drawing. You can move the red points anywhere on the drawing. Move both points with the left mouse button depressed to characteristic locations within the

Points dist. [m]

drawing and enter the distance between them in the field `10,00` ⬍. The drawing is re-scaled after clicking the "Scale" button.

You can specify the dimensions of the drawing in "X size [m]" and "Y size [m]" fields. These dimensions should be specified in an absolute scale in metres. If the "Proportions" box is checked, altering one dimension of the drawing forces the other dimension to change accordingly.

The last two fields in the right-hand side of the window have the following meaning:
- "Display scale" – scale in which the drawing is displayed,
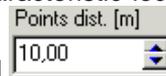- "Show size" – a blue broken line indicates the drawing border.

! The program enables importing graphic files in the DWG, DXF, WMF, EMF, BMP, JPG format and others. When a DWG/DXF file is selected for loading to the gallery, the import window is opened.

8.4.2. Saving a fragment of project as a drawing

The previous sections describe how to use drawings from other applications as graphic components in the program. The program, however, features some simple tools with which you can create your own drawings. There is also a function which enables to place such drawings in the Drawings gallery and then to use them elsewhere and in other projects.

♦   To save a drawing in the Drawings gallery:
1. Make the drawing. You can use all components, including those from the "Graphics" tab
2. Mark the components. You can use all the techniques described in sections 4.8.2 to 4.8.6,
3. Select "Modules / Save selected as drawing"
4. The marked components are saved as a drawing and placed in the Drawings gallery. Configuration of drawings in the Drawings gallery is described in section 8.4.1.

! Drawings in the Drawings gallery bear no design data. Only the drawing is saved and it will not be possible to modify it in the future.

♦ To save a fragment of the project in the form of a set of program objects, mark the components to be saved and select "Append to toolbar" in the "Modules" main menu.

8.4.3. Using drawings

You can insert drawings anywhere within the project without any link with other components of the project.

♦ To insert a drawing into the project:

1. Select "Drawing" in the toolbar of the "Graphics" tab
2. Move to workspace and place the drawing in the project
3. Open the list in the "Drawing" field in the data table and select the drawing or click "add new ..." to open the drawings gallery.

"Yes" in the "Maintain size" field of the data table means that the dimensions of the drawing specified in the "Drawings gallery" are preserved. When "No" is entered in that field, you can resize the drawing.

The "Displayed" field enables modifying the display of the drawing. When "No" is specified, the drawing disappears from the drawing area and only the olive oil frame remains.

8.5. Importing DWG/DXF file as plan without interpretation

A file generated with a CAD application can be used as a base plan, which can be loaded from a DWG/DXF file without interpreting its components.

The imported building drawing or map may form a background on which the structure of the building, installation or network can be plotted.

The file is imported into the current worksheet. Select "Import building plan from DWG/DXF file" in the "File" menu. A new window is opened, where the file on the disk can be selected (the contents of the file are displayed in the upper right corner). Click the "Open" button. When importing for the first time, you will be prompted to specify the font file used in the project (with .shx extension). You can indicate a file on the disk – it can be provided with the project or it can be saved in AutoCAD folders. If the file is not available, click the "Cancel" button.

The file import window opens. All drawing layers included in the loaded file are displayed on the left side. A preview of the imported file is displayed in the upper main window of the screen. Buttons for viewing and moving the drawing and for measuring the lengths of components in the plan are locayed in the upper left corner of the svreen – these functions are enabled only after selecting the drawing unit of measure.

The drawing unit of measure is selected from a drop-down list box under the buttons. The unit must be appropriate for the actual dimensions of the object. The dimensions of the drawing, its width and height in DWG/DXF file units and in metres, are displayed below.

! ***The selection of the drawing unit of measure, appropriate for the dimensions of the actual object, is important with regard to the graphical editor, which always displays and reads dimensions in metres. Therefore the imported drawing should be properly rescaled.***

When you press the right mouse button within the drawing preview area, a pop-up menu is displayed, which – in addition to viewing the drawing – enables selecting display options, that is the background colour and toggling between Paper <-> Model.
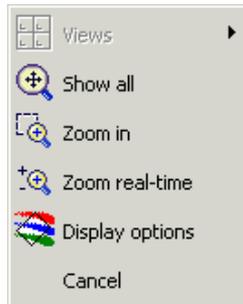
♦ To move the drawing preview press and hold the left mouse button and move the drawing in the desired direction. During this operation the mouse pointer takes on the following form: .

To move to the next import window click the "Next" button in the upper part of the screen. The next stage is the selection of layers that are loaded as a drawing. By default all layers are marked. Unmark the layers that are not necessary and that would unnecessarily complcate the drawing (e.g. room labels, numbers, titel blocks, etc.).

When you press the right mouse button in the list of layers the options "Select all", "Deselect all" appear. You can deselect all layers or select only those, which are required as the drawing background for the project.

To complete loading the drawing, click the "Next" button. The drawing is automatically loaded onto the "Base" layer, where the "DWG/DXF base" is placed with all selected layers presented in the form of a uniform drawing (without highlighting any object).

Components loaded without interpreting retain their structure from the DWG/DXF file. When you move to the "Base" layer and click within the area of imported base, an olive green frame appears and base data are displayed in the table.
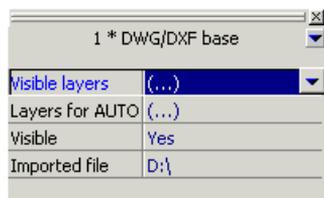
You can change the manner of displaying the entire base drawing or individual layers. You can decide in which layers the AUTO mode is to be activated (by default all

layers). This in practice means that when the AUTO mode is activated and you draw rooms or manually insert heating zones, the corners of these components are snapped to the marked layers. Similar operations can be performed by clicking the right mouse button on the base components. Then the layers which include those components are located and the display of those layers can be disabled and AUTO mode settings can be changed.

8.6. Exporting a drawing

The program enables exporting a drawing in the form of a plan, developed view, profile, system or network diagram into a file on disk. Drawings can be exported in the DWG, DXF, DXB, Windows Metafile (WMF files), Enhanced Metafile (EMF files), JPEG and other formats. DWG / DXF files can be opened in many professional graphics software packages, such as AutoCAD®. WMF and EMF files can be read by most Windows applications. The effect, however, can be unsatisfactory. Some applications may require appropriate graphical filter (e.g. for MS Office®, you can specify the graphical filters when installing the package).

♦ To export a drawing select "File / Export drawing ...". The export window is opened, where you specify the type of file into which the drawing is to be exported.

! *Export is not possible without specifying file type.*
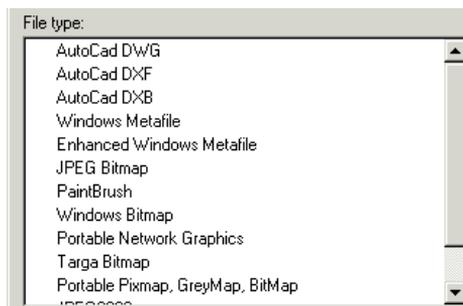
After selecting file type click the "Next" button. Another window is opened, where you select the worksheets to be exported and specify the type of export.

By default all worksheets are selected for export. If you don't want to export some worksheets, uncheck the ☑ fields next to those worksheet names.

Export with appending is performed only in the case of DWG / DXF / DXB files. This is characterised by the generation of a copy of the previously imported .dwg file and appending it with layers created by and saving it with a new name. The base on which the project was created does not need to be exported, as the entire structure of the .dwg file remains intact – it only includes additional layers with inserted components.

When the .dwg file was imported and it is on the disk, its name is displayed in the "Appended file" field. After marking a worksheet (highlighted in dark blue), additional icons appear and the appended file and the destination file may be changed. You can

change the appended file or find the appended file on the disk, if its location has been changed, by clicking the ![button] button.

! ***When the imported file is not on the disk or when it cannot be localised, a ◇ symbol is displayed. In this case you can look for the file manually or perform an export without appending.***
! ***If there is more than one imported.dwg file on one worksheet, none of them will be appended and no name will be displayed. You can append only one of these files.***
You can change the name of the destination file and its location on the disk with the ![button] button. By default the name consists of the project file name + name of the worksheet. The file is saved to the data folder.

To export a DWG / DXF / DXB file without appending click on the ![button] button. Both the layers generated by the program and the base layers are exported. However the objects of the base will only be graphical objects, such as a line or a polygon, and will have no properties.

Exporting of files other than DWG/DXF is done without appending any files. The name of the destination file can be changed.

To move to the last export window click on the "Next" button. Depending on the file type selected, appropriate options are displayed: file version, unit, exported lines and their thickness or DPI resolution.
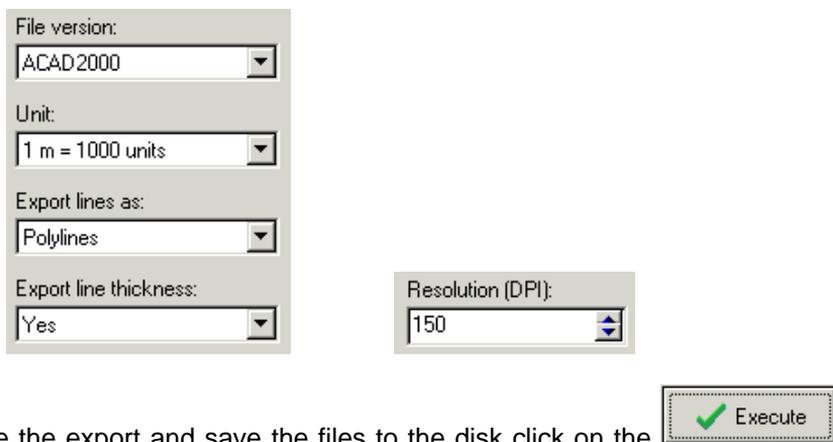
File version:
ACAD2000

Unit:
1 m = 1000 units

Export lines as:
Polylines

Export line thickness:
Yes

Resolution (DPI):
150

To complete the export and save the files to the disk click on the ![Execute] button. An "Exporting project as drawing. Please wait ..." message is displayed. Export is completed when the message disappears.
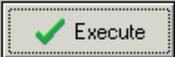
8.7. Duplicating fragments of the project

Fast creation of a project may be facilitated by efficient duplicating of existing fragments. There are four basic methods of duplicating fragments of the project:
– copying to the clipboard and pasting,

–   saving fragments of the system as modules to a toolbar (see section 8.8.2),
–   duplicating fragments of the project in specified direction.

8.7.1. Copying to the clipboard and pasting

This is the simplest method of duplicating fragments of the project. The fragment to be duplicated may have its data already completed. Preferably it should be correct in formal terms (e.g. with internal connections made), as this facilitates further graphical editing of data.

♦   To duplicate a fragment using the clipboard:
1. Mark the components you want to duplicate. You can use all the techniques described in sections 4.8.3 to 4.8.6.
2. Select "Edit / Copy" (**Ctrl+C**, 🗐). The components will be copied to the clipboard as a module
3. Select "Edit / Paste" (**Ctrl+V**, 📋)
4. Position the module in the required location
5. Press the left mouse button. The module is now inserted
6. You can insert subsequent modules by selecting "Edit / Paste" again or by selecting "Edit / Insert last item" (**F3**, 📑).

!   When pasting, you can make use of the possibilities of the AUTO mode described in detail in section 4.6.4.
!   Components copied to Windows clipboard can be pasted as drawings into other applications (e.g. MS Word®, Corel DRAW!®, AutoCAD®).

8.7.2. Duplicating drawing fragments in specified direction

This is a method of fast duplicating of a fragment of the project while retaining one of its co-ordinates, e.g. elevation (duplication with horizontal translation). In some cases automatic determination of the location of the duplicated module expected by the User is not possible. Then it is best to use the clipboard to duplicate.

♦   To duplicate a fragment of the project in specified direction:
1. Mark the fragments you want to duplicate. You can use all the techniques described in sections 4.8.3 to 4.8.6,
2. Press **Ctrl + Alt + Shift + <cursor control key>** or the respective button on the toolbar: ⬅ , ⬆ , ➡ .
3. Now you will be prompted for the duplication distance. Enter the distance between the existing fragment and the new duplicated fragment.
4. Upon entering the value and clicking 'OK' the module or the marked set of components is duplicated in the specified direction.

8.8. Modules

8.8.1. Modules - introduction

Although you can always create an installation using single components, using typical modules can speed up work considerably when fragments of the project are repeatable.

8.8.2. Creating and storing modules

Module is an arrangement of several components stored on the toolbar. The creation of a module consists in inserting components into the project, appending their data (if necessary) and storing on the toolbar. You can use existing modules to create new ones.

♦   To create a module and store it on the toolbar:
1. Using single components or other modules, create the fragment of installation you want to store as a module
2. Mark all the components that are to make up the module. The best way to do this is to mark components in a rectangle (see section 4.8.5)
3. Select "Modules / Add to toolbar". The marked components are stored as a module in the "User" tab. The module configuration window is displayed (see section 8.8.4).

8.8.3. Modules gallery

You can view and edit all created modules in the modules gallery.

♦   To invoke a modules gallery select "Modules / Modules gallery ..."

A modules gallery window appears:
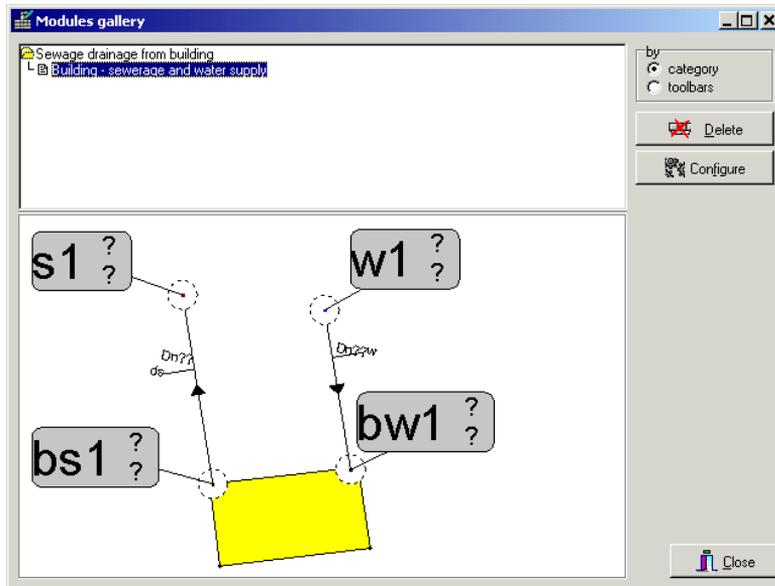
A list of modules, arranged according to categories, is displayed in the upper part of the window. After selecting a module in the list, the drawing of that module is displayed in the lower part of the window. Two buttons to the right of the list are used to delete and to configure a module.

You can set the size of the list area and the module view area by shifting the border between the two areas with the mouse.

8.8.4. Configuring a module

After clicking on the "Configure" button in the modules gallery window (see section 8.8.3) or in the "Configure" window (see section 8.3.3), the module configuration window is opened.
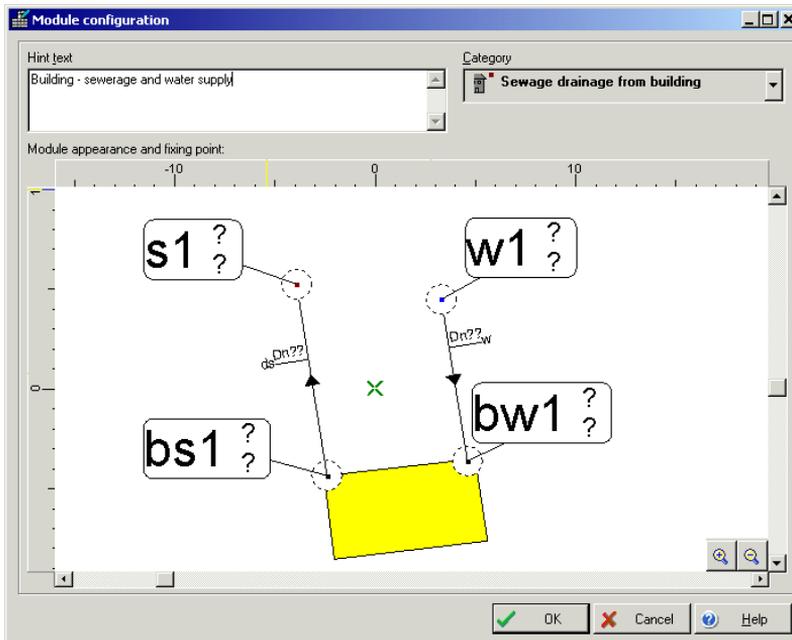
Here you can:
1. Enter the text of the hint displayed when the mouse pointer is positioned over the button on the toolbar
2. Specify the category of the selected module
3. Move the handle of the module. This is the point, where the mouse pointer is located when inserting the module. The handle is indicated by a cross in the configure window.

8.9. Graphical components and labels

Each project can have graphical components and additional labels (descriptions) added to it. Graphical components enable appending additional descriptions and notes.

Labels and drawings are introduced with the help of elements included in the „Graphics" tab of the toolbar:

, , – pipe-run labels: oblique, level and simplified. These are graphical components which are inserted and handled in the same manner as fitting components. They visualise diameter sizing results in the drawing; optionally energy flux provided by water in the pipe-run may be displayed.

! *A pipe-run label, configured as required, can be saved to a toolbar by selecting "Modules / Append to toolbar" from the main menu. It can then be used in other projects.*

⎍⎍⎍ – polyline. You can use the ORTO mode when drawing a polyline. Polyline type can be configured.

⎍⎍⎍ – polygon. The ORTO mode is used when drawing a polygon. You can specify the line type and fill colour of the polygon.

⎍Abc⎍ – text. Each text can consist of several lines – to insert a line break enter the "|" symbol. You can configure the font, text orientation and justification.

⎍Abc⎍ – rectangle (box) with text inside, if required. You can specify separately the text properties (may consist of several lines) and line type.

⎍(Abc)⎍ – ellipse (oval) or circle with text inside, if required. You can specify separately the text properties (may consist of several lines) and line type.

⎍⎍⎍ – project title block. The project title block is fully configurable – both in terms of size and fields used.

⎍⎍⎍ – drawing saved to the drawings gallery. More information on drawings and drawing galleries is given in section 8.4.1.

⎍⎍⎍ – note. You can place comments or notes in the drawing. This, for instance, may be used to indicate an area in the drawing where corrections or amendments have to be made. Upon entering a comment into the data table, it will be displayed in a help bubble when the mouse pointer is positioned over the "note".

You can change the size of some of the graphical components (e.g. a rectangular or an oval) and rotate them. All resizable components have black squares on their edges, which can be dragged to resize the component. All components that can be rotated have a square mark for rotating them. You can find more information in sections 4.8.8 and 4.8.9.

8.10. Grouping and ungrouping

Grouping of components in Net-san is used when editing network profiles to change the reference level in the middle of a profile.

♦ To group components:
  1. Mark the components. You can use all techniques described in sections 4.8.2 and 4.8.6.

  1. Select "Components / Group" (**Ctrl+G**, ⎍⎍).

! ***It is not possible to group one single component.***

A group is displayed surrounded by a brown frame:
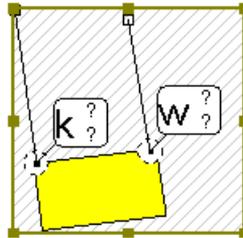
When you click within the group area where there is no component the group is marked and group data are displayed in the data table. The marking of the group is indicated by an appropriate caption in the status line, by thicker lines of the group frame, and the contents of the data table.

! ***Make sure whether a group or only one of its components is marked – this is particularly important when you intend to delete a group component only.***

You can create nested groups, which means that one group can be included within another group.

You can also ungroup components.

♦   To ungroup components:
1. Mark the group,

1. Select Components / Ungroup (**Ctrl+H**,),
2. The group (as a kind of logical structure) is dissolved; but its components remain in the drawing.

♦   To change the reference level in the middle of a profile:
1. Mark fragment of the profile for which you want to change the reference level
2. Group components
3. Change the local reference level in the data table of the group
4. Ungroup components (if necessary).

8.11. Calculate intermediate elevations

If the profile is drawn by the user, some fragments of it may require smoothing. You can also smooth the network profile and the ground above it. When this function is used, all network sections and the ground above it will have a uniform gradient calculated from the difference of elevations of terminal sections marked.

♦   To smooth a network profile (or network profile and ground level):
1. Mark the train of profile sections to be smoothed

142

2. Select "Component data / Calculate elevations of intermediate manholes and ground"
3. The selected fragments of profile are smoothed.

You can also move one or more elevations of duct bottom or pipeline axis by specified distance.

♦ To move duct or pipeline:
1. Mark profile section(s) you want to move
1. Select "Component data / Move duct (pipeline) elevations"
2. In the box displayed enter the absolute value of the shift in metres. precede the value with "+" to move the elevations up, or "-" to move them down
3. The selected sections are moved.

8.11.1. Finding a component

Finding components is a very useful function of program. A special window is designed for this operation:

– Select components/Find component (Ctrl+F)
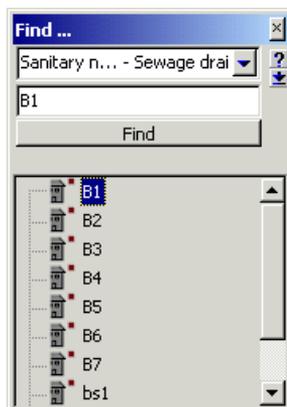
The upper field within this window specifies the type of component you are looking for. In the field below you enter the symbol or the number of the component. After pressing **Enter** or clicking the "Next" button, the component is located and marked on the screen.

If a component cannot be found, appropriate message is displayed.

You can display a list of project components of the specified type. To do this click on the arrow button under the hand and question mark buttons.

Clicking on an item in the list also starts a search.

8.12. Working on two or more projects simultaneously

Program Net-san is a single-document application, which means that at any given moment you can have only one document (project) opened. There is, however, no objection to running two or more instances of graphical editor at the same time. You can then open a different project in each instance of editor or you can open the same project in both instances, but activate other worksheet in each instance.

Running two instances of program facilitates the transfer of project fragments from one instance to the other with the use of the Windows clipboard.

♦   To transfer project fragments into another project:
1. Start two instances of program,
2. In one instance open the project from which you want to transfer the fragment,
3. Mark the fragment to be transferred,
4. Copy the fragment to the clipboard: "Edit / Copy" (**Ctrl+C**,),
5. Switch to the second program instance,
6. Paste the fragment from the clipboard: "Edit / Paste" (**Ctrl+V**,).

8.13. AUTOREAD file

The AUTOREAD file is loaded as a base when creating a new project. In this file you can store default project options, print settings and possibly a fragment of project (e.g. a building plan, map, title block, etc.). When you install program there is available a standard AUTOREAD file.

♦   To create your own AUTOREAD file:
1. Create a project, which will form the base for other projects. This project can include any fragment of an installation. It can also be an empty project,
2. Specify project options,
3. Select "File / Save as ...",
4. Type in AUTOREAD and press the **Enter** key.

After carrying out these operations, the next time you create a new project, this AUTOREAD file will be used as a template. As normally such file already exists, you will be asked to confirm overwriting of the previous version.

9. EXAMPLE

In this section we describe entering data for a calculation example included in the "sewerage example" file.

Net-san is opened from the package manager or directly from the Start Menu. The welcome window is displayed, where we select "New project". The general data window appears. Now we check that all catalogues necessary to create the new project are loaded. To do this we click on the "Catalogues" button and the "Catalogues" window opens. Now we verify lists of loaded catalogues on individual tabs (to load a catalogue mark it in the left-hand window and click on the button with blue arrow pointing to the right). Changes made are confirmed by clicking on the OK button in the lower part of catalogue selection window. Now we are back in the general data window where we specify the default pipe type for the network in the project.

We select the "Info" tab and enter data concerning the detailed description of the project, project developer and designer. On the "Edit" tab we check the correctness of entered data (horizontal and vertical grid, etc.). All changes in the general data are confirmed by clicking OK.

Our project consists of two worksheets: one map type sheet and one profile type sheet. First the network layout (map) is created, and the profile will be generated by the program.

The basis for drawing the network diagram is the base map of the area. We move to the "Base Map" tab and select "Map" component and load into the data table map file of ism format generated with the "Combine Maps" application. Then the scale of the map and editing colour are specified.

We now move to the "Networks" tab and start drawing the network diagram in the following sequence:

- mark buildings and other objects covered by the project by inserting rectangles or polygons at their locations
- insert terminal nodes of the network
- specify the functions of the nodes in the data table (receptor, source, etc.)
- plan the route of the network
- insert network junctions at points of connection
- connect the nodes and junction by means of sections (pipe-runs) of the network
- check connections and correct them, if necessary
- insert fittings and appurtenances
- introduce other networks that are not calculated by the program and may interfere with the network under design
- append and verify data of all components in the data table
- enter and configure component descriptions (labels).

In the example from the "Graphics" tab we select "Polygon" and draw buildings from which sewage is to be drained. In the data table we select the fill colour for these polygons so that they stand out against the base map background. Then we place "Sewage drainage from building" taken from the "M –Sewerage" tab on each building.

At the point where sewage is to be discharged (treatment plant or connection with other network) we put a junction with "Sewage receptor" function assigned to it.

While tracing the route of the network we also put in sewerage network junctions (manholes), which will form the connection points of drainage from buildings with the main network.

We connect junctions by means of gravity sewerage network sections using the AUTO mode.

At this stage of the design, the main structure of connections in the network should be complete. We check that by pressing **Shift+F2**. After checking connections it is a good idea to save the project to a disk file, unless it has been done before.

Now we put in the drawing the existing networks (heating and gas supply) that the planned network comes across.

We press **F12** to display the data table, where data of the components of the network under design will be appended.

To introduce the same type of pipe in all sections of the sewerage network, we select "Select all components of type ..." from the "Edit" menu, then we select "Sewerage network" and the "Section of gravity sewerage network" option. In the data table we select from the available catalogues the desired pipe type in the "Pipe type" field. We unmark all marked pipe-runs (e.g. by clicking on the drawing background or by selecting the appropriate icon in the "Functions" tab). In a similar manner we mark all manholes where we entered "?" in the field "Elevation of bottom of manhole".

We apply similar procedure to all network components. Then we specify the data for individual components.

For sewerage network sections these data include:
- designation,
- minimum and maximum gradient,
- minimum and maximum cover depth,
- maximum fill ratio.

For sewerage junctions these data include:
- designation,
- inflow at point,
- ground elevations.

We place labels taken from the "Graphics" tab on the individual pipe-runs. We configure the labels in the data table.

After specifying all data in the table, we run the data diagnostics. To do this we click on the calculator button on the "Program" tab (we can also press **F10** instead). After analysing the data, the program opens the calculation options. The following options should be considered with care, as they have great impact on the calculations:
- in the "Common options" tab, it is the vertical distances between networks at collision points,
- in the "Sewerage network calculation options" they include permissible fill ratios, minimum and maximum gradients and minimum and maximum cover depths.

As we move to the last tab the calculations are done. We check the results by reviewing the sub-tabs with tables of results.

After successful completion of calculations, we go back to the graphical editor, where the most important results are displayed in manhole and pipe-run labels. To make sure we don't lose the data, we save the project to the disk.

The next stage is automatic generation of profile of the calculated network. To do this we create a new worksheet ("File / Worksheets ...") of the "Profile" type. After opening the new sheet we open the "General data on profile" window by pressing the **F7** function key and we select the "Profile definitions" tab. In the table we select the worksheet from which the data will be taken and we define the "Sewerage" profile type. Then we define the following fragments of the profile (route) by selecting designations of the start and terminal junction. After defining the entire profile we close the project window and we generate the drawing of the profile by selecting "File / Create profile as defined" (**Shift+F11**).

We press **F7** again and specify in the project options the reference level, horizontal and vertical scale of profile and amend the appearance of components. Finally we save the project to the disk.

ANNEX A – DEFAULT OPERATIONS ASSIGNED TO KEYBOARD AND MOUSE BUTTONS

The table below lists operations assigned to the keyboard and mouse buttons. The tables below list default assignments; you can make changes in toolbars, pull-down menu and the assignments of keyboard and mouse buttons in "Options / Configure...".

A.1. Keyboard:

Key combination:	Operation executed:
Esc	refresh display on screen.
F1:	when a component is marked – displays help specific to that component.
	when a component type is being selected from a list – displays help specific to that type.
	when no component is marked – displays general help screen.
	in the error list and in the diagnostics results window – displays information about errors.
F2	repeat the last value entered in data table.
Shift+F2	check connections.
F3	insert the same component/module used last time.
Shift+F3	pipe-runs re-numbering.
F4	toggle display: calculated/not calculated installation.
F5	show the entire project – scale and position are set to display the entire project on the screen.
Shift+F5	display the entire active layer – set the scale and position to display all components in the active layer.
F7	project general data – display "Project options" window on the "General data" tab.
Shift+F7	project info - display "Project options" window on the "Info" tab.
F8	toggle on/off error list.
F9	toggle on/off data set list.
Shift+F9	display toolbar list.
F10	run calculations.
Shift+F10	run quick calculations without displaying options and tables.
F11	toggle on/off table of results in the graphical editor.

Shift+F11	create profile as defined.
F12	toggle on/off data table.
Alt+C	toggle on/off GRID mode – components snapped to grid.
Alt+Q	display pop-up menu.
Alt+V	toggle on/off AUTO mode – automatic display of possible component connection points.
Alt+X	toggle on/off LOCK mode – lock (protect against moving) all components.
Alt+Z	toggle on/off ORTO mode – place vertical and horizontal components only + optional defined angle.
Alt+B	włączenie/wyłączenie trybu POWT – powtarzalne wstawianie elementów konstrukcji, instalacji i armatury.
Ctrl+A	mark all components in the active layer.
Ctrl+B	lock (protect against moving) marked components.
Ctrl+C	copy marked components to clipboard.
Ctrl+D	unlock marked components – moving enabled.
Ctrl+F	toggle on/off search window.
Ctrl+G	group marked components.
Ctrl+H	ungroup marked components.
Ctrl+I	import plan view from DWG/DXF file.
Ctrl+L	disconnect marked components on the left side.
Ctrl+M	display macro editing window.
Ctrl+N	
Ctrl+O	
Ctrl+P	print current project section.
Ctrl+Q	divide elements.
Ctrl+R	disconnect marked components on the right side.
Ctrl+Shift+R	completely disconnect marked elements.
Ctrl+S	save project to disk.
Ctrl+V	paste fragment of project or module from clipboard.
Ctrl+X	move marked components to clipboard.

Ctrl+Z	undo last operation.
Ctrl+Y	redo last operation.
Ctrl+Enter	in data table – open a list in a field when available.
Ctrl+Ins	copy marked components to clipboard.
Ctrl+Page Up	toggle to previous worksheet (toggle "left" among tabs).
Ctrl+Page Down	toggle to next worksheet (toggle "right" among tabs).
Ctrl+„+"	zoom in.
Ctrl+„-"	zoom out.
Ctrl+„*"	enable zoom in.
Ctrl+„/"	enable realtime zoom.
Ctrl+ arrows (arrows pad)	move in specified direction to same type component nearest the marked component.
Ctrl+Alt+ arrows (arrows pad)	move in specified direction to a component of any type nearest the marked component.
Ctrl+ arrows (numerical pad)	move in specified direction to connected component.
Ctrl+Shift+Alt+arrows	duplicate marked fragment of project in specified direction.
Shift+Ins	paste module from clipboard.
Delete	delete marked components.
Enter	

A.2. Mouse:

Left button:	
click on a component	mark the component and unmark other components.
click on a component while pressing **Shift** key	mark the component without unmarking other components; if the component you are clicking on is already marked, then that component will be unmarked.
click on a component while pressing **Ctrl** key	mark the component underneath the component already marked.
double click on a component	configurable (see settings); default: mark the entire connection line (all its segments).
double click in data table	configurable (see settings); default: change entry in the field to the next item in the list (if available).
Right button:	
click	configurable (see settings); default: display pop-up

	menu.
double click	configurable (see settings); default: show data table if not displayed.
Middle button:	
click	configurable (see settings); default: display pop-up menu of connection line.
Mouse wheels:	
turn wheel	realtime zoom-in/out.
turn wheel while pressing right-hand **Alt** key	scroll project display. In a dual wheel mouse – scrolling accordant with default wheel mode – vertical or horizontal. In a single wheel mouse – scroll vertical.
turn wheel while pressing **Shift+Alt** keys	precise scrolling of project display (lower scrolling speed). In a dual wheel mouse – scrolling accordant with default wheel mode – vertical or horizontal. In a single wheel mouse – scroll vertical.
turn wheel while pressing **Ctrl+Alt** keys	scroll project display with swapped mouse wheels. In a dual wheel mouse – scrolling contrary to default wheel mode – horizontal or vertical. In a single wheel mouse – scroll horizontal.

ANNEX B – ERROR MESSAGES

In this section all error messages, warnings and hints used in Net-san are presented.

B.1. Connection error messages

Error: No flow
This error occurs when the component through which the medium flows is not connected on either end or when pipe-runs connected to that component are not connected elsewhere.

Warning: Cannot determine flow direction
This warning occurs when part of the system has no water supply source or no drain water receptor and the direction of flow cannot be determined.

Error: Component not connected
This error occurs when a component (pipe-run) is not connected on one end to junction.

Error: Component not connected at all
This error occurs when a component (pipe-run, junction) is not connected at all (stands alone in the drawing).

Error: Component number 0

Error: Two or more components with same number

Error: Components form a loop
This error occurs when ring-shaped systems are formed, not included in the calculation scope of the program (entire loop indicated).

Error: Component forms loop with other components
This error occurs when ring-shaped systems are formed, not included in the calculation scope of the program (single component of loop).

Error: Segment represented on drawing as single point
This error occurs, when due to shifting of pipe-runs, one of them is reduced to a point (its length in the drawing is zero). This is not apparent in the drawing, as the pipe-run is represented as a single point and it therefore may be taken as a point belonging to another pipe-run.

Error: More than two components of remote connection have the same symbol

Error: Unpaired remote connection component

Error: Missing 'water source' or 'drain water receptor' components'
     This error occurs when no component of that function is specified. Missing water source may cause many other connection errors (e.g. "missing source for receptor" or "cannot determine flow direction").

Error: Different configuration of remote connection outputs

Error: Segment included in drain water source
     This error occurs when more than one segment of system is connected to drain water source. Check junction functions.

Error: Segment included in water receptor
     This error occurs when more than one segment of system is connected to water receptor. Check junction functions.

Error: Elevation of existing component not specified
     This error occurs when a component is defined as existing component and has no elevation specified. Enter the missing data in the data table.

Error: Part of system not connected to source
     This error occurs when a water source is indicated in the system diagram, but part of the system is not connected to it. Connect the system part to the indicated source or specify another water source.

Error: Connection errors in sheet
     This message is displayed when connection errors occur in the sheet. More informative messages are displayed in the 'Diagnostic results' window.

B.2. Messages generated in diagnostic run and during calculations

Error: Critical error when arranging system
     This error indicates faulty program operation. Contact the software supplier and provide data file by electronic mail.

Error: No catalogues assigned to project

Error: No default pipe type specified
     This error occurs when there are pipe-runs in the project for which no pipe type has been specified and no default pipe type has been specified in general data. Specify pipe type in system segment data or in general data.

Error: Elevation of existing component not specified
     This error occurs when an existing component has no elevation specified. Enter the missing data.

Error: Undefined segment location
     This error may occur after loading a file created with older version of the program. Occurs when location has been specified in segment data.

Error: Incorrect minimum cover depth
This error occurs when the specified value of minimum cover depth for the segment is less than zero, more than maximum cover depth or more than 10 m.

Error: Incorrect maximum cover depth
This error occurs when the specified value of maximum cover depth for the segment is less than zero.

Warning: Minimum cover depth dubious
This warning is displayed when the specified value of minimum cover depth is less than 1.4 m or more than 6 m.

Warning: Maximum cover depth dubious

Hint: Undefined pipe type – default type applied
Message for the user informing that no pipe type has been specified for the pipe-run in the data table and the type indicated in general data is being applied.

Hint: Unlikely short pipe-run
This message informs that pipe-run length is out of probable range.

Hint: Unlikely long pipe-run
This message informs that pipe-run length is out of probable range.

Error: Pipe-run too short
This error indicates that the pipe-run length is out of correct range.

Warning: Pipe-run too long
This error indicates that the pipe-run length is out of correct range.

Error: Cannot match proper pipe-run diameter to flow rate
This error occurs when the pipe family specified for the pipe-run does not include pipes of required diameter. Modify flow rate or select other pipe family.

Error: Pipe type not in catalogue
Presumably the effect of changing pipe catalogue after saving data (e.g. withdrawing catalogue from project) or transferring data from other file or software version. Select pipe type again.

Warning: Maximum water velocity exceeded
This message may appear when pipe-run diameters are imposed by user. Specify larger diameters, or select a catalogue with a broader range of diameters.

Warning: Water velocity too low
This message may appear when pipe-run diameters are imposed by user. Specify smaller diameters, or select a catalogue with a broader range of diameters.

Error: rmax exceeded
This error indicates that the maximum permissible specific pressure drop has been exceeded. Change or resize pipe diameter.

Error: Incorrect value for calculating pressure drop
This error indicates incorrect value of Kv for valve (zero). Specify correct value in data table.

Error: Wrong fitting or pipe for specified temperature and pressure

Error: Wrong fitting type

Error: Fitting type not specified

Error: Valve type not in catalogue
Presumably the effect of changing valve catalogue after saving data (e.g. withdrawing catalogue from project) or transferring data from other file or software version. Select valve type in data table again.

Error: Incorrect value to calculate pressure drop across fitting
For example: zero value of Kv for "Known Kv" component type. Specify correct value in data table.

Error: No data to calculate valve (error in catalogue – contact the supplier)
This error indicates faulty program operation. Contact the supplier.

Error: Wrong pipe-run location
This error may occur after loading a file created with older version of the program. Occurs when location has been specified in pipe-run data. Specify pipe-run location again in data table.

Error: Incorrect system design – no pumping station
This error occurs when pressurized drainage system is not connected to a pumping station. Check the system diagram.

Error: Cannot determine minimum gradient
This error indicates that land configuration and minimum and maximum cover depths specified do not enable determination of minimum pipe gradient.

Error: Duct end bottom elevation to be determined
This error indicates that the system could not be arranged because the specified elevation of duct end bottom is not consistent with catch basin bottom elevation.

Error: Duct start bottom elevation to be determined
This error indicates that the system could not be arranged because the specified elevation of duct start bottom is not consistent with catch basin bottom elevation.

Warning: Cannot size elbow on pipe-run

Warning: Diameter of existing pipe–run not specified
This message appears when diameters of system parts other than those being designed have not been specified. Specify pipe type and diameter in data table

Warning: Diameter range in catalogue too narrow
This message may appear when no pipe of required diameter can be found in specified or default catalogue. Select a pipe family with broader diameter range.

Error: Diameter range in catalogue too narrow
This error occurs when no pipe of required diameter can be found in specified or default catalogue. Select a pipe family with broader diameter range.

Error: Cannot size fitting on pipe-run

Warning: Wrong fitting size specified

Error: Segment not connected to system junction
This error occurs when two segments are connected directly to each other (with no system junction). Insert a system junction.

Error: Gravity system segment fed from pumping station
This error occurs when a segment of gravity sewerage is connected directly to a pumping station. Replace with a pressurized system segment.

Error: Unsolved collision with other system
This error occurs when land configuration and distances between collisions do not enable collision-free arrangement of system segment. Check distances between system under design and other systems in "Calculation options".

Error: Cannot arrange segment
This error occurs when land configuration and specified minimum and maximum values of cover depths or gradient do not enable correct arrangement of system segment.

Error: Incorrect duct start bottom elevation specified
This error occurs when the elevation of the duct start bottom has been specified. Check its consistency with catch basin bottom elevation.

Error: Incorrect duct end bottom elevation specified
This error occurs when the elevation of the duct end bottom has been specified. Check its consistency with catch basin bottom elevation.

Error: Incorrect diameter specified
This error occurs when a diameter has been specified which is not consistent with set conditions. Check the diameter in the data table.

Error: Connection with next segment is not direct
This error occurs when two segments are connected to each other directly (with no system junction). Select catch basin type "conn. with no basin".

Error: Minimum cover depth exceeded
This error may occur when the specified elevation of water supply system junction bottom causes the minimum cover depth to be exceeded.

Error: Maximum cover depth exceeded
This error may occur when the specified elevation of water supply system junction bottom causes the maximum cover depth to be exceeded.

Error: Incorrect maximum gradient specified
This error occurs when the specified maximum gradient is less than 5 per mille or less than the value of minimum gradient.

Error: Maximum gradient exceeded
This error occurs when the maximum gradient specified in the data table is not maintained during the arrangement of the water supply system.

Error: Undefined elevation of area under design
System junction defined as under design has no area elevation specified. Specify the elevation in data table.

Error: Undefined elevation of existing area
System junction defined as existing has no area elevation specified. Specify the elevation in data table.

Warning: Junction not designated

Error: Flow out rate not specified
This error occurs when system junction defined as source of drain water or water receptor has no flow out rate specified in data table. Specify the flow out rate in data table.

Warning: Required pressure not specified
This warning appears when the value of the required water pressure upstream of the building supplied is not specified. Specify the pressure in data table.

Error: Incorrect elevation of catch basin bottom
This error occurs when the specified elevation of catch basin bottom does not enable proper arrangement of the system. Enter correct value of elevation in data table.

Error: Pump output too low
This error occurs when the specified output of pump does not enable arrangement of part of pressurized system. Increase pump output.

Error: Incorrect pump operation time
This error occurs when the specified time of pump operation is incorrect (should be within 1 – 60 minutes). Enter correct value in data table.

Error: Junction definition error

Error: Error in catalogue: kinet quantity <> 1
This error indicates faulty program operation. Contact the supplier.

Error: Specified catch basin bottom elevation inconsistent with catalogue value
This error occurs when the elevation of catch basin bottom is specified and the type of catch basin or pumping station selected from catalogue does not enable arrangement of system. Specify correct elevation of bottom or specify 'size' or select other type of catch basin from catalogue in data table.

Error: Cannot size catch basin
This error occurs when the group of catch basins indicated by user does not include one of appropriate size.

Error: Cannot arrange system downstream of pumping station
Part of system downstream of pumping station cannot be arranged under the conditions specified.

Error: Cannot size pumping station
This error occurs when the group of pumping stations indicated by user does not include one of appropriate size.

Error: Cannot size catch basin
This error occurs when the group of catch basins indicated by user does not include one of appropriate size.

Error: Not all junction components have user specifiable elevations
This error occurs when minimum gradient of system segment cannot be determined because of incorrect elevation of duct or catch basin bottom being specified.

Error: Incorrect catch basin bottom elevation specified (segment requires other elevation)
This error occurs when the specified elevation of catch basin bottom does not enable correct arrangement of system segments connected. Specify correct elevation in data table.

Error: Pressure outlet elevation too high in pumping station specified by user
This error occurs when the cover depth of pumping station outlet specified by user is inconsistent with minimum cover depth of system segment. Specify correct minimum cover depth of system segment or cover depth of pumping station outlet.

Error: Pressure outlet elevation too high in pumping station from catalogue
This error occurs when the elevation of pressure outlet of pumping station selected by program is too high to be consistent with minimum cover depth of system segment. Specify correct minimum cover depth of system segment.

Error: Junction forces excessive depth
This error occurs when the specified elevation of junction bottom does not enable arrangement of system segment within maximum depth. Specify correct maximum cover depth for the system segment or correct elevation of catch basin bottom.

Error: Junction precludes minimum depth
This error occurs when the specified elevation of junction bottom does not enable arrangement of system segment within minimum depth. Specify correct minimum cover depth for the system segment or correct elevation of catch basin bottom.

Error: Incorrect elevation of junction bottom
This error occurs when the specified elevation of junction bottom does not enable arrangement of system segment. Specify correct elevation of junction bottom.

Error: Pumping station output too high
This error occurs when the specified output of pumping station does not enable arrangement of part of pressurized system. Increase pumping station output.

Error: Wrong type of pumping station for specified flow rate
This error occurs when the selected type of pumping station is not appropriate for the flow rate specified. Select other type of pumping station in data table or change flow rate.

Error: No source for receptor
This error occurs when the system design is incorrect. Water source should be specified in system segment where the receptor is located.

Error: Water receptor inlet elevation not specified
This error occurs when the elevation of water receptor inlet, necessary for proper arrangement of the system, has not been specified. Specify the elevation in data table.

Error: Water source pipeline axis elevation not specified
This error occurs when the elevation of water source, necessary for proper arrangement of the system, has not been specified. Specify the elevation in data table.

Warning: No drain in junction
This warning appears when a spot is found in the water system where a drain should be located. This is just an information, as the program does not enable placing such components.

Warning: No vent in junction
This warning appears when a spot is found in the water system where a vent should be located. This is just an information, as the program does not enable placing such components.

Hint: Minimum angle between system segments not maintained
This hint appears when the maximum angle between system segments of pressurized or water system specified in calculation options has been exceeded.

Warning: Open basin level above elevation of duct end bottom
This warning appears when specified elevation of open basin level is above the elevation of drain water duct. Specify correct elevation of open basin in data table.

Error: Water source pressure too low
This error occurs when the specified pressure at water source is too low. Increase pressure at source.

160

Error: No receptors

This error occurs when the system design is incorrect. A system segment lacks receptors. Check junction functions in data table.

Error: Undefined system outflow equivalent

ANNEX C – COMBINE MAPS

C.1. Introduction

The "Scan Combine" application is an additional tool that facilitates scanning a large map or plan of building (base drawing), e.g. of A0 format, when you can use only a small size scanner, e.g. of A4 format. The program adjusts the settings of individual fragments, pastes them together and finally trims and scales the base drawing. The functional principle of the application is the identification of identical points characteristic of the entire base drawing and on each individual fragment.

Drawings are saved to files with an „.nts" extension. These files can be loaded onto worksheets in the graphical editor.

C.2. General scheme of using the application

When using the application, follow the steps listed below:

1. Load or scan the first part of base drawing
2. Align the first part to vertical
3. Optionally, specify the number of parts and the sequence of appending them ("Use navigator" in the right middle part of screen)
4. Begin appending subsequent parts
5. Load the next part
6. Indicate identical characteristic points in both parts of base drawing and append the loaded part
7. Repeat steps 5 and 6, depending on the number of parts
8. Trim the base drawing
9. Scale the base drawing
10. Save the drawing to disk.

! ***IMPORTANT! Try to save the drawing frequently, preferably after each completed stage.***

It is possible to undo only one step, as the previous steps take up a lot of memory.

C.3. Rules for scanning base drawing fragments

With "Scan Combine" you can scan fragments of the base drawing using TWAIN compatible scanners or load drawing fragments scanned previously with the use of other applications.

♦ To scan a base drawing fragment:
 1. Click "Scan"
  2. Follow instructions displayed.

♦ To load a base drawing fragment scanned with the use of other application:
   1. Click "Load"
   2. Select file to be loaded and click "Open".

   When scanning try to follow these guidelines:
   1. Drawing fragments should be scanned as monochrome images (1 bit) or 256-colour images (8 bit). This is perfectly sufficient and the file size is kept within reasonable limits.
   2. The drawings should be scanned with overlaps in each fragment, so that at least two common spots can be defined in every two adjacent fragments
   3. Try to scan fragments without turning the base drawing
   4. Usual scanning resolution is 150 or 300 DPI with multicolour drawings and 300 or 600 DPI with black and white drawings. Generated files have reasonable sizes. Of course, other DPI settings are allowed.

! ***To assure that the size of scanned file is as small as possible, it is necessary to perform the scanning of drawings is smallest possible resolution and in black and white version.***

   The scanned fragments of a base drawing may look like in the example below. Crosses indicate common characteristic points. Of course, you don't have to draw any crosses on the base drawing before scanning it. For a characteristic point you can select, for instance, a corner of a building, crossing point of lines or ground elevation designation.
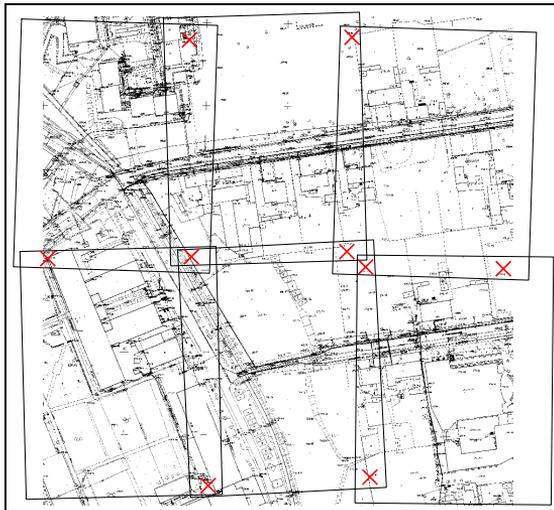
C.4. Operations carried out on the first fragment of base drawing

   After the first fragment of the drawing has been scanned or loaded, you can display it and move it on the screen using the mouse (with the left button pressed) and you can also do the following:
   1. Adjust the angular deviation of the fragment so that the left edge of the drawing or a line you have indicated is vertical

2. Specify the number of remaining parts and manner of their appending – this function, called the "navigator", makes the operations systematic and frees the user of specifying the type of appended parts.

♦ To adjust the angular deviation of the first fragment of the base drawing:
   1. Click "Deviation adjustment" on the right side of the screen
   2. Click on the upper and then on the lower point, which after rotating are to indicate vertical line in the drawing. The point to be marked is shown on the right side of the screen
   3. Click "Rotate" – the scan or drawing fragment is rotated so that the marked points are aligned vertically.

♦ To define the settings of the "navigator", i.e. to specify the number of appended fragments and their sequence:
   1. Check "Use navigator", if it is not checked yet
   2. Specify the number of fragments you want the base drawing divided in horizontal and vertical directions. In the example above the drawing is divided into 3 fragments in the horizontal direction and 2 fragments in the vertical direction
   3. Determine the sequence of appending the fragments. This sequence is represented by one of two selectable icons.

C.5. Appending subsequent fragments of the base drawing

To proceed with appending subsequent fragments of the base drawing, click on the "Next" button in the upper right corner of the screen. The entire base drawing is displayed on the left side, while the fragment being appended is displayed on the right side. If you use the "navigator", then the project status and position of the fragment being appended are displayed in the middle of the screen.

♦ To append the next fragment of the base drawing using the "navigator":
   1. Load or scan the next fragment of the base drawing. Make sure with the navigator (in the middle of the screen), which part is being appended
   2. Mark the same characteristic point on the entire drawing preview and on the fragment being appended. To see which point is expected – look at the "Mark point" frame. Points are marked with the **RIGHT** mouse button. In case of error you can change the setting in the "Mark point" frame and click again
   3. Mark the second characteristic point
   4. Mark the third characteristic point - this is required only when two sides of the part being appended adjoin the fragments of the base drawing already appended
   5. Click "Append".

Append the subsequent fragments of the base drawing using the "navigator". After appending all parts click "Next" to move to the last stage – trimming and scaling.

If the "navigator" is not enabled, then the procedure is similar to that described above, you only have to specify the "Append method" and "Append direction" for each fragment.

C.6. Trimming and scaling the base drawing

In the last stage of combining the scanned drawing, you can trim it (cut off unwanted margins) and scale it so that it can be loaded onto the graphical editor. There are two ways of scaling the map (base drawing):

1. Based on the known scale of the base drawing and selected DPI scanning resolution (DPI – dots per inch) – usually the following resolutions are available: 100, 150, 200, 300 and 600 DPI
2. Based on the specified actual distance between two marked points

♦   To trim the drawing:
1. Select the "Trim" tab in the right part of the screen
2. Mark with the **RIGHT** mouse button two points (upper left and lower right) of the final form of the base drawing
3. Click "Trim".

♦   To scale the base drawing based on known scale and DPI:
1. Switch to the "Scale" tab in the right part of the screen
2. Check the "Base scale and DPI" box
3. Specify the scale and DPI
4. Click "Scale" – the "m / pixel" conversion factor is then displayed.

♦   To scale the drawing based on the actual distance between two marked points:
1. Switch to the "Scale" tab in the right part of the screen
2. Check the "Distance between marked points" box
3. Mark two points on the base drawing (map) with the **RIGHT** mouse button
4. Enter the actual distance between the points in metres
5. Click "Scale" – the "m / pixel" conversion factor is then displayed.

After combining all fragments of the base drawing and saving it to the disk, click on the "End" button to exit the "Scan Combine" application.