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CP-System

User's guide

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Introduction

About CAD-Profi programs

CAD-Profi software consists of programs „CP-Manufacturers", „CP-Symbols" and „CP-System".

- **CP-Manufacturers** – the program is a kind of electronic catalogues that contain libraries of blocks of known manufacturers. It is distributed by manufacturers of devices. „CP-Manufacturers" is included in „CP-System" program.
- **CP-Symbols** – it is a program containing complete libraries of symbols for various branches as well as basic commands supporting schemas edition. Available modules: "HVAC", „Electrics" „Mechanics" and "Architecture".
- **CP-System** – a comprehensive application that supports computer designing of installations. There are available two basic modules: **"HVAC"** and **"Electrics"**, in which a designer can create any schemas, plans and sections. The following modules are also available: **„Architecture"** – it contains elements that support creating of architectural projects,
  „Mechanics" – it contains an extensive set of standardized elements used in mechanical and structural projects.

CAD-Profi programs can work as applications for various CAD programs. It means that **CAD program** is understood as AutoCAD or IntelliCAD program in this manual. Many commands in these programs work similarly. If there are any differences, it is described in details.

CAD-Profi Programs can cooperate with the following CAD programs:

- AutoCAD from 2000 version
- AutoCAD LT from 2000* version
- Most IntelliCAD distributions from 4.0 version.

*) in AutoCAD LT, there is necessary to install additionally LTX™ or LT-Extender program

Important information about working in CP-System

Designing in CP-System is compatible with general rules of designing in AutoCAD /IntelliCAD programs. The most important benefit of the program is that the program automates the most drawing activities and provides an extensive base of objects and solutions ready to use in created projects.

CP-System commands work in an intuitive way. However, it is strongly recommended to read this user's manual because it allows a user to use full possibilities of the program.

To work comfortably in CP-System it is necessary to get to know the most important assumptions of the program:

- **Compatibility with dwg, dxf formats**
  Designs that are drawn in CP-System are saved in dwg or dxf format. These drawings can be edited in CAD programs, even without loading CP-System application.

- **Standard drawing**
  CP-System inserts layers, text styles, dimension styles and linetypes definitions saved in a template file into every new drawing. A file normal.dwg/normal_lt.dwg is a template file for AutoCAD/AutoCAD LT, whereas a file normal_ic.dwg is a template file for IntelliCAD. These files are stored in a folder *..\Block\General*
  By changing template files a user can change or expand standard definitions of styles that are being loaded.

- **Drawing scale**
  A user should make all projects to a scale of 1:1 in model space and in selected unit of designing (mm, cm, m).
  A user specifies target scale of drawings either during printing or when he works in paper space.
  It is necessary to enter scale in some commands. The most often this scale is used to specify an appropriate size of texts.

- **Linetype scale**
  A user can design in various units (mm, cm, m) in CP-System. If he wants to change linetype scale, CP-System will change automatically a global scale factor.
  A default global scale factor is 99.9 for designing in mm, 9.99 in cm and 0.0999 in m. Default linetype scale is appropriate for building projects as well as projects of installations whereas it is often necessary to change linetype scale for mechanical projects.
  If a global scale factor is 1.0 in saved drawing, CP-System will change it for one of default values when a drawing is open. A user should set any linetype scale other than 1.0 or default values by himself to make CP-System not to change a global scale factor.
  Changing of linetype scale is done by the command _LTSCALE.

Once a command has been started, a user should enter a required value, e.g. 5. This value will be correct e.g. for most mechanical drawings drawn in mm. Here is an example of changing of linetype scale:

```
Command: _ltscale
Enter new linetype scale factor <99.9000>: 5
```

During starting some commands (in mechanical module) CP-System suggests turning off automatic matching of linetype scale. The automatic matching of linetype scale to units can be turn on again in any moment in the dialog box **CAD-Profi Options**.

- **When dialogs are not displayed** (e.g. in the **Save** command)
  Some CP-System commands block displaying standard AutoCAD / IntelliCAD dialogs. CP-System restores displaying of them after a user has finished such commands normally.
  Sometimes a user breaks a CP-System command, e.g. by pressing ESC keystroke. In such cases CP-System may not restore displaying of standard dialogs.
  If a user wants to restore displaying of them, he should set variables FILEDIA and CMDDIA to 1. In order to do that a user should enter a name of the variable in command line and set its value to 1.

  ```
  Command: filedia
  Enter new value for FILEDIA <0>: 1
  Command: cmddia
  Enter new value for CMDDIA <0>: 1
  ```

  A user can restore displaying of standard dialogs by means of **CAD-Profi Options** command.

- **Cartesian coordinate system**
  The most CP-System commands can be used in UCS. However, some commands require the coordinate system to be set to WCS. It is the default coordinate system using by AutoCAD/IntelliCAD programs.

- **Layers**
  A user can set an automatic managing of layers to unchecked in the CP-System settings. However, it is not recommended because some objects properties depend on layers. For instance, when a user makes specifications, he can choose automatically objects from given module (e.g. electrics symbols). If automatic managing of layers is unchecked, a command will not be able to identify symbols of a given module.
  Plot styles that specify optimal lineweights on layout are added to CP-System. These styles are matched to automatic layers structure.

- **Toolbars**

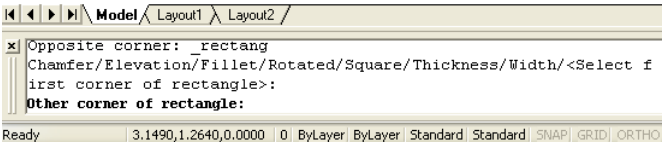
  In programs AutoCAD / IntelliCAD a user can adapt both menus and toolbars. Using **CAD-Profi Configuration** program he can attach and detach CP-System to or from the CAD application. When a user attaches an application to the program, a standard CP-System menu and toolbars are loaded. At the same time, any changes made in toolbars and menus will be overwritten.

- **Specification – print preview**

  A user can create specifications and professional plots in CP-System. He can save a print preview in pdf files or in his own format – in files with LL extensions. To browse LL files a user can use LLview13.exe file which is placed on the disk in main CP-System folder (e.g. *C:\CadProfi\xx*).
  A user can copy and distribute the LLview11.exe program for free. This program is translated into many languages, it has a small size and does not need any special hardware configurations. Thanks to it, a user can create files in LL format and make them available with a viewer. Both LL files and a viewer can be attached to an e-mail and sent between cooperating companies from any countries. No one should have any problems with opening and printing LL files.

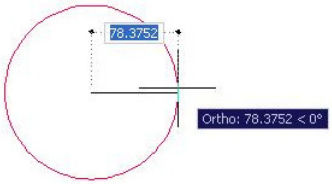
- **Command line**

  From AutoCAD 2006 and in IntelliCAD, it is a possible to turn off command line. AutoCAD / IntelliCAD and CP-System application display important information in a command line. Because of it, a user should turn on a command line with at least two visible lines.



Command line of AutoCAD program

A user can turn on a dynamic command line from AutoCAD 2006. A dynamic command line displays only one line, furthermore, it can slow down an application. (e.g. during editing many objects simultaneously). That is why a dynamic line is turned off by many commands of CP-System.



The dynamic command line of AutoCAD

Turning off both classical and dynamic command line is particularly inconvenient, because a user will not be able to see the state of currently processed command.

- **Expansion of libraries and catalogues of manufacturers**
  A user can expand libraries of symbols and objects of CP-System. Once the program has been installed, a user can install new libraries of manufacturers, e.g. the ones that have been created after buying CP-System. Both user's and manufacturers' libraries are stored in folder *Producers*. This folder is placed in main folder of the program (e.g. *C:\CadProfi*). If a user wants to save any changes that were done in manufacturers' libraries as well as in his own symbols' and objects' libraries, he should archive all or only selected libraries from the folder *Producers*.
  After possible re-installing of the CP-System or during transferring it to the other computer a user can transfer content of the *Producers* folder to new installation. There is also possible to copy libraries in one company.

- **User's database – folder UserData**
  A user can define various parameters or expand standard databases. He also can, for instance, specify additional kinds of installations and pipes or ducts; define additional parameters for objects (e.g. price, colour, currency etc.).
  Standard databases were separated from user's ones in CP-System. Thanks to that, a user can update the program without losing data that he saved by himself. During updating standard databases are overwritten or placed in the other folder and user's databases remain untouched.
  A user can save his data in the *UserData* folder. This folder is placed in main folder of the program (e.g. *C:\CadProfi*). If a user wants to save all changes in user's databases, he should archive all the *UserData* folder. When a user re-installs CP-System or transfers it to the other computer, he can copy content of the *UserData* folder to new installation. It is also possible to copy *UserData* data in one company.

Installation

In order to install CP-System program, a user needs to have rights either of advanced user or administrator. Once the application has been installed, it can be used by a user with limitations. It is important that a user has rights to save in a folder, where the program is installed.

Installation of the program

After inserting the CD to the CD-ROM driver a user will see a window, where he can either start an installation of the program or choose the other options. If the window does not appear, it will mean that a function of an automatic playing was turned off for a CD-ROM driver. Then a user should start the *autorun.exe* program placed in the '*Start*' folder on the CD-ROM driver. Once an installing program has been run, a user should follow instructions appearing on the screen.

Choosing installed components

All modules of CP-System program are placed on a CD-ROM disc. During installation of the program, a user can install all available components. They will work 30 day after first running of the program with no limitations. After that they will be disabled unless activation codes will be entered. A user receives activation codes for the modules that were bought.

There are a lot of producers' blocks, PDF files as well as other components in CP-System. A user can install the program remaining selected libraries on CD. However, CP-System will display an alert box with a request to insert appropriate CD every time a user wants to use these libraries.

Installation of additional components

CAD-Profi programs are developed all the time. Once a user has received the newest version he can add additional components.
To add new or updated components, a user has to run the installation program. This program will find a previous version and it will install new components in the same folder on the hard disk.

Configuration of CP-System to the CAD program

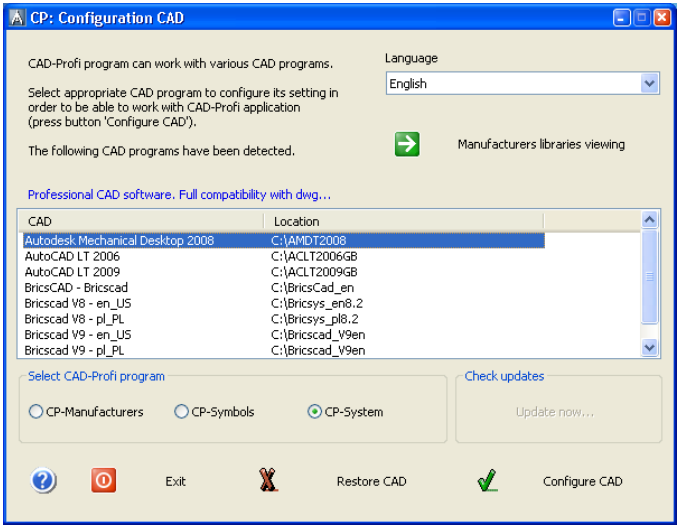
Configuration of settings of CAD program

When a user has finished an installation of CP-System , he should run “CAD-Profi Configuration” program to configure AutoCAD / IntelliCAD settings.

A user will be asked for choosing a required CAD program in main window of configuration program and then for pressing the **Configure CAD** button**.** Configuration program will do suitable changes.
In case of using AutoCAD any changes will be done in a current AutoCAD profile. A user can also create a new profile in AutoCAD program and later he can configure it using ”CAD-Profi Configuration” program. This may be necessary e.g. during using a few various applications. For more information about profiles, see AutoCAD user’s guide.

When a user has already finished the configuration, he can start AutoCAD / IntelliCAD (using either its icon or its shortcut). Additional main menu and toolbars will be activated in CAD program.

CP-System program is developed as an application of AutoCAD® program as well as the other solutions that base on IntelliCAD / AutoCAD® (Architectural Desktop®, AutoCAD Mechanical® and other). However, additional components (LTX™ for CP-System or LT-Extender) make CP-System to be used with AutoCAD LT®.



Restoring of CAD program settings

To restore previous settings of CAD program, the **Restore CAD** button is used. Once it has been done, it is recommended to rerun CAD program.

Only if CAD program is turned off can a user do configuration and restoring its settings. Otherwise, changing of setting may not give any effects.

Notes about IntelliCAD program

During configuration of CP-System to IntelliCAD, it is recommended to use „CAD-Profi Configuration" program. This program searches automatically the most popular distributions of IntelliCAD. If a user uses untypical distributions, he should add line

```
RegICADKey=NameICAD
```

in *C:\CadProfi\xx\cpabout.ini* file which is placed in [Setting] section.

'NameICAD' is a main name of a key of IntelliCAD program, which is used in HKEY_CURRENT_USER\Software branch in registry. This name can be obtained from a distributor of IntelliCAD program.

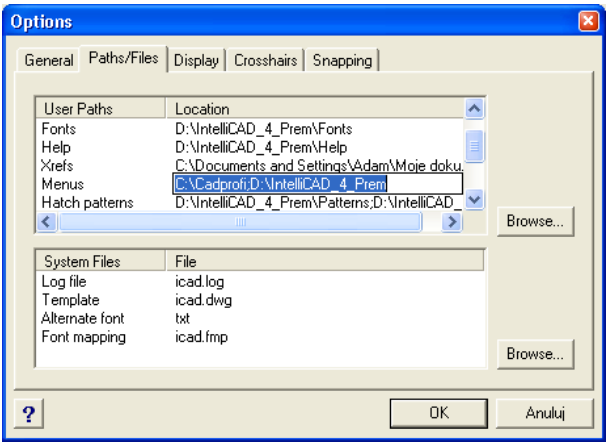
An exemplary entry for different versions of IntelliCAD:

- BricsCAD IntelliCAD – `RegICADKey=Bricscad`
- ProgeCAD IntelliCAD – `RegICADKey=Progecad`

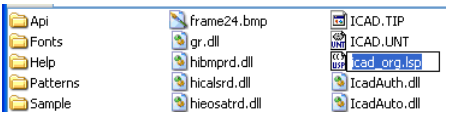
Once a user has added the entry „CAD-Profi Configuration" program will find given version of IntelliCAD and will make appropriate configurations. Some distributions of IntelliCAD program are not compatible with a kernel of IntelliCAD. Some problems can occur in these distributions by automatic configuration of CP-System and manual configuration is necessary.

There is also another possibility. A user can specify settings in IntelliCAD program by himself. To do this:

- Choose the **Options** command from the main menu **Tools**, and then activate tab **Paths/Files**. Add the path *C:\CadProfi\xx* (or other, depending on a folder where CP-System was installed) in the position **Menus**.



- If *icad.lsp'* file is in a folder where IntelliCAD is installed, its name should be changed for *icad_org.lsp'*. This file will be loaded automatically by CP-System application.



If a user specifies these settings, he should close IntelliCAD program. The next run of IntelliCAD will also load CP-System automatically.

Notes about using AutoCAD LT Assistant
Using CAD-Profi programs in **AutoCAD LT**, it is strongly recommended to close the Assistant.

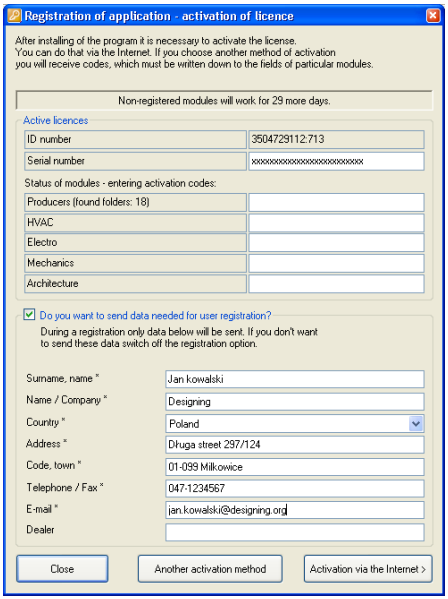
Registration and activation of the program

During starting of CAD-Profi program, dialog **Registration of the application** is displayed. A user can either activate the program or press the **Close** button to start the program.

Dialog **Registration of the application** is being displayed until the activation code is entered. A user has to enter the code within 30 days. Otherwise CAD-Profi program will not be run any more.

If a user has connection to the Internet, he can activate the program automatically by using the **Activate by the Internet** button. If the activation by the Internet is not possible, **Another activation method** button should be pressed. In this case a form will be displayed which needs to be sent by an e-mail. A user can also print it and send it either by fax or by mail.

After that he'll receive activation codes which he should enter in appropriate fields of the dialog.



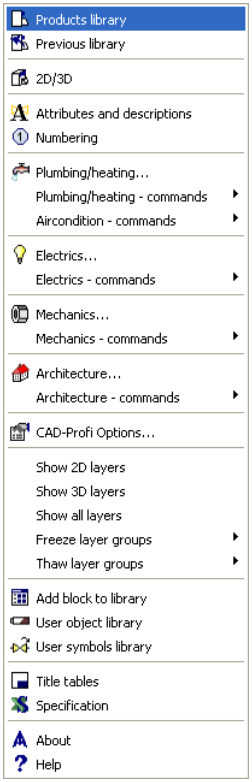
Notes:

- When the checkbox: "Do you want to send data needed for user registration" is checked, **only** information in dialog will be sent. We encourage you to register because only registered users are informed about new CAD-Profi versions. If a user is registered, he can also get access to updating or to take additional components by the Internet.
- Authorization codes are appropriate only for one computer. If a user wants to transfer the license to other computer, he needs to activate program once again.

Settings and interface of CP-System

Menu and toolbars

When a user has started CAD program additional toolbars and additional main menu of CP-System will appear.



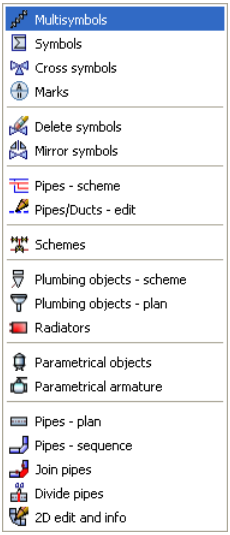
Main toolbar

- Products library
- Previous library
- 2D/3D
- Attributes and descriptions
- Numbering
- Tables/Frames
- Specifications
- Add block to library
- User's objects library
- User's symbols library
- CAD-Profi Options
- About
- Help

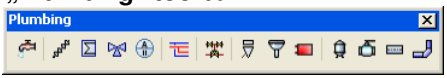
„CP-System Edit" toolbar

- Delete symbols
- Mirror symbols
- Pipes / ducts / conductors - edit
- Join pipes / ducts / cable trays
- Divide pipes / ducts / cable trays
- 2D edit and info

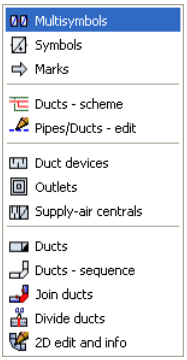
Commands of „CP-System Edit" toolbar are used to edit objects created in all modules.



„Plumbing" toolbar

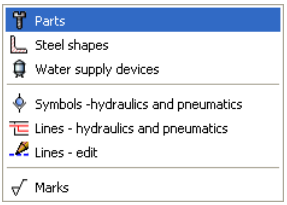


- Plumbing
- Multisymbols
- Symbols
- Cross symbols
- Marks
- Pipes - schema
- Schemas
- Plumbing objects – schema
- Plumbing objects – plan
- Radiators
- Parametrical objects
- Parametrical armature
- Pipes - plan
- Pipes - sequence



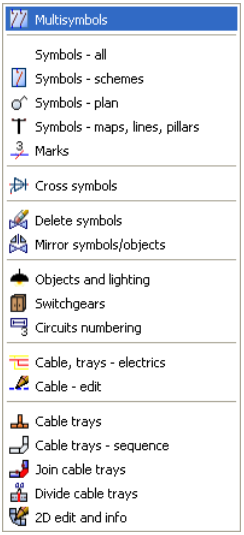
„Air condition" toolbar

- Multisymbols
- Symbols
- Marks
- Ducts - schema
- Duct devices
- Outlets
- Supply-air centrals
- Ducts - plan
- Ducts – sequence

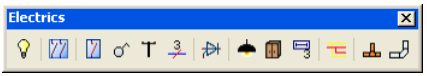


„Mechanics" toolbar

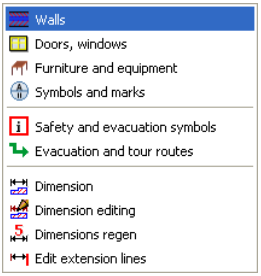
- Mechanics
- Parts
- Steel shapes
- Water supply devices
- Symbols – hydraulics and pneumatics
- Lines – hydraulics and pneumatics
- Marks



„Electrics" toolbar



- Electrics
- Multisymbols
- Symbols – schema
- Symbols – plan
- Symbols – maps, lines, pillars
- Marks
- Cross symbols
- Objects and lighting
- Switchgears
- Circuits numbering
- Cable trays - schema
- Cable trays - plan
- Cable trays - sequence



„Architecture" toolbar

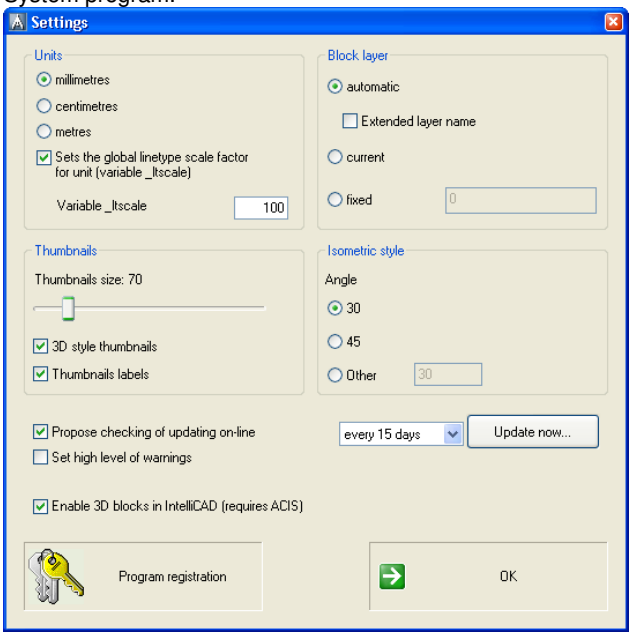
- Architecture
- Walls
- Doors, windows
- Furniture and equipment
- Symbols and marks
- Safety and evacuation symbols
- Evacuation and tour routes
- Dimension
- Dimension editing
- Dimension regen
- Edit extension lines

Commands repeating - CPX

Either a right mouse button or ENTER are used to repeat the last command in AutoCAD or IntelliCAD. It is often necessary to use alternately CAD program commands (e.g. copy) as well as CP-System commands (e.g. inserting symbols). That is why CPX command was introduced. It repeats the last command of CP-System application. To use this command, a user should enter CPX command in AutoCAD or IntelliCAD command line and press ENTER.

CAD-Profi - Options

The CAD-Profi Options command is used to specify basic parameters of CP-System program.



Units

It is recommended that a user creates drawings at a scale of 1:1 in CAD programs. The scale of the drawing is specified only during printing.

Example:
Real dimensions of inserting device are 600 mm x 400 mm
If a unit is specified as:
mm – 600 x 400 dimension object will be inserted (in drawing unit)
cm – 60 x 40 dimension object will be inserted (in drawing unit)
m – 0.6 x 0.4 dimension object will be inserted (in drawing unit).

When a user designs projects on building plans he needs to check the unit of a source drawing. The best way is to use the **Distance** (_DIST) command, to measure the distance between two points, for instance a door. Depending on that distance a user needs to set unit of designing for CP-System program. There is also possibility of rescaling the whole building plans to fit the size of drawing to required scale. For instance, if a drawing had been drawn in cm, a user can rescale it 10x and use mm to design in CP-System.
Sometimes building plans are drawn for example, at a scale of 1:50. CP-System cannot work in a scale, that is why a user has to rescale such drawings to standard units: mm, cm or m.

The **Set the global linetype scale factor for unit (variable _ltscale)** command activates automatic setting of density of dotted or dashed lines. Density of lines is appropriate to CP-System installation projects, architectural drawings, etc. In case of mechanic projects a user is recommended to specify optimal linetype scale by himself for instance using _LTSCALE.

Layers

A user can use automatic managing of layers in CP-System program by using of internal structure of layers. It is very comfortable property because a user does not have to create layers and place objects on individual layers. Layer structure of CP-System program is used to automatic creating of specifications, numbering and in commands that make possible quick turning on or off layer groups (e.g. quick hiding of all objects belonging to central heating system). These commands recognize objects among others by layers. That is why, it is recommended to use system of **automatic** managing of layers.
If a user turns on the **Extended layer name** option, CP-System program will create separated layers for objects that belong to various installations.
Example: symbols of plumbing are inserted on 2DH_SYM layer. When a user has turned on **Extended layer name** option, symbols inserted in pipes 'Supply', 'Return' etc. will be on layers 2DH_SYM_s, 2DH_SYM_r etc.

Using of the extended layers structure is necessary by creating separated specifications for individual kinds of installations (e.g. Inlet, Outlet, Supply, Return). However, a lot of layers will be created in a drawing in this case.
A user can turn off an automatic managing of layers in CP-System.
If the **Current layer** option is active, all objects will be inserted on a current layer that can be changed by a user.

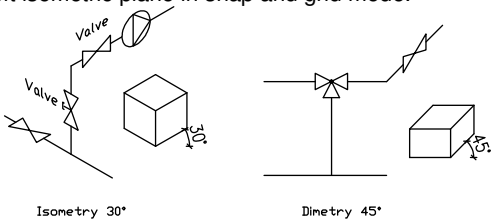
If **Fixed layer** option is turn on, all objects will be inserted on a fixed layer which name is specified in a dialog. The colour, the linetype and other properties of a fixed layer can be specified using appropriate commands of CAD program.

Note: CP-System identifies type of pipes by means of layers. That is why schematic pipes or ducts or cable trays and fittings are always inserted on automatic layers.

Additional information about layers a user can find on page 108.

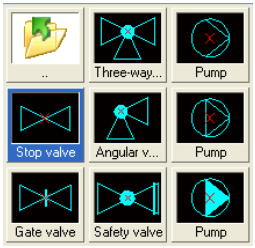
Style of isometry

A user can create isometric drawings in CP-System. It is possible to create rectangular dimetry and dimetry at any angle.
When inserting symbols in axonometric drawings CP-System match automatically an obliquing angle. It is done according to an angle that is specified in **CAD-Profi Options** dialog. Most often an angle 30° is used for isometry and an angle 45° is used for dimetry. A user can specify other angle between 5° and 85°.
While a user is creating an isometry, AutoCAD program (or IntelliCAD) makes a special snap and grid mode available. A graphic cursor is matched to an upper, a right or a left isometric plane in snap and grid mode.
For other kinds of an axonometric drawing CP-System program allows switching automatically a obliquing angle of a graphic cursor compatible with a current isometric angle.

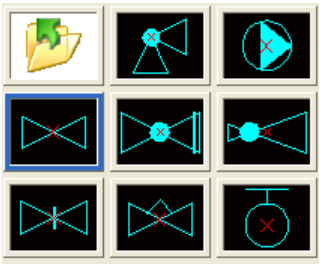


Miniatures

There is used a miniature view of symbols and objects in some dialogs. A user can change the size of miniatures and their style.



Miniatures with labels, 3D style, size 70



Miniatures without labels, flat style, size 80

Option – Enable 3D blocks in IntelliCAD

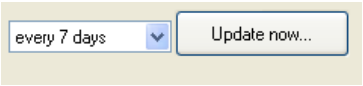
In IntelliCAD, there is an additional option – 'Enable 3D blocks in IntelliCAD'. **Only if** a user uses IntelliCAD version where edition of 3D objects is possible, can his option be turn on.

Option – Set high level of warnings

Many messages and warnings are displayed in a command line by CP-System. It is related to informational messages which are not critical. When a user has turned on high level of warnings, some messages will be displayed in a dialog and it will be necessary to close them.

On-line update

Manufacturers' libraries are updating systematically in CP-System program. Content is expanded and new functions are added. System of automatic program updating by the Internet was introduced in order to update new elements quickly. This program works similarly to Windows updating.



When a user has turned on the **Propose checking of updating on-line** option CP-System will be reminding about possibility of on-line update every 7, 15 or 30 days.
If a computer is connected with the Internet at that particular moment, a user can run Updating program. This program allows a user to check available updates and download them as well as installing selected updates.
The **Update now…** button allows a user to run Updating program in any moment.

Program update runs in a few steps:

1. **Checking of updates** –the program is connecting with CAD-Profi Server, checks and displays all updates that can be downloaded for the given computer.

2. **Selecting updates** – A user can select updates that he wants to download in particular moment. If a selected update is not selected, a user will be able to download it later.
   Total size of files that will be downloaded from server is displayed under the list of available updates.
3. **Downloading files** – a user downloads all available updates. Updated program has an option of transfer resuming. If a connection with server is disconnected, downloading updated files will be resume from disconnected place. During downloading big updates, the program can work in the background and a user can continue his work in CAD program.
4. **Installation** – after downloading files a user has to install them.
   It is necessary to close CAD program during the installation. When a correct installation has finished, **CAD-Profi Configuration** program can start. This program configures CP-System to a CAD program with all changes that were downloaded in updates.

By installing of updates, it is possible to turn on the **Make a copy of updated components** options. It guarantees that copies of all updated files will be saved. Later these copies can be used to restore previous version of the program using the **Restore previous version** button.

Notes:

- After restoring previous version of the program, it is recommended to start CAD-Profi Configuration in order to configure Cp-System to CAD program with parameters of previous version.
- **CAD-Profi on-line Update** program requires connection to the Internet using http protocol.
  If it is corporate network, administrators may sometimes introduce limitations on the Internet access. Then updated program will not be able to connect with the CAD-Profi Server.

Policy of keeping privacy of users during on-line update
To save user's privacy, the program neither send nor collect any user's data. The only elements which are sent:

- A serial and an ID number of CAD-Profi program
- information about installed components of CAD-Profi program

These data are necessary to specify availability of updates for a computer. They are also necessary to check rights to download updates as part of a program subscription with a given serial number.

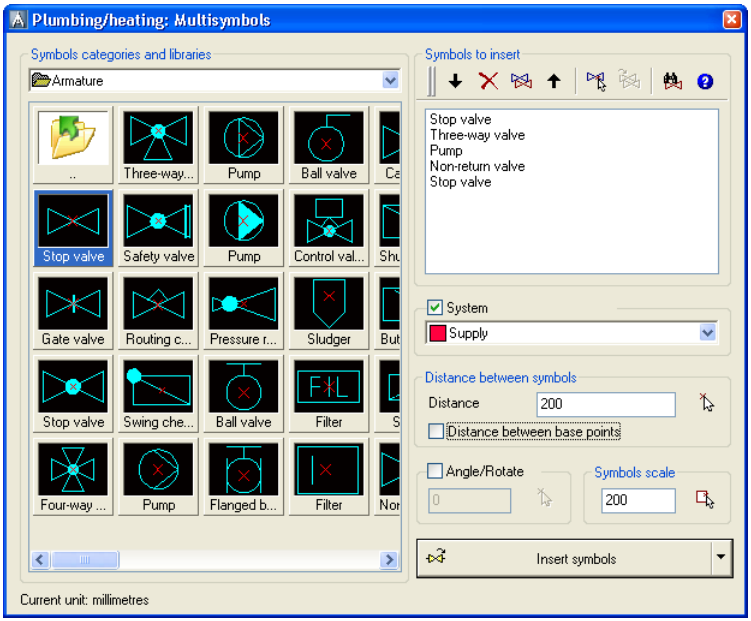
HVAC, Electrics

Edition of schemas

A user can create any installations schemas in CP-System program. Designing of schemas a user can use intelligent symbols, pipes and functions that are used to describe drawings and to create specifications.
All CP-System symbols have specified properties and that is why they behave in an intelligent way, for instance, match automatically an inserting angle to a pipe and break automatically a pipe in a connection points. Symbols can be described automatically as well as can be used to automatic creation of specifications.
Multisymbols, **Symbols** and **Marks** commands are used to insert symbols into drawings.

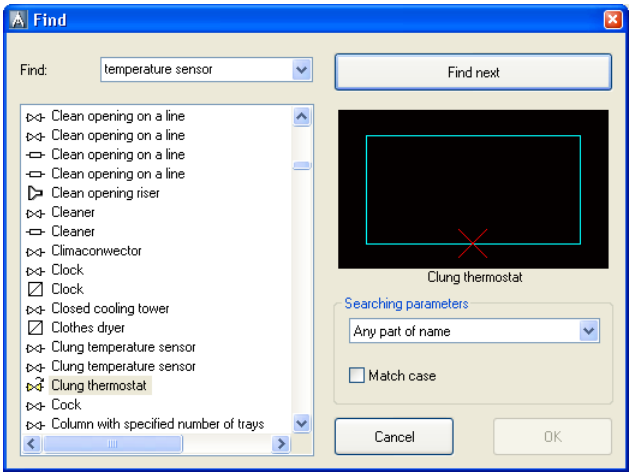
Multisymbols

Multisymbols command allows a user to select many symbols that will be inserted into a drawing at the same time.
Symbols are grouped in categories. It makes easier quick choosing symbols used to designing specified kind of installation.



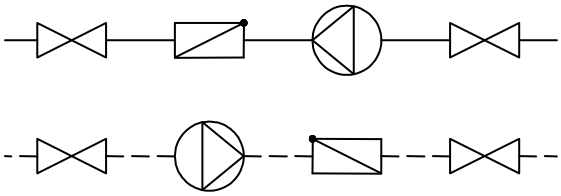
First a user has to choose a category of symbols and later select (by clicking) miniatures of demanded symbols. Selected symbols will be added to a list of symbols. This list can be organized in an easy way, for example, by dragging or by using of buttons : (**Move down, Delete, Copy, Move up**). A user can also choose symbols by using **Select symbols, Most recently selected symbols** or **Find** commands:

- **Select symbols** command allows a user to select symbols from a current drawing. During selecting symbols a dialog is hidden. Selecting symbols directly on the drawing makes designing faster. It also allows a user to do intelligent copying of symbols with an ability to change their order, size, angle or distance.
- **Most recently selected symbols** option allows a user to select symbols that have been used recently during using **Multisymbols** command.
- **Find** option displays additional dialog where a user can find symbols according to given words.



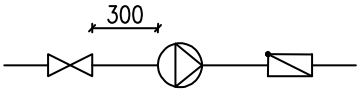
Multisymbols option

System – during inserting a few symbols CP-System is drawing pipes between particular symbols. If a user inserts single symbol in a drawing, a pipe will not be drawn. Using extended structure of a layer, a selected pipe specifies a layer where a symbol will be inserted. For instance, the ‘Supply” pipe symbol will be inserted in 2DH_SYM_s layer, the ‘Return’ pipe symbol will be inserted in 2DH_SYM_r layer. If ‘System’ is turned off or if extended layer name is not used, symbols will be inserted in 2DH_SYM layer.

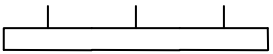


Various pipes drawn between symbols

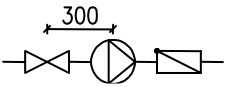
Distance between symbols – inserting a few symbols a user should specify distance between them. He can do this in two ways. First one: specifying the length of sections that are drawn between symbols. Second one: specifying the distance between base points of particular symbols.
If a user set distance 0, symbols will be drawn one after the other. This option is used, for instance, during inserting a few symbols of distributors.



Distance between symbols

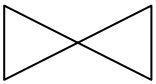


Distance '0' between symbols



Distance between base points

Symbols scale – specifying a size of inserted symbols.
Designing schemas, dimensions of symbols does not have to be the same as real dimensions. If a user wants to have readable drawings, he should keep an appropriate ratio between other symbols and objects. In CP-System, there are schemas where 200-scale symbols were used. It is recommended to use 'mm' unit for this scale. If a user uses 'cm' units then the recommended scale will be 20, and for 'm' units the recommended scale will be 0.2.

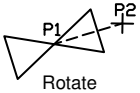


Scale 200

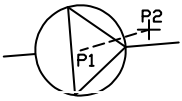
Scale 100

Angle / **Rotate** – a user can specify a fixed angle of inserting symbols or turning on an option rotate and insert symbols at any angle.
When symbols are inserting in pipes, an angle of inserting depends on an angle of a pipe (an angle specified in dialog is ignored).
When inserting symbols in pipes, a user can use **Rotate** option. This option is used to dynamic specifying a rotating of symbol regarding a pipe. It is very comfortable during inserting symbols of pumps, valves etc.

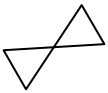
0 angle



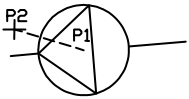
Rotate



Inserting in a pipe with rotation

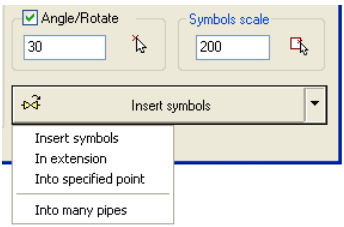

30° angle

Inserting in a pipe without rotation



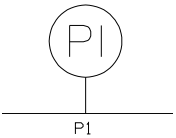
Inserting symbols

A user can use following options to insert symbols:



- **Insert symbols** – it is a default option which can be used to insert symbols in pipes or in any points. Inserting a symbol in a pipe causes automatic breaking of pipe in connected points.

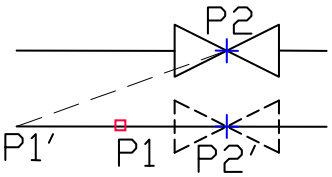
- **Into specified point** – this option allows to insert a symbol in any point. When a user specifies a point lying on a pipe, the pipe will not be broken. This option is used, for instance, to insert sensors etc.



- **In extension** – this option makes possible of precise specifying a point of an inserting symbol towards other objects that are in the drawing.

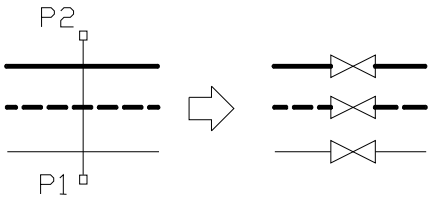
Example of using 'in extension' option: inserting symbol in a pipe in the point that was indicated on the base of other symbol.

1. When a user has selected a command, he should specifies a pipe where the symbol will be inserted (P1).
2. CP-System indicates the end of the pipe which is a point of reference to set the distance (P1').



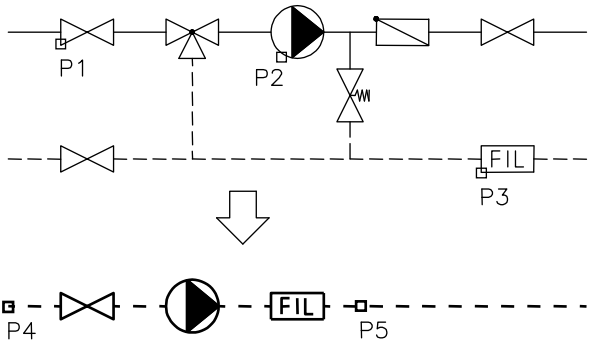
3. Next, a user can specify the distance or specify a point (P2), which sets a new point of reference (P2'). This point is set as a perpendicular projection of P2 point.
4. When P2' point has been set, a user should press ENTER.

- **Into many pipes**– thanks to this option a user can insert quickly a selected symbol in many pipes. After choosing this option it is necessary to select points P1 and P2 that will set line breaking of demanded lines.
  This option works only if one element is selected.



Intelligent copying of symbols

To make a designing quicker CP-System program allows a designer to do an intelligent copying of symbols from a current drawing.
In order to copy symbols, it is necessary to start the **Multisymbols** command, and then select the **Select symbols** option. Program will hide a dialog and a user will be able to select copied symbols (P1, P2, P3). Symbols will be copied in his order as they were selected in a drawing. It is possible to select the same symbol many times. If a designer wants to finish selecting symbols, he should press ENTER (or right mouse button). Once symbols have been selected, the **Multisymbols** dialog will be displayed again. A user can specify all options of inserted symbols in this dialog. Inserting symbols takes place the same way as for symbols selected from program libraries (e.g. inserting in a point or in a pipe – P4, P5).



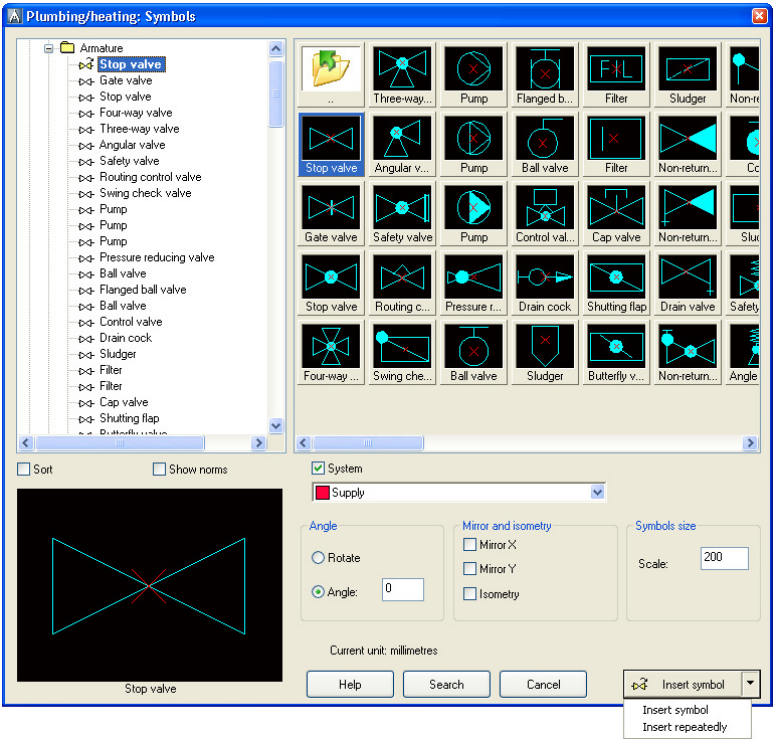
Copying of three symbols with inserting them in new pipe

Note:
If a selected object is not compatible with the **Multisymbols** command, it will not be added to a list of copied symbols. Copying symbols from different modules, for example, symbols from plumbing/heating and electrics is not allowed.

Symbols

The command inserts single symbols.

When a user start the command, a dialog with libraries of symbols will be displayed. To make searching symbols easier, symbols that are available in **Multisymbols** command are also available in **Symbols** command.
All symbols are grouped in categories in the dialog. When a designer has selected a required category, miniatures of all symbols from a given category will be displayed. Moving a cursor over a miniature of a symbol a user will see its full name. After checking 'Sort' checkbox, a list of categories and symbols will be sorted.

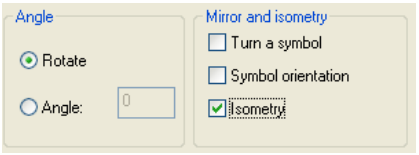


Symbol options

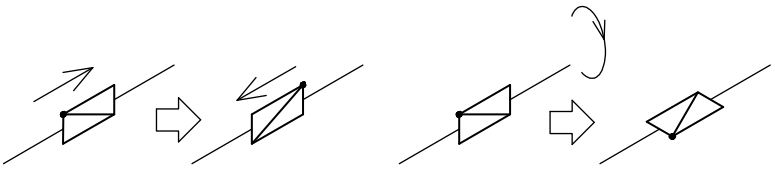
- **Scale** – it specifies size of inserted symbols (see page 23). Some symbols' size has to be fixed, so possibility to change a scale is disabled for them. In this case a default scale is set.
  An example of such symbols can be views of plumbing objects that are

used on plans. The scale suits to vertical dimensions of devices has to be kept on plots. Because of that the program sets automatically appropriate size of symbols and disables possibility to change it.

- **An angle of rotation** – If the **Angle** option is active, a symbol will be inserted with a specific angle. If the **Rotate** option is active, it will be possible to rotate a symbol during inserting it in a drawing. (look page 23).
  Angle and **Rotate** options are switched for many symbols by the program. For instance, **Rotate** option is set for symbol of a pump. Thanks to that during inserting of the symbol it is possible to specify a direction of supply of a pump.
  - **Mirror** – this option allows a user to insert a mirror image of a symbol. After a user has marked the **Mirror X** or **Y** option a preview of a selected symbol is updating systematically. After turning on the mirror X or Y some texts in preview are displayed backwards or upside down. In spite of that preview, during inserting in a drawing texts are corrected automatically (see additional information about attributes at page 30).
- **Isometry** – designing axonometric drawings CP-System makes suitable transformation of inserted symbols. Symbols are given a slope that suit to a current angle of isometry.
  When a designer has marked the **Isometry** option, meaning of others options related to rotating and mirroring is changed. An angle of isometry is a default angle of inserting symbols. This angle is specified in **CAD-Profi Options** dialog (page 18).
  Turn symbol allows a user to insert reverse symbol, e.i. directed in the opposite direction.
  Symbol orientation allows a user to insert symbol that is rotated towards a selected isometry axis.



Symbol orientation can be different in particular isometry axes. It means that by the same settings a given symbol inserted in a right axis may have a horizontal orientation, however, inserted in a left axis may have a vertical orientation. So, if a user wants to design efficiently, he should gain experience designing in CP-System.



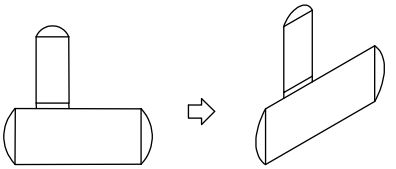
Turn symbol

Symbol orientation

Isometric views for symbols are generated dynamically. Some symbols were not expected to be used in axonometric drawings. For such symbols isometric views can be generated incorrectly.

A user can add new symbols to CP-System by himself (see page 111). These symbols can also be used in axonometric drawings. It is sufficient to add usual symbols to the program and isometric views will be created automatically by the program. During defining his own blocks a designer should avoid using complex objects like: solids, multilines, hatching etc. During defining his own blocks a designer should avoid using complex objects like: solids, multilines, hatching etc.

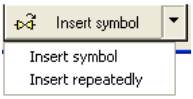
Additional information about creating drawings are available in the **Pipes** chapter (page 36) as well as in the **Attributes and descriptions** chapter (page 92).



User's symbol Automatic isometric view

Inserting symbols

A designer can insert symbols in any point or in existing pipes. When a user has selected a symbol and specified its options, he should press the **Insert symbol** button to insert the symbol in a drawing.



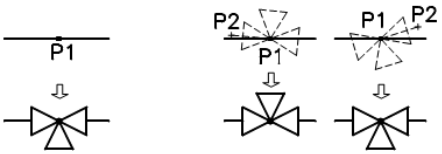
It is also possible to insert repeatedly a selected symbol. In order to do that a user should press the button and choose the **Insert repeatedly** option. To finish the operation **Esc** button must be pressed.

All symbols which are in a library have specified properties that make a designing easier. For example:

- symbol of a stop valve can be inserted in a pipe with automatic fitting an angle and breaking a pipe;
- symbol of clung temperature sensor can be inserted in a pipe but will not break a pipe;
- some symbols do not have possibilities to fit to a pipe;
- symbols with visible texts (attributes) display additional dialog that is used to edit attributes.

Inserting symbols in pipes

When a user wants to insert a symbol in a pipe, he should specify P1 point on a pipe (or any line). If **Rotate** option is active a designer should specify the second point to specify an angle of inserted symbol. An angle of symbol inserting is always fitted to an angle of pipe.



Without rotate With rotate

Note:

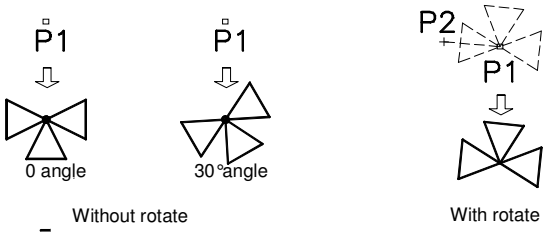
During inserting symbols in pipes a user can use extended structure of layers in which a symbol layer depends on a name of a pipe layer.
Additional information are available in the **Layers in CP-System** chapter on page 108.

Inserting symbols in any points

In order to insert a symbol a designer should specify a P1 point.

If the **Angle** option is active, a symbol will be inserted at angle specified in the dialog.

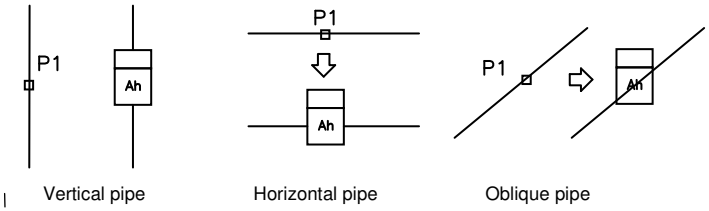
If the **Rotate** option is active, a user should specify a second point P2 to specify an angle of an inserted symbol.



Using symbols inserted with a fixed angle.

Some symbols are inserted with a fixed angle in order to keep correctness of a project. Examples can be electricity meters, sludgers, separators etc. These symbols should be inserted at angle 0°.

In CP-System, there are many symbols that have a fixed angle of inserting. Inserting this kind of symbols in horizontal or vertical pipes, causes automatic breaking of pipes. In other cases pipes are not breaking.



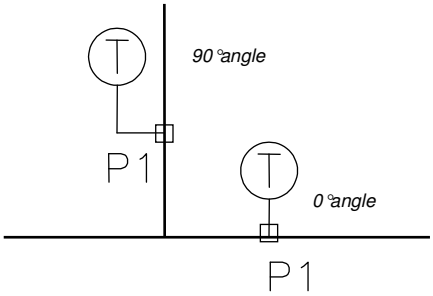
Notes:

- **The Angle** and **Rotate** options are ignored for symbols that have a fixed angle of inserting.
- Connection points for symbols that have a fixed angle of inserting are fitted to vertical and horizontal pipes.

Using of multivariant symbols

Appearance of some symbols should be different depending on an angle of inserting. Such symbols have got prepared variants in CP-System. Inserting these symbols at various angles, the program selects automatically appropriate variant of a symbol.

Example: during inserting a symbol of thermometer in a vertical pipe (an angle 90°) or in horizontal pipe (an angle 0°) various variants of a symbol are chosen.



Note:
During inserting, a basic variant of a symbol is visible. When an angle has been specified, an appropriate variant of symbol is selected automatically.

Symbols including texts (visible attributes)

For most symbols that include texts (visible attributes) CP-System program sets automatically an angle of the texts. Thanks to that texts are always displayed correctly.

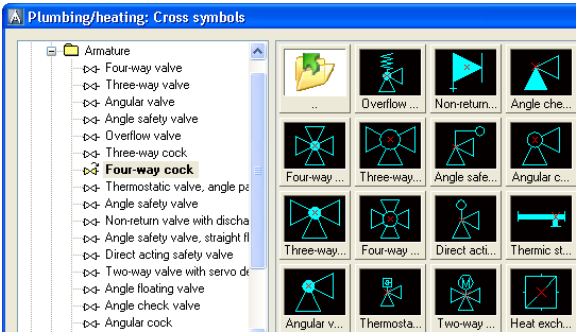
Ways of configuring texts:

- Without fitting – texts can be inserted at any angle.
- A fixed angle 0°.
- An angle 0° or 90°. If a symbol is introduced at a 90°angle, text will also be rotated at an angle of 90°. In other cases an angle of a text is 0°.
- Automatic angle from -30° to +120°. When an angle is specified from -30° to +120° the texts are readable enough.

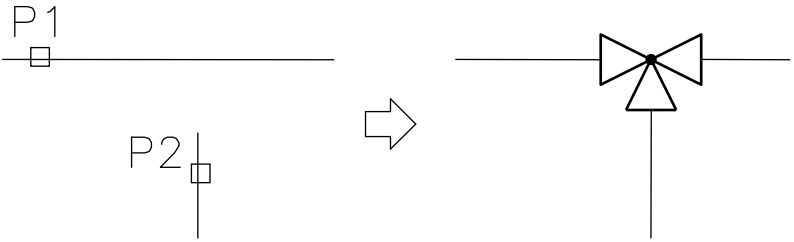
Matching of an attribute angle

Cross symbols

The **Cross symbols** command is used to insert symbols in a cross of two pipes.



When a user has selected a symbol and specified his size (scale), he should press the button OK. Then he should specify a first pipe (P1) and a second pipe (P2). The symbol will be inserted automatically in the point of crossing of specified pipes.



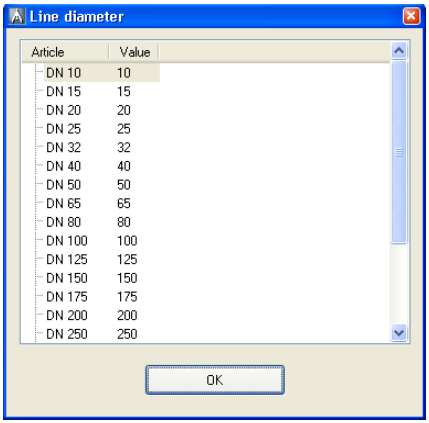
Notes:

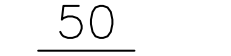
If specified pipes are crossing at a right angle, automatic fitting and breaking of pipes will occur.

Marks

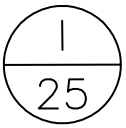
The command allows to insert various marks that are used by designing installations.

During inserting some marks, a list that consist of values suitable for a given symbol occurs, for example, for a symbol of mark of pipe a list that contains standard sizes of pipes occurs.



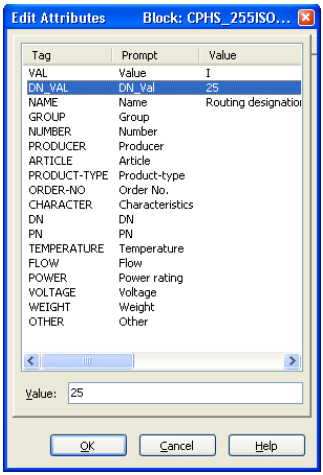


Marks of pipes



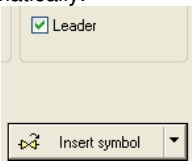
Routing designation

In order to change texts that are contained in symbols of marks a user should use a command of AutoCAD/IntelliCAD: **Attributes edition** - _DDATTE or CP-System command: **Attributes and descriptions** (see page 92).



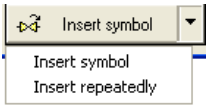
Automatic marks

In CP-System, there are symbols which can be filled automatically. These symbols copy data from described objects. That is why during automatic inserting marks a designer should select a described object and then specify the point of mark.
For some symbols it is possible to turn on the option of inserting symbols with leaders.



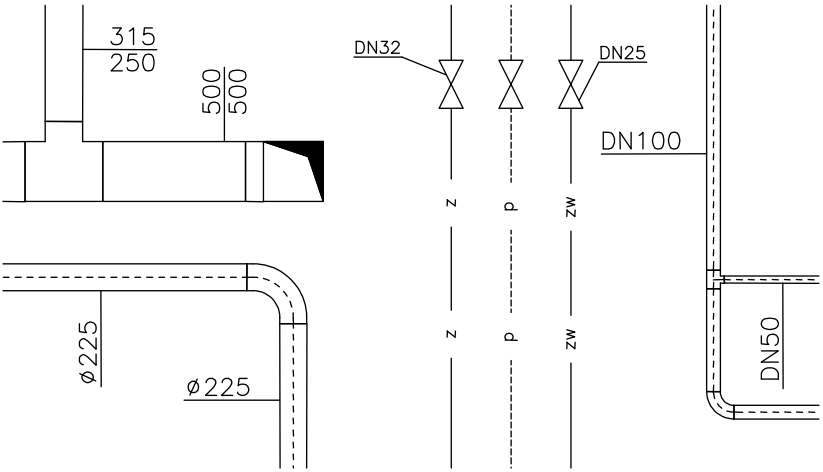
A user should use appropriate symbols to describe various objects. For example: the „Mark of rectangular duct" symbol is used to describe a size of diameter of an air duct. If a designer uses this symbol to other objects, Cp-System will not be able to read expected data. In this case, a symbol will be described in wrong way.

In case of marking many objects (e.g. description of air ducts) the best solution is to choose the **Insert repeatedly** option. In order to do this a user should open options available under the **Insert symbol** button.



Examples of automatic marks:

- Diameter of air ducts,
- Types of pipes (symbols of marks inserted in pipes),
- Size of armature (symbols of marks with automatic leader),
- Nominal size of pipes



Removing symbols and armature

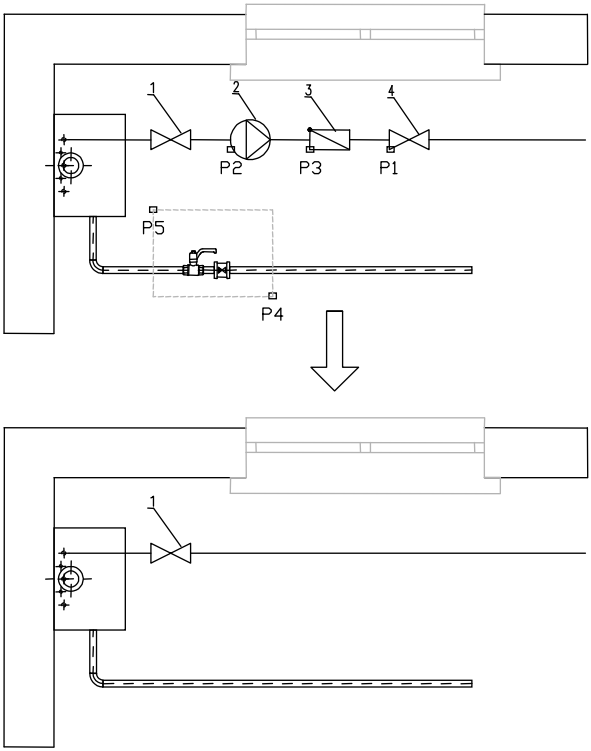
The **Delete symbols** command is used to automatic removing symbols and armature.

If removed symbols were inserted in pipes, CP-System merges these pipes. During removing numbered symbols, numbers are also removed.

A user can select many objects during removing but CP-System will remove only these symbols which are treated as armature (views of valves, pumps that were inserted in 2D pipes etc.).
In order to remove symbols a user should start the command and then select symbols (P1, P2, P3…). The program will be removing symbols until a user press the **Enter** or the **Esc** button. An additional options for removing are available in command line (restoring symbols).

```
Specify a symbol to delete or [Multiple/Undo]:
```

When a designer has chosen the **Multiple** (button **M**) option, he should select required symbols (P5-P6) and press the **Enter** button. During selection, he can select other objects (e.g. pipes etc). All elements which are not symbols or armature will not be removed.

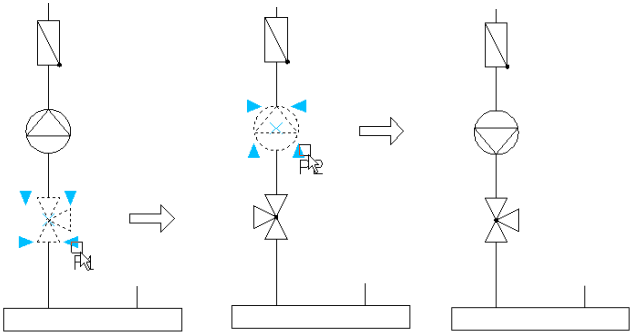


Mirroring symbols and armature

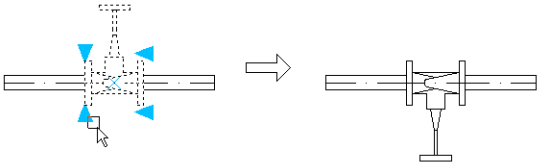
During edition of schemas or plans a user often has to mirror a direction of inserting symbols. He can use the **Mirror symbols** command to do that quickly.
After starting the command it is necessary to select a required symbol. Special markers are displayed around the selected symbol. To mirror a selected symbol a designer should click an appropriate marker. When the symbol has been mirrored, a next symbol can be selected. The operation ends when **Enter** or **Esc** button is pressed.

The **Mirror symbols** command can be used only for symbols and objects created by CP-System. When a user selects other elements, markers of mirroring are not displayed.

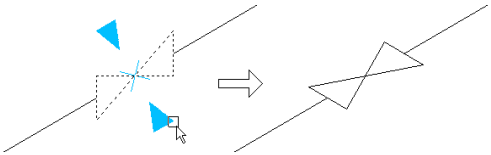
Example:
Using the **Multisymbols** command it is possible to insert many symbols at once in a drawing. It is possible that some symbols are inserted “improperly”. The easiest way to change their position is to use the **Mirror symbols** command.



Mirroring armature



Mirroring isometric symbols



Pipes

Schematic pipes

There are different kinds of pipes in the program. Each one has specified colour, linetype and lineweight on plots.

These parameters are compatible with the norms. Drawn pipes are placed automatically on particular layers.

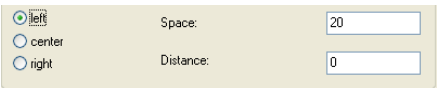
It is possible to draw a single pipe or many pipes at once. In the right part of dialog there is a list of pipes that are being drawn. When a user wants to add a pipe to this list, he should select a required element from a list of all pipes and press the **Add an element to a list** button . He can also add pipes by double-clicking or by selecting from context menu (a right mouse button). Choice of more than one pipe causes that pipes are being drawn with a given distance and with a specified order. That is why the list of selected pipes can be organized using buttons **Move up**, **Move down**, **Delete** and **Copy** .

After clicking the **OK** button a designer should specify points between of which the program will draw pipes. The way of drawing is similar to drawing lines in CAD program.

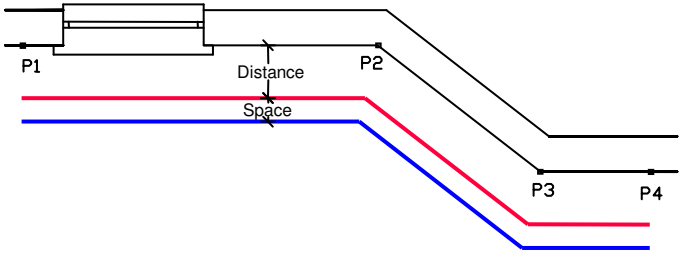
Pipes can be drawn in two views: “Type of installation” or “Type of pipe”. When a user wants to draw pipes in a view “Type of installation” he should uncheck an option “Draw by type of pipe”. In such case selected pipes will be placed on appropriate layers according to data included in internal data-bases. On the other hand if a user wants to draw pipes in a view “Type of pipe” he must check an option “Draw by type of pipe” and in column ‘Type of pipe’ of the list of selected pipes specify required type of a given pipe (e.g. DN50). In such case pipes will be drawn with linetype determined by entered text.

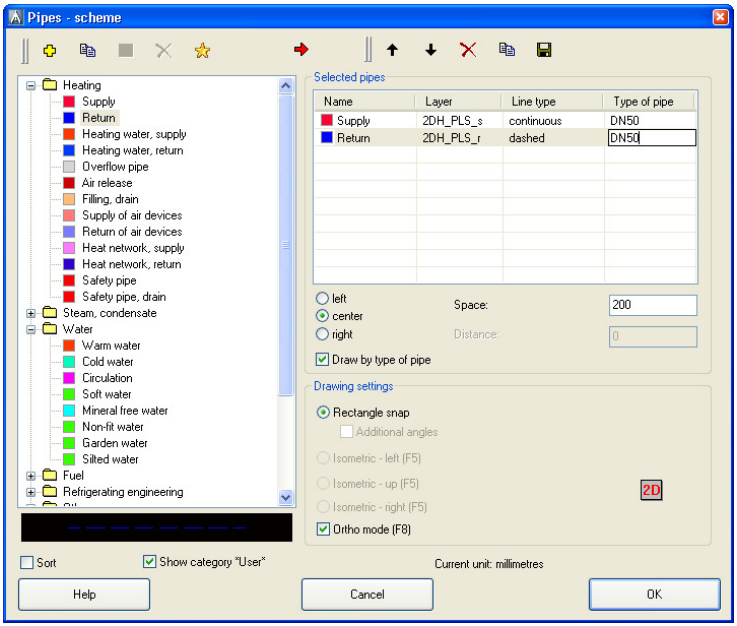
Example – drawing pipes moved away from a wall

- „Supply” and „Return” should be added to the list of selected pipes.
- Specify the distance from a wall and the space between pipes.
- Select drawing on the left edge.



When a user has clicked the **OK** button, he should select points lying on the edge of a wall (P1, P2, P3, P4). Pipes will be drawn automatically with a given distance from a wall.





A user can add new definitions of pipes and installations CP-System. He can also create sets that consist of several pipes.

Define new. A designer can add his own types of pipes or installations. For this purpose colour, linetype, and lineweight should be determined in an appropriate dialog. New pipes are added automatically to the ***User*** category.

Copy pipes. Copying selected pipe to the ***User*** category.

Edit pipe style. Changing of parameters of pipes (colour, linetype etc.).

Remove a pipe. Removing of selected definitions of user's pipes. When a user has removed an element by mistake, he can retrieve it. In order to do this, he should select the **Restore removed** option from a context menu **directly** after removing a pipe.

Add to favourites. Copying a pipe to the ***Favourites*** category. In this way a designer can create a list of the most often used pipes.

Note:

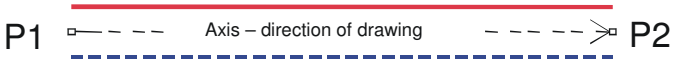
Only pipes defined by a designer and placed in the *User* or *Favourites* category can be removed and edited.

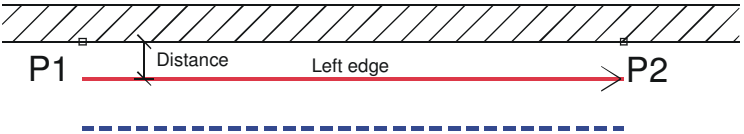
Add element to a list. Adding a highlighted pipe or set of pipes to a list of pipes.

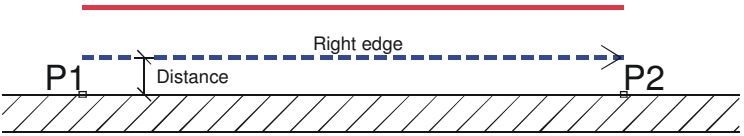
Save as set of pipes. Sometimes it is necessary to draw specified pipes next to each other. CP-System allows to save such pipes as set. Adding a set of pipes to a list of selected pipes causes removing all previously selected pipes.

The edge of drawing, distance

It is possible to draw pipes by axis, by left or by right edge. Left and right edge are determined by the direction of pipes drawing (points P1, P2…). While a designer is drawing by left or right edge, he can specify the distance from drawn points. Thanks to that, it is possible to draw pipes e.g. along the walls.

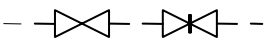




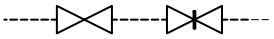


Linetype scale

Particular pipes are drawn with various types of lines, for instance, „Return" is drawn with dashed line (DASHED). Global scale factor is set using the CAD command _LTSCALE. Default the factor is set default to 1.00 in new drawings. If the value of this factor is too low or too high, the pattern of this line can be not seen and the whole line seems to be continuous line.
A designer can set units to millimeters, centimeters or meters in CP-System program. And he has to match the global scale factor to the edited schema. For instance, during drawing schemas in mm and with size of symbols 200 a user should set a global scale factor to a value from 50 to 200.



_LTSCALE = 200

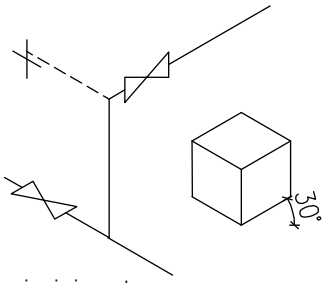


_LTSCALE = 50

Pipes in isometric drawings

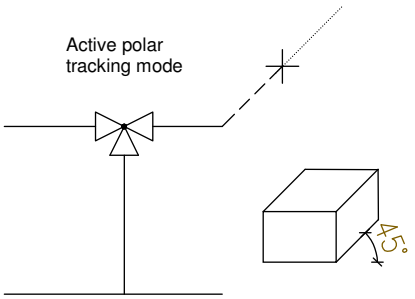
In order to design comfortably usual drawings (2D) or various types of axonometric drawings a user can select optimal CAD settings in the **Drawing settings** section. Only if one pipe is selected, will mode of isometric drawing be available.

- **Isometry** – a special mode of snap and grid is available in AutoCAD or IntelliCAD program. A graphic cursor in this mode is adapted to design the isometry in which axes are set at an angle 30°, 150° and 270°. In order to turn on this mode a user should select the **Isometry** option with an activation of a left, upper or right plane. A designer should turn on the ORTO mode during drawing pipes in isometry. To shift quickly between planes during drawing, the F5 key can be used.



Drawing in isometry

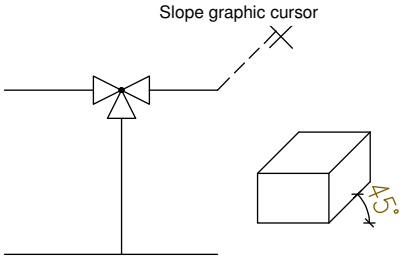
- **Dimetry** – there is a polar tracking available in AutoCAD program. The polar tracking makes drawing at specified angle easier. Having selected rectangular Snap and **Additional angle for dimetry**, polar tracking will be activated which is adapted to design drawings in angle dimetry. During drawing in isometry, ORTO mode should be turned off.



Drawing dimetry

- **Dimetry** – Polar tracking option is not available in IntelliCAD program. That is why a **cursorObliquity** option is available in the command line. This option is used to change an oblique angle of graphic cursor compatible with current angle of dimetry. An angle of dimetry is specified in **CAD-Profi Option** dialog (page 18). During drawing pipes in dimetry in IntelliCAD a user should use ORTO mode which can be turned on or turned off by F8 key.

  During drawing a pipe a user should specify the next points. The **cursorObliquity** option is available in the command line. In order to select the option a user should write letter 'O' in the command line and press Enter. An oblique angle of cursor will be changed. If a designer wants to restore normal settings of a cursor, he should again select the **cursorObliquity** option.



Drawing dimetry in IntelliCAD

An exemplary text from the command line during specifying another points of a pipe:

```
Specify the first point of a pipe or [cursorObliquity]:
Specify the next point or [Undo/cursorObliquity]:
Specify the next point or [Close/Undo/cursorObliquity]:
```

Note about using the command line:
In order to select options, a user should enter a letter or several letters which are capitalized in name of options. In order to choose the **Undo** option in the example, a user should enter a letter 'U', and press Enter. He can press a right key of a mouse or a spacebar instead of the Enter key.

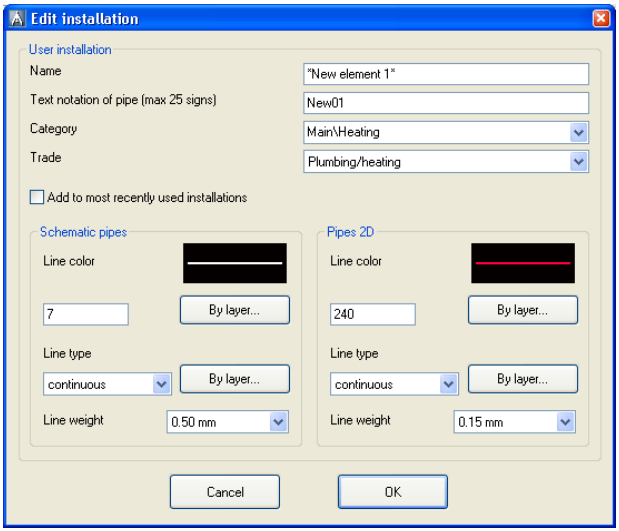
Layers of pipes

Every kind of pipe is drawn on a different layer. In the main menu, there are additional commands that make managing layers easier. A user can use these commands to quick hide or show all layers that belong to the various installations.
Plot styles that are included in *C:\CadProfi\xx\PlotStyle* folder are matched to structure of program layers. A user can choose either files to monochromatic plots (*CadProfi Mono.ctb*) or to colour plots (*CadProfi Color.ctb*). *'*.ctb'* files should be copied to appropriate folders of AutoCAD or AutoCAD LT program. Plot styles of IntelliCAD program have been defined in *.pcp files. A user can find more information about layers and plot styles in the **Layers in CP-System** chapter at page 108.

Defining and edition user's pipes or installations

A user can specify additional types of pipes or installations. They are saved in *UserData* folder (see page 8 and111).

After pressing the **Add** button a dialog will appear where a user should specify the name and the style of an added installation. A style of an installation contains colour definition, linetype definition as well as a definition of lineweight for schematic pipes (one-line pipes) and for 2D pipes.

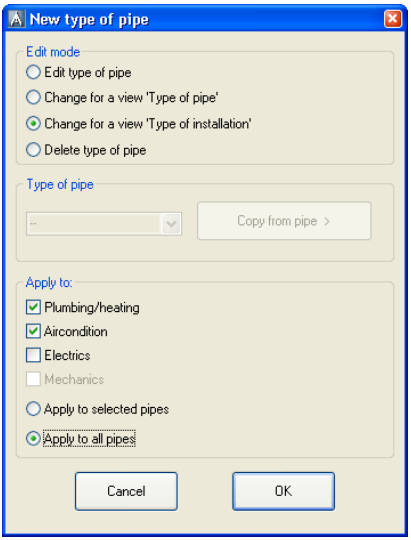


The colour has an influence on the lineweight in plots in CP-System program. That is why after changing of a colour, the program will suggest an appropriate lineweight. Unless styles of plots (files *.ctb) of CP-System program (see also information at page 108) are used during printing, a user can select other lineweight.

Text mark – is used during creating a name of a pipe. It should be unique for all pipes.

Add to most recently used installations – including an added element in a short list of installation in the **2D Pipes** command.

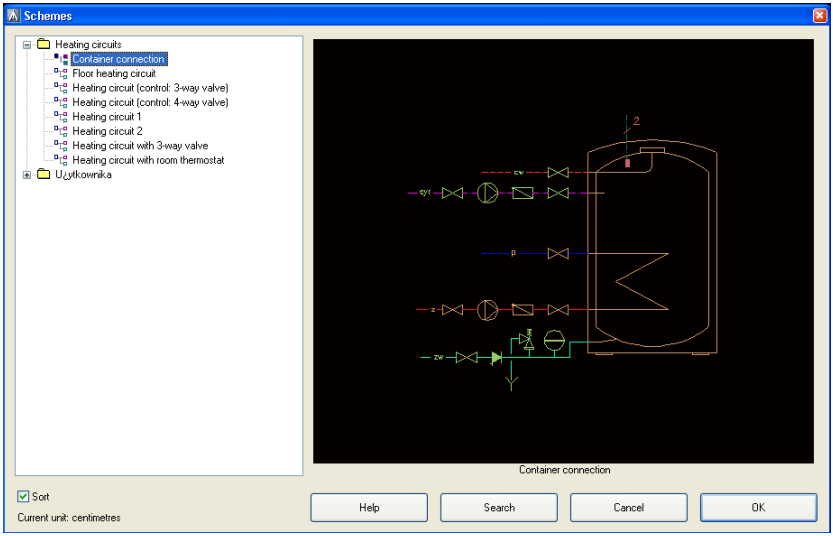
Specifying and changing type of pipe



Schematic pipes can be drawn in two views: “Type of installation” or “Type of pipe”. To change a view of pipes a user can use a **Pipe/Ducts/Conductors - edit** command. There are a few edit modes for specifying type of pipes. A user can change an existing type of pipe for another type either by selecting it from a list or by entering a new name of the type. It is also possible to copy the type from a pipe in a drawing by clicking ‘Copy from pipe’ button. A user can also change view of pipes either for “Type of installation’ or for “Type of pipe”. To do this, an appropriate button should be checked.
To clear type of pipe a button ‘Delete type of pipe’ must be on. Clearing type of pipe means that it is impossible to change a view for such pipes. Only pipes belonging to installations existing in a drawing can be changed. For example, if a drawing does not contain plumbing pipes, an appropriate option is grayed. All changes can be applied to all or selected pipes.

Libraries of typical schemas

A user can use the **Ready schemas** command to insert typical fragments of schemas such as: a container connection schema, various schemas of distributors, a supply schemas of heating circuits, typical schemas of a boiler rooms, schemas of furnace rooms in drawing etc.



A schema that is inserted in a drawing is usually exploded into its components objects (blocks representing devices are not broken). Thanks to that a user can edit a drawing, add descriptions etc.
A user can add his own and most often used schemas to the program. To do this a user should:

- Save a required schema using the _WBLOCK command. Block should be saved in the folder: *C:\CadProfi\UserData\Components\Sanit\Block_schem*
- Using the _MSLIDE command a user should create a slide (a file that contains a preview of a schema). The file of the slide should be saved in the: *C:\CadProfi\UserData\Components\Sanit\Slides* folder
- Add an entry to the file *C:\CadProfi\UserData\Components\Sanit\Schem_hs.uuc* which contains definitions of available schemas. Details about edition of a file are described at the beginning of this file.

Notes: In order to edit *'*.uuc'* files, a user can use e.g. a system notepad. The folder *C:\CadProfi* is a default folder of an installation of CP-System program.

Many ready schemas are also placed in manufacturers' libraries which are an integral part of CP-System.

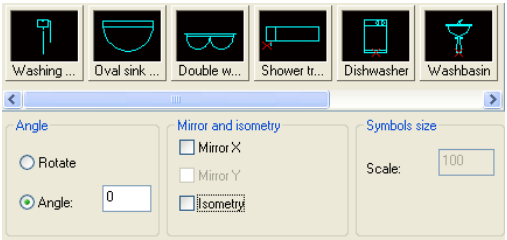
Plumbing objects - schemas

The plumbing objects are special kinds of symbols that are used on schemas of plumbing installations.

It is necessary to keep the vertical scale on plots. That is why symbols of plumbing objects are inserted in standard size. The size only depends on the unit of designing (mm, cm, m), therefore, an option of changing of a symbol scale is disabled.

Plumbing objects can be inserted on the right or left side of verticals. For some symbols **Mirror X** option must be turned on.

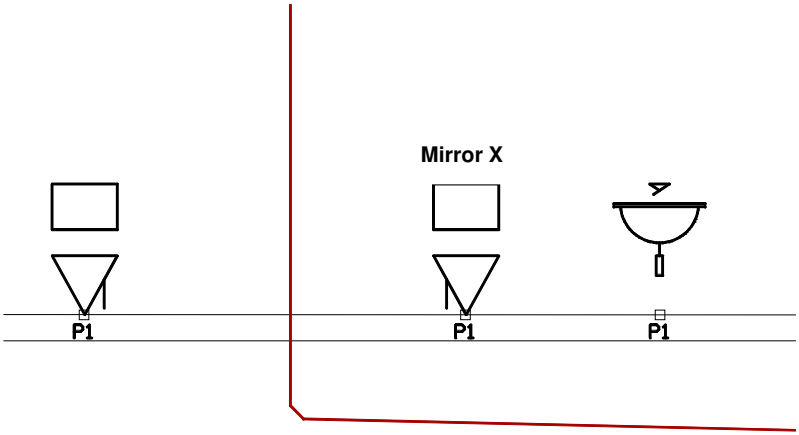
A user should insert symbols of plumbing objects at an angle 0.



Inserting symbols on schemas

The inserting point for all plumbing objects is defined on the level of the floor. Therefore, before inserting the first symbol a user should draw a section of ceilings.

Inserting symbols of plumbing objects on schemas a user should always select a point lying on an upper edge of a floor.(P1).



Plans and sections

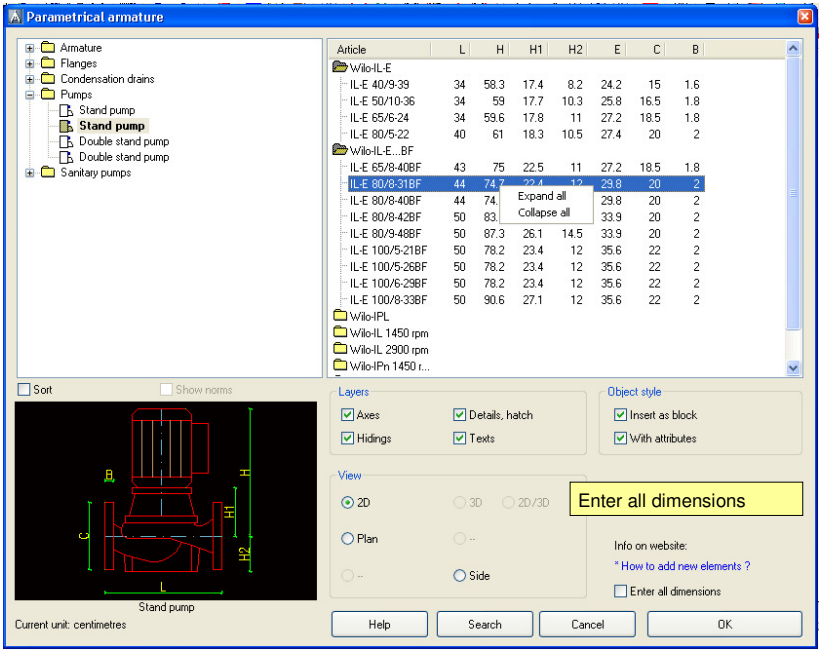
A designer can design various plans and sections of installations in CP-System program. During designing a user can use objects as well as pipes and 2D fittings.

Various types of objects are included in CP-System:
- Heaters – are available in the **Radiators** command. It is possible to use radiators on plans and schemas.
- 'Usual' objects – are available in the **Plumbing objects** command. 'Usual' objects present views of devices in a very general way (e.g. views that are compatible with norms or simplification of views). For 'usual' objects it is possible to specify only the external dimensions – length, width, height.
- Parametrical objects – are available in the **Parametrical objects** command. These objects display real views of devices such as: containers, distributors, pumps etc. A user can specify many dimensions for parametrical objects.
- Parametrical armature – is available in the **Parametrical armature** command. The armature, similarly to parametrical objects, displays real views of devices (armature, pipe equipments, flanges, pumps in in-line system etc.). These objects can be inserted in 2D pipes.
- Objects from manufacturers' libraries – are available in the **Products library** and **Previous library** commands. Products' libraries were created together with manufacturers of devices. Therefore, they contain extensive and current source of objects (blocks) used during designing. Products' libraries are being updated all the time and are offered by manufacturers also in the Internet. After a user has installed CP-System program, he can install new libraries which will be added automatically to the structure of the program.

It can happen that similar devices often can be found in different libraries or commands in CP-System. For example, pumps are in commands **Parametrical objects**, **Parametrical armature** and **Products libraries**.
If a user wants to insert views of devices in a project, he should use rules presented on examples of various pumps:
- Wilo Stratos 40/1-8 pump. Wilo catalogue is in manufacturers' libraries, so a user should look for pumps in manufacturers' libraries.
- X company pump that can be built on pipes. Unless X company is in manufacturers' libraries, a user should look for the pump in the **Parametrical armature** command. In this command, there are various objects to built on pipes (e.g. in-line pumps).
- Block pump of X company. Unless the company is in manufacturers' libraries, a user should look for it in the **Parametrical objects** command because there are objects which are not inserted in pipes (e.g. block pumps, or pumps with foundation board).
- During edition of schemas, symbols of pumps are available in the **Symbols** as well as **Multisymbols** commands.

Parametrical objects

In **Parametrical objects**, **Parametrical armature**, **Switchgears**, **Steel shapes** commands and others, there are available objects for which a user can define many dimensions. A flange valve is an example for which a user should define both length, width, height and dimensions of flanges and other. Databases containing standardized elements as well as elements prepared on the base of types of some manufacturers are available for every parametrical object. Inserting parametrical objects a user can enter dimensions manually. That is why parametrical objects can be used for any devices even for these that are not in standard databases of CP-System program.



Option of parametrical objects

Layers – eliminating layers that contain axes, details, hidings and others a user can specify a precision of drawing of parametrical objects.

Style of an object – Parametrical objects can be inserted as blocks that contain attributes. Thanks to this a user can edit technical parameters, insert descriptions and create specifications automatically. Parametrical objects can be inserted without attributes or as an exploded object.

Dimensions of objects

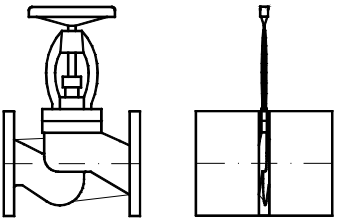
Once a user has selected a view of a device, a list containing sizes of devices appears. All dimensions are shown in a current unit. If a user wants to select a required dimension, he should mark a particular type on the list.

For some objects, like distributors, are available databases specifying all dimensions with exception of the length of connections (every time can be different length of a connection). Inserting these objects a user should specify essential dimensions on his own.

If a device that is not in a program databases must be inserted, a user should select a parametrical object that is the most similar to an inserted device and select the **Enable all dimensions** option.

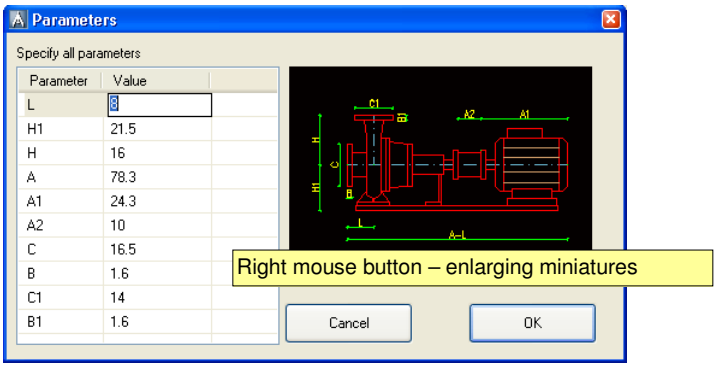
After a user has pressed the **OK** button, the program will demand entering appropriate dimensions. The dimensions should be entered in a current unit (m, cm or mm).

Note:
It is impossible to check dimensions before inserting an object. When a user specifies wrong data, an object can be deformed. It is also possible that an object will not be inserted in a drawing.



An example of inserting of an object with a too large flange.

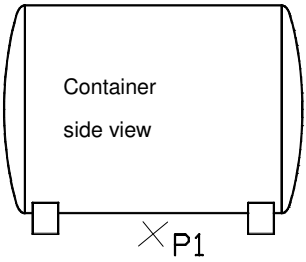
Before a user inserts an object, he can see an enlarged preview of it. A user should click a right mouse button on the miniature of an object to see an enlarge view.



Inserting of parametrical objects

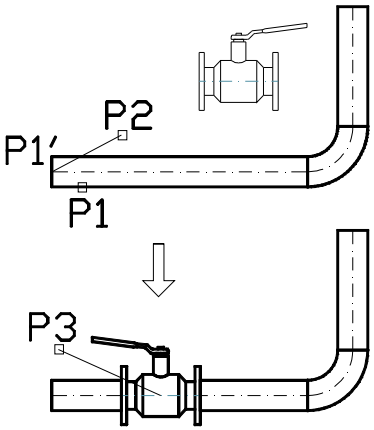
Inserting an object in any point

In order to insert an object, a user should specify a point P1 and then the second point P2 specifying an angle. Instead of specifying the point P2 a user can enter a value of an angle in the command line.

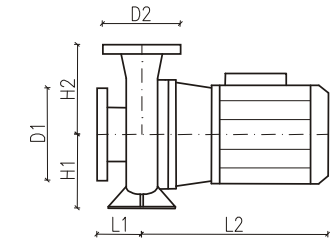


Inserting of an object in 2D pipe

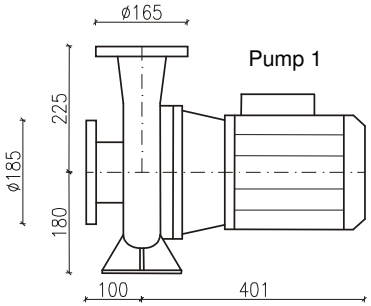
In order to insert an object in a pipe, a user should specify a P1 point that is placed close to the end of the 2D pipe. CP-System defines automatically the end of the P1 pipe' that is a reference point. Then a user should specify a point of inserting of an object. He can also enter a distance from the reference point (from the beginning of a pipe) or specifying a P2 point (a point of inserting is a perpendicular projection of the P2 point) in the command line. For many objects a point P3 can be specified. It specifies a way of inserting of an object e.g. position of a handle of a valve.

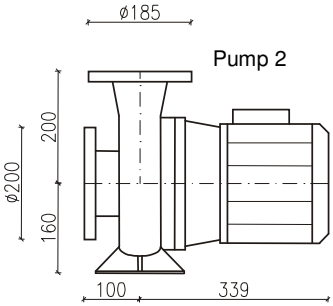


Examples of parametrical objects inserted with its own dimensions



Frame of dimensions		
	Pump 1	Pump 2
	Flanges	
D1	ø185 (DN65)	ø200 (DN80)
D2	ø165 (DN50)	ø185 (DN65)
	Other dimensions	
H1	180	160
H2	225	200
L1	100	100
L2	401	339





Other commands to insert objects

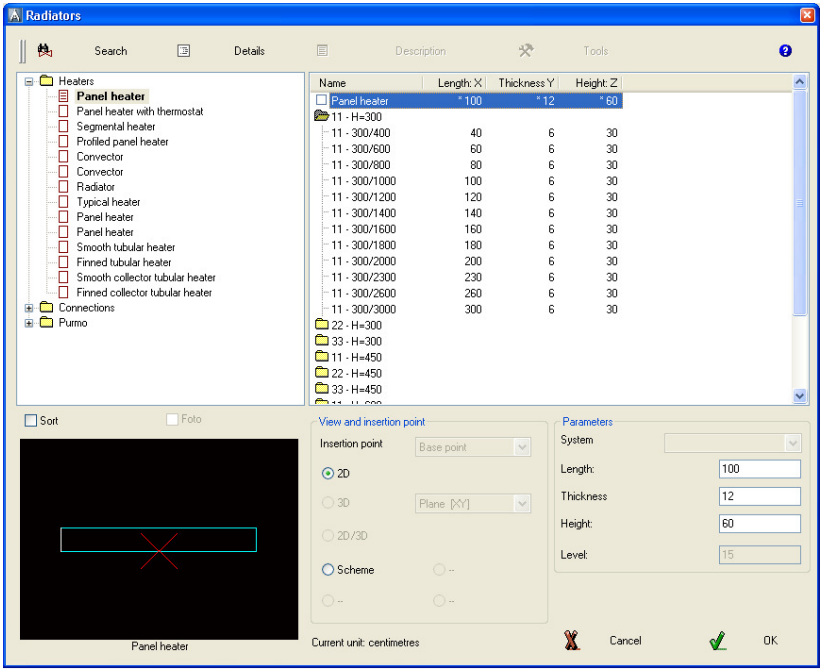
In CP-System program, there are additional commands that enable a user to insert objects: **Plumbing objects - plan**, **Radiators**, **Duct devices** etc.

Example:
Duct devices can be inserted in 2D ducts from which they copy properties of the system (supply air, exhaust air etc.).

Dialogs can be little bit different than described before, thanks to that a user can select and insert required object as easily as possible.

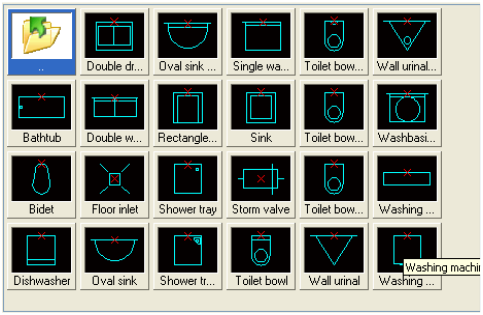
Radiators

In the **Radiators** command, there are databases with catalogues of radiators. They are displayed in a dialog. A user can enter dimensions of radiators on his own only for the first element on the list. The rest of radiators have specified a type and that is why their dimension should not be changed.



Plumbing objects

In the main dialog, there are miniatures of views of objects. They enable quick selection of a required element.



Pipes, ducts, cable trays and fittings

A user can design plans and sections of plumbing installations, air condition installation and cable trays.
During drawing installations, pipes/ducts/cable trays and fittings (elbows, reductions, tees) are available. Each fitting is a parametrical object that is drawn with the use of appropriate procedures. Thanks to that a user can draw any fittings according to specified dimensions. For fittings, there are available databases containing definitions of standardized elements as well as elements of selected manufacturers.

Fittings are available in the following commands:

- **Pipes** – this command contains typical pipes and fittings used during designing of plumbing/heating installations.
- **Ducts** – this command contains ducts and fittings used during designing air conditioning installations.
- **Cable trays** – this command contains elements of trays used to designing electrics.
- **Pipes/Ducts/Cable trays - sequence** – this command enables a user to draw many sections of pipes/ducts/cable trays and insert elbows automatically.
- **Join pipes/ducts/cable trays** – this command allows to join easily pipes/ducts/cable trays with automatic inserting suitable fittings (e.g. elbows, bends, reducers, fullers, tee-pieces, crosses etc.).

All these commands display dialog with a list, where a user can specify parameters for particular fittings. A set of available parameters depends on the type of a selected fitting.

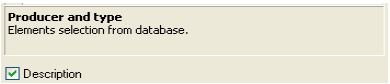
The meaning of parameters is similar for many fittings. Therefore it is strongly recommended to read the rest part of this chapter.

Inserting 2D pipes/ducts/cable trays and fittings

Pipes/Ducts/Cable trays command allows to insert a single fitting into a drawing.
Once the command has been run, a user can select a required fitting. A miniature with all dimensions for selected fitting is displayed under a list of available fittings. In order to enlarge the miniature a user should click a right mouse button on it.

A list with parameters of selected fitting is on the right side of the window. A list of parameters consists of a few sections. Marks [+] or [-] are used to open and close particular sections.

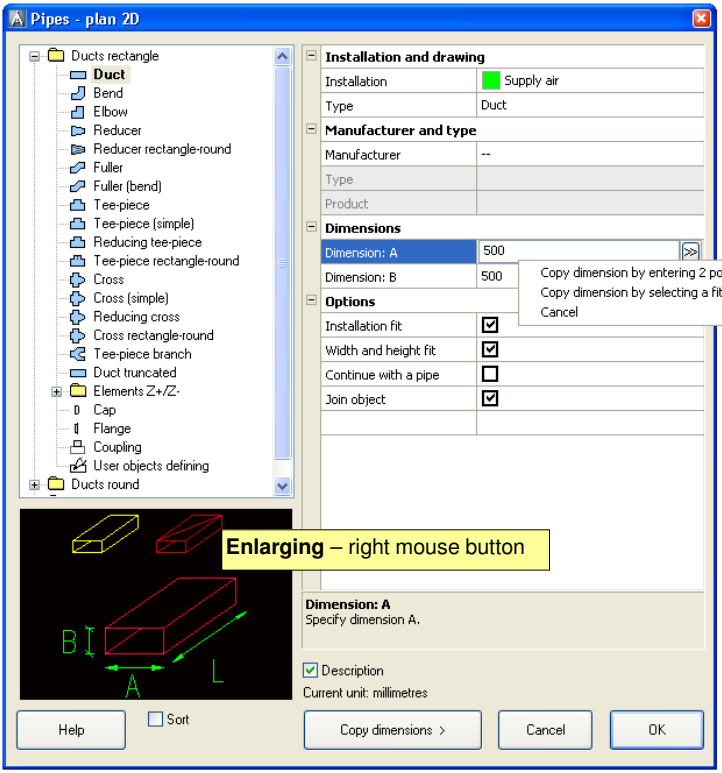
A description which explains in details the meaning of an active parameter is below the list. A user can turn it on or off by means of 'Description' option.



A user can select parameters from the databases or from lists:

- choosing a parameter from the databases.

- selecting parameter from a list.



A dialog with a selected rectangular duct

Copying dimensions

During drawing a user often needs to insert elements similar to the elements that have already been inserted in a drawing. In this case copying dimensions by using button **Copy dimensions** is the easiest way to do this. Once a user has pressed the button, he must specify a required fitting. It is important not to specify fittings with completely different parameters than an element selected in a dialog, because ambiguity of interpretation of data can occur.

For example: if an elbow is selected in a dialog and during copying dimensions a reducer will be specified then CP-System will not be able to specify the radius or the angle of the elbow because a reducer does not have such parameters.

It is also possible to copy single dimensions. To do that a user must click an appropriate field in the list of parameters to activate it (e.g. a diameter D).

Then he must press right mouse button to display context menu. A selected dimension can be copied either by specifying two points or by selecting a fitting. When “Copy dimension by selecting a fitting” option is used a user has to select a fitting to copy dimension from. If the selected fitting does not have a given dimension, an appropriate message is displayed in the command line and a user can continue selecting fittings. For example: a tee-piece does not have dimension E, so that if a reducer was selected in a dialog (before a dialog was hidden), a dimension E was to be copied and a tee-piece was selected, a message

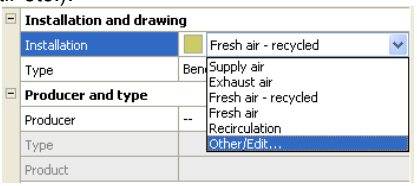
```
*** Cannot copy a dimension E from selected fitting. ***
```

will be displayed and a user will be able to continue selecting or to finish command by pressing ESC.

Parameters of fittings

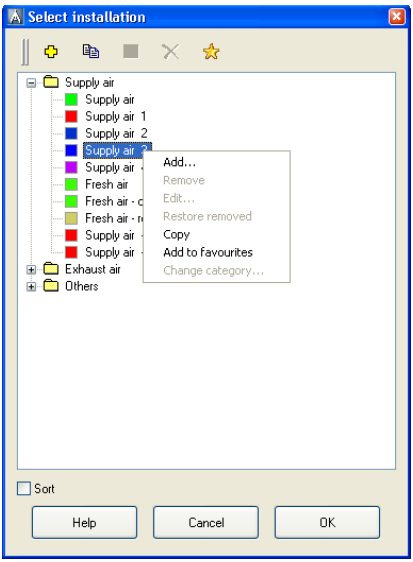
Installation and drawing

Installation – selecting of an installation or a system.
Installation can specify a flowed factor (e.g. fuel oil, gas, milk) or system (supply, return, supply air, exhaust air etc.).

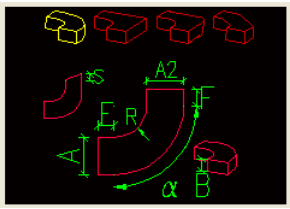


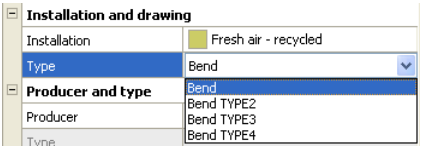
In the list of available installations, there are the most often used types of installations. If a user wants to select other types of installation, he should select the ‘**Other/Edit...**’ item. Selecting this item causes the dialog with all defined types of installations to be displayed.

A user can specify an additional types of installations. After selecting the **Add** button a dialog will be displayed where a name and a style of added installation should be specified. The style of an installation includes definitions of colour, linetype and lineweight. Defining installations - see page 41.



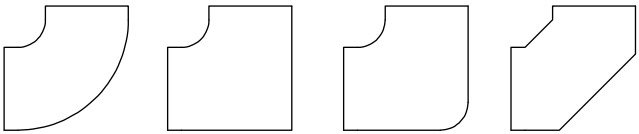
Type – for some fittings a user can select a type.
A selected type of a fitting with dimensions is displayed on a preview.



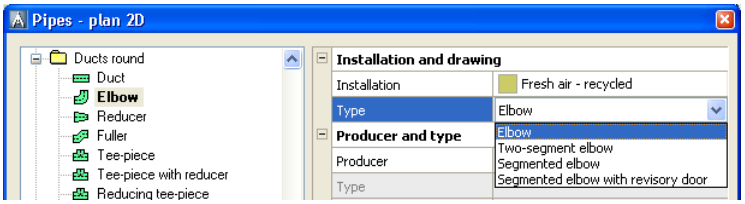


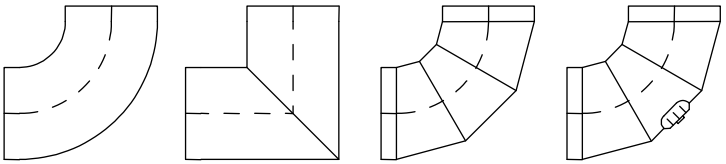
Examples of 'typed' fittings:

- Rectangular bends: 'Bend', 'Bend TYPE2', 'Bend TYPE3', 'Bend TYPE4'.



- Round elbows: 'Elbow ', 'Two-segment elbow', 'Segmented elbow' and 'Segmented elbow with revisory door'.



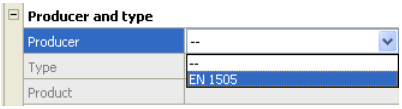


Manufacturer and type

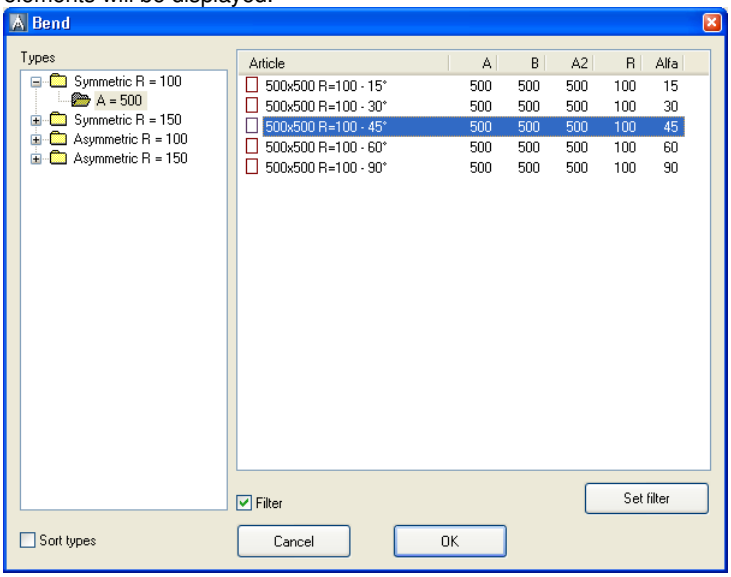
Manufacturer and type – selection of elements based on databases of manufacturers. After a user has selected a given manufacturer, an appropriate dialog containing available dimensions of fittings will be displayed.

CP-System draws fittings in two modes:

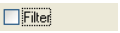
- Drawing based on norms or databases of manufacturers. If a user wants to activate this mode, he should select a required manufacturer (or a norm). After that, it is impossible to enter user's dimensions, because they are disabled.
- Drawing based on user's dimensions. In order to activate this mode, in the **Manufacturer** field a user should select ' '--'.



When a manufacturer (norm) has been selected, a database with available elements will be displayed.



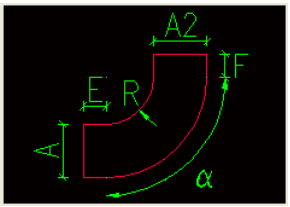
If a button 'Filter' is checked only main dimensions e.g. diameter of a fitting are displayed. To display all elements a user should uncheck the button.

- To turn off filtering a user should uncheck the 'Filter' option 

- To establish user's criteria of filtering a button 'Set filter' should be pressed.

Dimensions

A user can draw fittings with any dimensions. In order to do this a user should select an option '--' in the **Manufacturer** field.



While a user is drawing using his own dimensions, the fields in which he can enter required dimensions are active. He can also select typical dimensions.

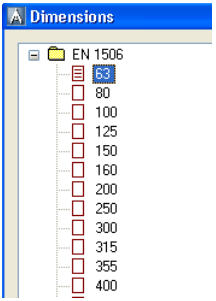
All dimensions are visible on miniatures of particular elements. Therefore only exemplary dimensions and options will be described.

Cross-section for fittings

A user should specify diameter D, D2… for round fittings. For rectangular ones, width (A, A2…) as well as height (B, B2…) should be specified.

CP-System creates 2D drawings. However, a designer should specify correct heights of fittings. It is necessary to get correct data for subsequent specification of elements.

If a button occurs on the right side of the given field (when the field is active) then a user can choose a typical dimension from a database.



Typical dimensions

Dimensions	
Dimension: A	50
Dimension: A2	50

Lengths of tee-pieces, reducers and other fittings

Lengths of fittings are specified automatically. After turning off the 'Length automatically' option, a user can enter any lengths.

Lengths automatically	☑
Length: L	35.2
Length: L3	30.1

Angles of elbows, tee-pieces and other fittings

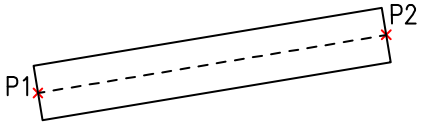
- For elbows angles must be different from 0°. The maximum angle for elbows is 180°.
- Only 90° angle is available for some fittings, e.g. reducing elbows.
- For tee-pieces and crosses the angle must be greater or equal 5° and less or equal 175°.

Angle: Alfa	90
Radius: R (75)	1.5
Use multiplier	☑

Inserting fittings

Pipes/Ducts/Cable trays

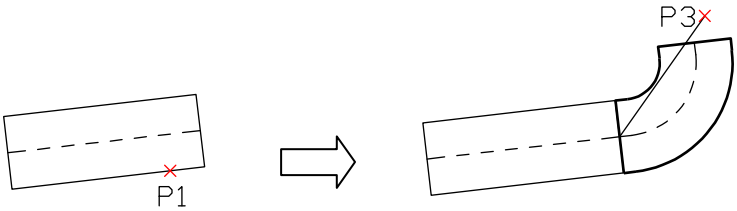
When inserting pipes/ducts/cable trays, a user should specify an insertion point (the beginning of an object) - P1, and then a second point (the end of an object) - P2. The object will be drawn between these two points.



Fittings

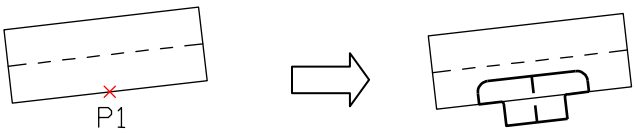
Most fittings can be inserted with automatic joining other fittings or in any point.

- If a fitting is to be joined another fitting a user should specify the object inserted previously (P1).
  a) Most fittings (e.g. elbows, tee-pieces, crosses, reducers, fullers…) are joined closer end of an element.

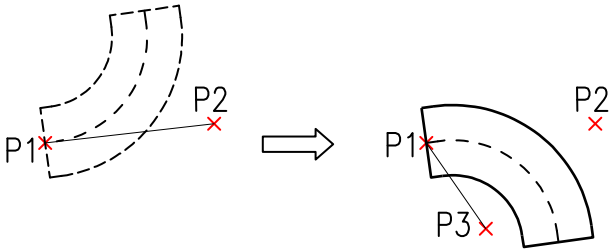


For some fittings, e.g. elbows, tee-pieces, a user should specify a point (P3) as a direction of inserting.

b) A part of fittings (e.g. couplings) are joined a side of a selected pipe/duct/cable trays.



- If a fitting is to be inserted in any point, a user should specify the insertion point (the beginning of an object) - P1, and then a second point (the end of an object) - P2 or specify value of an angle.

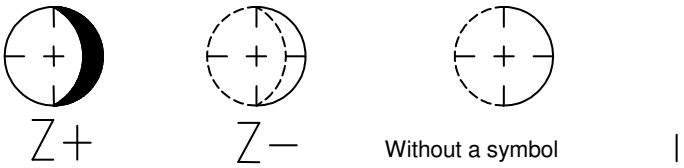


For some fittings (e.g. elbows, tee-pieces) a user should specify a point (P3) as a direction of inserting. For a given type of fitting point P3 sometimes is necessary, sometimes not, e.g. a reducer can be either symmetric or asymmetric. An asymmetric reducer requires point P3 to specify a direction of inserting, while a symmetric one doesn't require P3.

Fittings Z+/Z-

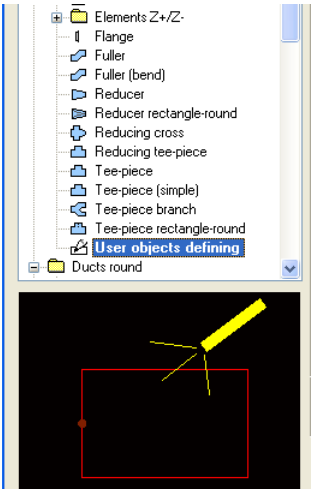
Z+/Z- pipes/ducts/cable trays represent vertical segments (along Z axis). Only a symbol of a cross section is seen on plans for these objects. That is why their length should be specified in a dialog before inserting them into a drawing.

For Z+/Z fittings a user can also specify a type of a symbol of a cross section (Z+, Z- or without a symbol) as well as a point of inserting (the beginning, the axis, the end).



Defining user's fittings

When designing, a user sometimes needs to use untypical fittings. In this case he should define his own fittings. CP-System will consider them during numbering as well as creating specifications.



In order to define an additional fitting it is necessary to draw it and to start the **Pipes/Ducts/Cable trays** command. The next step is selecting 'User objects defining' option. It is necessary to specify a new name for a new fitting. A user can also enter other parameters e.g. manufacturer or type.

After a user has pressed the OK button, he should select objects which a new fitting consists of and specify a base point.

If a given fitting is often used, it can be add to the libraries of the program with the use of a **Add blocks to the libraries** command (see page 111)

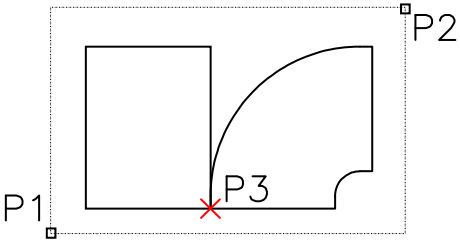
An example of defining of rectangular tee-piece (branch)

A defined fitting consists of a rectangular duct and a bend. The easiest way to draw a new fitting is to use the **'Ducts'** command. Using this command a designer should insert a rectangular duct and a bend at an angle 90°.

In order to use a new fitting in various installations, a user should move all elements on a layer 0. Thanks to that colour of a newly defined fitting and colour of an installation will be compatible. Then **'Ducts'** command should be run and a user should select a position 'User objects defining'.

A user should enter e.g. 'Tee-piece TYPE X' in a field 'Name' and he also should set a required installation (e.g. supply air).

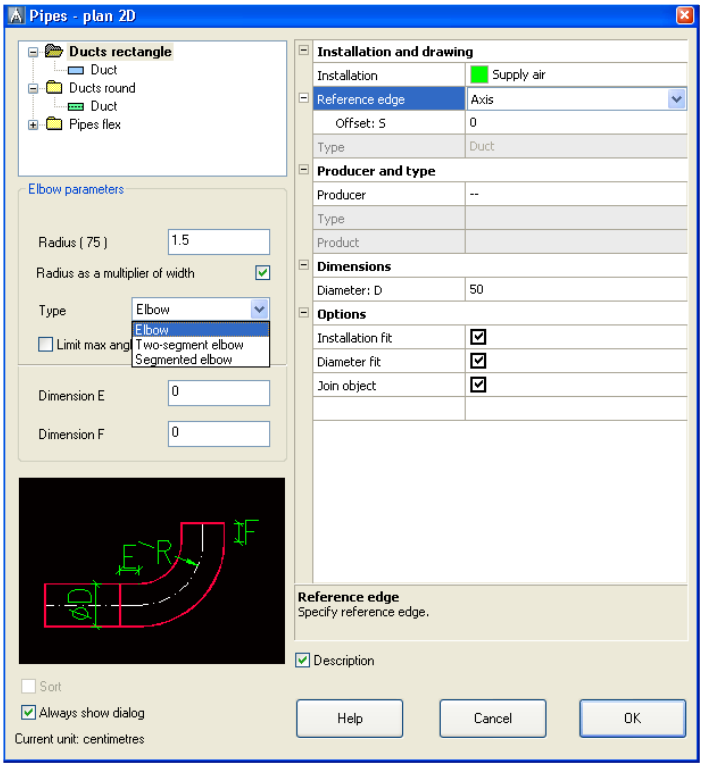
The next step is selecting elements that belong to a newly defined fitting (P1-P2) and specify the base point of the fitting (P3).



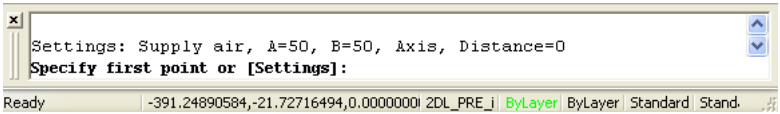
During defining the fitting is inserted automatically on a layer belonging to the selected installation.

Drawing a sequence of 2D pipes/ducts/cable trays

The '**Pipes/Ducts/Cable trays - sequence'** command enables a designer to draw many segments of pipes/ducts/cable trays with automatic inserting elbows or bends.
When a command is running first time, a dialog will be displayed in which a user can specify all parameters of a drawn sequence.



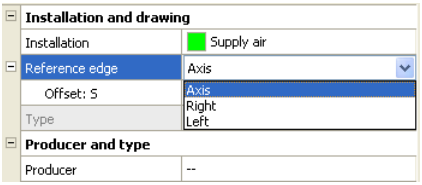
Subsequent running a command makes a user to begin drawing with current settings or to select an option **Settings.** It allows to display dialog with all parameters of a drawn sequence. The **S** letter should be entered in the command line and the 'Enter' key should be pressed to run the dialog.



Parameters of a pipe/duct/cable tray are placed in the right part of a dialog. The meaning of all parameters is the same as in '**Pipes/Ducts/Cable trays**' commands.

Parameters of bends and elbows are specified in the left part of the dialog but some of them (e.g. installation, manufacturer, type) are copied from pipes/ducts/cable trays.

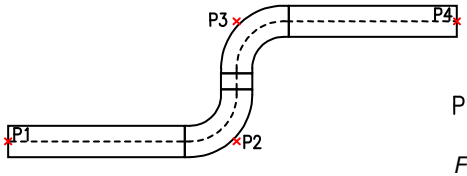
The reference edge and the offset



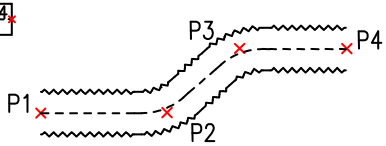
A sequence of pipes/ducts/cable trays can be drawn by axis, by left or by right edge.
The **Offset** parameter specifies the distance of a pipe/duct/cable tray from specified points.
The distance is counted from the axis or from a selected reference edge.

The manner of drawing

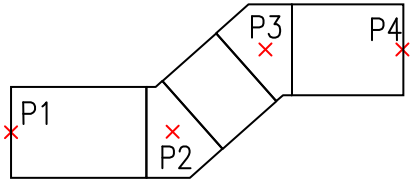
During drawing a user should specify points P1, P2 etc. In case of drawing by axis with no distance, pipes/ducts/cable trays as well as appropriate elbows and bends are inserted along the path of drawing.



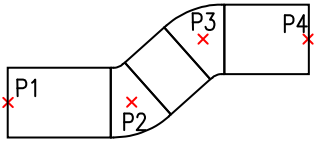
Round pipes



Flex pipes

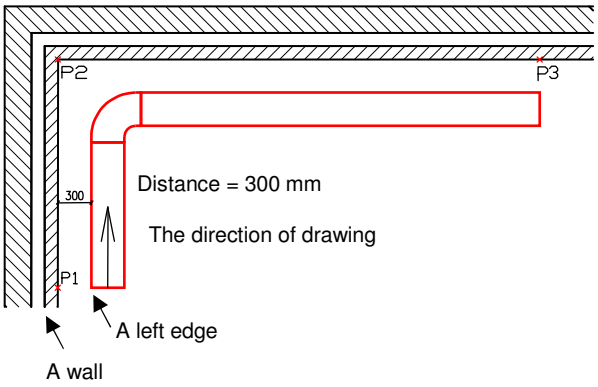


Rectangular pipes with elbows

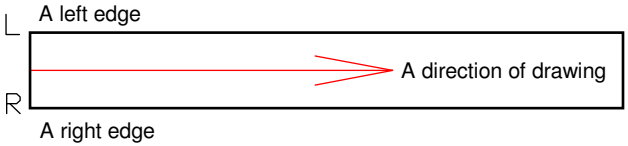


Rectangular pipes with bends

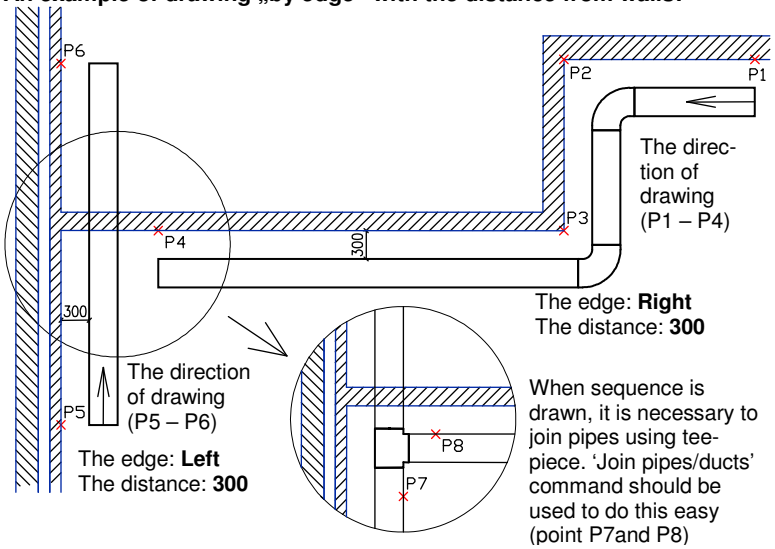
In case of drawing with a given distance pipes/ducts/cable trays as well as appropriate elbows and bends are inserted in a distance from specified points.



A left and a right edge are set in the direction of drawing of pipes/ducts/cable trays.



An example of drawing „by edge" with the distance from walls:



Joining pipes/ducts/cable trays

To create installation quickly and easy a user should draw only pipes/ducts/cable trays or sequences of pipes/ducts/cable trays and then join them using a versatile command **Join pipes/ducts**. CP-System lengthens or shortens joined pipes/ducts/cable trays intelligently as well as inserts optimal fittings automatically.

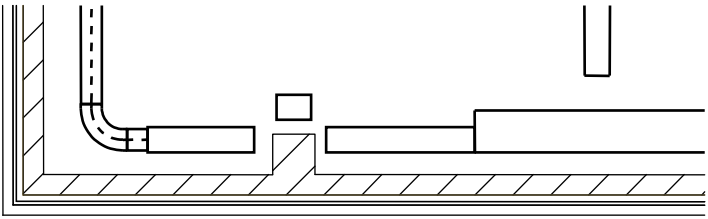
Once a user has run the **'Join pipes/ducts'** command, he should select pipes/ducts/cable trays to join them. Depending on how they are placed on the drawing, appropriate fittings will be suggested to insert.

A dialog presenting parameters of appropriate fittings is the same as in the '**Pipes/Ducts/Cable trays'** command. Some parameters and dimensions are disabled. For example, joining pipes/ducts/cable trays by means of reducer disables both diameters and distance of axes of reducer, because these parameters must be compatible with diameters and position of pipes/ducts/cable trays. On the other hand length of a reducer and dimensions E and F can be changed.

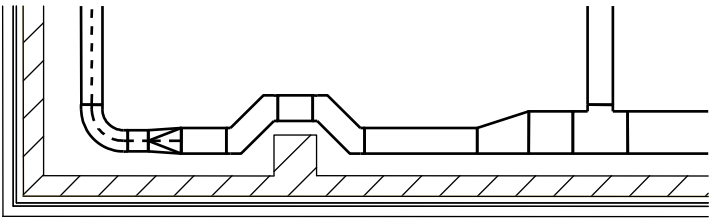
It is also possible to join pipes/ducts/cable trays and fittings as well as joining fittings. If a fitting is one of the selected objects to join it can happen that a fitting has too little room to be inserted. A message is displayed in such case, but a user can still insert a fitting. It must be moved e.g. by means of CAD command _MOVE subsequently.

Example:

- Step 1: pipes are drawn.



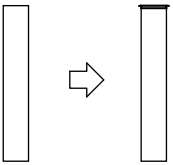
- Step 2: pipes are being joined by using appropriate fittings (reducers, fullers, tee-pieces).



The rules of joining.

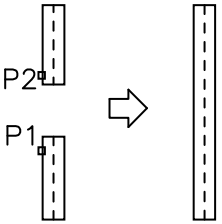
One pipe/duct/cable tray:

A cap, **a flange**, **a muff** or **a nipple** is inserted and a pipe/duct/cable tray may be lengthen/shorten.



Two, three or four coaxial pipes/ducts/cable trays with the same diameters:

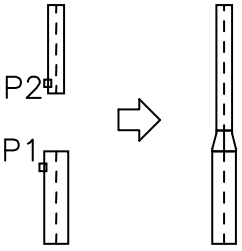
No fitting is inserted.

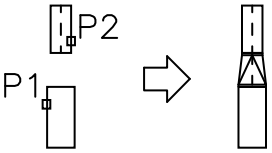


Two coaxial pipes/ducts/cable trays with different diameters, inserted at the same angle:

A symmetrical reducer is inserted.

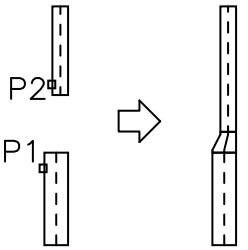
By inserting a reducer, length of a first selected pipe/duct/cable tray is not changed.





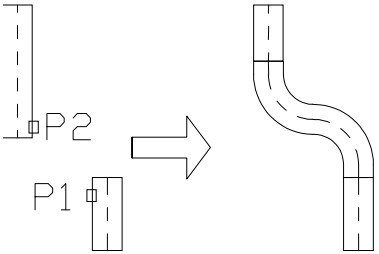
Two pipes/ducts/cable trays that are not coaxial and with different diameters, inserted at the same angle:

An asymmetrical reducer is inserted.



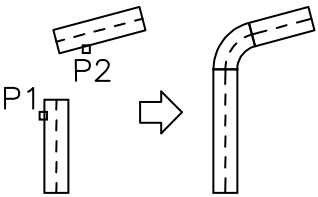
Two pipes/ducts/cable trays that are not coaxial and with the same diameters, inserted at the same angle:

A fuller is inserted.

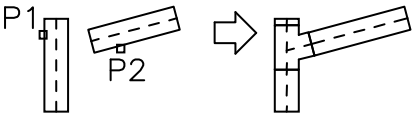


Two pipes/ducts/cable trays with the same diameters inserted at different angles:

An elbow or **a tee-piece** are inserted.

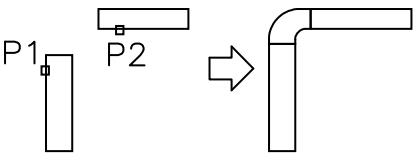


For tee-pieces, main pipe/duct/cable tray should be selected as a first one.

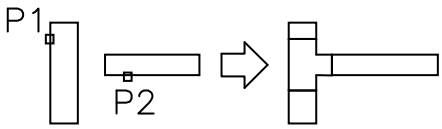


Two pipes/ducts/cable trays with different diameters, inserted at a 90° angle regarding to themselves:

A reducer elbow or **a tee-piece** are inserted.



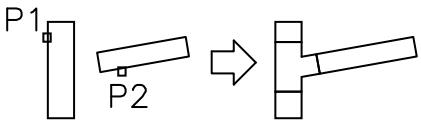
For tee-pieces, main pipe/duct/cable tray should be selected as a first one.



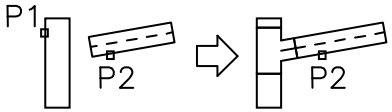
Two pipes/ducts/cable trays with different diameters, inserted at an angle other than 90° regarding to themselves:

A tee-piece is inserted

A main pipe/duct/cable tray should be selected as a first one.

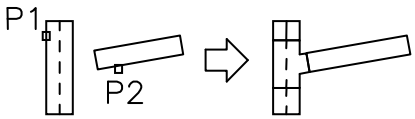


Two pipes/ducts/cable trays with different forms, inserted at various angles regarding to themselves:



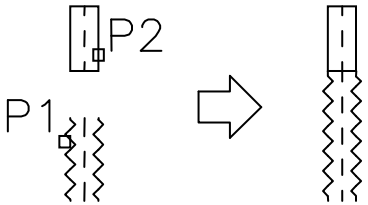
A round-rectangular / **rectangular-round tee-piece** is inserted.

A main pipe/duct/cable tray should be selected as a first one.



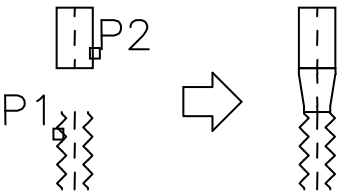
Joining of coaxial flex pipes/ducts with round pipes/ducts:

The same diameters – **no fitting** is inserted.



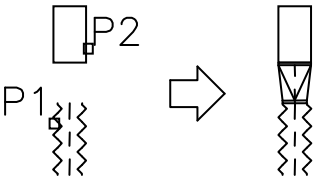
Joining a flex pipe/duct with a round pipe/duct. Both are inserted at the same angle:

A round reducer is inserted.



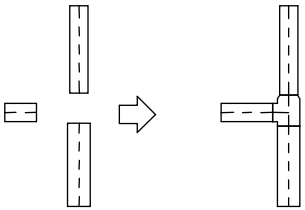
Joining a flex pipe/duct with a rectangular pipe/duct that are inserted at the same angle:

A rectangular-round reducer is inserted.



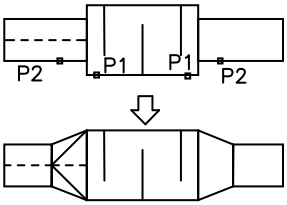
Three round pipes/ducts with a different diameters. Two pipes/ducts are coaxial and the third one is inserted at a different angle:

A tee-piece with reducer or **a reducing tee-piece** are inserted.



- Joining flex pipes/ducts with other pipes/ducts that are inserted at different angles - rules as for round pipes/ducts.
- Some joining cannot be done because there are no appropriate fittings to do them, e.g. there are no flex tee-pieces.
- It is impossible to join pipes/ducts/cable trays belonging to different systems. If this joining is necessary, a user should do it by inserting needed fittings manually.
- Duct device and rectangular or round pipe/duct (both are inserted at the same angle):

  A rectangular, round or rectangular-round reducer is inserted (depending on a form of both duct device and pipe/duct)



Edition of fittings

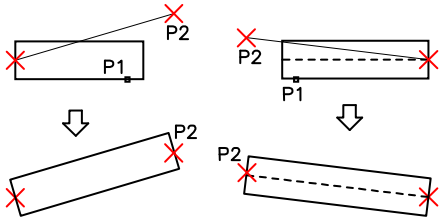
The **2D edit and info** command is used to obtain information about pipes/ducts/cable trays and fittings in a drawing.

Once a user has run a command and specified a required fitting, a dialog will be displayed in which all parameters and dimensions of a selected element are presented.

If a user selects a pipe/duct/cable tray, he can change its length directly.

A direct change of the length of a pipe/duct/cable tray

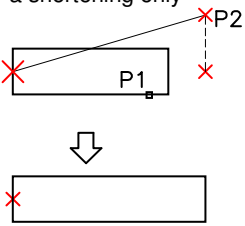
In order to change the length of a pipe/duct/cable tray a user should select the end of the object (P1) and then specify a new point as the other end of the object (P2).



Lengthening or shortening a pipe/duct/cable tray

Except for a direct change of the length a user can lengthen or shorten pipes/ducts/cable trays.

Unlike direct change of the length, during a lengthening or a shortening only the length of the pipe/duct/cable tray is changed while a position of the element is not changed.



During lengthening or shortening a user can:
- set the length by specifying a point (P2)
- specify completely new length
- specify the change of the length in percentages
- specify increase of the length

After a user has selected a command, he should select a pipe/duct/cable tray.
To specify a new length, a user should select the **changeLength** command in the command line. Then he should enter a required value. The length can also be set by selecting any point (P2).

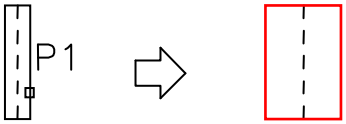
Alerts and options displayed in a command line:

```
Specify CAD-Profi fitting:
Specify point or [changeLength/Edit]:L
Specify point or enter new length or [Delta/Percent]:5000
```

The **Delta** option indicates change of increase of the length of an object. In case of entering a negative value the object will be shortened.

The **Percent** option indicates change of the length of an object in percentages in relation to a current length. If a user enters value 200%, the length will be doubled, but if he enters value 50%, the length will be reduced by half.

The **Edit** option enables a user to change a cross-section and other parameters. After a user has selected the **Edit** option, a dialog with all current parameters of pipe/duct/cable tray will be displayed.



An example of changing of diameter of a pipe

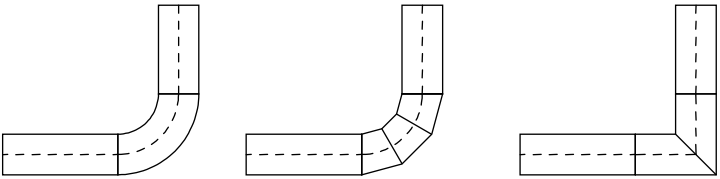
Edition of fittings

To edit fittings a user should start the **2D edit and info** command, select an edited fitting and change required dimensions and parameters in a dialog.

All elements of the dialog that is used to edit are the same as in the command used to insert fittings. An important difference between this two commands is that options used to automatic setting of dimensions (e.g. the length) are default turned on in a command to insert fittings. These options are default turned off in an edition command – it allows a user to specify all dimensions easily.

When editing, a user can specify single dimensions of fittings or select typical fittings from databases of manufacturers or norms.

He can also change a type of a fitting, e.g. an elbow can be changed for a segmented elbow, a two-segment one or an elbow with revisory door.

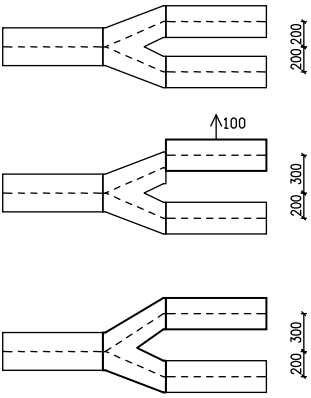


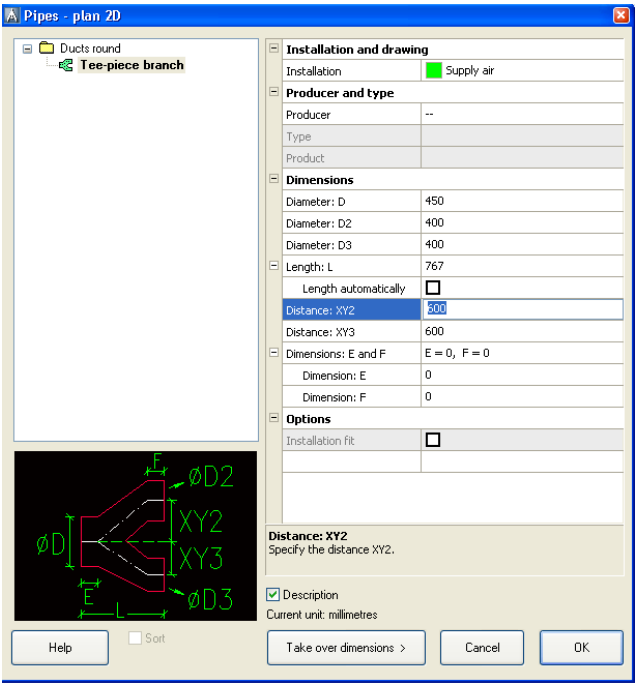
An example of changing type of a fitting

During editing of fittings the **Copy** option is available. It behaves the same way as in the command used to insert fittings.

Example of edition of a tee-piece branch:

- Pipes were joined using a tee-piece branch. The distance between branches is 200 mm.
- During designing, it was necessary to move one of the pipes 100 mm up – it caused the distance of the branches to be 300 mm now.
- In order to correct a drawing a command to edition of fittings was used. In this command, a new distance between axes was entered: XY2 = 300





A dialog for edition of fittings

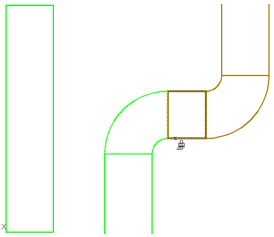
Change of the system for fittings

All elements that belong to the given system or to the installation in the program CP-System are placed automatically on a separated layers. If a user wants to change the system for selected elements, he can move them on the other layer.

In AutoCAD / IntelliCAD program, there are many ways of changing a layer of objects. Using a versatile command to match properties is a very convenient way:

Buttons **Match properties** on the standard toolbar of AutoCAD and **Match** in the program IntelliCAD.

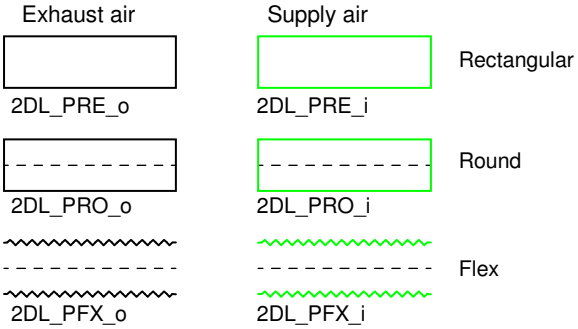
Example:
In order to change the system (e.g. from supply air for exhaust air in air conditioning installation) it is necessary to start the **Match properties** command, select a source object and then select all object for which a layer is to be changed.



Note:
Round, rectangular and flex objects that belong to the same system are placed on different layers.

Example:
For air conditioning layers are as described below:
- round objects are placed on 2DL_**PRO**_xxx layer
- rectangular objects are placed on 2DL_**PRE**_xxx layer
- flex objects are placed on 2DL_**PFX**_xxx layer

The beginning of a name of a layer (2DL) denotes air conditioning installation, middle part (PRO, PRE, or PFX) denotes a form of an object and the last part (xxx) means a system or installation (supply, return, etc.).



Note:
A user can turn off an automatic structure of layers for symbols and objects in CP-System. For pipes/ducts/cable trays and fittings layers specify the system or the installation. Therefore these elements are placed on various layers even if an automatic structure of layers is off.

Dividing 2D pipes/ducts/cable trays

When designing installations, a user can draw pipes/ducts/cable trays without concentrating on lengths of particular segments, because it is possible to divide a long sections of pipes/ducts/cable trays into shorter elements.
The **Divide pipes/ducts** command allows a user to do that quickly and easy. A user can divide one or many pipes/ducts/cable trays depending on a used option.

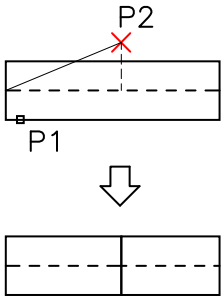
Once a user has run the command, he can select a pipe/duct/cable tray or one of available options in the command line:

```
Select pipe or [Layer/All/Select/byPipe]:
```

If a user selects a pipe/duct/cable tray then CP-System will activate **a mode of dividing of a single pipe.** The following message will occur in the command line:

```
Select point, enter distance or [segmentLength/Dynamically]:
```

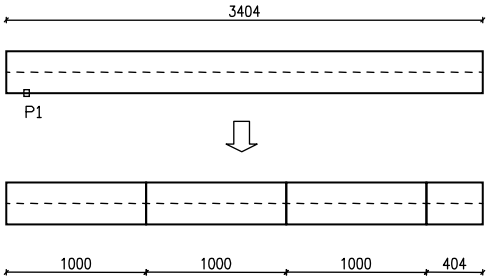
It is possible to **select directly the point of dividing** (P2) or enter the distance of a dividing measured from the beginning of the selected object (P1).
The **Dynamically** option enables a designer to divide an object into variable-length sections. Successive points should be specified to make dividing.



The **segmentLength** option is used to divide a whole selected object into fixed-length sections. A new length must be entered to make dividing. CP-System divides the object starting from the end lying closer to the selected point (P1).

Example of using the **segmentLength** option:

```
Select point or enter distance or [segmentLength/Dynamically]:L
Specify length of segment <1000.000>: 1000
```



Note:
During dividing of numbered objects, numbers are removed. Therefore it is recommended to number objects after ending edition of installation.

Dividing many pipes

Using **Divide pipes/ducts** command a user can divide many pipes into sections.

Once the command has run, a user should select an option: **Layer, All, Select, byPipe**. These options make pipes/ducts/cable trays to be selected in a different way.

```
Select pipe or [Layer/All/Select/byPipe]:
```

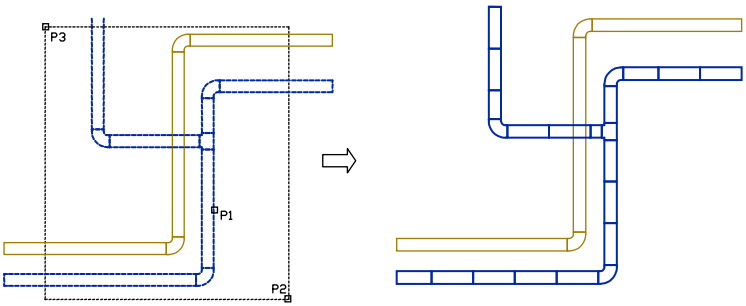
The **Layer** option is used to divide all objects lying on a given layer. After choosing the option, an object lying on a required layer should be selected. Afterwards, a user should specify the length of sections. Since pipes/ducts/cable trays belonged to the given system or installation are lying on a specified layer, the **Layer** option can be used to divide quickly all objects from a given installation (e.g. a user can divide all ducts lying on 2DL_PRE_o layer).

The **All** command is used to divide all pipes/ducts/cable trays in a whole drawing. After choosing the option, the length of sections should be specified. All objects (including not visible currently on the screen) are divided in this option. If there are many objects to process, an operation can be long-lasting. Therefore a status of processing is displayed in the command line e.g. Done: 70%.

The **Select** option allows a user to select any pipes/ducts/cable trays to divide. After choosing the option, the length of sections should be specified.

The **byPipe** option enables a user to select pipes/ducts/cable trays intelligently. After choosing the option, an object belonging to a given system should be selected. Afterwards, the other objects should be selected e.g. using window selection mode. Only elements compatible with a previously selected object will be selected (elements belonging to the same system or installation), the other will be filtered out.
An example of using the **byPipe** option:

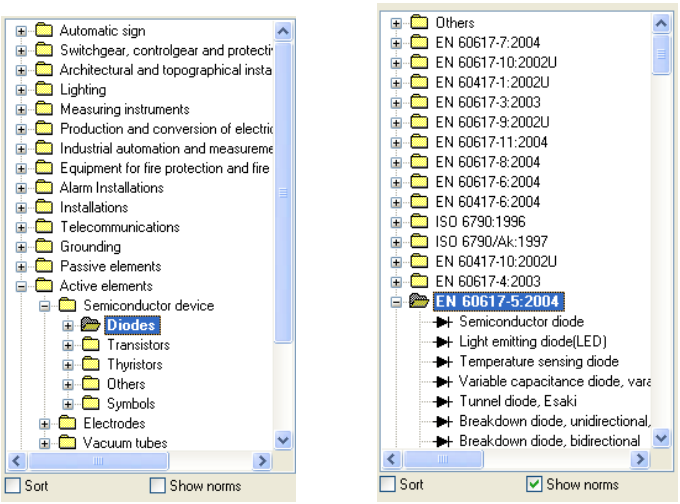
```
Select pipe or [Layer/All/Select/byPipe]:P
Select 2D pipe: (a user should select an object from a required system)
Select 2D pipes: (a user should select other objects)
Specify length of segment <500.000>: 1000
```



Electric schemas

Schemas of electrics are created similarly to schemas of plumbing/heating or air conditioning installations. Intelligent symbols, cable trays as well as functions for descriptions are available.

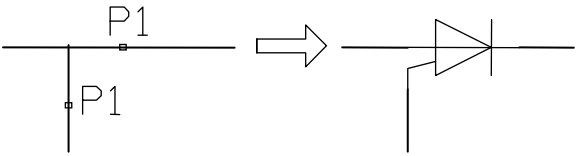
In electric module, there are a lot of symbols. They were divided into two basic libraries: symbols used during designing schemas and symbols used on architectural plans. Many symbols can be used both on plans and on schemas. Since the beginners can have problems with finding required symbols, the **CP-System** ->**Electrics - commands** -> **Symbols - all** command is available in main menu. In order to find a symbol it is recommended to use the **Search** button, which displays a dialog with all currently available elements. Particular symbols are grouped in thematic categories.
It is possible to sort the elements of libraries and to display them by norms.



Symbols are also available in **Multisymbols**, **Marks**, **Cross symbols** commands. These commands enable a user to insert many symbols at once, automatic inserting symbols in cross of lines as well as inserting marks. Some marks are filled automatically on the base of properties of described objects.

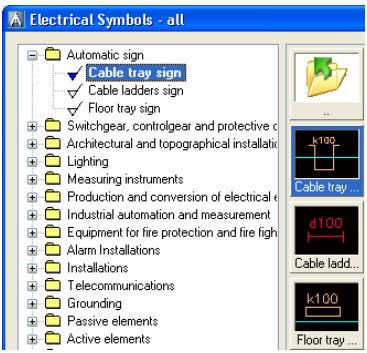
All properties of symbols and all options of particular commands were described in details in previous chapters:

- **Multisymbols** (page 21). We recommend to familiarize with the properties of symbols, ways of inserting many symbols, inserting symbols in lines, inserting a symbol in many lines and with an intelligent copying symbols.
- **Symbols** (page 26). In this chapter, there are described options of symbols, inserting symbols, inserting symbols in lines, inserting symbols repeatedly. It was also explained how to insert symbols at a constant angle, multivariant symbols or symbols containing texts.
- **Cross symbols** (page 31). Automatic inserting symbols in a cross of two lines. Using this option a user can insert easily thyristors, switchers, transistors, etc.

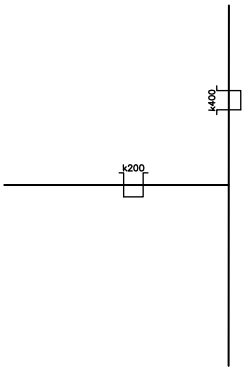


- **Marks** (page 32). Special symbols used to describe drawings. Automatic marks and symbols are described. For them a user can select parameters from appropriate databases.

  Below, there is presented an example of automatic marking the size of schematic cable tray.



selecting of a symbol

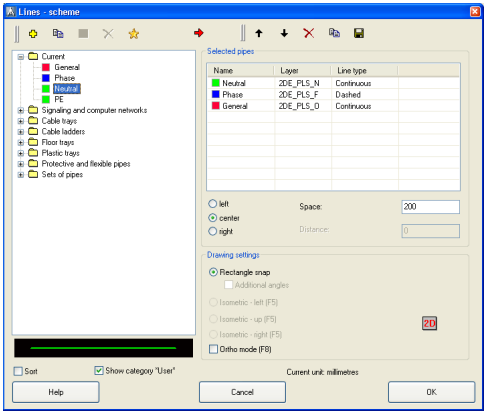


automatic mark of cable trays

- **Removing symbols** (page 34). A user can remove one or many symbols.
- **Mirror symbols** (page 35). A convenient way to mirror symbols inserted in lines. This command can be also used to other objects of CP-System program.

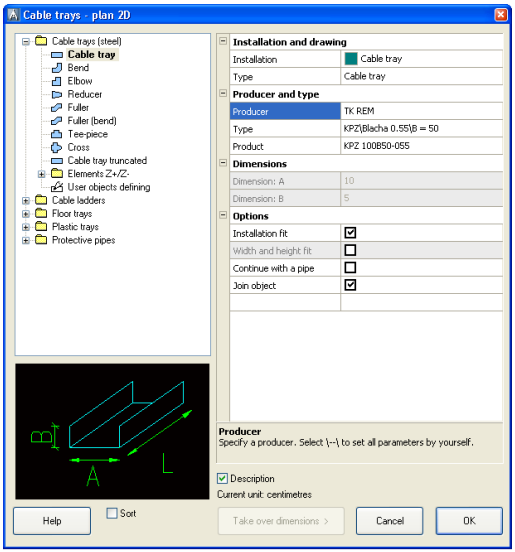
Cable trays

The „**Cable trays (schematic)**" and "**Cable trays**" commands are used to draw cable trays (see page 36). Cable trays, cable ladders, floor trays, plastic trays, protective and flexible pipes and much more elements are available. They are drawn by means of lines with the same colour but lying on different layers. In order to change a type of cable, a layer of this element must be changed. Cables can be described automatically with the use of special symbols of marks.



The **Cable trays (schematic)** command enables a user to define any types of cable trays. The newly defined elements are added to the *User* category. A user can copy and edit their style. It is impossible to edit a style of original cable trays.
If their definition must be changed, they must be copied to *User* category and edited in any way by the user.

Commands „**Cable trays**" as well as „**Cable trays - sequence**" enable a user to draw 2D cable trays. Drawing of them is very similar to drawing plumbing pipes or air conditioning ducts (see page 51).
It is possible to specify any dimensions for drawn cable trays or to select them from databases.



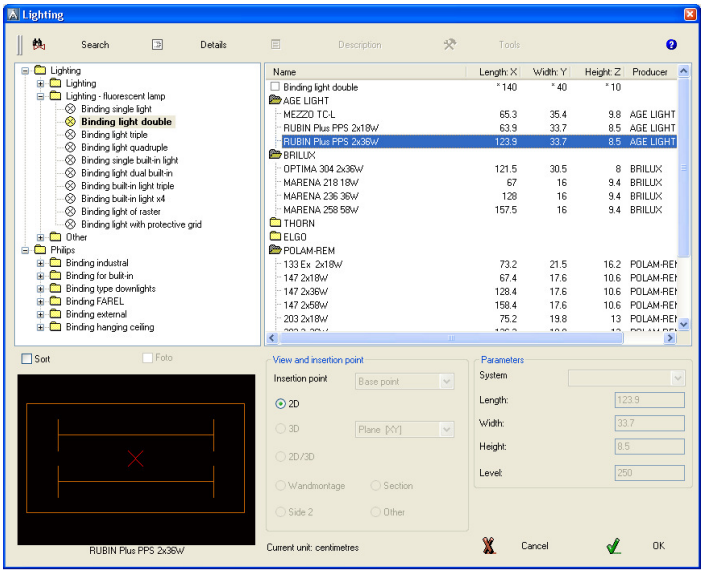
Note:
Although cable trays, cable ladders, floor or plastic trays and other 2D elements look similarly on plans, joining automatically elements of various systems is not possible.

Lighting and other objects

Lighting

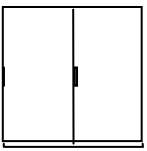
The **Objects and lighting** command enables a user to select and to insert binding lights into projects. Catalogues of binding lights were prepared according to catalogues of known manufacturers. In the program, there are also general binding for which a user can specify any dimensions. After a user has inserted an object in a drawing, he can specify their parameters (manufacturer, type etc.). These data will be considered for specifications.

Binding lights as CAD objects are inserted in the same way as other objects described in details on the page 46.

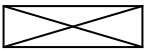


Switchgears

In CP-System, there are many types of cabinets and distributors. It is also possible to specify user's dimensions of cabinets and their types.



NKVS2/1100/SV
Front view



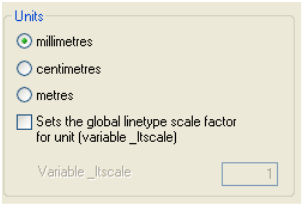
NKVS2/1100/SV
Up view

Switchgears are used in the same way as other parametrical objects in CP-System. Detailed description of parametrical objects is on the page 46.

Architecture

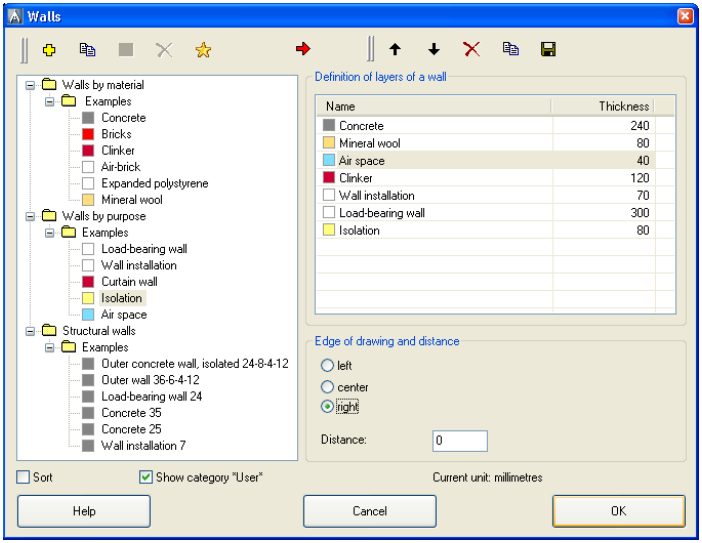
CP-System Architecture is intended for designers of trade installations. This module is sufficient supplement to design or edit building plans much faster.
Architects can treat it as a basic program which allows to create architectural documentation using walls as well as other elements more quickly and easy.

After installing CP-System, a default unit is mm. Most architectural projects are designed in cm. In order to change the unit a user should start the **CAD-Profi Options** command and mark “centimeters” button



Walls

A user can draw single-layer walls or structural walls in CP-System program. During starting designing in CP-System it is necessary to define a required materials or structural elements of walls because of very big variety of materials used in buildings. It is also possible to save definitions of structural walls which a user will use in his projects.



Before drawing walls, it is necessary to specify layers of walls or select a previously saved definition of a structural wall.

Selection of layers of a wall

In order to add a single layer (material) to a list that contains a definition of a drawn wall, a user should highlight a required element in the right part of a dialog and then click the **Add an element to a list** button . Layers of a wall can also be added to the list by a double click or selection of the **Add to a list** option from context menu (a right mouse button). An element that is being added is inserted at the end of the list with default thickness. It is possible to change thickness of every layer of a wall. An order of elements added to a list can be organized with the use of buttons **Move selected rows up**, **Move selected rows down**, **Remove selected rows** and **Copy selected rows** .

The **Save as a structural wall** button enables a user to save a whole definition of a wall. It is also possible to save walls that consist of single layers. It is used to store various thickness for walls of specified types, e.g. a supporting wall 18 cm, a supporting wall 24 cm etc.

In case of adding to a list structural walls, whole list of layers is replaced with elements from an added wall.

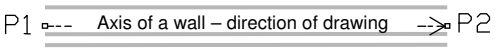
Drawing of walls

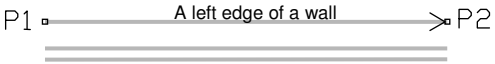
After a user has selected layers of a drawn wall and clicked a button **OK**, he should specify points between of which the program will draw segments of a wall. The way of drawing is similar to drawing lines in CAD program.
During drawing, it is possible to use various options in the command line:

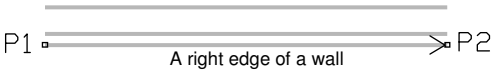
```
Specify first point for a wall:
Specify next point or [Undo]:
Specify next point or [Close/Undo]:
```

The **Close** option enables a user to close boundary of a building or room automatically. The **Undo** option enables a user to move back to a previous point. Thanks to that it is possible to correct designed walls without cancelling of drawing.

Walls can be drawn by axis, by left or by right edge. Left and right edge are determined by the direction of pipes drawing (points P1, P2…).
While a designer is drawing by left or right edge, he can specify the distance from drawn points. Thanks to that, it is possible to draw walls in distance from other elements in a drawing. Drawing with the distance is used e.g. for partition walls.

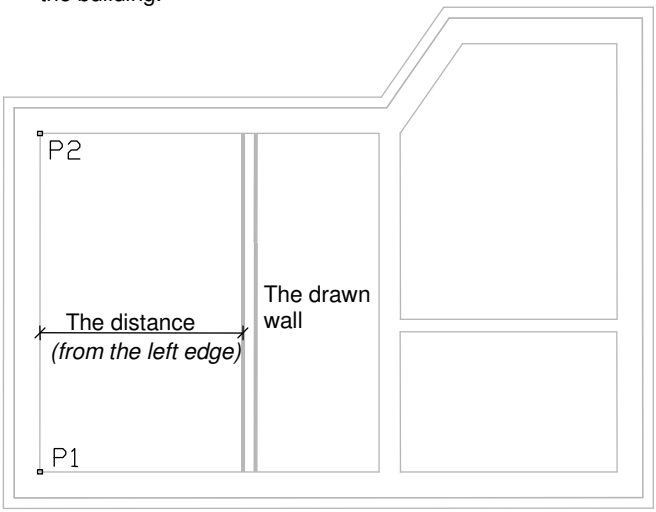






An example of drawing a partition wall
In order to draw a partition wall, a user should:

- Select **the left edge of drawing** in the dialog.
- Specify the distance which suits the width of the room (e.g. 360 cm).
- During drawing select points P1 and P2 that are lying on the side wall of the building.



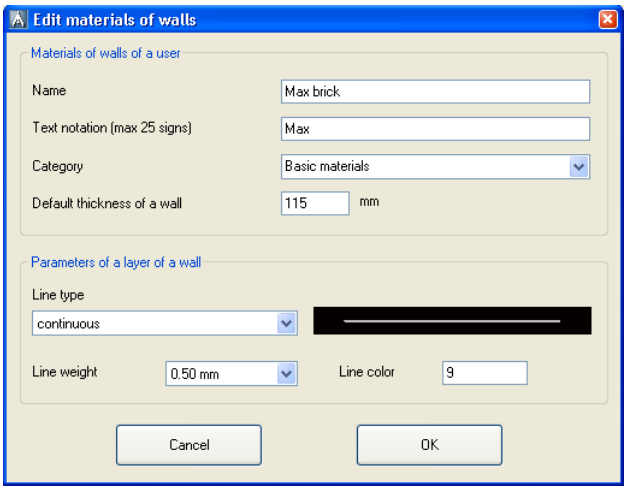
Defining and editing user's walls

After a user has installed the program, predefined materials and structural walls are available. These elements can neither be changed nor removed by a user. A user can define his own elements of walls on the basis of those materials. Defined elements are stored in *UserData* folder (see page 8 and 111).

After pressing the **Add** button a dialog will appear where a user should specify a name and a style of an added material or a wall. He should specify colour, linetype, lineweight and default thickness of the wall in the style.

The colour has an influence on the lineweight in plots in CP-System program. That is why after changing of a colour, the program will suggest an appropriate lineweight. Unless styles of plots (files *.ctb) of CP-System program (see also information at page 108) are used during printing, a user can select other lineweight

Text mark – is used to name a layer on which a wall will be placed.



Elements defined by a user can be copied, removed and edited with the use of appropriate buttons or context menu. This menu occurs when a right mouse button is pressed.

Copy. Copying selected element to the ***User*** category.

Edit. Changing colour, linetype and other parameters of a wall.

Delete. Removing of selected definitions of user's walls. When a user has removed an element by mistake, he can retrieve it. In order to do this, he should select the **Restore removed** option from a context menu **directly** after removing an element.

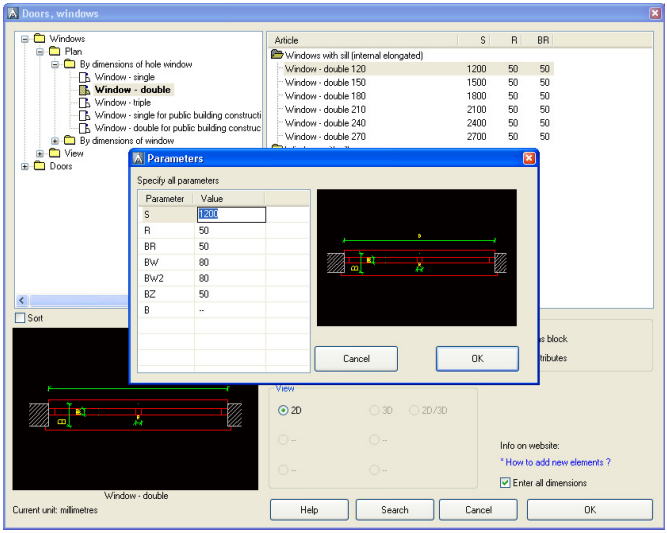
Add to favourites. Copying an element to the ***Favourites*** category. Thanks to that a user can create a list of the most often used materials or types of walls.

Note:
Only walls defined by a designer and placed in the *User* or *Favourites* category can be removed and edited.

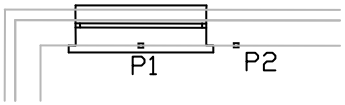
Architectural woodwork, parametrical objects

There are typical windows and doors in CP-System. It is also possible to insert elements on plans (horizontal sections) as well as in views of elevations or vertical sections. In case of inserting windows on plans, a user can specify the thickness of a wall. Thanks to this windows are fitted to the thickness of walls. After inserting windows or doors in walls, it is necessary to „cut out" manually redundant segments of walls using of the _TRIM command. Walls will be corrected automatically in the future releases of CP-System.

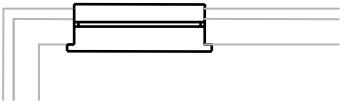
Windows and doors are parametrical objects for which specifying dimensions is possible. Databases containing elements prepared on the base of norms and products of known companies are available for windows and doors. During inserting parametrical objects, a user can enter dimensions manually. In order to do this he should mark the **Enter all dimensions** option before clicking the **OK** button.



During inserting a window, a user should specify the point of inserting (P1 – centre of a window) as well as the second point to specify an angle of inserting (P2). After the window or the door has been inserted, it is necessary to remove redundant parts of walls using _TRIM command.



A view after inserting the window

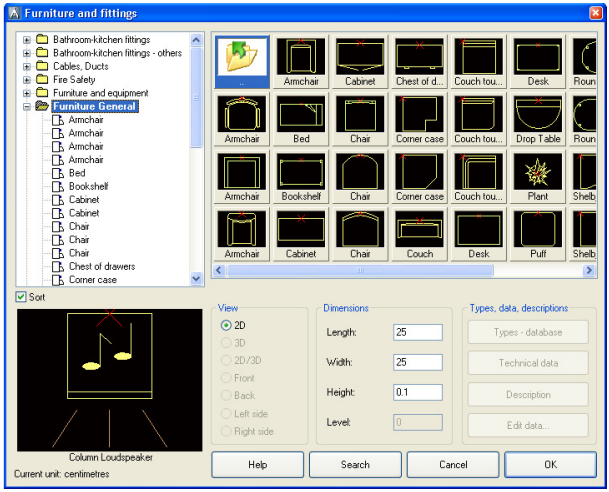


A final view of a wall

Furniture, equipment

The **Furniture and equipment** command enables a user to select and insert furniture, plumbing objects etc. A user can specify any dimensions or select types from available databases for most objects. After a user has inserted objects in the project, he can specify their parameters (manufacturer, type, etc). These data will be considered during creating specifications.

Detailed description of work with objects is discussed on page 46.

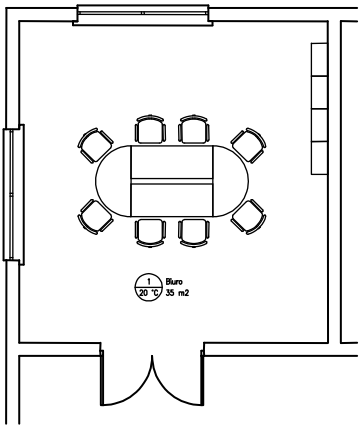


Symbols and marks

This command enables a user to describe objects. There are available level indicators, marks of architectural woodwork as well as a label of rooms.

The **Marks** command is described more detailed in chapters about edition of schemas of installation (see page 32).
In architectural module this command works in the same way.

A user can create his own libraries with the use of the **Add block to library** command. User's symbols are added automatically to a special library „Working_symbols" (see page 111).



Mechanics

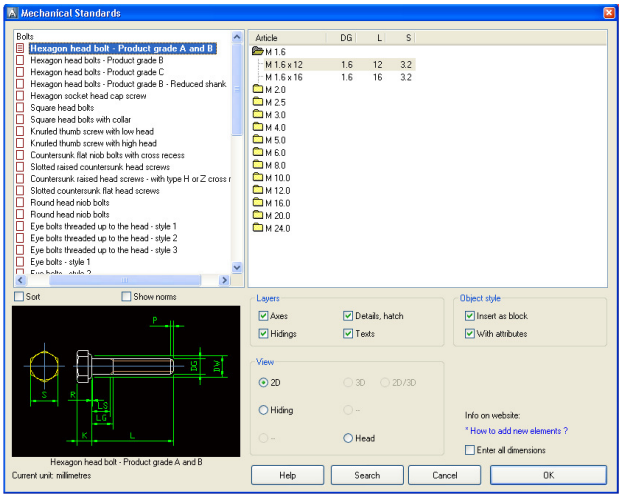
CP-System Mechanics can be used by designers during creating detailed drawings.
In CP-System, there is a extensive set of standardized elements that is essential for every designer.

Most of CP-System modules are intended for designers who create projects in architecture. Because of that, a default a global scale factor for lines is matched to architectural projects. In mechanical designs, it is essential to change the linetype scale factor. A default value of factor is 99.9 during designing in mm, 9.99 during designing in cm and 0.0999 during designing in m in CP-System.
If a global scale factor is 1.0 in saved drawing, CP-System will change it for one of default values when a drawing is open. A user should set any linetype scale other than 1.0 or default values by himself to make CP-System not to change a global scale factor.
Changing of linetype scale is done by the command _LTSCALE.
Once a command has been started, a user should enter a required value, e.g. 5. Here is an example of changing of linetype scale:

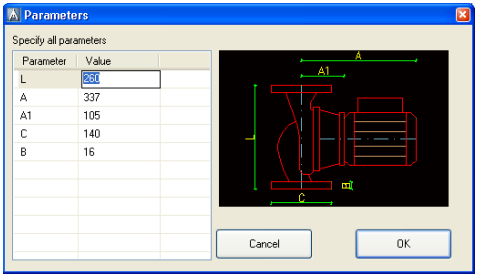
```
Command: _ltscale
Enter new linetype scale factor <99.9000>: 5
```

Parts

In CP-System, there is an extensive set of standard elements for which various views are available. In case of necessity of using element with non standard dimensions, a user should mark the **Enter all dimensions** option and then specify required dimensions in a dialog.

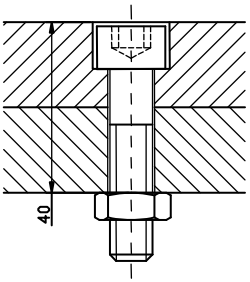


If a user specifies dimensions by himself then the program does not check whether they are correct, so it's possible that a "deformed" object will be inserted.



Contents of a catalogue can be looked through by types of elements or norms.

Many objects from the **Parts** command require „manual" specifying one or more dimensions. In this case a dialog is displayed in which a user should enter necessary dimensions.



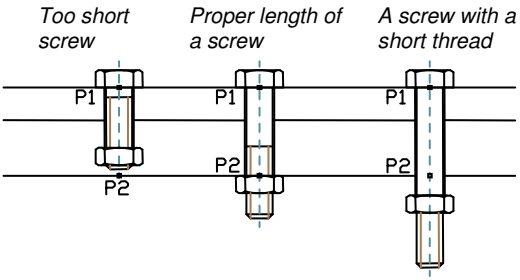
During inserting some objects, e.g. „screw joining" two points should be specified. The first point is a base point of an inserted block while by means of the second one, the automatic length (i.e. the distance from the base point to a nut) is calculated.

For screw joining CP-System controls minimum and maximum length of a joining for selected type of screws.

An example of a screw joining

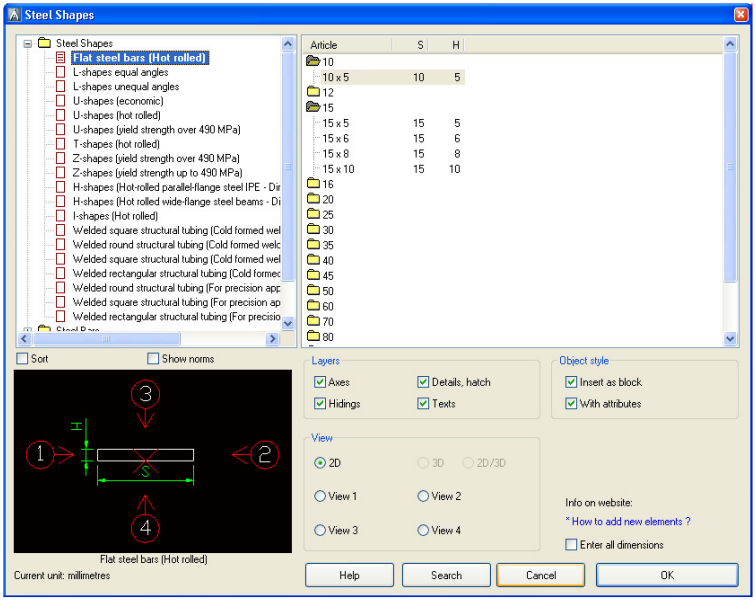
In order to insert a screw joining (i.e. a screw and a nut) a user should select an appropriate type of joining in a dialog and a required size of a screw. Then he should specify a point of inserting (P1) and the thickness of a joining (P2) in a drawing.

Note: A user should select the appropriate length of a screw in a dialog. Selecting, for instance, 30mm-length screw for 40mm-joining is a mistake.

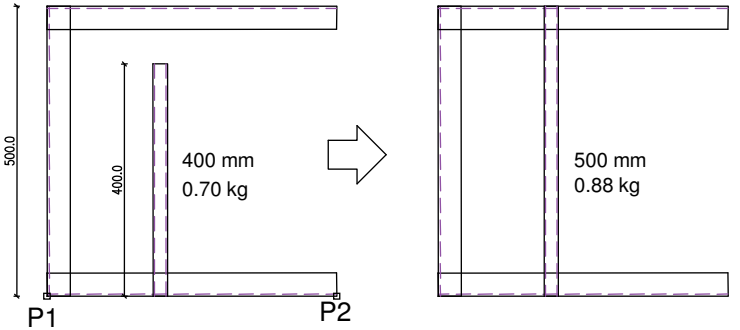


Steel shapes

The command works in the same way as the **Parts** command.
During inserting steel shapes in 2D view, specifying the length of a drawn shape is necessary.
Similarly to other objects a user can specify his own dimensions. It allows to create very untypical solutions.



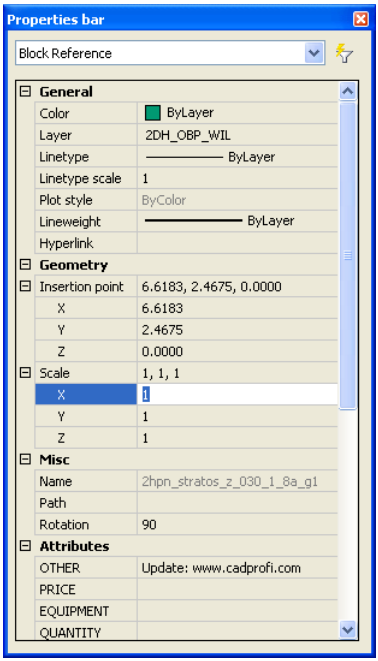
An example of drawing steel shapes
After a user has selected a shape, he should specify the beginning point P1 as well as the length and an angle of a drawn element (P2).



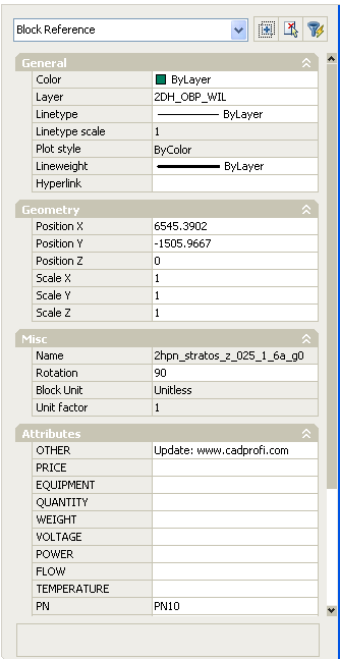
400mm-long closed shape.

Change of the length of shape for 500mm, i.e. change of scale X = 500

Steel shapes are inserted as blocks in CP-System program. Scale X for side views suits the lengths of shapes. In case of cross-sections, the length of shape suits scale Z of block. Therefore by changing scale X or scale Z a user can change the length of a shape inserted in a drawing, e.g. using CAD command **Properties**.
In order to display the **Properties** in AutoCAD, a user should click combination of **Ctrl + 1** buttons or select an appropriate command from main menu of AutoCAD. The **Properties** command is in main menu **Change** in IntelliCAD. The **Properties** is also displayed after double click a required object.



The Properties dialog in Bricscad

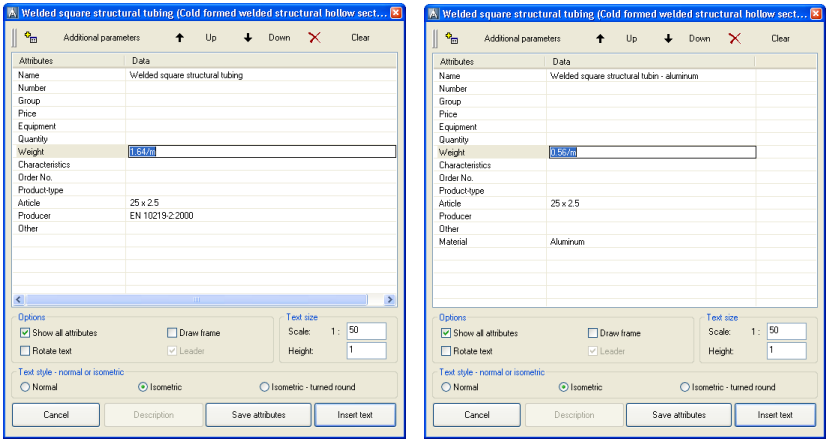


The Properties dialog in AutoCAD

Fittings have defined a unit mass. Thanks to this a user can obtain information about the length and the mass of elements that are inserted in a drawing. He can also obtain automatically full specifications of steel (or other metals).
Most CP-System shapes are steel elements prepared according to specified norms. A user can specify any dimension. It allows to insert untypical shapes, aluminum shapes, copper ones and other. If untypical sections are inserted, it is necessary to correct the type, the kind of material and unit mass or total mass of inserted elements. These parameters can be corrected easily with the use of the **Attributes and descriptions** command. This command enables change data for many selected objects simultaneously (see page 92).

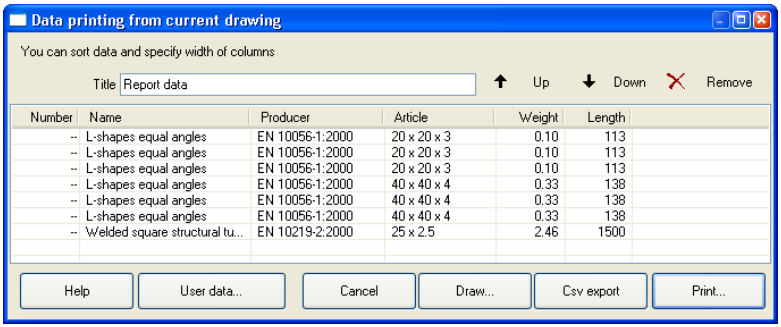
An example of marking the mass of shapes:

- The mass „1.76/m” denotes a unit mass 1,76 kg per one meter of length of a given shape. During creating specifications the program counts the mass of an element taking into consideration his length.
- The mass „3.4” denotes a mass of a given element.



Changing parameters of steel shape for aluminum shape

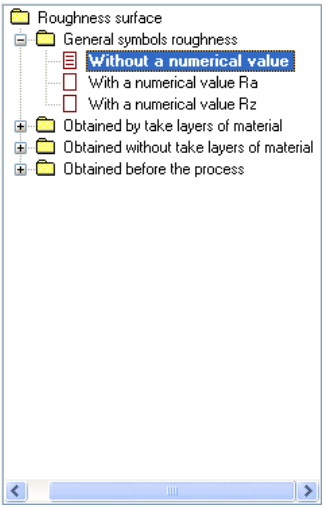
The specification of shapes can be created with the use of the **Specification** command.
„Mechanics – steel profiles...” should be selected as a type of a specification. A user can create summary specifications as well as complete lists of elements.

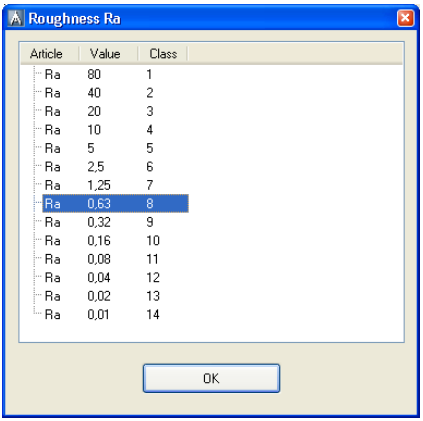


More information about creating specifications is in chapter **Specifications** on page 101.

Symbols and marks

Typical marks of surface roughness are in CP-System.
After a user has selected a symbol with numerical value, a window is displayed in which typical values are contained.





The **Marks** command is described in details in chapters about edition of schemas on page 32.

A user can add his own symbols and marks to the program with the use of the **Add block to library** command. User's symbols are added automatically to special „Working_symbols" library (see page 111).

General commands

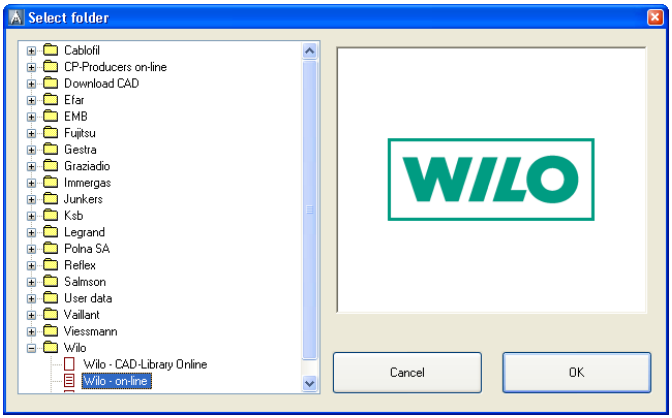
Except for commands and objects used directly during designing of installations, commands and functions supporting designing of installations are also available.

Manufacturers' libraries

Objects from libraries of manufacturers – CP-System includes "CP-Manufacturers libraries" module. Because of that, devices of known manufacturers are available, not only general and parametrical objects. All objects from manufacturer libraries have views that present in details particular devices. Most devices includes technical data as well as catalogue cards.

Manufacturers' libraries can occupy much space on hard disk and that is why during installation of the program a user can choose only some libraries. He can also install selected manufacturers' libraries without copying data on a hard disk (in such case objects from manufacturers' libraries will be inserted directly from CD-ROM). Libraries are also distributed by manufacturers that have license for „CP-Manufacturers" program. It allows a user to add new catalogues to CP-System at any time. He can do this by installing of new libraries obtained from manufacturers.

Many libraries of manufacturers are available in the program. **Products library** command enables a user to select an active library. Once a command has been run, a dialog is displayed in which a user can select a required library.



After a user has selected a library, a dialog is displayed in which he can look through devices or schemas of selected manufacturers.
Structure of libraries can be different for every manufacturer. Because of that, dialogs with elements from libraries can be also completely different. There are the same options as e.g. in **Plumbing objects - plan** or **Parametrical objects** commands in every library.

- The '**Working folder**' library is intended to create new CAD libraries and to extent the contents of the program by a user (see page 111).

Command 2D/3D

A user can use 3D objects in his own designs. A very convenient way is to create 3D model based on 2D plans. 3D model is used to obtain e.g. perspective as well as isometric views. 3D technology also allows to detect potential mistakes and conflicts easier and makes the documentation to be more readable and clear.

The **2D/3D** command generates 3D model based on 2D plans. It is also possible to make the reverse operation (inserting 2D blocks based on 3D model).

The command can only be used for devices that have defined 2D and 3D blocks simultaneously.

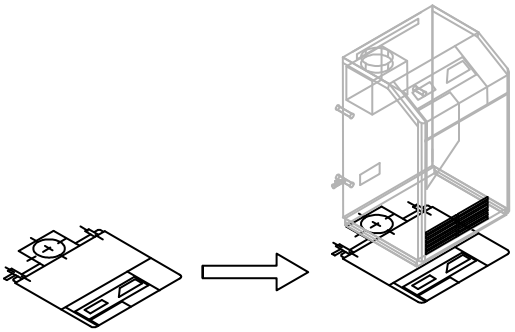
Once a user has run the command, he should select the object(s). The next step is either to enter a value of level on which inserted objects will be placed or to choose one of options: **Layer** or **Point**.

```
Enter level or [Layer/Point] <1300>:
```

Selecting the **Layer** option causes all objects on a given layer to be processed. The objects are placed on a level specified in triangular brackets. Selecting the **Point** option enables a user to specify 3D point. The Z coordinate of this point specifies value of level on which a new 3D object will be placed.

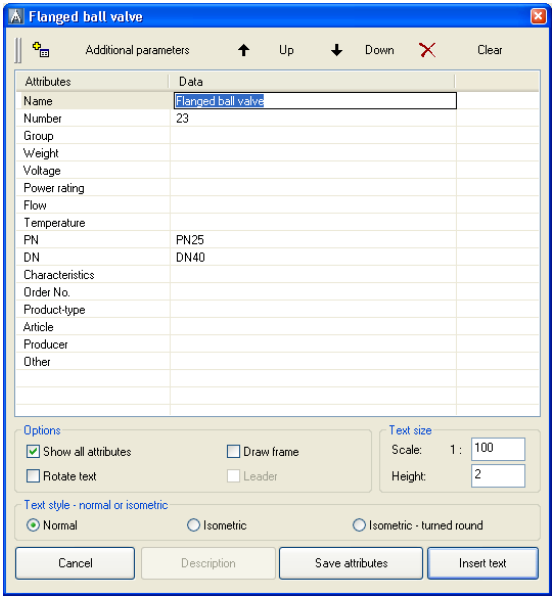
Note:

- The 2D/3D command **is not** available in IntelliCAD program.



Attributes and descriptions

During inserting symbols and objects, CP-System adds to them descriptions, parametrical data etc. These data are stored as an attributes. The **Attributes and descriptions** command enables a user to edit attributes and place them in a drawing as a text. Attributes are also used during creating specifications. It is possible to edit only objects created by CP-System. After a user has run a command and select a required object, a dialog is displayed in which particular attributes can be edited. The **Save attributes** button saves data and ends the command. The **Insert text** button also saves data and allows to insert selected texts into drawing.



In order to insert one line of a text, a user should select (highlight) a required attribute. However, in order to insert a few lines of a text, a user should select all required attributes. During selecting, a designer should click on names of attributes and keep the **Shift** or **Ctrl** buttons pressed.
The text is inserted in the same order as in the dialog (omitting empty lines).
The **Up**, **Down** buttons enable a user to organize the list. Organizing the list does not change the order of attributes saved in a given object. Because of that, the order of attributes will not be changed when the same object will be processed once again.
A click in a value of attribute (in a column Data) enables a user to edit a selected attribute directly. The **Clear** button erases data for selected attributes.

Global edition of attributes

The **Attributes and descriptions** command allows a user to edit attributes of many objects simultaneously. After a command has been started, the following text will be displayed in the command line:

```
Select CAD-Profi object or [Multiple]:
```

The Multiple (letter M) allows to edit many objects simultaneously. After selecting objects a dialog box is displayed in which all attributes from selected blocks are available. If values of attributes are various, the ***Varies*** text occurs. If the text is not changed then the given attribute will not be changed for any object (an attribute is omitted during saving data). The remaining data are saved in all selected objects.

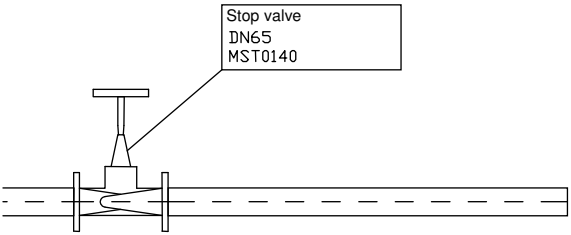
Attributes	Data
Name	***Varies***
DN	***Varies***
Manufacturer	Armature

Inserting texts

The size of a text on plots (in mm) as well as target scale of plots should be specified in dialog. After pressing the button **Insert text** a user should select a point specifying the beginning of the first line of a text (P1) as well as a point specifying an angle (P2). Instead of a second point a user can enter the value of an angle (e.g. “0” or “90”). In order to insert texts more quickly, the **Rotate text** option can be turned off. In this case text will be inserted horizontally.

The style of inserted text depends on current style in CAD program. Inserted text can be edited subsequently with the use of appropriate commands of CAD program.

It is also possible to insert texts in frames. In order to do this a user should select the **Draw frame** option. A leader can also be added to a frame.

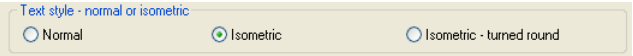


Note:

- The **Descriptions** button is used to display descriptions and catalogue cards for selected object (if a description is available in database). Most often catalogue cards are attached to objects from manufacturers’ library.

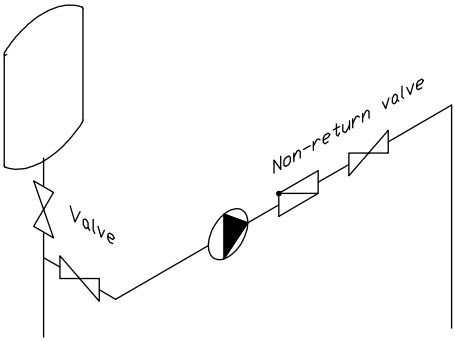
Descriptions in axonometric drawings

The **Attributes and descriptions** command enables a user to insert texts in axonometric drawings. In order to insert text in isometry or dimetry a user should select the option **Text style** – **Isometric** or **Isometric turned round**.

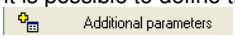


When inserting text in isometry CP-System activates isometric mode of a graphic cursor. Thanks to this specifying position and an angle of inserting text along demanded axis is easier (axes are placed at an angle 30°, 150° and 270° in isometry).
Only if a current angle of isometry is set on 30°,will isometric cursor be set automatically.

If a user wants to insert text in dimetry, he can enter an angle of inserting in the command line. It is the most convenient way. This angle has to be compatible with an angle specified in a dialog **CAD-Profi Options** (page 18).
If specified angle is other than 30° then CP-System assumes that the dimetry will be designed. Therefore a graphic cursor is switched in normal mode (rectangular snap and grid).



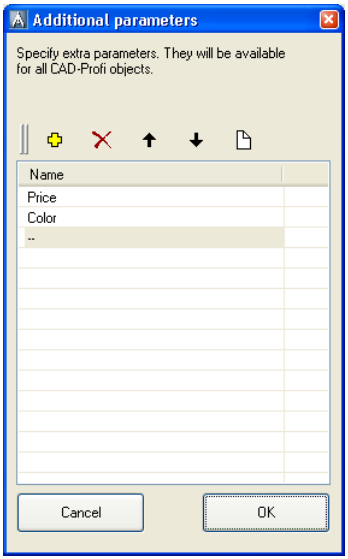
Defining and using additional parameters

Most objects have standard attributes in which a user can specify basic parameters. If additional parameters are necessary for description of objects, it is possible to define them. In order to do this a user should select a button 

and then specify names of additional parameters.

These parameters will be available for every object of CP-System.

Data included in additional parameters can be used in specifications or descriptions.



Notes:

- Additional parameters are not stored in attributes, so they are invisible in standard commands of AutoCAD / IntelliCAD used to edit attributes.
- If additional parameters for CP-System objects were filled then these data would be stored even in case of removing definitions of these parameters.
  However they will not be taken into account in specifications.
- When defining parameters, CP-System assigns automatic identifiers to them. After removing a given parameter and adding it again to the program, other identifier will be assigned to it. Because of it a user should avoid redundant operations of removing or adding parameters.
- Definitions of removed parameters can be retrieved at any time. If an object has parameters that are not on the list then the **Recover parameters** button is visible. After pressing this button, identifiers of retrieved parameters will occur. A user should enter required names of parameters in place of identifiers.

Drawing frames and tables

Both a frame and a table are important elements of every drawing.
The **Frames and tables** command allows a user to insert typical frames and tables easy. There are available horizontal and vertical frames as well as tables grouped in categories.

A user should fill tables by edition of their attributes with the use of the _DDATTE command.

Categories of tables:

Tables	typical drawing tables.
Specifications	tables used to make specifications in a drawing.
Legends	typical legends that contain recommended layers of pipes for plumbing and air conditioning installations.
User's	empty tables that can be defined by a user.

User's tables should be created in *Tab_usergb1.dwg* to *Tab_usergb5.dwg* files (*gb* denotes language version of the program, e.g. for English - *gb*).

Files of user's tables are stored in folder *C:\CadProfi\xx\Blocks\Tables_gb* (*C:\CadProfi\xx* it is a default folder of CP-System).

After creating tables a user should create slides for them. The slides should be saved in the *Tab_usergb1.sld* to *Tab_usergb5.sld* files in folder *C:\CadProfi\xx\Slides.*

The CAD command _MSLIDE is used to create slides.

Note: Legends and specifications can be also created by means of a Specification Wizard

Numbering of objects

Numbering system enables a user to assign numbers to symbols and objects created by CP-System. Objects data as well as numbers are used in descriptions and specifications.

During an automatic numbering, a separated numbering for various installations and objects is used.

Examples of sequences of automatic numbering:

- Numbering of plumbing pipes and fittings – a unique numbering for installations: supply, return, gas etc.
- Numbering of air conditioning ducts, fittings and objects – a unique numbering for particular systems: supply air, exhaust air, etc.
- Numbering of plumbing symbols, objects and armature – common numbering is used for all objects.
- In case of using additional libraries of symbols, a unique numbering will be used for particular modules:
  - CAD-Profi Electrics
  - CAD-Profi Architecture
  - CAD-Profi Mechanics.

After a user has started the **Numbering** command, he should select an object or a symbol that has been drawn in CP-System program. If a selected object does not have any number yet, then a dialog with options of numbering is displayed. If a selected object has already got a number, a user can switch the program to mode of removing numbers.

Assigning numbers

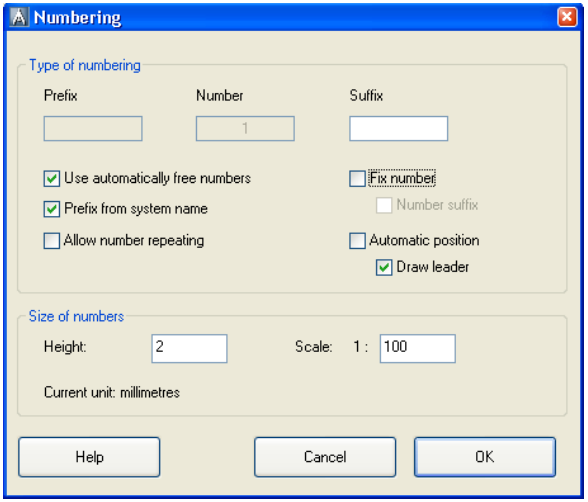
In a dialog, a user should specify the way of numbering as well as the size and appearance of inserted numbers.

The way of numbering

A number can consist of three parts:

- **Number** - can only be a whole number (integer) from 0 to 32767.
- **Prefix** – any text. Prefix is most often used to mark an installation or a system, e.g. V4 can denote fourth outlets system. The 'Prefix from system name' option is used to automatic assigning prefixes based on names of systems.
- **Suffix** – any text. In case of using multilevel numbering, a user can number automatically suffixes. In order to turn on this way of numbering, a user should mark the 'Fix number' option as well as 'Number suffix'. Then CP-System will number automatically the last segment in a whole number.
  An example of multilevel numbering:

W4-7-1,	W4-7-2	W4-7-3
W4-7-1.2.1,	W4-7-1.2.2	W4-7-1.2.3
7-1.2.1,	7-1.2.2	7-1.2.3

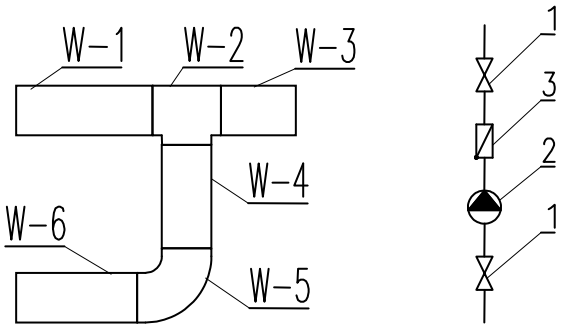


During numbering a user can assign automatically next free numbers. After turning off the 'Use automatically free numbers' option, a user can number objects starting from any given number. CP-System monitors process of numbering thanks to that assigned numbers are unique in a given sequence of numbers. If a user turn on the 'Allow number repeating' option, he can repeat numbers in a given sequence.

During numbering of objects a user can insert texts of numbers with a leader. The point of inserting of numbers is specified by a user. It is possible to turn on the 'Automatic position' option in which texts of numbers will be inserted directly in numbered objects.
The size of texts of numbers depends on height, target scale of plot and the unit of designing.

After a user has specified settings of numbering, he should select objects as well as the position of texts with numbers. In order to finish numbering a user should press the ENTER or Esc buttons.
After starting numbering, it is possible to number objects that can be assigned to a given sequence of numbering. It means that if a user starts numbering for example outlets then he can only use pipes, fittings and objects of outlets to finish the command. In order to number other installations, he should start again the '**Numbering**' command.
In case of selecting an object that does not belong to a given sequence of numbering or a selected object has already got a number then the following message is displayed in the command line, e.g.:

```
> Object is not compatible with previously selected object
> This object has already got a number: 1
```

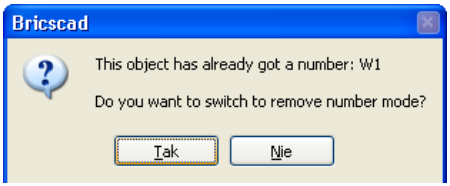


An example of various sequences of numbers in one drawing

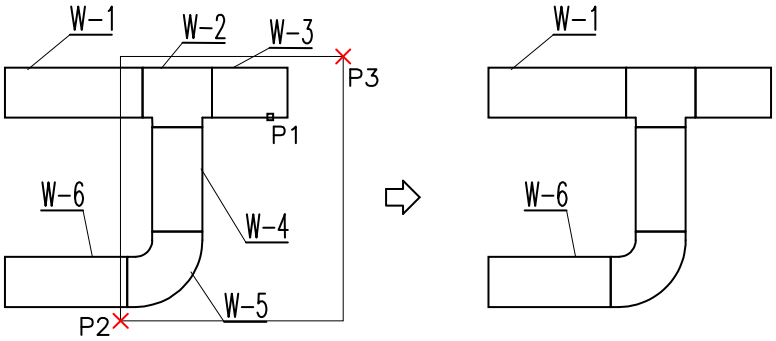
Removing numbers

During numbering, numbers are assigned to objects. Additionally, texts of numbers with a leader (if necessary) are inserted. An assigned number is stored directly in an object. Removing the text with a number will not cause a number from an object to be removed. Therefore the '**Numbering**' command must be used to remove numbers from objects totally.

In order to remove numbers a user should start the '**Numbering**' command and then select an object that has got a number (P1). It is also possible to select a text with a number. CP-System will suggest a mode of removing numbers.



After a user has activated a mode of removing of numbers, he should select required objects or numbers (P2-P3). Assigning of numbers as well as texts and leaders are removed for selected objects.



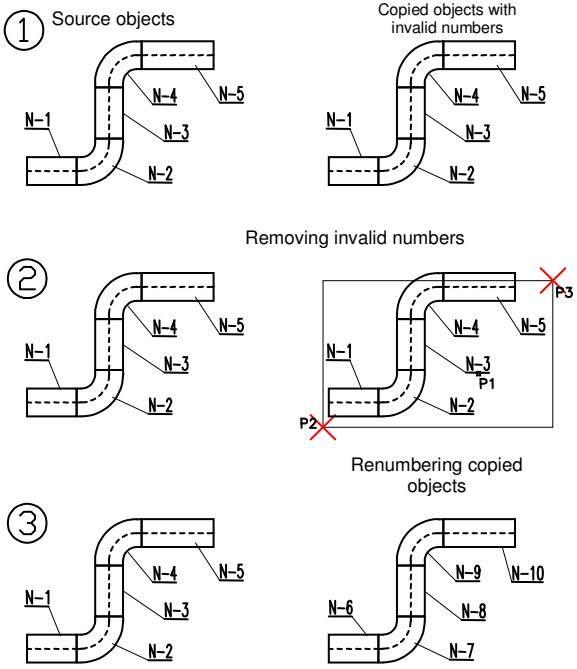
Copying objects with numbers

During copying numbered objects, target elements are treated as objects without numbers!

Copied objects should numbered again, if necessary.

If texts of numbers are copied (during copying e.g. by means of CAD command “Copy”) then these copies are only graphical objects with no meaning. They are invalid from CP-System’s point of view!. The invalid texts of numbers should be removed from a drawing!!!

To do that, a user can use the mode of removing numbers in the ‘**Numbering**’ command. After the command has been started, a user should select invalid text and then select removed texts by the window of selection. Other objects being inside a window of selection will not be removed.



Edition of numbered pipes/ducts/cable trays and fittings

Every command that make pipe/duct/cable tray to be divided into two separated elements, causes the assigned numbers to them to be removed. Numbers are removed during:

- inserting parametrical armature or duct devices in pipes or ducts;
- some joining, e.g. by means of tee-pieces;
- in the **Divide pipes/ducts** command;
- during removing symbols by means of the **Remove symbols** command.

Specifications

The '**Specifications**' command enables a user to obtain various specifications of data. To obtain specification easy and quickly a user can select one of prepared templates and direct printing.

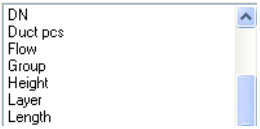
In case of creating specifications in AutoCAD 2007 (or higher) or in Bricscad v8.x (or higher) the program bases on Unicode characters. Thanks to this a user can obtain multilingual plots (e.g. Russian, Polish, German and other texts can be placed on one specification). Specifications created in previous releases of AutoCAD/IntelliCAD program are compatible with codepage of a current language of CP-System or operating system.

After the command has been started, a dialog is displayed in which a user should select a type of specification. There are many prepared templates that are used to obtain various specifications.
A user can make separated or summary specifications for symbols, objects as well as pipes/ducts/cable trays and fittings of appropriate installations.
Examples of templates:

- 'CAD-Profi – selected objects, symbols and pipes' – creating summary specification with all types of elements inserted by CP-System. When creating specification, a user can select objects from a current drawing.
- 'CAD-Profi' – all symbols ' – automatic creating specification with all symbols inserted by CP-System. When creating specification, a user has not to select objects in a drawing (all objects other than symbols will be filtered out).
- 'CAD-Profi – all symbols - summary' – automatic creating specification with summing the amount of the same symbols.
- 'Selected CAD objects' – a very versatile template that enables a user to create specification with any objects. CAD objects like: lines, arcs, circles etc. are taken into account as well.
- 'Plumbing/heating – pipes 2D, fittings 2D' – a template to create specifications with elements of plumbing/heating installations.

For every template various data are available. They can be presented in tables. All available data are placed in a left panel of a dialog. Columns to print are placed in a right panel of a dialog. I is also possible to add additional data to a right panel and to organize them. Specifications can be printed in a vertical or a horizontal order. More columns can be selected for a horizontal order.

Using additional parameters



If additional parameters were defined during describing objects, they will be available in a left panel of a dialog now. These parameters can be added to a right panel of course and thanks to this they will be taken into account in a specification.
Defining additional parameters is described on the page 95.

Options of specifications

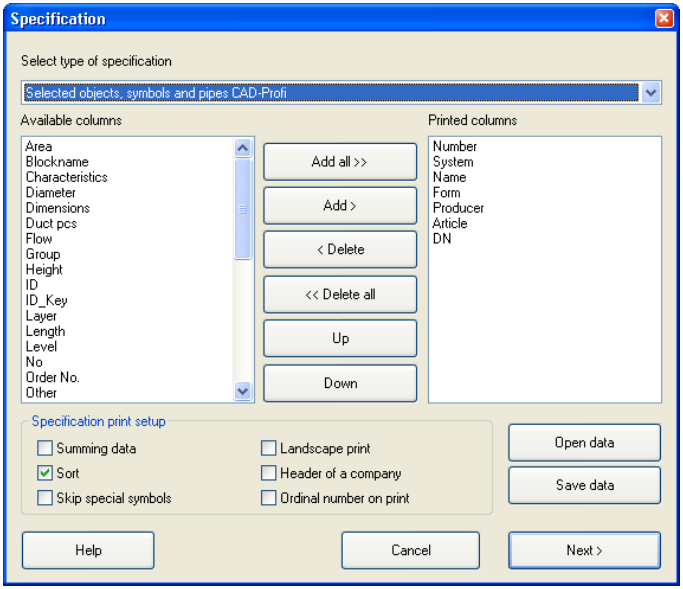
The '**Summing data**' option enables a user to make summary specifications with summing of amount of the same objects. The lengths of pipes, area of ducts and fittings etc. can also be included.

The '**Sort**' option is used to sort data on plots initially. Data can also be sorted in the next dialog containing initial preview of plot.

The '**Skip special symbols**' option causes blocks with no NAME attribute or with empty NAME attribute not to be taken into account in specifications. Blocks that are used to describe a drawing (symbols of pipe slope, marks of amounts of electric pipes) are not taken into account in specifications as well.

'**A column with ordinal number**' – adding a column with an ordinal number.

'**Header of a company** – this option enables a user to print a header of a company on the first page of specification. Data for a header as well as company logo are specified in another dialog containing initial preview of plot.

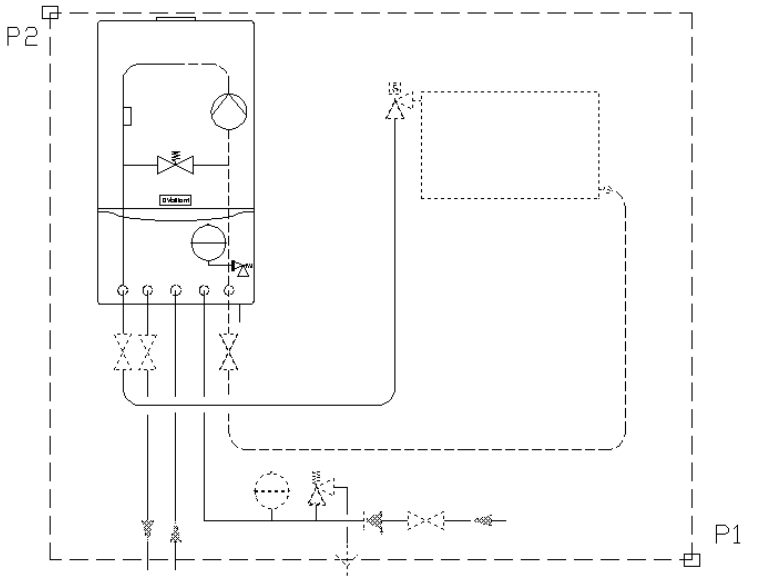


Summary specifications

Except for creating specifications from a current drawing, a user can also create specifications for designs that consists of several drawings.
In order to create a specification from several drawings a user should create a temporary file in which data from every drawing will be saved. Then this file can be printed as a summary specification.
The '**Save data**' button is used to create a file with temporary data. After pressing this button, a user should specify a file in which data will be saved. If an existing file is specified then data will be added to existing data. **In order to create summary specification** a user should start the '**Specifications**' command in every drawing, select the same template and save data to the same file. The '**Open data**' button is used to print data from a temporary file.
The temporary file has structure that is compatible with csv format, so this file can be used to export data to MS Excel or to MS Access databases.
Data of CP-System can be exported to various file formats (xls, pdf, rtf...). The way of exporting is described in a further part of this chapter.

Selecting objects to a specification

After pressing the '**Next>**' button, a user should select objects that are to be considered in a specification. All CAD methods can be used to select objects. CP-System filters selected objects. For instance, when creating specifications for symbols, CP-System selects only symbols. After a user has selected objects, a window that contain obtained data is displayed.



Window of data

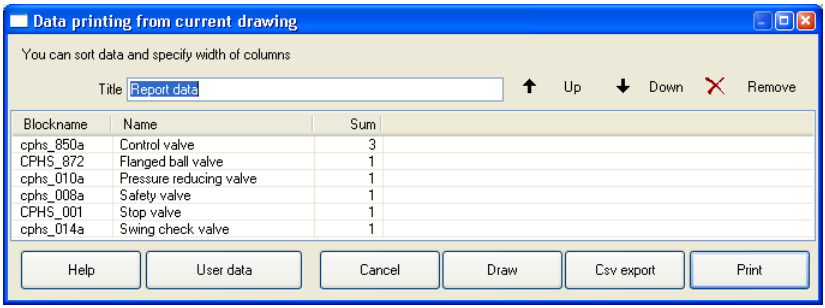
Sorting data, setting the width of columns in plots as well as removing selected rows or hiding redundant columns is possible in a window.

In order to sort data, a user should click a title of a required column. Re-click causes data to be displayed in inverse order. It is possible to sort many columns. In this case, a user should sort the lowest priority columns as a first.

In order to remove all rows of data, a user should mark a required rows and press the Delete button.

If a designer wants to change the width of a column, he should place a cursor between columns. A cursor will change itself for two-way arrow that enables change of the width of column by dragging. Selected columns can be hidden in this way. This possibility is useful for hiding empty columns or columns that are redundant on plots but were important by sorting data.

A title of plot should be specified in a window of preview. It is also possible to define a header of a plot: address as well as logo of company.



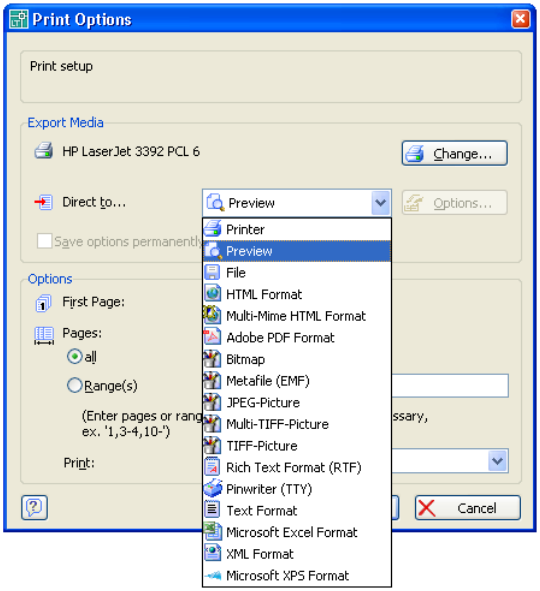
The ‘Print’ button enables a user to print or export data to many formats.

The ‘Csv export’ button enables a user to export data to a ‘csv’ file, that can be loaded to MS Excel, Open Office or other applications.

The ‘Draw’ button is used to create a graphical legends and also to fill drawing specifications tables.

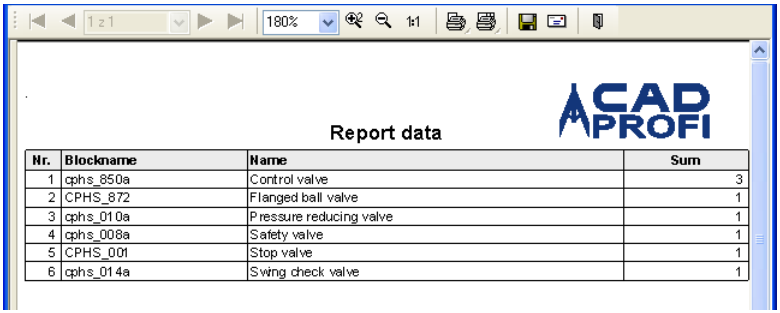
Printing data

After pressing the ‘Print’ button, a dialog with options of printing is displayed.



A user can print data directly on a printer or export them to various formats (xls, txt, rtf, pdf, html, xml, and many other).

During direct printing, it is comfortably to display a final preview thanks to what making any corrects will be possible. An interesting option is a possibility of sending the first page of specification on the other printer than the rest of pages. This option can be used to print the first page consisting of logo of company on colour printer and the rest pages on a monochromatic printer.



Nr.	Blockname	Name	Sum
1	cphs_850a	Control valve	3
2	CPHS_872	Flanged ball valve	1
3	cphs_010a	Pressure reducing valve	1
4	cphs_008a	Safety valve	1
5	CPHS_001	Stop valve	1
6	cphs_014a	Swing check valve	1

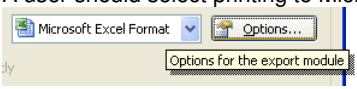
During exporting data a user can specify parameters of export in details, e.g. only export of data or export data with keeping formatting.

A program to display LL files
A preview of plot can be saved in LL files. To browse LL files a user can use LLview13.exe file which is placed on the disk in main CP-System folder (e.g. *C:\CadProfi\xx*).

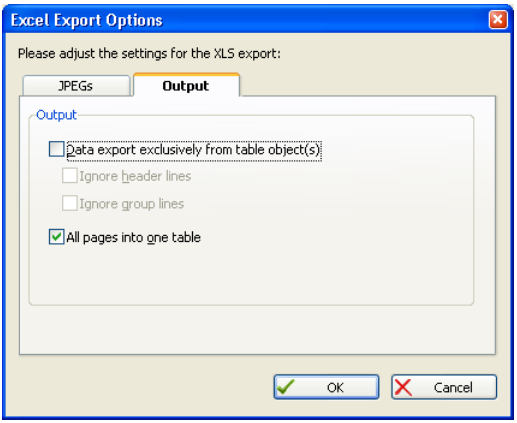
Example:

Export data to xls format (Excel) with keeping formatting.

- A user should select printing to Microsoft Excel format.



- Additionally, he should specify options of module of data export.



In order to export with keeping formatting a user should turn off the 'Export data only from objects of a table' option.

Export data to xls format without keeping formatting.

- A user should select printing to Microsoft Excel format.
- Additionally, he should specify options of module of export data.
  In order to export data only, without keeping formatting, a user should mark the 'Export data only from objects of a table' option.

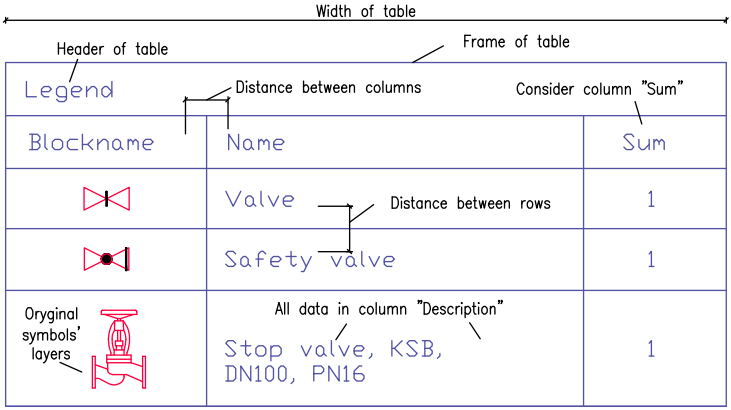
In order to create a drawing legend or specification using Specification Wizard, a user should press the 'Draw' button.
The command works in two modes: 'Legend' and 'Specification'.

- The '**Legend**' mode is used to make any defined legends or tables. A

user can specify a total width of a table, distances between particular columns and rows as well as the size and the style of the text. A legend can contain a header with a title of a specification and with titles of particular columns.
The 'Description' column has a special meaning. After a user has turned on the "All data in a 'Description' column", a legend will be created where all data from a drawing will be consolidated to one column.
When turning off this option it is possible to create tables that contain many columns.
When a user selects the '**Block**' column in a Specifications Wizard, legends containing views of symbols and objects will be created. Particular symbols in a legend, can be placed on layers that are compatible with layers of symbols in a drawing. Thanks to this, symbols in a legend will have colours and other properties of original objects from a drawing.

Note: *Legends containing symbols are created only on the base of a current drawing (summary graphical legend containing data from several drawing cannot be created).*

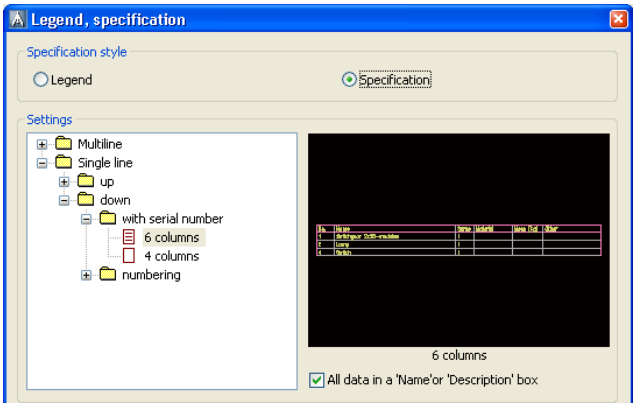
The width of particular columns of legends are specified proportionally to the width of columns that are set in a main window of data. After setting a cursor of a mouse between columns, the cursor will change for two-way arrow that enables a user to change the width of selected columns.



- The ‘Specification’ mode is used to fill template tables of drawing specifications. Tables of specifications are prepared on the base of the suitable norms. That is why tables of specifications have precisely specified order (the width of columns, the size and the style of texts etc.). When inserting tables of specifications, a user should pay attention to choose the same columns in Specification Wizard and a given type of tables. Otherwise, tables will not filled in a correct way. The ‘All data in a column Name’ option enables a user to consolidate various data to a column containing various descriptions.

  Note: Specified system of names of particular parameters is established in the program, such as ‘Name’ ‘Type’ ‘DN’, ‘PN’ etc. Using other names is required in some tables, e.g. a parameter ‘Type’ is equivalent to a parameter ‘Symbol’ or ‘Mark’.

  Example: A table contains columns ‘No’, ‘Symbol’ (Type), ‘Description’, ‘Sum’. In this case a user should select columns ‘Type’ and ‘Name’ in Specification Wizard. The ‘Sum’ column will be filled when the ‘Summing’ option is active. A column ‘No’ is filled when the ‘Ordinal number on print’ option is active.



Layers in CP-System

Layers of AutoCAD are being created automatically during designing in CP-System program. In order to manage these layers easy, a lot of commands in main menu of the program are available.
Using the **CAD-Profi Options** command**,** a user can specify on which layers the program is to place various elements.
It is recommended to turn on automatic managing of layers. It is also possible to insert all elements on a current layer (manual managing of layers by a user) or on a layer with a fixed name.

Automatic managing of layers allows a user to keep order in a created drawings.

In order to obtain proper colours and lineweight on plots of drawings created in CP-System, a user should use "CadProfi Color.ctb" and "CadProfi Mono.ctb" plot styles. These files should be copied from a CAD-Profi folder e.g. *C:\CadProfi\xx\PlotStyle* to a CAD folder e.g.: *C:\Program Files\AutoCAD\Plot Styles*.
Files *.pcp are used in IntelliCAD program.

C:\CadProfi\xx – it is a default folder of CP-System program.

Structure of layers

Name of created layers consists of several segments (segments are separated by signs "_").
A prefix of layer name specifies the trade of designing and a type of created drawing: 2D, 3D, isometry etc.

- The second segment specifies a type of an object, e.g. symbol, pipe, duct etc.
- Another segments of name of layers specify a specific installation, type of material, type of pipe etc. (e.g. S – supply, R – return). These segments are created when the **Extended layer name** option is checked in the dialog **CAD Profi Option** (page 18).

There are also special layers for special purposes (details, axes, hidings). On special layers objects such as texts of tables etc. are placed.

Colours of layers

Colours of automatic structure of layers in CP-System are adapted to **black background of graphical screen** of AutoCAD. Only some colours in range from 10 to 249 are used for keeping ergonomics.
If a colour is specified by a norm, it is recommended to use colours compatible with a norm in drawings, e.g. supply pipe in plumbing installations should be red. For objects which have colour independent of a norm, a user should use colours that enable to assign lineweight on plots.

Plot styles "CadProfi Color" and "CadProfi Mono" have 4 lineweights defined. A table of colours for particular lineweight is displayed below.

Colour of print	0.15 mm	0.35 mm	0.5 mm	0.7 mm	szer. obiektu
black	13, 23...	**11, 21...**	15, 25...	17, 27...	19, 29...
colour	16, 26...	14, 24...	**10, 20...**	12, 22...	18, 28...

Colours 18, 28... 19, 29... do not have defined the width of print (i.e. they have set the width of an object) and are not used in CP-System program. That is why a user can use them in any way in CP-System (no conflict with a structure of program layers will occur).

Show 2D/3D layers, Show all layers

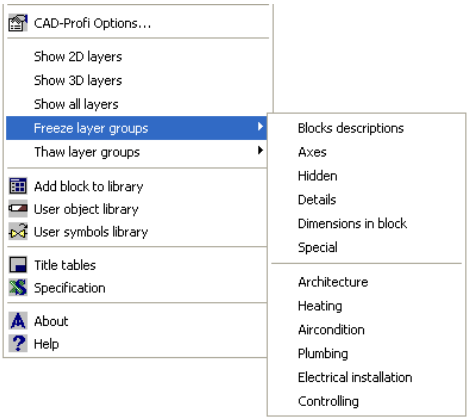
These commands enable a user to show or hide quickly 2D or 3D layers. The Shows all layers command turn on visibility of all layers in a drawing.

Note: The commands work correctly in drawings in which structure of layers of CP-System program is used.

Freeze layer groups, Thaw layer groups

These commands allow a user to lock and unlock all layers of a given trade e.g. plumbing/heating, air conditioning or electrics installations quickly.

Note: The commands work correctly in drawings in which a structure of layers of CP-System program is used.



Extending contents of the program

The program has an open structure. Thanks to this a user can add new elements as well as extend available databases. It is also possible to add user's pipes/ducts/cable trays, symbols, objects, parametrical devices, tables and frames etc. Command '**Add block to library**' is a very convenient tool to do that. This command adds automatically blocks to the working folder.

Extending databases of the program

There are available buttons that enable a user to extend and edit databases of the program in many dialogs:

Add. The button enables a user to add a new element to database. The most often this element is added to the ***User** * category.

Copy. Copying a selected element to the ***User*** category. With the use of copying a user can create a subcategory **– Favourites -**, in which he can place the most often used elements.

Edit. The button enables a user to change selected elements.

Remove. Removing selected elements from databases of a user.

Note:
A user can remove and edit only elements that have been added by himself.

User's data are saved in the *UserData* folder. This folder is placed in main folder of the program (e.g. C:\CadProfi\xx).
Particular data are saved in *'*.uuc'* files. A structure of these files is similar to a structure of *csv* files. Because of that, advanced users can edit them in any text editor e.g. notepad, while the other should use only appropriate commands of CP-System to edit '**.uuc*' files.
In order to keep user's changes, a user should archive a whole *UserData* folder. Thanks to this he will be able to restore all data when CP-System will be moved to the other computer or will be reinstalled.
During updating, user's data are attached automatically to every new version of CP-System program.

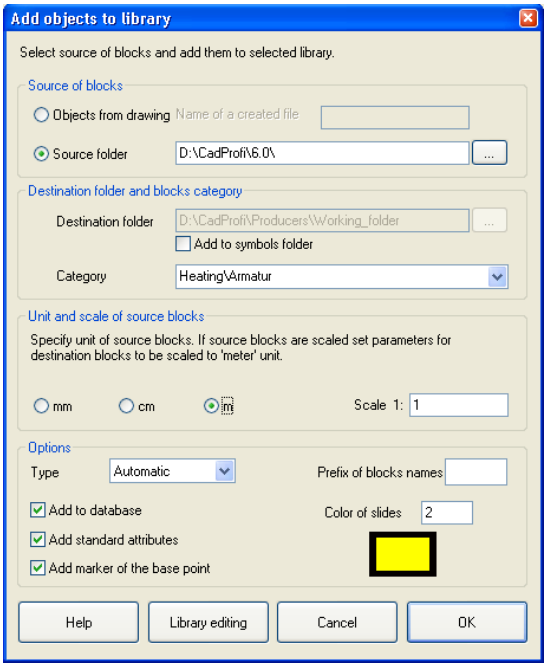
Adding blocks to library

Every CP-System library consists of CAD blocks as well as a database in which names and other parameters are defined. Blocks are saved in **.dwg* files that must have an appropriate structure. During adding new blocks, there are also created files of **.sld* slides in which are miniatures (previews) displayed in dialogs.

The **Add blocks to library** command automates the process of preparing all files which CAD library consists of, and adds them to a special 'Working folder – objects' or 'Working folder – symbols' library.

In order to add blocks saved in dwg or dxf files to CP-System, a user should start the **Add blocks to library** command in a new drawing and then specify a source folder. A button '...' enables a user to select a folder on a disk. If a current drawing is not empty, a user can add a block that consists of elements in a drawing. For a created block a user should also specified the name of a file in which the block will be saved.

By default blocks are added to library 'Working folder – objects' (folder *Working_folder* on a disk). If a user marks an option **Add to symbols folder**, blocks will be added to a library 'Working folder – symbols' (folder *Working_symbols* on a disk). Names of target symbols can be changed only in special version of the program that is intended to create catalogues of manufacturers.



In a dialog there should be specified parameters of added blocks:

- **Unit and scale** – unit of source blocks should be specified. If source blocks are drawn in scale then a user should set parameters for blocks to be rescaled to a unit 'meters'.

  Example: real device has got dimensions 20 cm x 30 cm.
  A block presenting this device has got dimensions 200 x 300.
  It means that the block has been drawn in mm. Because of it a user should mark unit **mm** and scale 1:**1** in a dialog

  If new symbols are added to the program, a user should enter such a value of scale as if target symbols have dimensions about 1x1. A lot of original symbols included in CP-System have such dimensions.

  Example: Symbol on a drawing is 200 long.
  A target symbol is to be 1 long. A user should rescale source block 200 times. So an entered scale must be: 1 / 200 = 0.005

- **Category** – specifying the name of a category of added blocks.
- **Type** - 2D, 3D, Front, Back, Left side, Right side, Schema – specifying the type of added blocks. One device can have a several separated blocks (*dwg* files) with various views in CP-System program.
  After a user has finished the command, the names of *dwg* files added to a library will be preceded by prefixes indicating a type of a block:
  2D – 2D block, plan, (e.g. *2D_ju001.dwg*)
  3D – 3D model, (e.g. *3D_ju001.dwg*)
  PD – front view (e.g. *PD_ju001.dwg*)
  TD – back view (e.g. *TD_ju001.dwg*)
  RD – right side view (e.g. *RD_ju001.dwg*)
  LD – left side view (e.g. *LD_ju001.dwg*).

  The 'Automatic' option enables automatic recognizing of type of added blocks (on the base of prefixes of name files: 2D, PD, 3D etc.).

  In case of adding symbols to the program, a user should select the 'With no adjustment' option thanks to what names of destination blocks will not changed.
- **Prefix of block names** – possibility of specifying a fixed prefix that will be added to every name of a block. Various blocks have to have unique names in AutoCAD and IntelliCAD programs. Using prefixes allows a user to change names of blocks easily (names of files) without manual edition of every block.
- **Add to database** – saving information about added blocks to databases. In case of turning off the 'Add to database' option, **.dwg* files will be processed as well as files of **.sld* slides will be created. Processing blocks without saving in database can be used e.g. to update blocks that have been added previously to the library.
- **Add standard attributes** – adding attributes to blocks. Attributes are used to describe objects as well as to create specifications in CP-

System. In case of adding complex blocks (e.g. schemas) to the library, the option 'Add standard attributes' should be turned off.

- **Add marker of the base point** – marker of the base point is visible as a red cross. This marker is added only to slides (miniatures) of blocks. This marker specifies a point of inserting blocks in drawings.
- **Colour of slides** – choosing of colour for slides of created blocks. Colour of slides is set only for elements of blocks which were drawn on layer 0 with the colour 'ByLayer'.

After a user has pressed the **OK** button, the program runs. If blocks are added from a source folder then particular blocks from this folder are inserted into drawing. These blocks are modified and saved in a library.

If source blocks already exist they will be overwritten.

If the **Objects from drawing** option is marked, a user should select objects and specify a base point for created block after pressing **OK**. A slide (a miniature) for a created block is being created simultaneously. A view of a slide is compatible with a current view. Therefore a user should set view and zoom as if only a block being created was visible on the screen.

Files created during adding blocks to library of objects:

Blocks are saved in folder:
C:\CadProfi\Producers/Working_folder\Block

Slides are saved in folder:
C:\CadProfi\Producers\Working_folder\Slides

Files of data about blocks:
C:\CadProfi\Producers\Working_folder\cp_DevicesN.uuc.
A user can edit these data after pressing 'Library editing' button in dialog **Add blocks to library.**

Files created during adding blocks to library of symbols:

Blocks are saved in folder:
C:\CadProfi\Producers/Working_symbols\Block

Slides are saved in folder:
C:\CadProfi\Producers\Working_symbols\Slides

Files of data about blocks:
C:\CadProfi\Producers\Working_symbols\SymbWorkN.uuc.
A user can edit these data after pressing 'Library editing' button in dialog **Add blocks to library.**

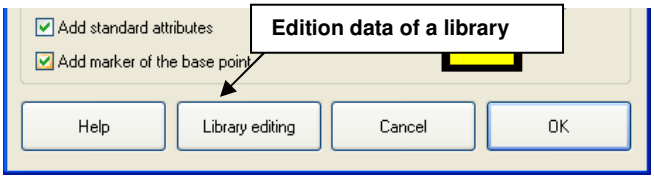
Notes:

- *C:\CadProfi* – a default folder of CP-System program.
- Some releases of IntelliCAD program create slides (preview) only with a part of a screen. Therefore by trial and error method, a user should specify an optimal view during adding blocks to libraries. In case of errors closing and reopening of IntelliCAD should be made.

Edition of data in libraries

The program enables a user to edit basic data such as: name of devices, catalogue numbers, dimensions etc.

To edit a user should run a command **Add blocks to library** and press a button **Library editing**.



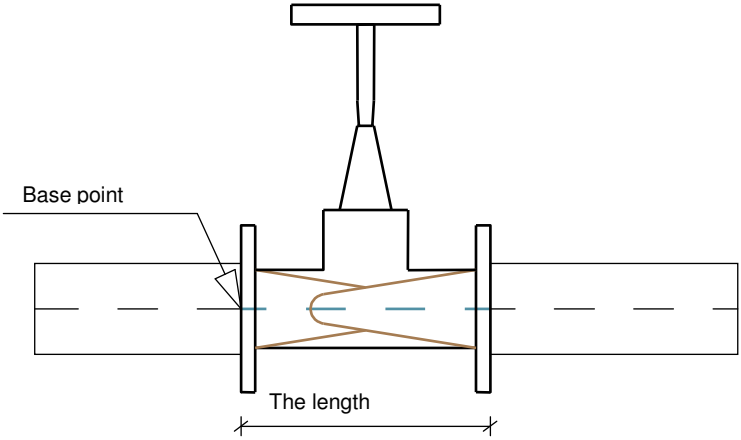
A user should specify in dialog:

- **Name** – name of an object that will be displayed in a dialog as well as in specifications.
- **Short name** – up to 10-15 characters. It is displayed below miniatures of blocks in dialogs.
- **Category** – name of category in which a device will be available.
- **Catalogue number** – if added objects present a specific device, a user can enter a catalogue number that will be taken into account in specifications.
- **Dimensions** – (length, width, height). Dimensions can be entered or measured. In order to measure e.g. the length, a user should select the most convenient view and then press the button **'…'**

  A dialog will be closed and a block with a selected view will be inserted in a drawing. Two points should be selected – distance between them determines a value of length. After selecting the second point, dialog will be redisplayed. The height of a device can be measured on front view or on 3D model. Dimensions are not available for symbols.
- **Possibility of scaling** of a block during inserting in a drawing. Parametrical blocks can be added to CP-System. When inserting these blocks in a drawing, a user can specify dimensions of a device by himself.
  An example of this object is bathtub. One block presenting a bathtub can be inserted with the various lengths (e.g. 170 or 200 cm).
  Before inserting blocks a user should check out appropriate options in a column **Fix XYZ** in order to make specifying dimensions to be possible.
  This option is not available for symbols.

- **Armature** – a user can specify in this column whether a given view (e.g. 2D) can be treated as an armature or not. Armature can be inserted in 2D pipes/ducts in CP-System. To do this properly, a block has to be prepared in special manner:

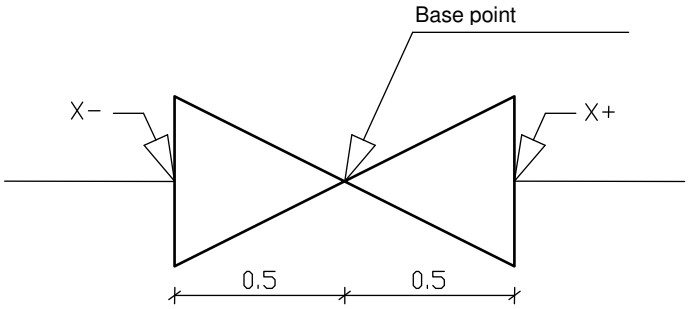
  Blocks of armature have to have a base point on the left side – this is a first point of joining of a pipe. The second point of joining of a pipe is specified by the length that should be entered in the **Length – original** field.



Edition of data of symbols

For symbols' library, a user can specify additional properties:

- **Break, X-, X+** – if the **Break** option is turned on, a symbol will be inserted in a pipe with automatic breaking in points of joining. Points of joining are specified in fields X-, X+. A designer should enter the distance of joining from a base point of selected symbol.



Notes:

There are additional parameters in a dialog. These parameters are grayed because they are reserved only to develop the program as well as for manufacturers of devices that prepare new libraries to CP-System.

List of 2D fittings

The most often rectangular steel air conditioning fittings and ducts are joined by means of flanges. Air conditioning fittings and ducts are drawn on the base of EN as well as DIN norms in CP-System. According to these norms flanges do not have to be displayed on a drawing. Thanks to this pipes/ducts and fittings from CP-System program can be used in any projects of installations that use various technologies, materials and ways of joining (e.g. installations with steel pipes/ducts, plastic installations etc).

There should be specified kind or type or technology of used air conditioning ducts in a project. An appropriate note should be placed e.g. in a describing part.

It is possible to insert flanges by means of an appropriate command of CP-System, if necessary. A designer is also responsible for decision whether flanges inserted in a drawing are to be taken into account in specifications or not.

Rectangular elements in CP-System are drawn without an axis to distinguish them from objects with other cross-section.

Most elements have various views in CP-System program (e.g. 2D view, side view). Thanks to this, it is possible to draw both plans and cross sections with pipes or ducts and fittings that are placed horizontally or vertically in relation to a plane of a given drawing. Vertical views of fittings or pipes/ducts are marked as Z+/Z- views in CP-System. All names of Z+/Z- fittings and pipes are used only to distinguish views. In specifications, all Z+/Z- elements are treated as 2D elements.

In specifications, a surface of pipes, ducts and fittings can be calculated. Calculations concerning the surface are based on EN 14239 norm.

A user can also include dimensions or types of fittings or pipes/ducts. Only elements selected from databases of manufacturers have got the type. In other case the type is not specified and main dimensions of elements are included in specifications. These dimensions are compatible with dimensions on drawings in the **list of 2D fittings**. We recommend to print and to attach it to a specification of elements. The list is placed in file *Manual.pdf*. in subfolder *File_gb*.

Cable trays, protective pipes, cable ladders and other elements are available in Electrics module. Some fittings presented in a list of 2D fittings are not used in electrics and therefore they are not available in this module.

Flex pipes or flex protective pipes (from Electrics) are drawn by means of flex pipes and flex fittings. However, flex fittings are treated as flex pipes so in specifications all flex elements are treated as flex pipes with specified length.

Creating specifications of 2D fittings

Pipes, ducts, cable trays

In order to take into account the length and amount of particular segments of pipes/ducts/cable trays in a specification, this specification should be created with **Length** and **Duct pcs** columns.

In order to export only the total length of pipes/ducts/cable trays with specified cross-section, a user should not take into account columns **Number**, **Length** and **Duct pcs** in specification.

There is amount of segments pipes/ducts/cable trays with given length in a column **Duct pcs**.

e.g. 6 segments 1800 mm long

There is the length of elements in mm in a column **Length**.

e.g.. 1800mm long segments

System	Name	Article	Duct pcs	Length	Area	Sum
Supply air	A pressed saddle	D2=160, D=200, L=200, a=90?°		200	0.151	1.00
Supply air	Duct	D=160	1	918	0.461	0.92
Supply air	Duct	D=200	2	1800	2.262	3.60
Supply air	Duct	D=300	1	1675	1.579	1.68
Supply air	Duct	D=300	1	814	0.767	0.81
Supply air	Duct	D=300	1	904	0.852	0.90
Supply air	Duct	D=300	6	1800	10.176	10.80
Supply air	Duct	D=300	1	911	0.859	0.91
Supply air	Duct	D=300	1	1036	0.976	1.04
Supply air	Elbow	D=300, R=450, a=90?°, E,F=50			1.884	2.00
Supply air	Tee-piece	D=300, D3=300, L=400, L3=200, a=90?°		400	0.565	1.00
Supply air	Tee-piece branch	D=300, D2=200, D3=200, L=423, XY2=250, XY3=300,...		423	0.961	1.00

The **Area** column contains the area of an element in **m^2** or total area of all elements exported in a given row of a specification.

e.g.. 15.209 m^2 for ø300 duct

The **Sum** column contains total length of pipes/ducts/cable trays **in meters**. A column **Sum** is used only in specifications with an option **Summing data.** markered

e.g. the total length of ø300 ducts is 16.14 m

System	Name	Article	Area	Sum
Supply air	A pressed saddle	D2=160, D=200, L=200, a=90?°	0.151	1.00
Supply air	Duct	D=160	0.461	0.92
Supply air	Duct	D=200	2.262	3.60
Supply air	Duct	D=300	15.209	16.14
Supply air	Elbow	D=300, R=450, a=90?°, E,F=50	1.884	2.00
Supply air	Tee-piece	D=300, D3=300, L=400, L3=200, a=90?°	0.565	1.00
Supply air	Tee-piece branch	D=300, D2=200, D3=200, L=423, XY2=250, XY3=300, E,F=20	0.961	1.00

Fittings

Type or dimensions of fittings are exported in a column **TYPE**. All dimensions are compatible with dimensionns on drawings in the list of 2D fittings.

The **Area** column can be considered in air conditioning installations. This column contains the area (in m^2) of elements calculated according to EN 14239 norm.

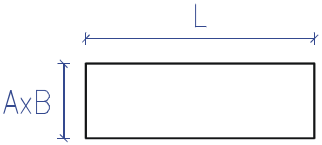
For fittings, the column **Sum** contains total amounts of fittings of a given type.

e.g. there are two 90ºelbows with ø300 diameter and the radius R=450 (1,5D).

Rectangular fittings

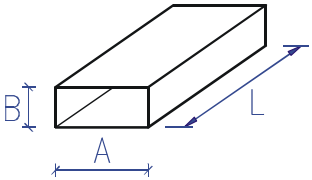
A rectangular duct

- Air conditioning duct
- Cable duct
- U cable tray
- Closed-profile cable tray

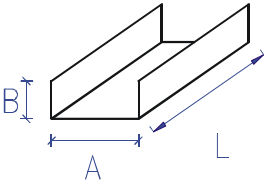


Examples:

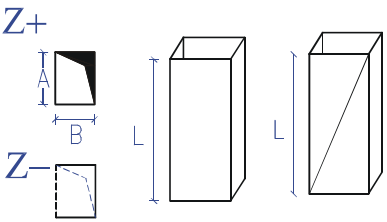
An air duct, cable tray or closed cable tray



U profile cable tray in Electrics

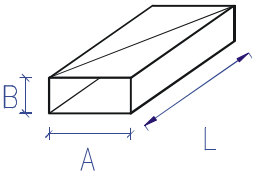


Z+ or Z- view (vertical pipe/duct).
Duct-Z+/Z is exported as 2D pipe/duct or rectangular cable tray in specifications.



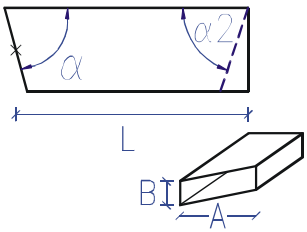
A rectangular duct (fit)

The length of fitting element is exported with 10% extra length in specifications.

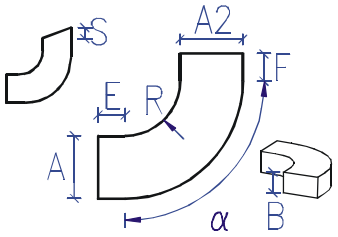


A truncated rectangular duct

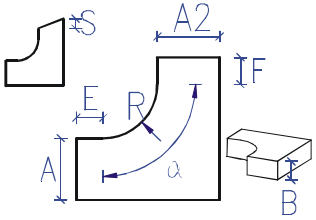
A truncated duct (on one or both ends) is only used in drawings during „attaching" to other ducts or objects at specified angle. In specifications, the truncated duct is exported as 2D duct which a total length (including cuts).



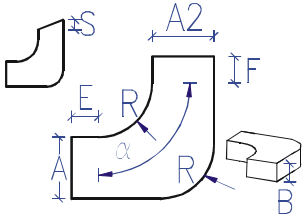
A rectangular bend
For truncated bends S dimension is greater than 0. S dimension is not exported for 2D bends.



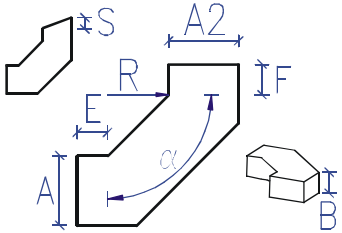
A rectangular bend TYPE2



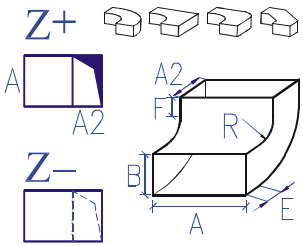
A rectangular bend TYPE3



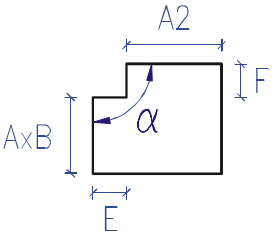
A rectangular bend TYPE4



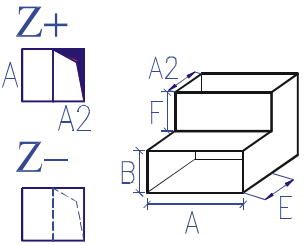
A rectangular bend
Z+ or Z- view (directed up or down).
Z+/Z- bend is exported as 2D rectangular bend in specifications.
Presented view is also used for Z+/Z- TYPE2, TYPE3 and TYPE4 bends



An elbow

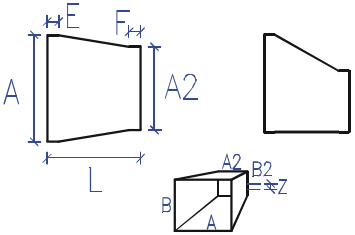


An elbow Z+/Z-

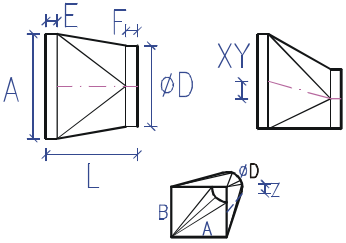


A rectangular reducer

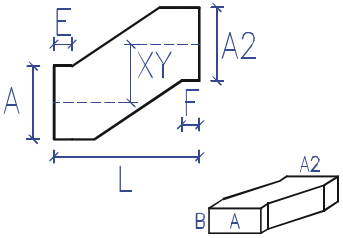
- A symmetric reducer
- An asymmetric reducer
- An asymmetric reducer with displacement by Z axis (Z>0 in specifications)



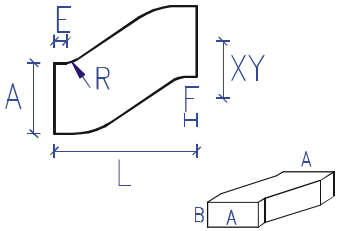
A rectangular-round reducer



A fuller

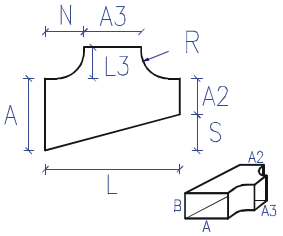


A fuller (bend)

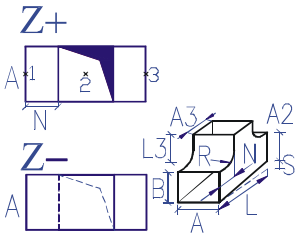


A view for fullers Z+/Z-
Symbol '+' on higher side

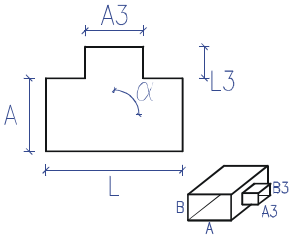
A tee-piece



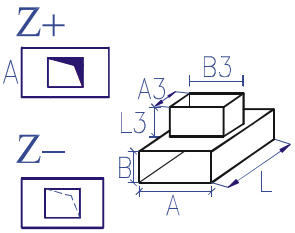
A tee-piece Z+/Z-



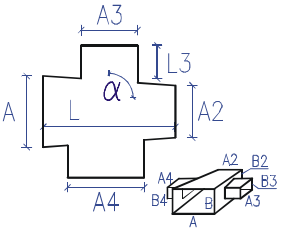
A tee-piece (simple)



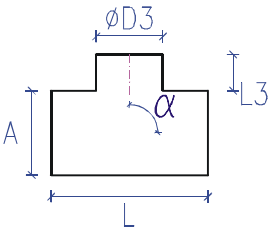
A tee-piece (simple) Z+/Z-



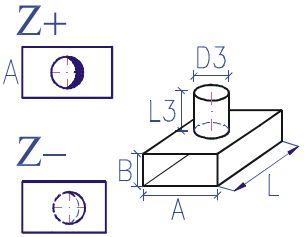
A reducing tee-piece



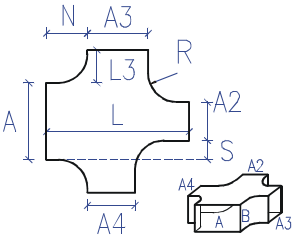
A rectangular-round tee-piece

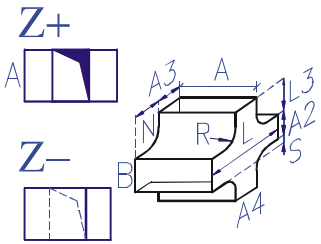
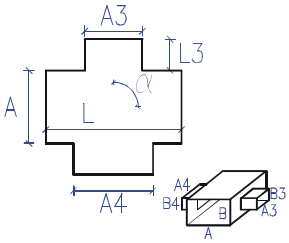
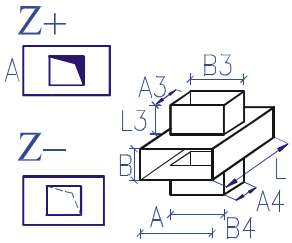
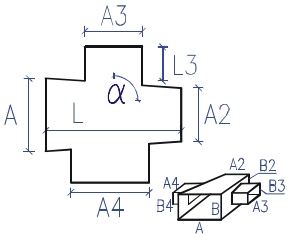
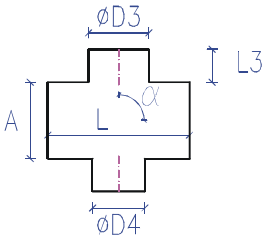


A rectangular-round tee-piece Z+/Z-

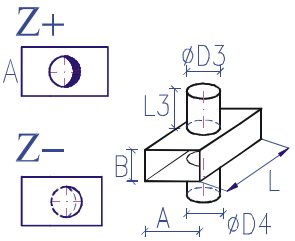


A cross

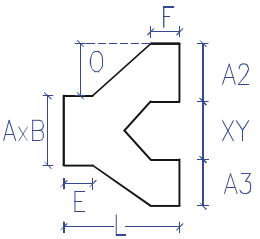


A cross Z+/Z-	
A cross (simple)	
A cross (simple) Z+Z-	
A reducing cross	
A rectangular-round cross	

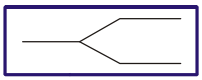
A rectangular-round cross Z+/Z-



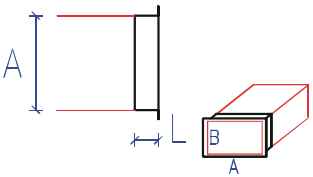
A tee-piece branch



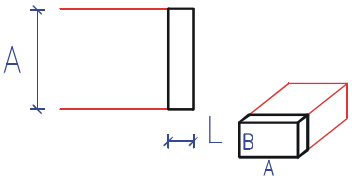
A view for tee-pieces branch Z+/Z-



A cap external

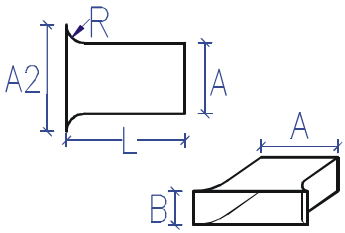


A cap internal



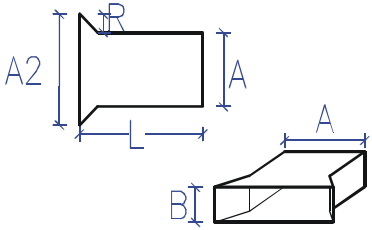
A coupling

- Filleting with zero radius
- Filleting with nonzero radius (on one or both sides)



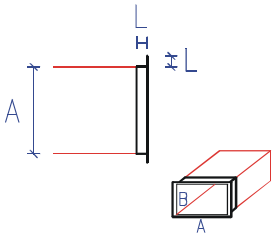
A coupling TYPE2

- Chamfer distance zero
- Chamfer distance nonzero (on one or both sides)



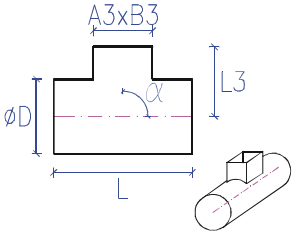
A flange

During inserting flanges a user can turn off considering them in specifications



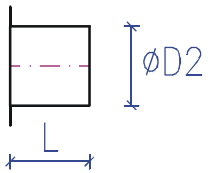
A round-rectangular tee-piece

This element is available in a list of round fittings.



A round coupling

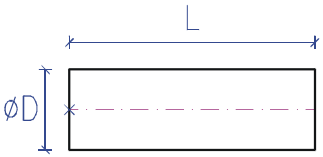
This element is available in a list of round fittings.
A round coupler is used to join round pipes to rectangle devices or pipes.



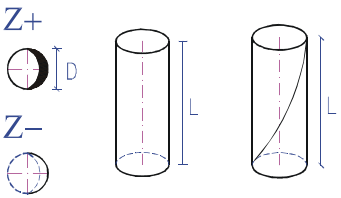
Round fittings

A round pipe

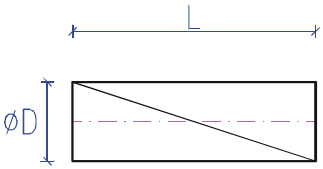
- Round air conditioning duct
- Pipe
- Protective pipe (in Electrics)



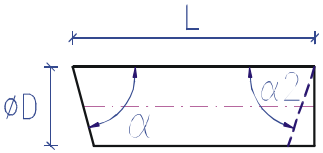
View Z+ or Z-
(rectangular pipe/duct).
Duct Z+/Z- is exported as 2D pipe/duct in specifications.



A round duct (fit)
The length of fitting element is exported with 10% extra length in specifications.

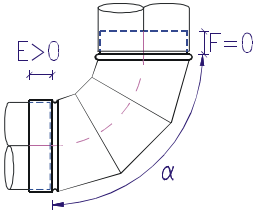


A round truncated duct
A truncated duct (on one or both ends) is only used in drawings during „attaching" to other ducts or objects at specified angle. In specifications, the truncated duct is exported as 2D duct which a total length (including cuts).

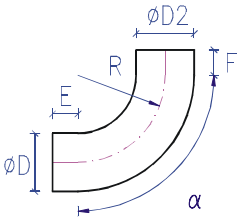


Endings of fittings
Round fittings of air conditioning installations have “nipple” endings on which pipes can be put (round ducts).
A user can also select „muff” ending that enables a user to put directly a fitting on other fitting.
Notes about E and F dimsensions:
For all fittings when E>0 and/or F>0 do „muff” ending.
If E=0 and/or F=0 do standard long “nipple” ending.

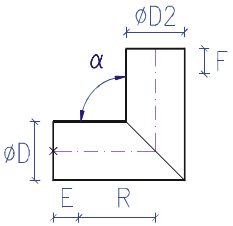
Real fitting with “nipple” ending (F=0) and “muff” ending (E>0)



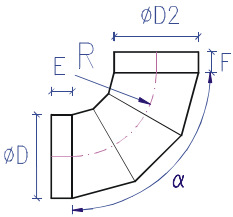
A round elbow



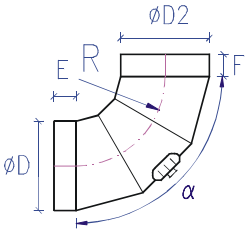
A two-segment elbow



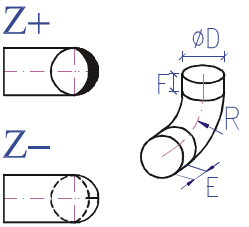
A segmented elbow



A segmented elbow with revisory door

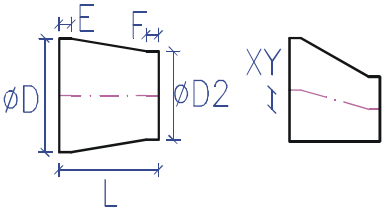


An elbow
Z+ or Z- view (directed up or down).
Z+/Z- elbow is exported as 2D round elbow in specifications.
Presented view is also used for all types of round elbows (segmented etc.).

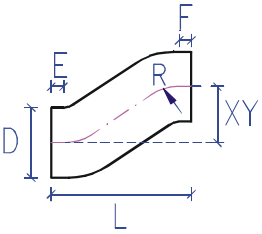


A reducer

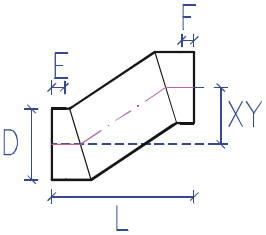
- A symmetric reducer
- An asymmetric reducer
- An asymmetric reducer with a shift on Z axis (Z.0 in specifications)



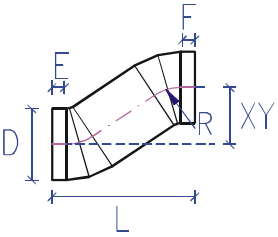
A fuller



A fuller TYPE2 (three segmented)

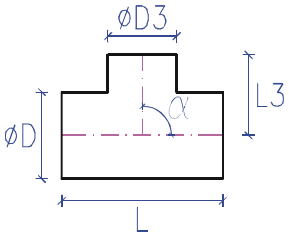


A segmented fuller

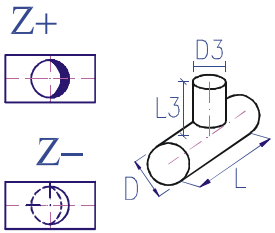


View for fullers Z+/Z-
Symbol '+' on higher side.

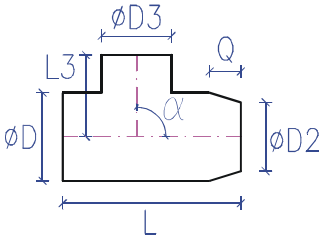
A tee-piece



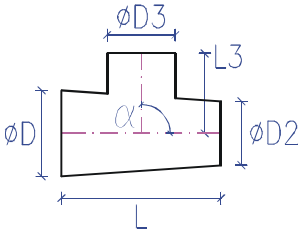
A tee-piece Z+/Z-

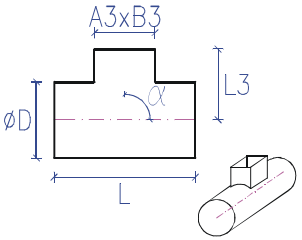
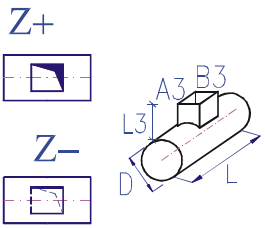
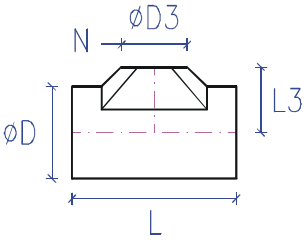
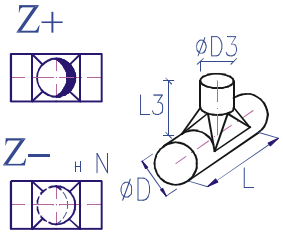
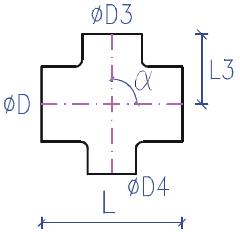


A tee-piece with reducer

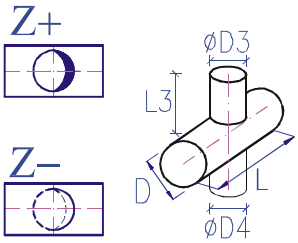


A reducing tee-piece

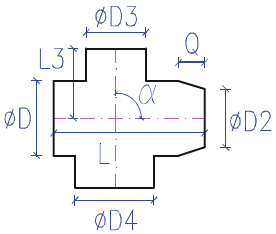


A round-rectangular tee-piece	
A round-rectangular tee-piece Z+/Z-	
A taper tee-piece	
A taper tee-piece Z+/Z-	
A cross	

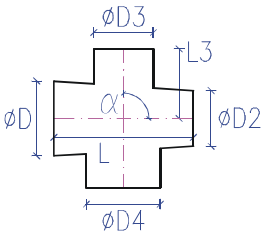
A cross Z+/Z-



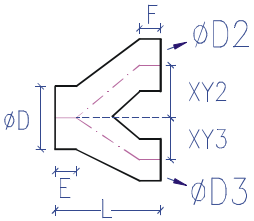
A cross with reducer



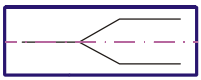
A reducing cross



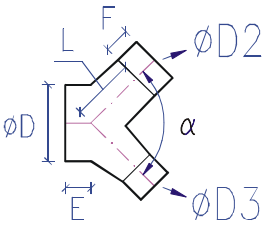
A tee-piece branch



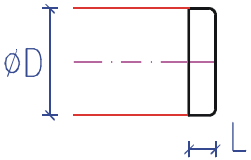
A view for tee-piece branch Z+/Z-



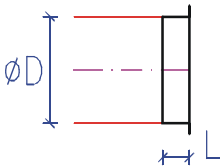
A tee-piece branch (oblique)



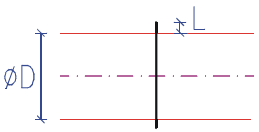
An internal cap



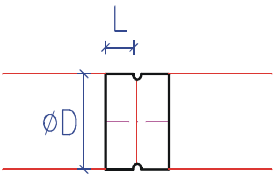
An external cap



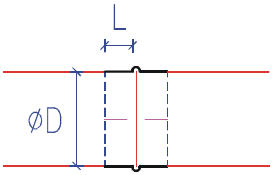
A flange
During inserting flanges a user can turn off considering them in specifications.



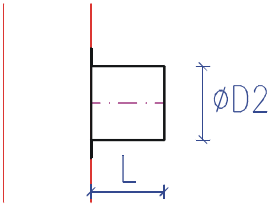
A muff



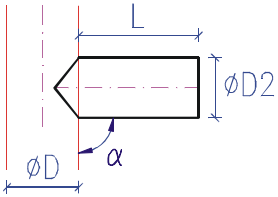
A nipple



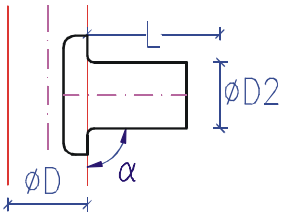
A round coupling
A round coupler is used to join round pipes to rectangle devices or pipes



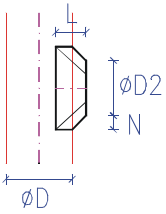
A formed saddle



A pressed saddle



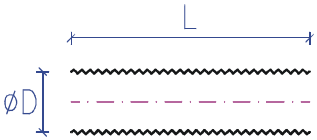
A taper saddle



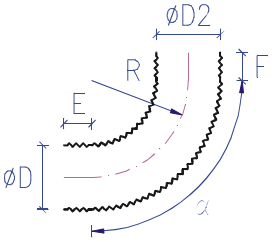
Flex pipes

A flex pipe

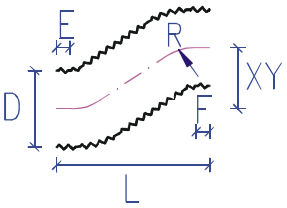
- Flex air conditioning pipe
- Protective pipe (in Electrics)



A flex elbow
Flex elbow is exported as a flex pipe with specified length in specifications

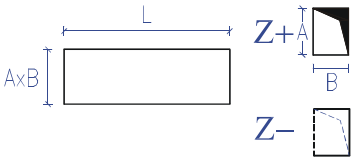


A flex fuller
Flex fuller is exported as a flex pipe with specified length in specifications

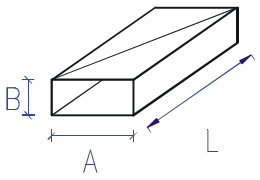


Rectangular fittings – an annex to specifications made in CP-System

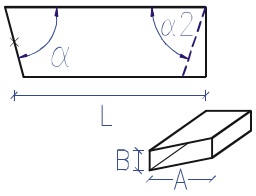
A rectangular pipe



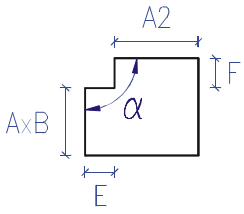
A rectangular duct (fit)



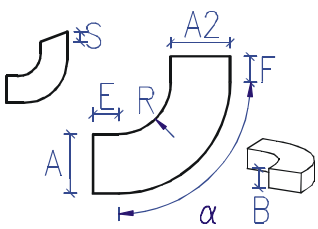
A rectangular truncated duct



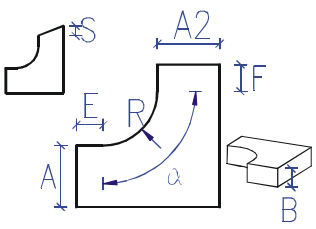
An elbow



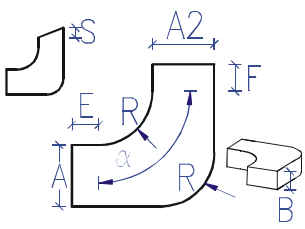
A rectangular bend



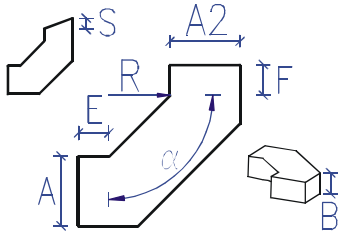
A rectangular bend TYPE2



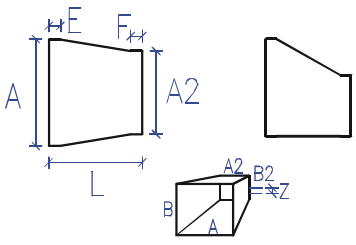
A rectangular bend TYPE3



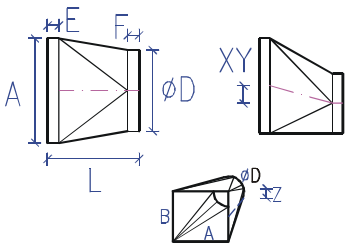
A rectangular bend TYPE4



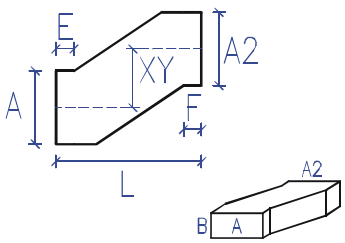
A rectangular reducer



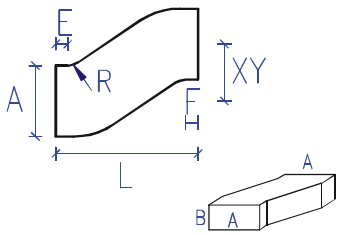
A rectangular-round reducer



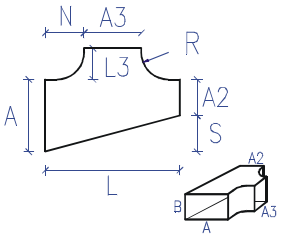
A fuller



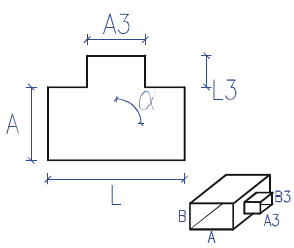
A fuller (bend)



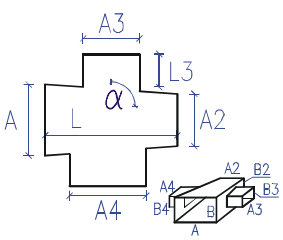
A tee-piece



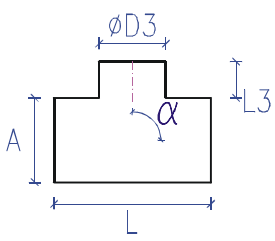
A tee-piece (simple)



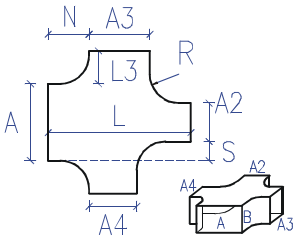
A reducing tee-piece



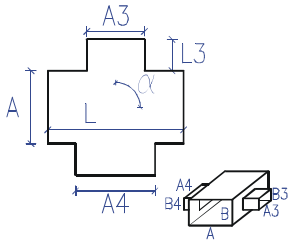
A rectangular-round tee-piece



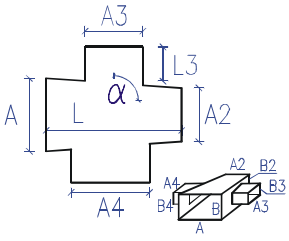
A cross



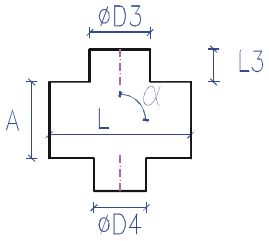
A cross (simple)



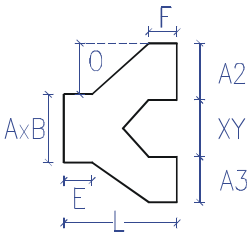
A reducing cross



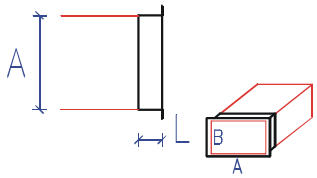
A rectangular-round cross



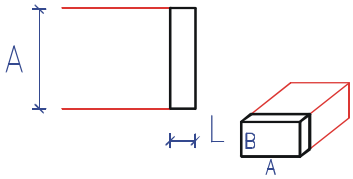
A tee-piece branch



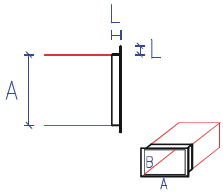
An internal cap



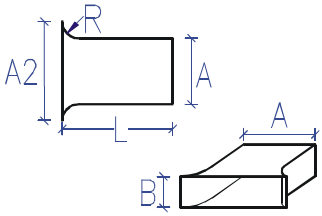
An external cap



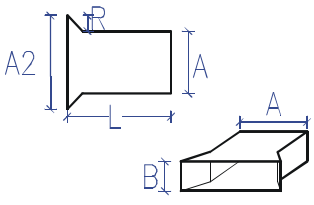
A flange



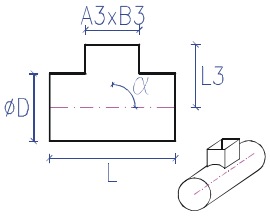
A rectangular coupling



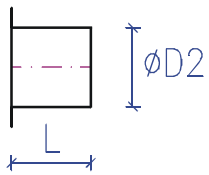
A rectangular coupling TYPE2



A round-rectangular tee-piece

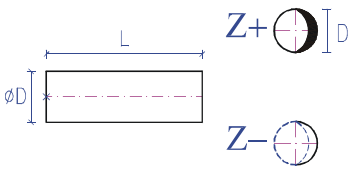


A round coupling

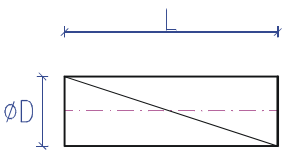


Round fittings – an annex to specifications made in CP-System

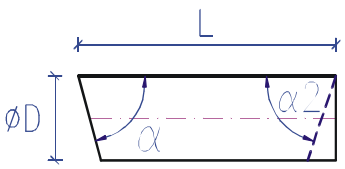
A round pipe



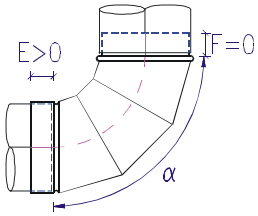
A round duct (fit)



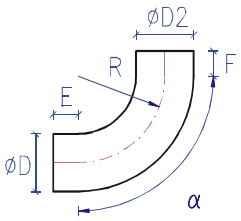
A truncated round duct



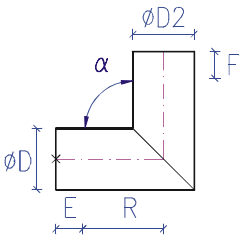
A real fitting with a 'nipple' ending (F=0) and a 'muff' ending (E>0)



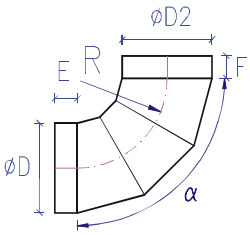
An elbow



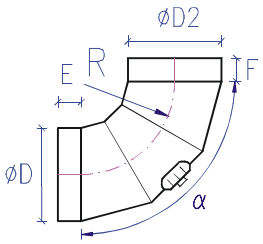
A two-segment elbow



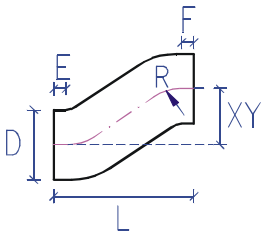
A segmented elbow



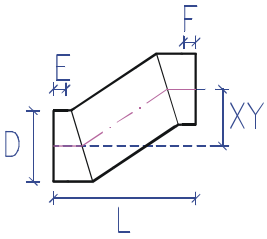
An elbow with revisory door



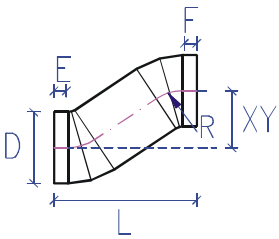
A fuller



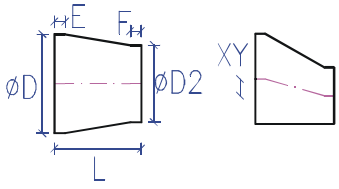
A fuller TYPE2 (three-segmented)



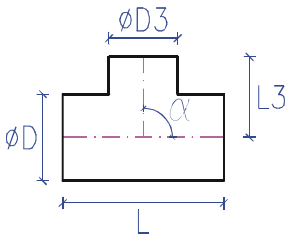
A segmented fuller



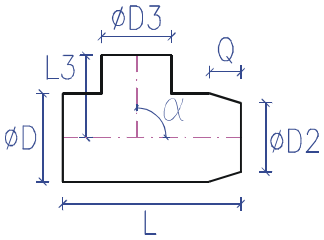
A reducer



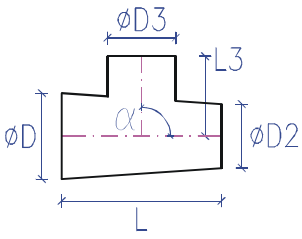
A tee-piece



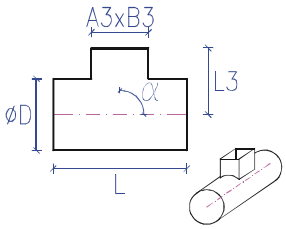
A tee-piece with reducer



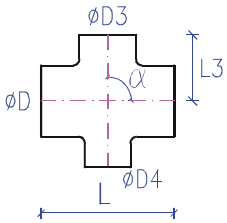
A reducing tee-piece



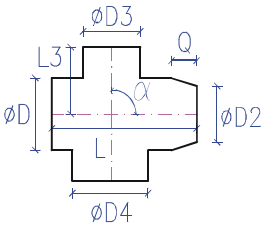
A round-rectangular tee-piece



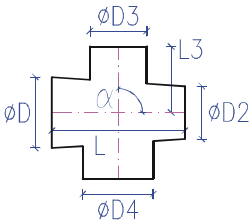
A cross



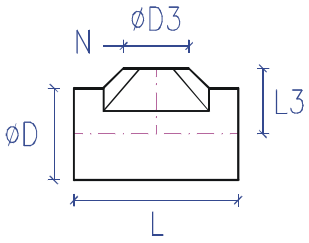
A cross with reducer



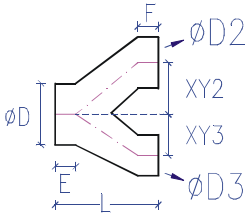
A reducing cross



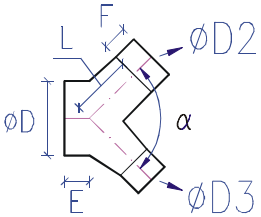
A taper tee-piece



A tee-piece branch

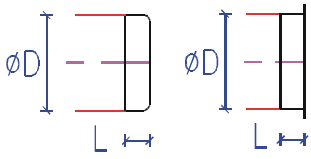


A tee-piece branch (oblique)

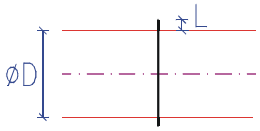


A round cap

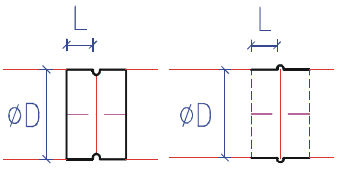
internal **external**



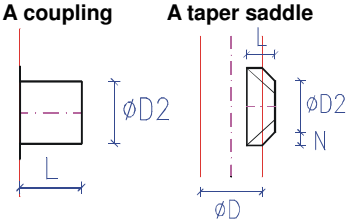
A round flange



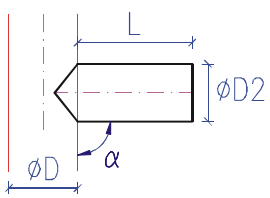
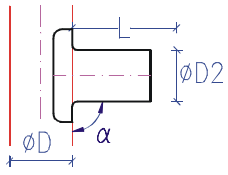
A muff **A nipple**



A coupling **A taper saddle**



A formed saddle



A pressed saddle