

USER GUIDE

nanoCAD

Version 3

Nanosoft
2011

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| Node note | Error! Bookmark not defined. |
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To create a layout from template Error! Bookmark not defined.

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General information

Introduction

nanoCAD – is a universal editor, graphic platform for CAD in different fields.

A friendly and usable traditional interface and the capability to use traditional working methods makes nanoCAD a universal tool, almost every designer can work with it.

nanoCAD is used for the creation and editing of vector primitives (graphic elements) and more complex objects (blocks, dimensions and etc.). You can insert and use a raster image as file backing,. nanoCAD contains built-in display tools to view different 3D geometry from *.dwg files. nanoCAD has the functionality to create 3D frame and Polyface Mesh models. The user can design working documentation according to the drawings and projects of 3D models created in different 3D CADs.

nanoCAD can be used by an individual user and also by project teams, which use the functionality of external references and of integration to the engineering document workflow, including PDM/PLM systems.

nanoCAD allows the user to:

- Create and edit different 2D and 3D vector primitives, texts, objects of drawing preparation, settings of graphic display and printing of technical documentation.
- Create and use any types of tables and specify drawing elements by block attributes and objects of drawing preparation.
- Adjust the working environment to prepare working documentation to different standards.
- Work in 3D model space and 2D paper space using viewports.
- Display, create and edit 3D meshes. Create a User Coordinate System to edit geometric snap-to 3D objects.
- Cooperate and collaborate with your colleagues who create drawings in other popular CADs by using the common file format - *.dwg.
- Use any technical documentation created previously and stored in raster file format (scanned drawings, texts, tables, photos).
- Print technical documents using any installed plot devices in the operational system.

nanoCAD uses Teigha core, consisting of set of Teigha program libraries, developed by Open Design Alliance (ODA) international consortium. They allow reading and writing to *.dwg format files used by many CADs. Teigha program libraries support all current versions of the *.dwg format.

nanoCAD supports the import and export of vector data of *.dxf format. Using the *.dwg file format allows integrating solutions on the nanoCAD base with almost any CAD. Structurally, the program consists of functionally linked parts:

- **Universal vector editor** – has the same functionality as most popular CAD packages have.
- **Universal table editor** – has the technology to translate the graphic, attribute data of objects into table form, with the ability to perform any mathematical calculations in tables; has the tools for the import/export of table data.
- **Plot master** – creates and edits plot settings from model space and paper space.
- **Functionality of developer tools** – available only for registered developers.

System requirements

| | |
|----------------------|--|
| Operating system | Microsoft® Windows® 7 Enterprise, Ultimate, Professional or Home Premium edition. Microsoft Windows Vista (SP1 or later) Enterprise, Business, Ultimate or Home Premium edition. Microsoft Windows XP Professional or Home edition (SP2 or later). Microsoft Windows 2000 (SP4 or later). |
| Processor | Intel Pentium 4/AMD Athlon or higher. |
| RAM | From 512 Mb. 2 Gb is recommended if you have big projects. |
| Hard disk free space | About 300 Mb is required for full program installation. About 1-3Gb is required for working (according to the difficulty of projects). |
| Monitor | Required resolution: 1024×768. Recommended resolution: 1280×1024 or higher. |
| Graphics | Video adapter with OpenGL-compatible hardware 3D acceleration. |
| Additional hardware | DVD-ROM (if the program is installed from the corresponding device). Internet access (If the program is installed on-line). Mouse or other devices. |

| | |
|---------------------|--------------------|
| Additional software | MS Excel, MS Word. |
|---------------------|--------------------|

Program installation

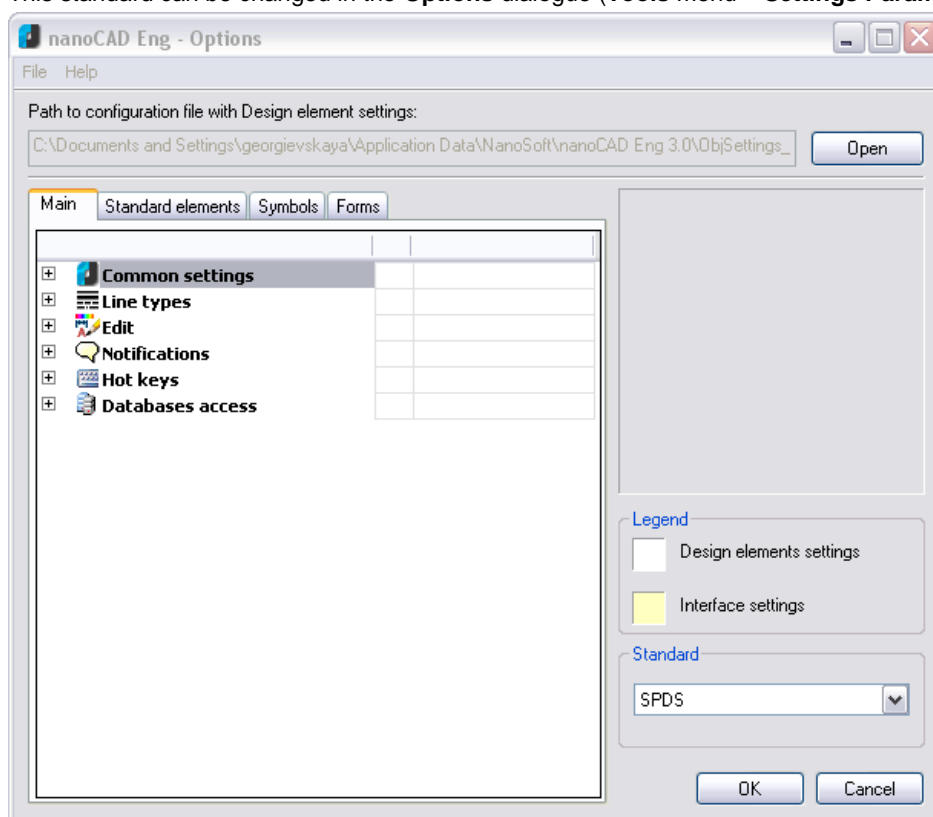
You must have administrator rights to install the program.

You do not need to be an administrator to work with the program; it can be launched by a user with reduced authority.

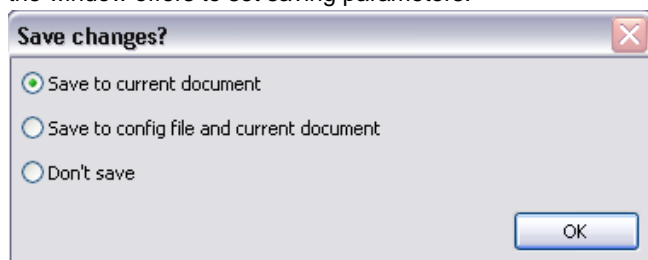
As nanoCAD is a universal graphic platform, during the first launch of the program you have to make a choice which standard (SPDS or ESKD) you want to use:



This standard can be changed in the **Options** dialogue (**Tools** menu – **Settings Parameters**).



You can change this standard for one document or make it the default for all documents. After selecting a standard, the window offers to set saving parameters:



Launch of nanoCAD

To launch the program:

- Click twice on the nanoCAD icon on the Windows desktop or
- On the taskbar select **Start > All programs > Nanosoft > nanoCAD > nanoCAD**.

Help



Menu: **Help** –  **About...**



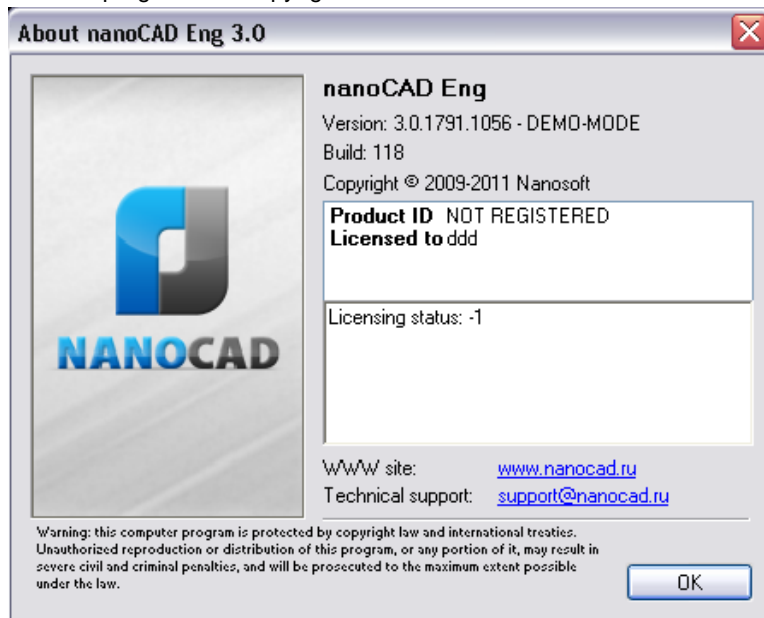
Toolbar: **Main** – 



Command line: **HELP, HELPCONTENTS**

There are buttons in the dialogue boxes to open the Help file.

The **About** command opens the **About nanoCAD 3.0** dialogue with information about the version and build of the installed program and copyright:




. The Help dialogue box shows the licence number, who registered the program, a link to the developer's website – www.nanocad.ru and an email link for technical support – support@nanocad.ru

The **Help** menu also contains links to the nanoCAD support forum, learning videos on YouTube and the official site of Nanosoft for additional information.

If you have an Internet connection, you can load the links from nanoCAD.

Exit from nanoCAD



Menu: **File** –  **Exit**



Hotkeys: **CTRL+Q**

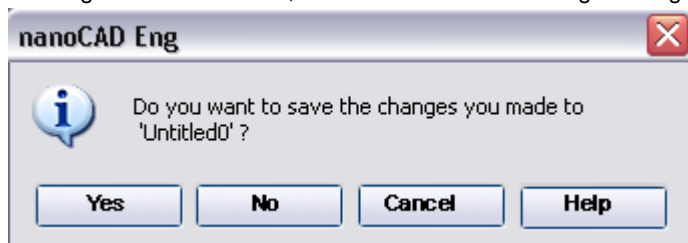


Command line: **EXIT, QUIT**

nanoCAD can be closed by clicking on the  icon in the top right corner of the program.

If all changes in opened documents were saved, no additional messages are shown.

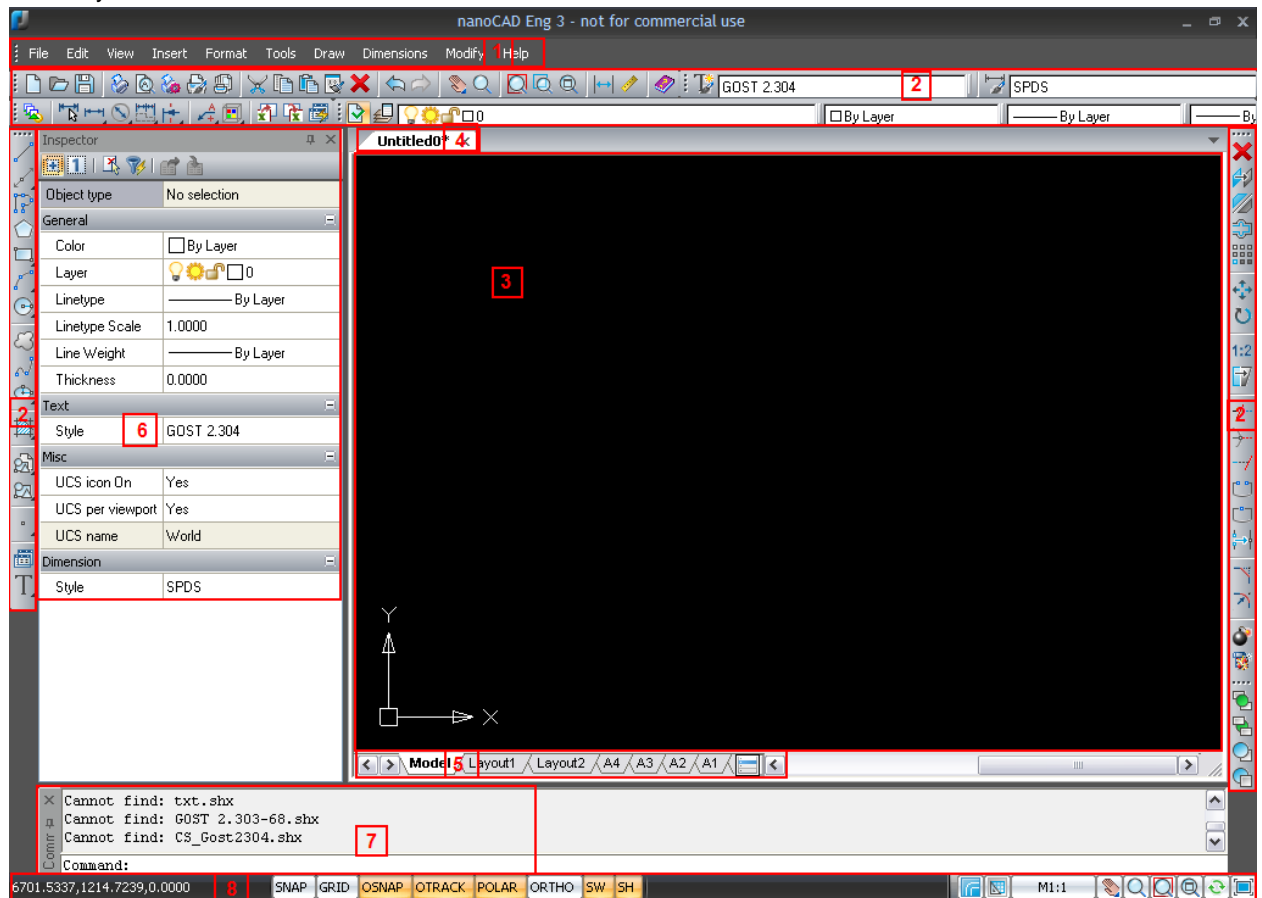
If changes were not saved, nanoCAD shows a warning message:



where a user can save changes, refuse to save changes or select the **Cancel** button and continue to work in the program.

nanoCAD user interface

With nanoCAD you can organise user work space according to the user's needs and tasks. The standard set of tools is usually used:



The nanoCAD window consists of the following interface elements : menu bar, toolbars, document windows, Properties window, command line and status bar. Most interface elements can be moved to other places; they can be fixed or floating. Some elements can be auto hidden.

Main menu

The Menu bar (1) is placed at the top of the nanoCAD window and consists of drop-down menus containing all the basic commands.

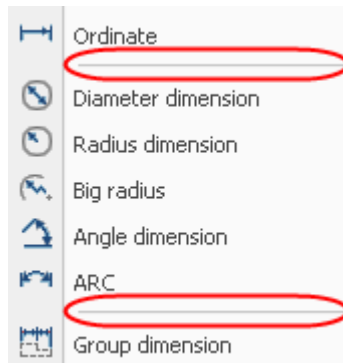
Commands from the drop-down menus are grouped by functionality:

| | Description |
|------------------------|---|
| File menu | Commands to work with files and documents: create, open, save, export and import files, print and print settings. There are also some useful utilities: audit, recover and purge. |
| Edit menu | Undo and redo commands, exchange buffer commands, select commands, find and replace command. |
| View menu | Zoom commands, creation of named views and viewports, views and visual styles, display of scroll bars, status bar and toolbars. |
| Insert menu | Insert blocks and external references, also insert rasters, commands to work with layouts (create, save, delete and rename). |
| Format menu | Commands to work with layers, line types, text and dimension styles, point styles, units and drawing limits. |
| Tools menu | Display order commands, edit blocks and external references, drafting settings and program options. |
| Draw menu | Commands to draw objects. |
| Dimensions menu | Commands to set dimensions and manage dimension styles. |
| Modify menu | Commands to edit drawing objects. |

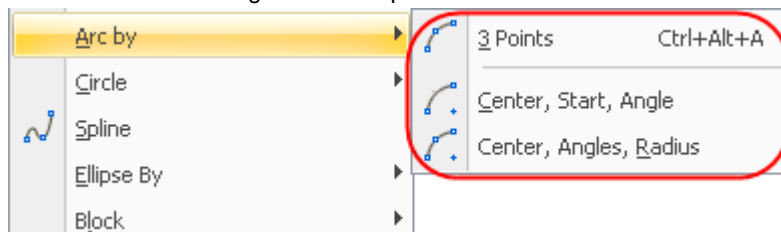
| | |
|------------------|------------------------|
| Help menu | Help and useful links. |
|------------------|------------------------|

The graphic interface of the drop-down menus contains set of symbols to make work with the menus easier:

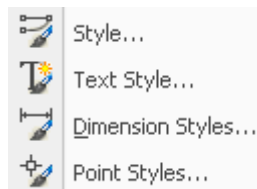
- commands from each functional group are separated by lines in the drop-down menu:



- a small black triangle in the left part of the menu means that selection of the item opens a cascade menu:



- three dots at the end of the menu item means that the item opens a dialogue box:

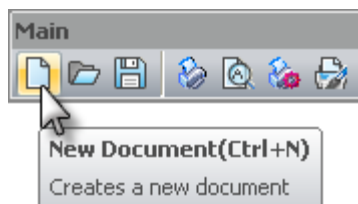


- there are hotkeys to the right of the most used commands:



Toolbars

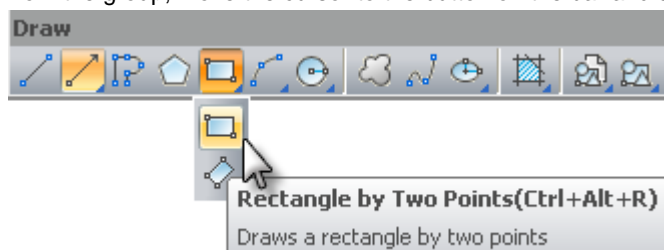
Buttons on the toolbars (2) are used to start the commands. When you move the cursor over the button the tooltip is shown:



On some toolbars the commands are grouped and you can see only one button of command from each group.

The buttons from the command groups are marked with small arrows in the bottom right corner: .

When clicking such a button, a toolbar opens showing the tools of the specified group. To select the required tool from the group, move the cursor to the button on the bar and select it.



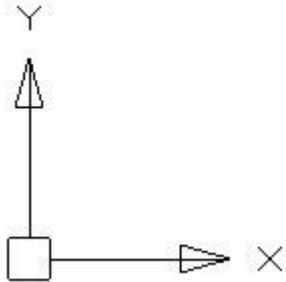
Show or hide toolbars by selecting **View – Toolbars > nanoCAD** toolbar; or from the context menu of any toolbar. Toolbars can be moved to any part of the program.

A toolbar in the document area is called floating. You can change its shape and location. To make it fixed, move it out of the document area.

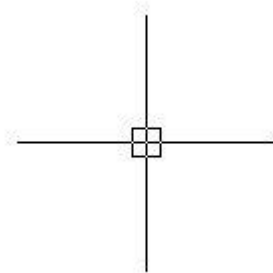
nanoCAD document window

There are documents in the nanoCAD work space (3). If several documents are opened in the program, use the tabs (4) to select the required document.

User coordinate system icon (UCS) is placed in the default coordinates: X=0; Y=0; Z=0 (in the bottom left corner). To show/hide the icon, select the **View menu – Display > UCS icon**.





Cursor – the main instrument for selection and specifying in the graphic area. The cursor has a crosshair shape with a square sight at the intersection point:



The shape and size of the cursor can be adjusted in the **Cursor** section of the **Options** dialog (**Tools** menu – **Options**).

Note: To change the cursor's colour, deselect the **Use default color** box in the **Category Name and Color** dialogue box.

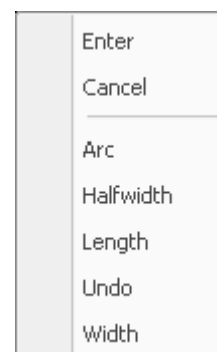
Outside the graphic area, the cursor has an arrow shape: .

The document's tabs (5) in the bottom part of the window are used to switch between layouts and to switch from model space to paper space and back. It can be more useful to use the  button located at the end of tabs row. The button allows switching between layouts and named views in the document. Scroll bars in the bottom and left parts of the graphic area are additional tools for panning. The **Scroll bars** command from the **View** menu shows/hides the vertical scroll bar. The horizontal scroll bar is always shown.

Context menu

In nanoCAD, as in many other Windows applications, the context menu appears when you click the right button. The content of the context menu depends on the current context – object type, location of the cursor and what command is being run at the moment.

Examples of the context menu



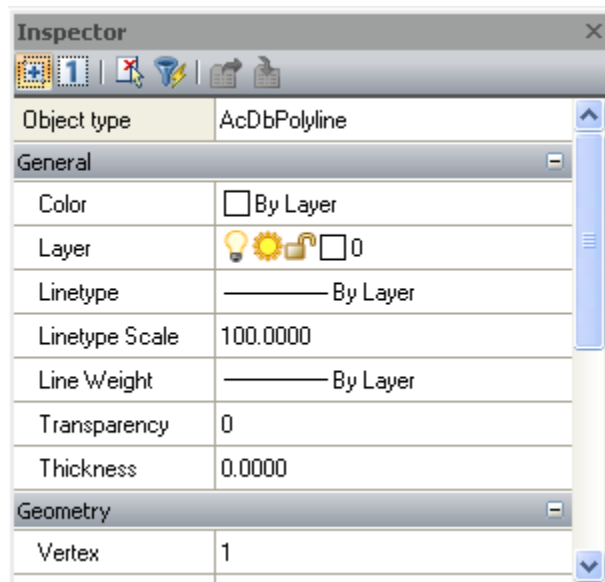
| |
|------------|
| Enter |
| Cancel |
| Area |
| Dimensions |
| Rotation |

Note: Right button clicking with **SHIFT** or **CTRL** pressed during the execution of a command for creating or editing objects opens the context menu of the **Object snap**:

| |
|---------------|
| Endpoint |
| Midpoint |
| Center |
| Node |
| Quadrant |
| Intersection |
| Extension |
| Insertion |
| Perpendicular |
| Tangent |
| Nearest |
| Parallel |
| None |

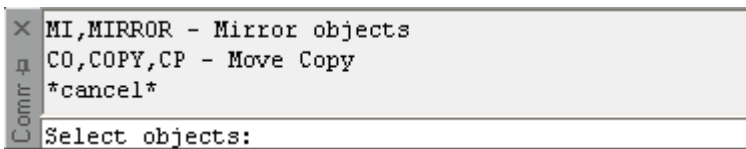
Inspector window

The **Inspector** window (6) is used to display information about selected objects, to change objects' properties, to set selection mode and to launch selection commands.



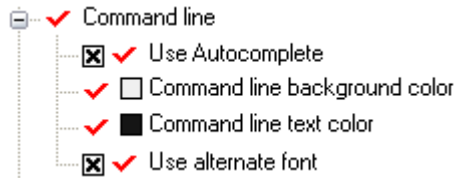
Command line

The command line (7) is used to enter commands from the keyboard, to show tooltips and nanoCAD messages, to select the options of the selected command:



The default command line height is set to display 5 lines of command history. To move from one line to another, use the scroll bar in the right part of the command line window. The height of the command line can be changed by dragging the top edge and dropping it in the required place.

In the Command line section of the **Options** dialog you can change the text and background colour of the command line, switch on/off **Use Autocomplete** and **Use alternate font** options:



If the command line was closed it can be opened by clicking in the document window.

Input of commands, aliases and shortcuts

To enter a command from the keyboard, type the full command name in the command line and press **ENTER** or **SPACEBAR** to execute it.

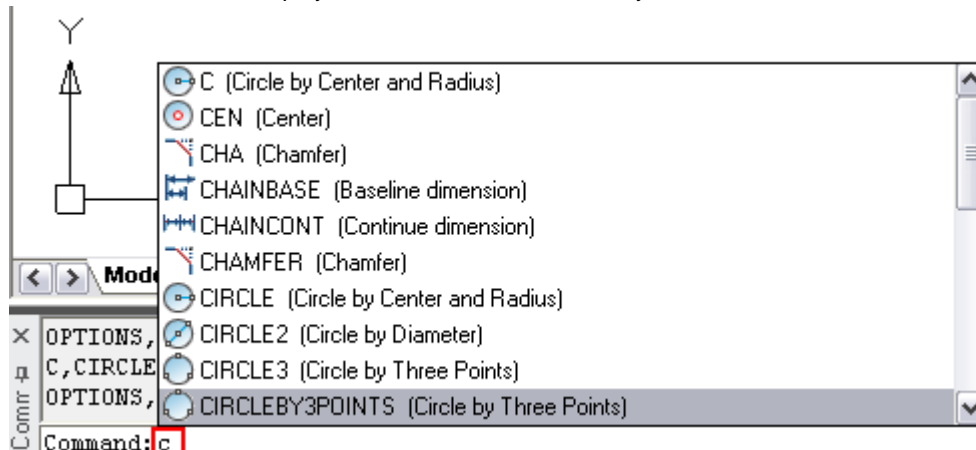
Instead of the full name of the command you can use a command abbreviation, the so-called alias of command. For example, instead of typing CIRCLE, you can just type C.

Several aliases can be assigned to one command. However, a particular alias can only be assigned to one command. Aliases are specified in *nCAD.pgp* file.

Shortcuts are similar to aliases, but launch a set of commands grouped by a certain criterion.

Note: It does not matter what case is used when typing in the command line.

With the **Use Autocomplete** mode (*Command line* section of the **Options** dialogue box) you can choose a command from the list displayed in the command line after you have entered several letters:

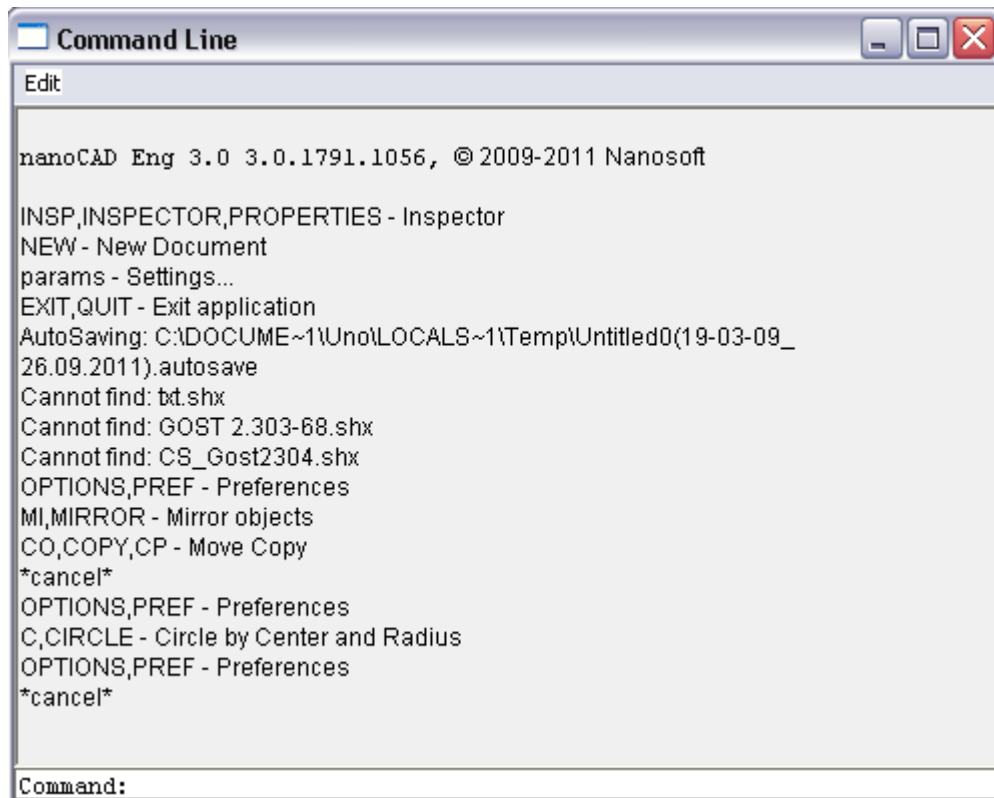


To select a command from the list use **TAB**, **ARROW DOWN** and **ARROW UP** buttons and the **ENTER** button to confirm a selection.

You can see a full list of registered commands, aliases and shortcuts in the command line by entering an apostrophe.

Text window

To display the command history, press F2 to open the text window:



You can close and open the window at any time.

The window is opened at the same location and size as set previously.

Command Line prompt

The default command line prompt looks like:

Command:

but it varies depending on the current command or program state.

The command prompt can include descriptive messages that are displayed in the status bar when the command is running. For example, the prompt during a creation of a circle is:

Specify center point for circle or [**3P**/**2P**/**TTR**]:

It can also include interactive keywords in square brackets separated by a slash, highlighted in a blue colour and underlined.

Note: To prevent confusion with hyperlinks, interactive key words in prompts are shown in a red colour and bold font.

Specify center point for circle or [**3P**/**2P**/**TTR**]:

You can type any keyword manually or just click on it. A keyword specified in triangular brackets is a keyword “by default”. For example:

Enter number of sides <4>:

or

Extend <Inscribed> or [**Inscribed**/**Circumscribed**]:

To choose it press the **ENTER** key.

As an answer to the prompt, type the keyword, or you can just type the capital letter of the word:

Specify next point or [**Arc**/**Close**/**Halfwidth**/**Length**/**Undo**/**Width**]:

For example, to select the **Close** keyword, type **C**.

Note: The capital letter can be in the middle of the key word.

Format of input data

You can enter different formats of data, text, digital values, coordinates, in the command line.

To use coordinate values to specify a point, enter an X value and a Y value separated by a comma <X,Y>:

23.45,-6.98

If the angle units are degrees/minutes/seconds, the following syntax is used:

32d20m10s or 32-20-10

Mathematical processor

The command line allows for the input not only of commands, numbers and points, but also mathematical expressions. It is convenient when the coordinate or number is not known beforehand but should be calculated from some conditions.

For example, instead of calculating the coordinate of a point using the $\text{SQRT}(349.56-275.90)*2$ formula and only then typing it in the command line, like here:

Command: L,LINE - Line by points

First point: -12.45,17.17

You can use a mathematical expression instead of coordinates in the command line:

Command: L,LINE - Line by points

First point: -12.45,SQRT(349.56-275.90)*2

To calculate a mathematical expression at any time type the ? sign before this expression. In the following example, a multiplication operation is performed during the line creation process.

Command: ?25.7*5

After pressing ENTER the result is shown in the command line:

?25.7*5 = 128.5

Mathematical expressions can be calculated during the execution of some commands. The following example shows a multiplication expression calculation during drawing a line:

Command: L,LINE - Line by points

First point: 25.7,41.32

Next point: 25.7*5,93.77

After you press ENTER, the line with first point coordinates (25.7, 41.32) and end point coordinates (128.5, 93.77) will be drawn.

The complete list of registered operations, functions and constants can be viewed in the **Command Line** window by entering the ? sign.

The Mathematical Processor supports the following arithmetical operations:

| | |
|------------------------|---|
| Constants: | |
| E | The constant <i>e</i> , the base of natural logarithms. |
| PI | The constant <i>pi</i> . It represents the ratio of the circumference of a circle to its diameter. |
| Arithmetic operations: | |
| () | Groups expressions. |
| + - | Adds, subtracts. |
| * / | Multiplies, divides. |
| % | Remainder. |
| ** | Power. |
| Functions: | |
| COS | Returns the cosine of a number. |
| SIN | Returns the sine of a number. |
| TAN | Returns the tangent of a number. |
| ACOS | Returns the arccosine of a number. |
| ASIN | Returns the arcsine of a number. |
| ATAN | Returns the arctangent of a number. |
| ABS | Returns the absolute value of a number. |
| EXP | Returns <i>e</i> (the base of natural logarithms) raised to a power. |
| LN | Returns the natural logarithm of a number. |
| LOG | Returns the base-10 logarithm of a number. |
| SQRT | Returns the square root of a number. The number must be non-negative. |
| FLOOR | Returns the largest integer less than or equal to the given numeric expression |
| CEIL | Returns the smallest integer greater than or equal to the given numeric expression. |
| FRAC | Returns a locale-specific count of the number of digits to display to the right of any decimal point. |

| | |
|----------------------------|--|
| TRUNC | Returns the integral digits of the specified number. Any fractional digits are discarded. |
| ROUND (x,precision) | Returns a number rounded to a specified number of decimal places (precision). Precision value indicating how many places to the right of the decimal are included in the rounding. |
| NEG | Returns the negative value of a number. |
| SGN | Returns an integer indicating the sign of a number: 1 - Greater than zero; 0 - Equal to zero; -1 - Less than zero. |

The user defined variables can be used in calculation expressions. The variable is created using the following syntax:

?<variable>=<value>

For example:

?base=100

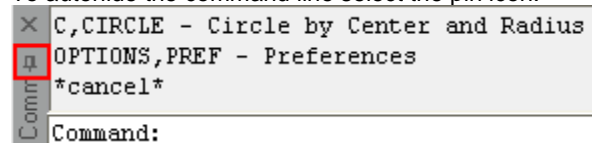
Command: L,LINE - Line by points

First point: base+11.56,base-5

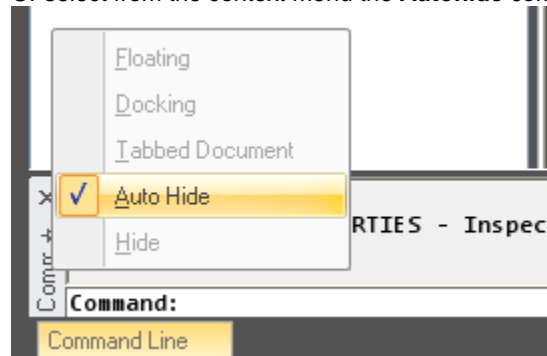
Auto hide mode in the command line

Now the command line can be placed in autohide mode. In this mode the command line is reduced to a tab. In the tab name you can always see the command line's message and entered values.

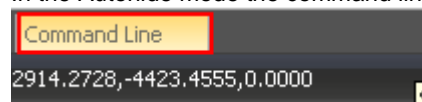
To autohide the command line select the pin icon:



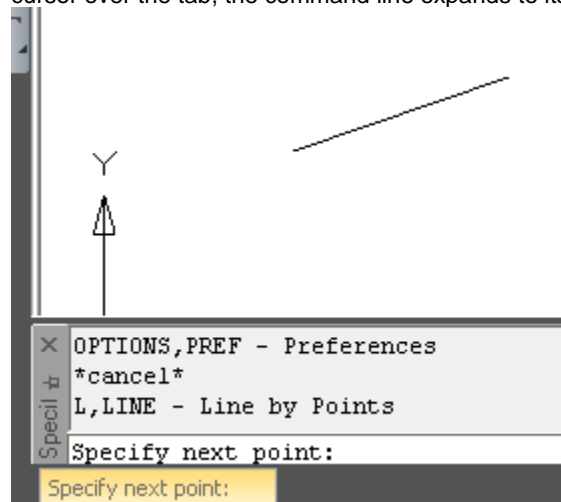
Or select from the context menu the **Autohide** command:



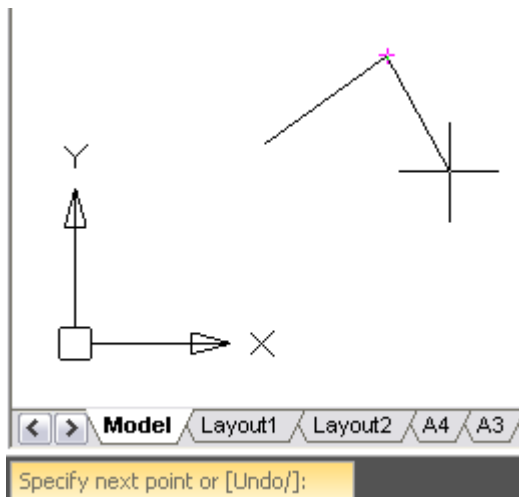
In the Autohide mode the command line folds into a tab.



The entry of commands and required values is performed without showing the command line. When moving the cursor over the tab, the command line expands to its full size, and when moving the cursor away it folds again.



The command line is hidden when working with a drawing, but you can see the current messages of the command line. Command and values entry is performed without expanding the command line.




Working with commands

Commands are used to create and edit objects, open dialogue windows and start any other operations. Commands can be launched:

- from the main menu;
- on the toolbars;
- from the context menu;
- in the command line;
- using hotkeys;

For example, to open the **Options** dialogue box:

- from the main menu: **Tools – Options**;
- on the **Settings** toolbar: the  **Options** button;
- in the command line: **PREF** command;
- using hotkeys: **CTRL+9**.

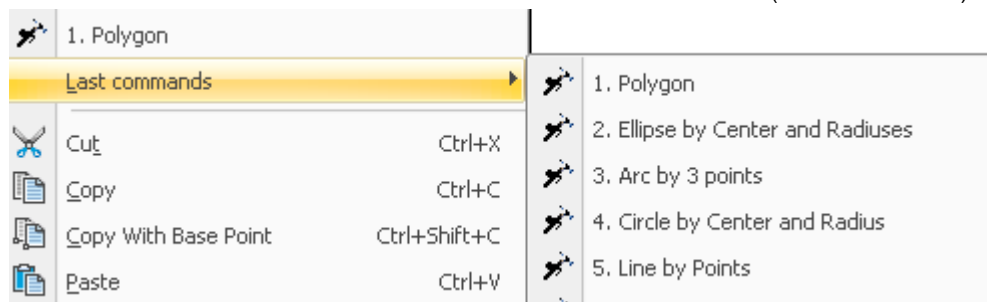
Transparent commands

Some commands can be launched while other commands are executed. Such commands are called transparent. Most of these commands are settings commands for display and document.

Command recall

The last launched command can be started by pressing the **ENTER** or **SPACE** buttons.

The last nine launched commands are available from the context menu (**Last commands**):



Last commands are shown in the command line.

Use **ARROW DOWN** and **ARROW UP** buttons to select one of the last commands and press **ENTER** to launch it.

Cancellation of commands

nanoCAD creates protocols for all used commands and changes in the drawing, one or several commands can be undone to get to the previous step. Undone command can be launched again.

Cancellation and recall of commands can be made using the **Undo** and **Redo** commands.

Undo



Menu: **Edit –  Undo**



Hotkeys: **CTRL+Z**



Toolbar: **Main** –



Command line: **UNDO, U**

The command cancels all previous commands and operations.

Redo



Menu: **Edit** – **Redo**



Hotkeys: **CTRL+Y**



Toolbar: **Main** –

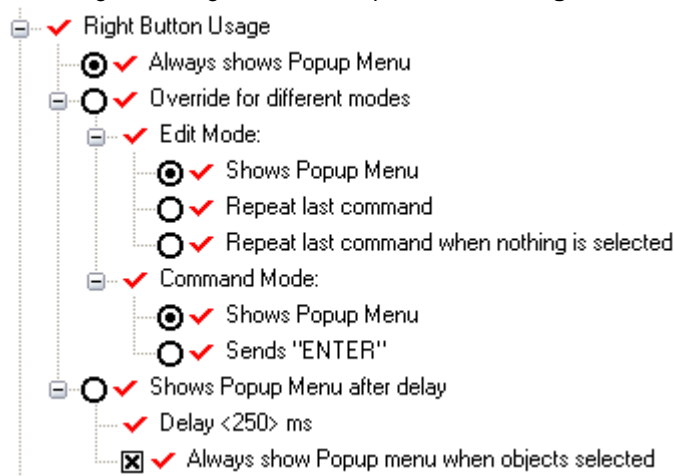


Command line: **REDO**

The command recovers all cancelled commands and operations. It becomes available after the **Undo** command is used.

Settings of the right button

The settings of the right button are specified in the **Right Button Usage** section of the **Options** dialog:



Parameters:

Always shows Popup Menu

Switches on the only context menu launch.

Override for different modes:

Switches on the redefining of right button functions for different working modes in the document window.

Edit Mode:

The sections redefine the right button functions when it is pressed in the editing mode, i.e. there are no running commands.

Shows Popup menu

Switches on the only context menu launch.

Repeat last command

Switches on the **ENTER** key imitation function, when clicking the right button again starts the last command if objects were selected.

Repeat last command when nothing is selected

Switches on the **ENTER** key imitation function, when clicking the right button again starts the last command if no objects were selected.

Command mode:

The sections redefine the right button functions when it is pressed in the mode of any running command.

Shows Popup menu

Switches on the only context menu launch.

Sends “ENTER”

Switches on the **ENTER** key imitation function.

Shows Popup menu after delay:

Switches on the mode which considers the duration of right button holding:

- short clicking – repetition of command or **ENTER** according to the mode,
- long holding – opens the context menu.

Delay <250> ms

Specifies, in milliseconds, the duration of right button holding to open the

context menu. By default the value is 250 ms.

To change the duration of button holding:

- click twice on the delay value,
- enter a new delay value.

Always show Popup menu when objects selected

Switches on/off the mode, where the context menu launches every time the right button is clicked during selection of objects.

Status bar

There are interface elements in the status bar (8):

| | |
|-------------------------------------|--|
| | Current coordinates of the cursor. Display modes: <ul style="list-style-type: none">- dynamic display of cursor absolute coordinates when the cursor is moving,- display of relative distance (distance<angle) when the cursor is moving: Switching to the relative distance mode is made automatically when specifying two or more points is required. |
| | Button to switch to viewport. It is not used in Model space. |
| | Button to switch to Layout from Model space or viewport. |
| | Button to set scale of objects. |
| | Button to start regeneration mode. |
| | Button to switch on/off full screen mode. |
| Buttons to switch on/off modes: | |
| | Snap (F9). |
| | Grid (F7). |
| | Object snap (F3). |
| | Object tracking (F11). |
| | Polar tracking (F10). |
| | Ortho mode (F8). |
| | Show line width. |
| | Hatch |
| Buttons to manage the graphic area: | |
| | Pan. |
| | Zoom. |
| | Zoom all |
| | Zoom window. |

To switch the status bar on/off, use the **Status bar** command from the **View** menu..

Managing elements of the Status bar

Use the **Status Bar Configuration** menu to set elements of the Status bar:

| Status Bar Configuration | | |
|-------------------------------------|--------------------|-------------------------------|
| <input checked="" type="checkbox"/> | . | 83039.9160,-39647.1639,0.0000 |
| <input checked="" type="checkbox"/> | SNAP | SNAP |
| <input checked="" type="checkbox"/> | GRID | GRID |
| <input checked="" type="checkbox"/> | OSNAP | OSNAP |
| <input checked="" type="checkbox"/> | OTRACK | OTRACK |
| <input checked="" type="checkbox"/> | POLAR | POLAR |
| <input checked="" type="checkbox"/> | ORTHO | ORTHO |
| <input checked="" type="checkbox"/> | SW | SW |
| <input checked="" type="checkbox"/> | SH | SH |
| <input checked="" type="checkbox"/> | | |
| <input checked="" type="checkbox"/> | Switch to Model | Switch to Model |
| <input checked="" type="checkbox"/> | Switches to Layout | Switches to Layout |
| <input checked="" type="checkbox"/> | Scale | |
| <input checked="" type="checkbox"/> | PAN | PAN |
| <input checked="" type="checkbox"/> | Zoom | Zoom |
| <input checked="" type="checkbox"/> | Zoom All | Zoom All |
| <input checked="" type="checkbox"/> | Zoom Window | Zoom Window |
| <input checked="" type="checkbox"/> | Regen | Regen |
| <input checked="" type="checkbox"/> | FULLSCREEN | FULLSCREEN |

To open the context menu:

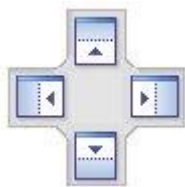
- right button click on the Status bar,
- select/deselect the required elements in the context menu.

Managing interface elements

To change the location of interface elements, drag their titles.

To change location:

- click on an element's title and drag it to an area of the window where you want to attach the element. The icon appears in the centre of the window, showing the place of attachment in the window.



- holding the left button, place the cursor over one of the pictograms. The central pictogram is used to attach the object as a tab, others attach to the sides of the window.

The new location of the element is highlighted blue.

If you are satisfied with the new location, release the button.

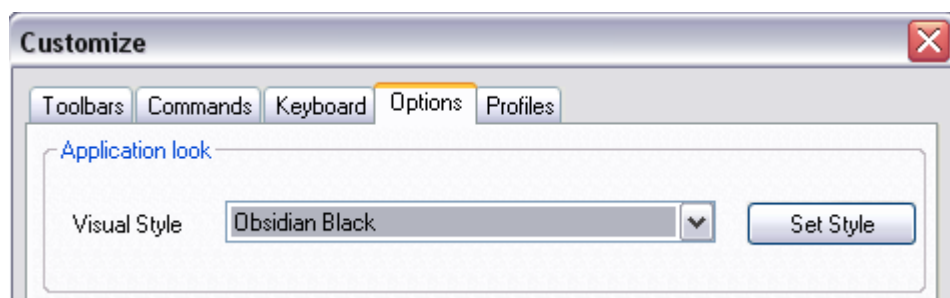
The attached element can be made into a pop-up tab. Select the pin in the element's title:



For some elements, the context menu is available with various ways of attachment:

Colour schemes

Besides full reorganisation of the working area for user needs, nanoCAD has a set of colour schemes for the interface. You can change the colour scheme in the **Options** tab of the **Interface** dialogue box (**Tools – Customize – Interface**):

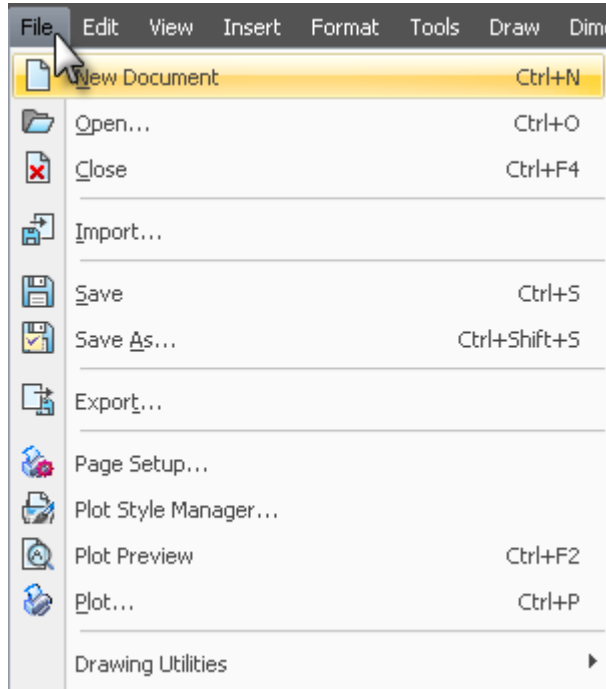


Working with documents

nanoCAD allows documents to be saved with ***.dwg** (drawing format), ***.dwt** (template format) и ***.dxf** (graphic data exchange format) extensions. There is support of file formats created in the following AutoCAD versions:

- AutoCAD 2010-2012/LT2010-2012
- AutoCAD 2007-2009/LT2007-2009
- AutoCAD 2004-2006/LT2004-2006
- AutoCAD 2000-2002/LT2000-2002
- AutoCAD R14/LT98/ LT97
- AutoCAD R13/LT95
- AutoCAD R11

Commands to work with documents are in the **File** menu:



and on the **Main** toolbar:



Creation of new document

Menu: **File** –  **New Document**

Toolbar: **Main** – 

Hotkeys: **CTRL+N**

Command line: **NEW**

A new document, created at first launch, has the name **Untitled0**; all subsequent documents have the names **Untitled1**, **Untitled2** etc. Documents which have been created and not saved have a star symbol (*):



Template usage

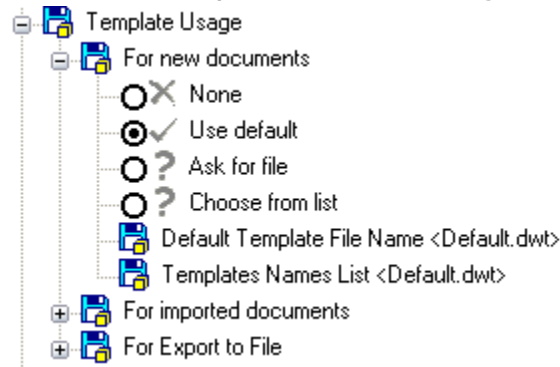
To create a new document you can use a template (files with ***.dwt** extension). The template is a drawing used for the creation of new drawings and contains some base settings (type and accuracy of units, drawing limits, settings of

SNAP and **GRID** modes; managing of layers; dimension and text styles; types and weights of lines etc.) and graphic objects (main titles, borders and logos).


Changes in a drawing created using a template are not applied to the template.

Note: You can create a template not only for new documents but also for imported documents.

Templates are managed in the **Template Usage** section of the **Options** dialog:




In the **For new documents** subsection you can select:

| | |
|-----------------------------------|---|
| None | Prevents the use of a template for new documents. |
| Use default | Uses the file specified in Default Template File Name . |
| Ask for file | Opens the dialogue box File → Open . |
| Choose from list | Opens the Choose Template dialogue box. |
| Default Template File Name | Sets the Default Template File Name . |
| Templates Names List | Specify the template files if the Choose from list is selected: <ul style="list-style-type: none">Click twice on the Templates Names List,Press the  button,In the Open dialogue box select template files, select Open. |

Opening a document

Menu: **File** –  **Open...**

Toolbar: **Main**– 

Hotkeys: **CTRL+O**

Command line: **OPEN**

To open a document you can drag it into the working area of nanoCAD.


Note: If a document contains a font, which is not included in the program, when you open the document it is automatically replaced with an alternative font. There is a message in the command line like:

```
Командная строка:
Модуль загружен
Загружается модуль: mcDDLloader3.dll
Модуль загружен
Шрифт ST.shx заменён на txt.shx
Шрифт ST.shx заменён на txt.shx
Команда:
```

This illustration
and needs rep
English

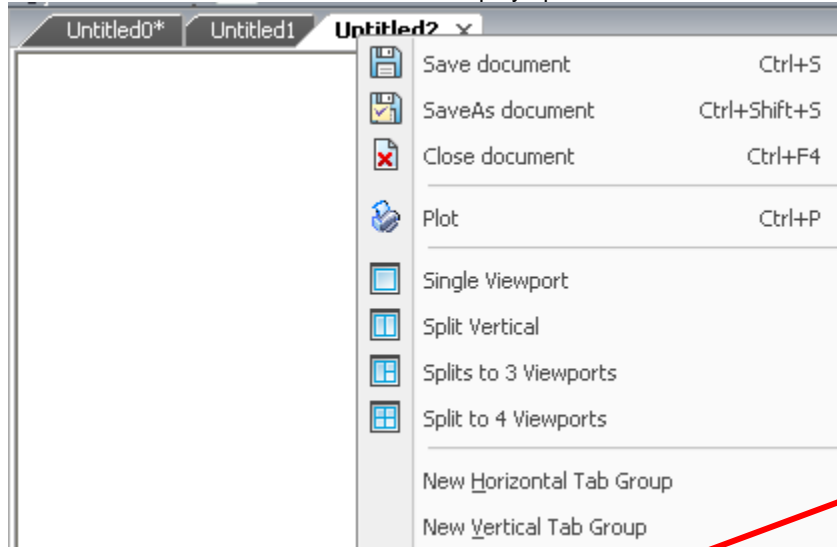
nanoCAD uses a default *txt.shx* for replacement. You can change it in the **Options** dialog.

Text
Alternate font name <txt.shx>

When many documents are opened, every document is opened in a separate window with a tab in the top part of the working area. To switch between them, select the tab or click the  **Active files** button:



The context menu allows the selection of display options for the documents:



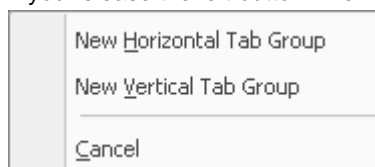
Should this say Print?

Commands of the context menu:

| | |
|---------------------------------|--|
| Save | Saves a document. |
| Save As... | Saves a document with a new name. |
| Close | Closes a document. |
| Plot | Prints a document. |
| Single Viewport | Shows a document in one Viewport |
| Split Vertical | Splits document to 2 Viewports. |
| Splits to 3 Viewports | Splits document to 3 Viewports. |
| Splits to 4 Viewports | Splits document to 4 Viewports. |
| New Horizontal Tab Group | Horizontal display of a tab in the working area. |
| New Vertical Tab Group | Vertical display of a tab in the working area. |


You can change the location of tabs by dragging them into other windows.

If you release the left button when dragging a tab, the context menu with tab display options will be shown:




Closing a document



Menu: **File** –  **Close**



 button on the document's window



Hotkeys: **CTRL+F4**



Command line: **CLOSE**


When closing an existing document you should save the changes to it; a document is saved with its name.
When closing a new document, the program offers to save changes and opens the **Save Document File** dialogue box.

Saving a document

Documents can be saved in the ***.dwg** drawing format, in the ***.dxf** graphic data exchange format and in the ***.dwt** template format. **Save**, **Save As** and **WBLOCK** commands are used.

Saving of an existing document.



Menu: **File** –  **Save**



Toolbar: **Main** – 



Hotkeys: **CTRL+S**

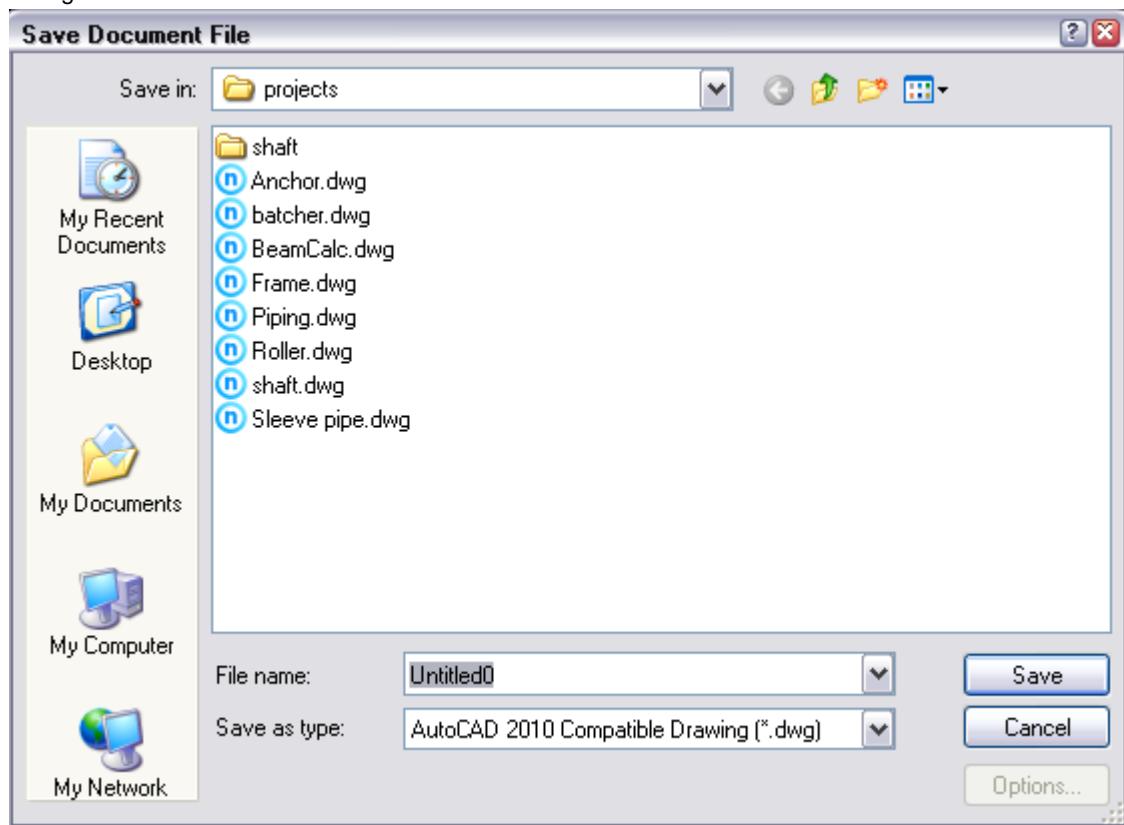


Command line: **SAVE**, **BS**

The command saves a document with its name.

Saving a new document

The **Save** command opens the **Save Document File** dialogue box, where you can specify a path for a document and change the default file name:



To save a document:


- specify the path to the file in the **Folder** drop-down list,
- type the name in the **File name** field,
- select the format in the **File type** field,
- select the **Save** button.

Saving the file will be done without specifying the path and file name in the future.

Note: Format, specified in the **File type** field, is also used to save all documents with **Save**, **Save As** and **WBLOCK** commands.

Saving a document with another name



Menu: **File** –  **Save As**



Hotkeys: **CTRL+SHIFT+S**



Command line: **SAVEAS**

Every time you select the **Save As** command, nanoCAD will offer to specify a path to a file and/or change a file name in the **Save Document File** dialog. The selected path and entered file name are set by default, and when the **Save** command is used, a file will be saved with the path and name specified earlier.

The **Save As** command can be used to change the format of a file, specified to save all documents using **Save**, **Save As** and **WBLOCK**.

Saving a document or its part using the WBLOCK command



Command line: **WBLOCK, W, ACADWBLOCKDIALOG**

The **WBLOCK** command is started from the command line and allows saving of an entire document and also part of a drawing using the **Write Block** dialog:

The **Write Block** dialogue box complements and extends the capabilities of the **Save** and **Save As** commands, allowing the saving of an entire document (**Entire drawing** parameter) and also part of a drawing (**Block** and **Objects** parameters).

Auto saving and Backup

To minimise losses caused by software or hardware failures and any other consequences, nanoCAD has auto saving and backup functions.

Auto saving

If Auto saving mode is switched on, the data is saved at the specified intervals.

The time and date of the auto saving is added in round brackets to a file name with ***.autosave** extension, for example: **<file name>.dwg(20-29-44_15.08.2010).autosave**.

Note 1: the time of auto saving is the time of the first auto saving and it is not updated further.

Note 2: If the program is closed normally, auto saved documents are removed. If there is a software or hardware failure and any other consequences, auto saved files are not removed.

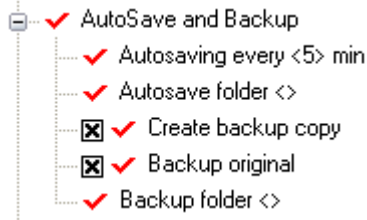
Note 3: You can restore the previous session of a document from the auto saved file. Do not close the program and file, copy it to another folder and rename as described below.

A default folder for auto saved files

C:\Documents and Settings\Account Name\Local Settings\Temp.

To open and work with an auto saved file, change its extension to ***.dwg** (delete the additional information in round brackets and the ***.autosave** extension).

The auto saving parameters are specified in the **AutoSave and Backup** section of the **Options** dialogue box (**Tools** menu > **Options**):

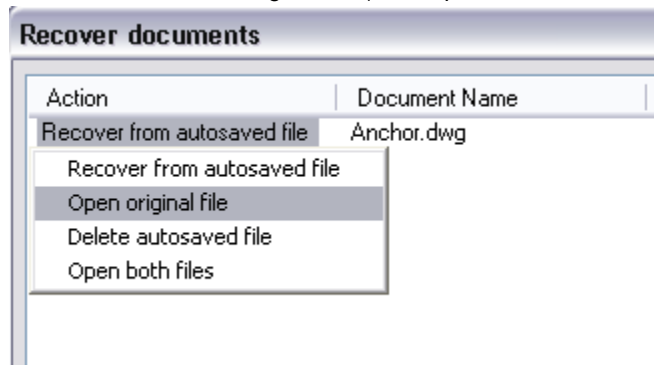


Parameters:

Autosaving every <XX> min Time interval between savings. To switch off auto saving, set 0 for this parameter.

Auto saving folder Folder to save files.

The first launch of nanoCAD after a failure is started from the **Recover documents** dialogue box, containing information about the original file (name, path and creation time) and last auto saving time.



To recover document data, click on the file name and select one of the following in the context menu:

Recover from autosaved document Opens the autosaved file.

Open original file Ignores the automatically saved version and opens the original document (if it exists).

Delete autosaved file Deletes the automatically saved versions

Open both files Opens both documents.

Backup

nanoCAD creates 2 backup copies of the file with the same name, but with different extensions.

Original backup copy (file with ***.original** extension) saves document data as it was when the file was opened for editing (*original backup* is created at the beginning of the session of file editing);

Backup copy (file with ***.bak** extension) is often updated during the document editing process (at every saving by the user). Different contingencies can occur with a document during the working process, for example, malfunctions of hardware, failure of the software or improper actions of a user. Backup copy allows the user to recover information and minimise losses of information by means of periodic savings of the file by the user.

Backup copies are saved in the folder where the document file is saved.

Backup parameters are specified in the **AutoSave and Backup** section of the **Options** dialogue box (**Tools** menu > **Options**):

Parameters:

Create backup copy Switches backup mode on/off.

Backup original Switches original backup mode on/off.

Backup folder Location to save backup copies.

To recover data from a backup copy, change the file extension to ***.dwg**.

Import



Menu: **File**—  **Import...**

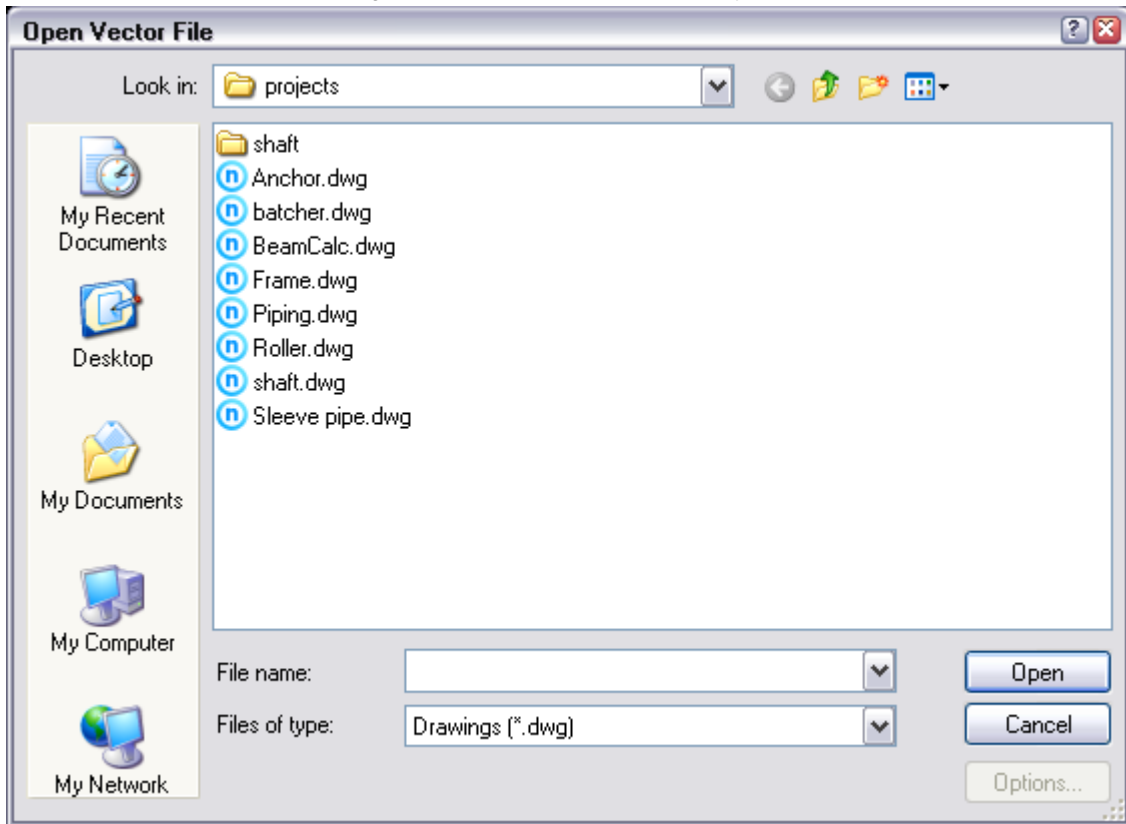


Command line: **IMPORT**

You can import data in AutoCAD format(*.dwg) and data exchange format (*.dxf) into nanoCAD documents.

To import data:

- Start the **File – Import** command;
- In the **Open Vector File** dialogue box select a format and specify a file name:



- Select **Open**.

The ways to use templates for imported documents are set in the **Options: Template Usage > For imported documents**.

Export



Menu: **File – Export...**

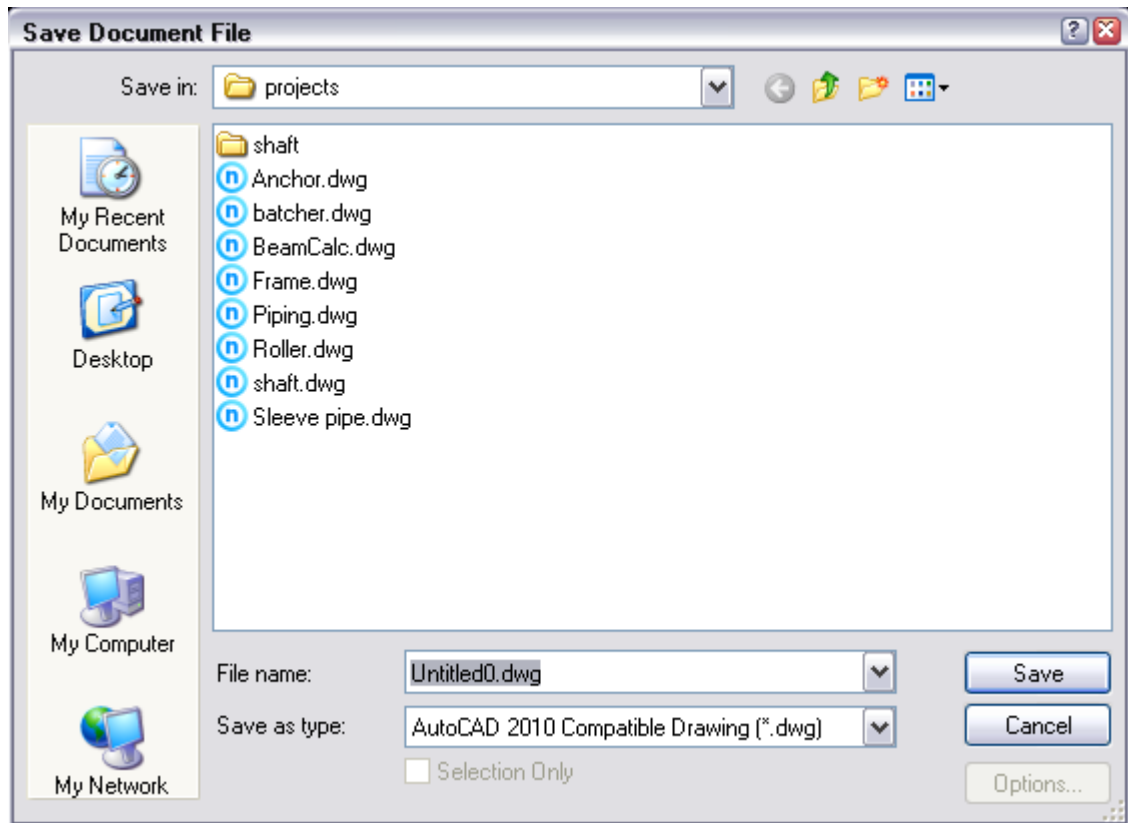


Command line: **EXPORT**

A nanoCAD document's data can be exported to AutoCAD format(*.dwg) and data exchange format (*.dxf). Either the entire document or a part of it can be exported.

Export of all data

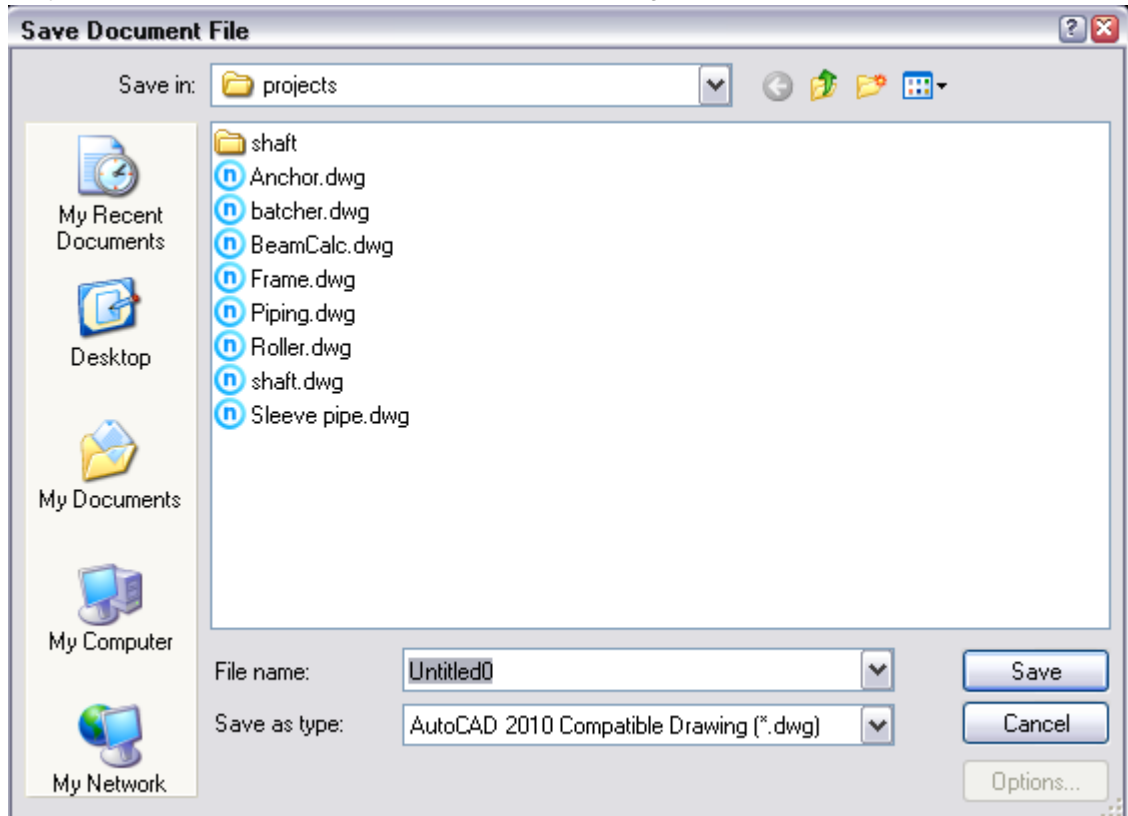
In the **Save Document File** dialogue box:



- specify the path for the saved file in the **Folder** drop-down list,
- type a name in the **File name** field,
- select a format in the **File type** field,
- select the **Save** button.

Export of selected data

The export procedure for selected data is the same as the export procedure for all the data, except that the **Selection Only** checkbox is checked in the **Save Document File** dialogue box:



Note: If the objects were not selected before saving, the **Selection Only** checkbox is unavailable.

Utilities

Errors and damage can occur in document files due to disruptions in power supply, malfunctions of computer hardware, programs crashes, transfer of documents on electronic media and transferring over the network etc.

Utilities find and correct errors and damage in problem files as far as possible. Full recovery of data is not guaranteed.

Purging of a document allows deletion of unused elements from the finished document to reduce document size.

Document audit



Menu: **File – Drawing utilities > Audit**



Command line: **AUDIT**

The command allows auditing of an opened document in nanoCAD for errors and correcting some of them. If a document cannot be opened due to damage, use the **Recover** command.

Correction of all errors is not guaranteed.

After starting the command, in the inquiry *Do you want to fix errors ? or [Yes/No]*: select **Yes**.

Audit protocol is like:

```
AcDbLine(117689). Error:: Inactive XData handle (1157FE) was found.
```

```
Audit: Invalid .
```

```
Fixed:: Replaced: Set to Null .
```

```
AcDbArc(11768A). Error:: Inactive XData handle (1157FE) was found.
```

```
Audit: Invalid .
```

```
Fixed:: Replaced: Set to Null .
```

```
AcDbCircle(119E1F). Error:: Inactive Z coordinate - 1e+100 was found.
```

```
Audit: Invalid .
```

```
Fixed:: Replaced: Set to 0 .
```

To see the protocol press **F2**.

Recovery of document



Menu: **File – Drawing utilities > Recover**



Command line: **RECOVER**

The command allows auditing an opened document in nanoCAD for errors and correcting some of them. I

Full recovery and correction of all errors are not guaranteed.

After command launch, a standard dialogue box to open a file appears, you have to select the folder and the name of the damaged file and after that select the **Open** button.

The protocol is shown in the command line and is similar to the protocol of the **Audit** command:

```
AcDbLine(117689). Error:: Inactive XData handle (1157FE) was found.
```

```
Audit: Invalid .
```

```
Fixed:: Replaced: Set to Null .
```

```
AcDbArc(11768A). Error:: Inactive XData handle (1157FE) was found.
```

```
Audit: Invalid .
```

```
Fixed:: Replaced: Set to Null .
```

```
AcDbCircle(119E1F). Error:: Inactive Z coordinate - 1e+100 was found.
```

```
Audit: Invalid .
```

```
Fixed:: Replaced: Set to 0 .
```

To see the protocol press **F2**.

Purging of document



Menu: **File – Drawing utilities > Purge ...**





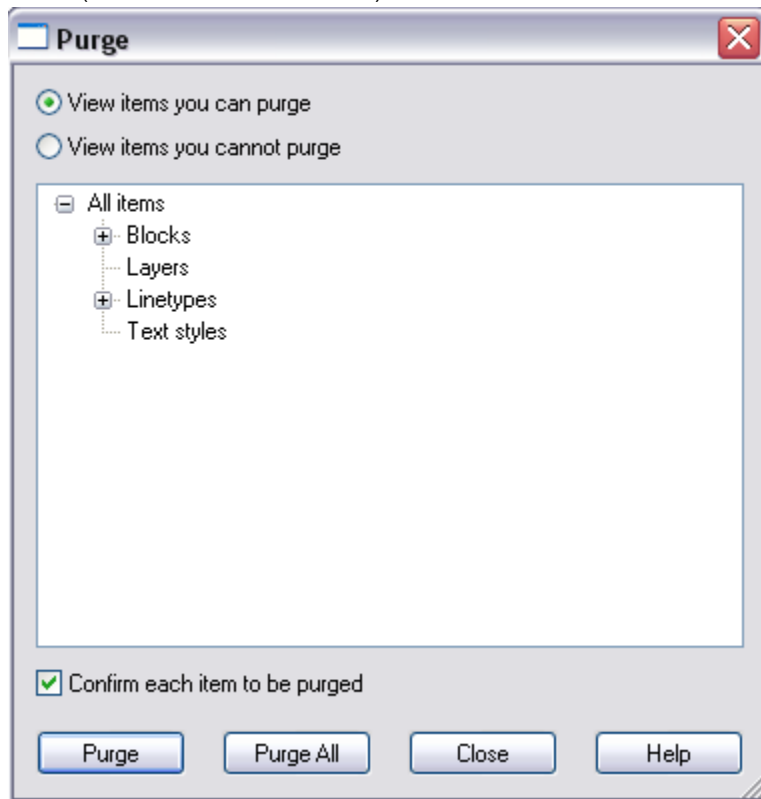
Command line: **PURGE, PU**

This command is used to purge unused named elements from a document: blocks' descriptions, layers, text styles and line types.

Using this procedure you can reduce file size after finishing your project.

Note: Deleted elements cannot be recovered.

The **Purge** dialogue box appears after the command is launched. There is a list of elements available for deletion and a list of elements not available for deletion. The list is organised as a tree: to open a list of elements of one type select the  sign to the left of the name. If there is no  sign, it means that there are no elements of this type to delete (or which cannot be deleted).



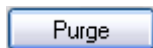
Parameters:

View elements you can purge

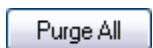
Switches on the mode for viewing elements which can be deleted from a document.

View elements you cannot purge

Switches on the mode for viewing elements which cannot be deleted from a document. Elements, used in a document or standard items, which cannot be deleted.


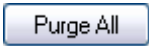


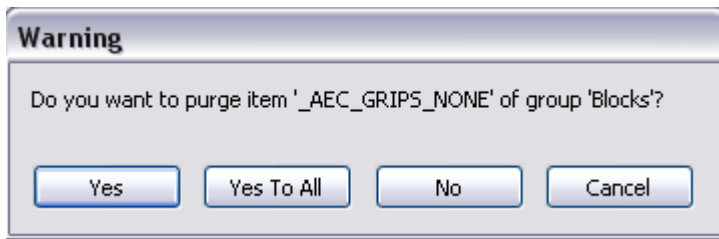
Purges the selected elements.



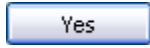
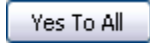
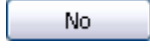
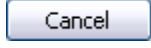
Purges all elements from the list.

To delete elements with confirmation:

- Switch on the **Confirm each item to be purged** mode.
- Select the  button or .
- In the opened dialogue box confirm purging of the elements:



Parameters:

- | | |
|---|--|
|  | Confirmation for every selected element. |
|  | Confirmation for all elements. |
|  | Refuse to delete a selected element. |
|  | Cancels the purging mode. |

Tuning nanoCAD

User can change different parameters of nanoCAD.

Tuning program parameters



Menu: **Tools – Options...**

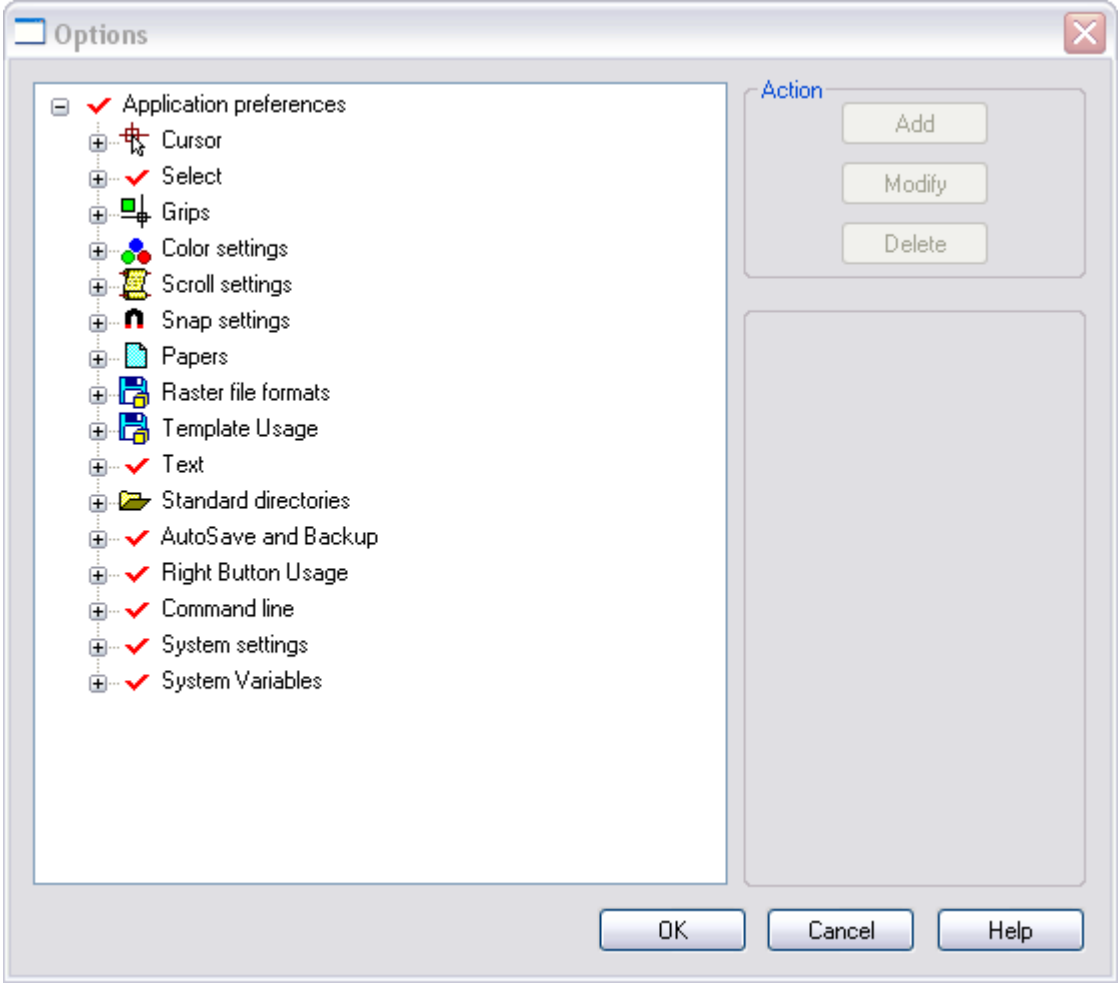


Hotkeys: **CTRL+9**



Command line: **OPTIONS, PREF**

Tuning program parameters is performed in the **Options** dialogue box:



There is a tree of Options, grouped by sections, in the left part of the dialogue box.

To see the parameters of the group, click twice on the icon to the left of the section name. If you click the icon, the section (subsection) will be closed.


















The **Add**, **Modify** and **Delete** buttons of the **Action** section become available; this means that the selected parameter from the tree can be edited.




































To change a selected item, click twice on it or select the button and specify a new value.

There is brief information about the selected section, subsection or parameter in the bottom right field.

The tree of parameters contain the following elements: (adjustable parameters are bold):

| | | |
|--|---------------------------------|---|
| | Application Options | |
| | Cursor | Changes the appearance and size of the graphic cursor. |
| | Crosshair size<5%> | Sets the size of the Crosshair as a percentage of the display size. |
| | Pickbox | The Pickbox size. |

| | | |
|---|---|---|
|  | PickBox size <4> | Pickbox size in pixels. |
|  | 3D style colors | 3D colour style for the crosshair and pickbox. |
|  | Cursor color | Cursor colour |
|  | Show Object and Snap Tooltips | Switches object tooltips on/off. |
|  | Select | Parameters of frames to select objects. |
|  | Color frame | Colour frame |
|  | Color fence | Colour fence. |
|  | Transparency <70> | Transparency of frame. |
|  | Grips | Parameters of the grips of the selected objects. |
|  | Grip size <10> | Grip size in pixels. |
|  | Grip max objects count <100> | Limits of grips shown for selected objects. |
|  | Grips color | Grips colour. |
|  | Hovered grips color | Grips colour under cursor. |
|  | Selected grips color | Selected grips colour. |
|  | Color | Colours of the program components. |
|  | Preview frame | The frame colour that indicates an image preview area inside the preview window. |
|  | Background | The colour of the program window background. |
|  | Print Area | The frame colour that indicates a print area |
|  | Grid Color | The colour of Grid points. |
|  | Layout Paper | The colour of the Layout paper. |
|  | Scroll settings | Defines the scroll parameters of a document in the program window. |
|  | Mouse Wheel scale factor <1.5> | Scale factor used to scale with the Mouse Wheel |
|  | Snap settings | Defines the Snap settings. |
|  | Hold Aperture Size <10> | The size of a cursor frame in snap mode. |
|  | Show Aperture Box | Switches the aperture box on/off in the snap mode. |
|  | Snap Marker size <5> | Snap marker size. |
|  | Show Tooltips | Turns on the display of a snap name. |
|  | Vector Marker Color | The colour of the snap marker when snapping to a vector object. |
|  | Papers | Contains standard paper formats. Allows the modification of an existing format or addition of a new one. Formats from this section are used for printing. |
|  | Raster File Formats | Raster file formats which can be inserted with Image from File command. |
|  | TIFF | |
|  | Windows Bitmap | |
|  | JPG | |
|  | JPEG | |
|  | Portable Network Graphics | |
|  | Template Usage | Section to specify templates for new documents and for import and export of documents. |
|  | For new documents | Actions on the File > New command. |

| | | |
|---|---|---|
|  | None | No action taken. |
|  | Use default | Opens the file specified in the Default Template File Name . |
|  | Ask for file | Opens the File > Open dialogue box. |
|  | Choose from list | Opens the Choose Template dialogue box. |
|  | Default Template File Name <Default.dwt> | Shows and allows changing of the Default Template File Name . |
|  | Templates Names List | List for Choose Templates . |
|  | For imported documents | Actions for imported documents. |
|  | None | No action taken. |
|  | Use default | Opens the file specified in the Default Template File Name . |
|  | Ask for file | Opens the File > Open dialogue box. |
|  | Choose from list | Opens the Choose Template dialogue box. |
|  | Default Template File Name <Default.dwt> | Shows and allows changing of the Default Template File Name . |
|  | Templates Names List | List for Choose Templates . |
|  | For Export to File | Actions for exported documents. |
|  | None | No action taken. |
|  | Use default | Opens the file specified in the Default Template File Name . |
|  | Ask for file | Opens the File > Open dialogue box. |
|  | Choose from list | Opens the Choose Template dialogue box. |
|  | Default Template File Name <Default.dwt> | Shows and allows changing of the Default Template File Name . |
|  | Templates Names List | List for the Choose Templates . |
|  | Text | Text parameters. |
|  | Alternative font name <txt.shx> | Name of font file to replace an missing font in an opened document. |
|  | Standard directories | Sets the folder in which the miscellaneous system files are stored. |
|  | Shx file location: < C:\Documents and Settings\All Users\Application Data\Nanosoft\nanoCAD 2.5\shx > | |
|  | Templates file location: < C:\Documents and Settings\User_name\Application Data\Nanosoft\nanoCAD 2.5\Templates > | |
|  | Samples file location: < C:\Program Files\Nanosoft\nanoCAD 2.5\Samples > | |
|  | PlotStyles files location: < C:\Documents and Settings\User_name\Application Data\Nanosoft\nanoCAD 2.5\PlotStyles > | |
|  | PlotConfigs files location: < C:\Documents and Settings\User_name\Application Data\Nanosoft\nanoCAD 2.5\PlotConfigs > | |
|  | Pat files location: < C:\Documents and Settings\All Users\Application Data\Nanosoft\nanoCAD 2.5\shx > | |
|  | AutoSave and Backup | Autosaving and Backup parameters. |
|  | Autosaving every < 5 > min | Saving interval for the current document. Zero value switches off autosaving. |
|  | Autosave folder < > | Folder for autosaved files. Default folder is TEMP . |
|  | Create backup copy | Switches backup copy mode on/off. |
|  | Backup original | Switches original backup copy mode on/off. |
|  | Backup folder < > | Folder to save backup files. By default, backup files are |

| | | |
|-----|--|--|
| | | saved in the same folder as the original file. |
| ✓ | Right button usage | Functions of the right button. |
| ⊙ ✓ | Always shows Popup Menu | Switches on the context-only menu launch. |
| ○ ✓ | Override for different modes: | Switches on the redefining of right button functions for different working modes in the document window. |
| ✓ | Edit Mode: | This section redefines the right button functions when it is pressed in the editing mode, i.e. there are no running commands. |
| ⊙ ✓ | Shows Popup menu | Switches on the context-only menu launch. |
| ○ ✓ | Repeat last command | Switches on the ENTER key imitation function, where clicking the right button again starts the last command if objects were selected. |
| ○ ✓ | Repeat last command when nothing is selected | Switches on the ENTER key imitation function, where clicking the right button again starts the last command if no objects were selected. |
| ✓ | Command mode: | This section redefines the right button functions when it is pressed during a running command. |
| ⊙ ✓ | Shows Popup menu | Switches on the context-only menu launch. |
| ○ ✓ | Sends "ENTER" | Switches on the ENTER key imitation function. |
| ○ ✓ | Shows Popup menu after delay: | Switches on the mode which considers the duration of right button holding: <ul style="list-style-type: none"> – short clicking – repetition of command or ENTER according to the mode, – long holding – opens the context menu. |
| ✓ | Delay <250> ms | Specifies in milliseconds, the duration of right button holding to open the context menu. By default the value is 250 ms. To change the duration of button holding: <ul style="list-style-type: none"> – click twice on the delay value, – enter a new delay value. |
| ☒ ✓ | Always show Popup menu when objects selected | Switches on/off the mode where the context menu launches every time the right button is clicked during selection of objects. |
| ✓ | Command line | |
| ☒ ✓ | Use Autocomplete | |
| ✓ ☐ | Command line background color | |
| ✓ ■ | Command line text color | |
| ☒ ✓ | Use alternate font | |
| ✓ | System settings | System setting section. |
| ✓ | Graphics hardware acceleration settings | Settings of the platform component serving video card and OpenGL subsystem. |
| ⊙ ✓ | Setting 1 (OpenGL) | |
| ○ ✓ | Setting 2 (OpenGL) | |
| ○ ✓ | Setting 3 (OpenGL) | |
| ✓ | Print preview mode settings | Section is used to control platform components, used for creation of preview if printing is in metafile format (WMF) or raster image (BMP). |
| ⊙ ✓ | Metafile (WMF) | |
| ○ ✓ | Bitmap (BMP) | |
| ☐ ✓ | Separate display cache for Model Space views | Creates display cache for every viewport of Model Space. |
| ☐ ✓ | Separate display cache for Paper Space views | Creates display cache for every viewport of Paper Space |
| ✓ | System variables | Changes values of system variables. |

| | | |
|---|---------|--|
| ✓ | CMDDIA | |
| ✓ | FILEDIA | |

Tuning the interface



Menu: **Tools – Customize >**  **Interface...**



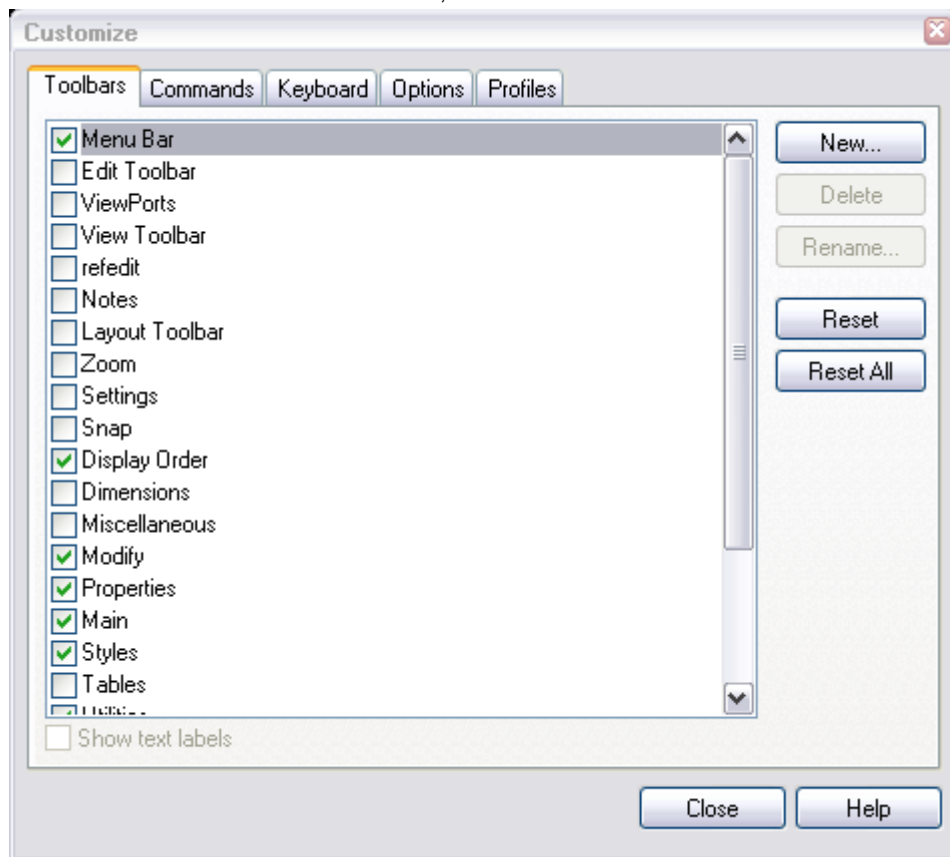
Command line: **CUSTOMIZE**

In the **Customize** dialogue box you can change the settings of the working environment, create new menus and toolbars, add or delete commands to menus or toolbars, save all settings in one profile and manage profiles.

The **Customize** dialog contains 5 tabs: **Toolbars**, **Commands**, **Keyboard**, **Options** and **Profiles**.

Toolbars tab

The toolbar contains a list of instruments, used in nanoCAD.



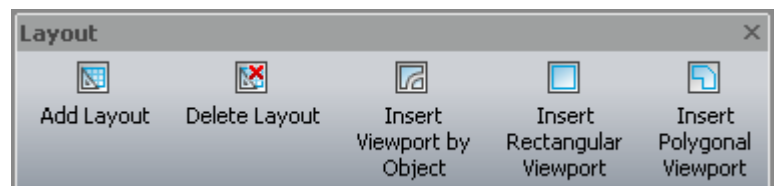
The ☒ checkbox to the left of the toolbars manages toolbar display.

The **Show text labels** checkbox switches on/off the display of text labels on the buttons of the selected toolbar. The **Layout** toolbar view:

Show text labels is switched off




Show text labels is switched on




The **Toolbar** tab allows the creation of custom toolbars and placing any button there, and adding buttons to standard toolbars.

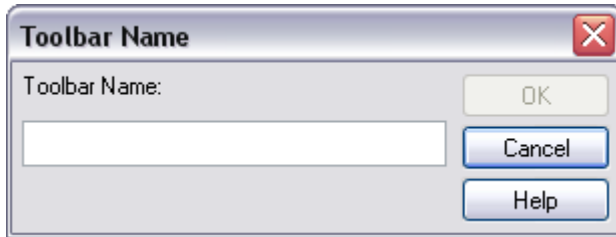
The  button resets the menu bar.

The  resets all menus and toolbars.

To create a new toolbar:

- Select the  button.

- In the **Toolbar Name** dialog specify a name and select **OK**:



The ☒ checkbox for the created toolbar is selected automatically.

- Select the **Close** to close the **Customize** dialogue box.

To place a button onto a new toolbar:

- Open the **Customize** dialogue box.
- Switch to the **Commands** tab.
- In the *Categories* section select **All Commands**.
- In the *Commands* section select the required command and drag it to the new toolbar.
- Select the **Close** button.

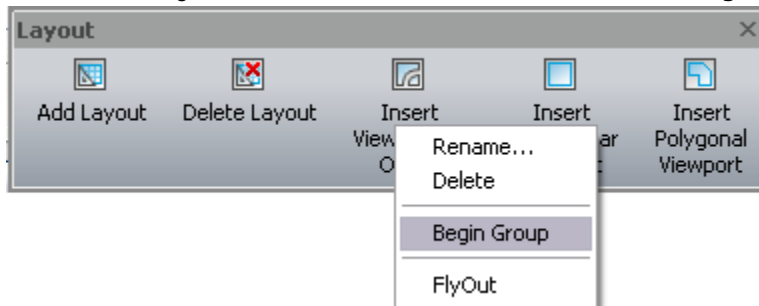
You can move and copy buttons from the existing toolbars to a new toolbar.

The toolbar from which you want to take a button must be displayed.

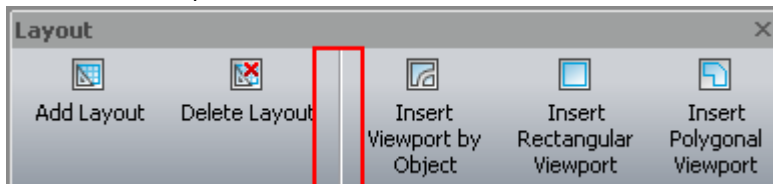
To copy a button drag it with **CTRL** pressed.

To create a new group of commands on a toolbar:

- Open the **Customize** dialogue box.
- Place the cursor over the button which will start a group.
- Press the right button and from the context menu select the **Begin Group** command:

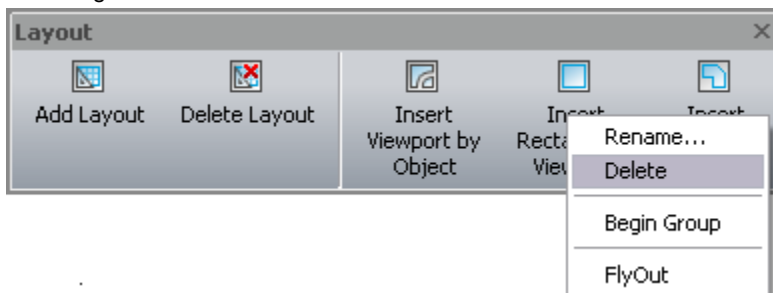


- There is a separation line to the left of the button:



To delete a button from a toolbar:


- Open the **Customize** dialogue box.
- Place the cursor over the button, which you want to delete.
- Drag the button off the toolbar or from the context menu select the **Delete** command:

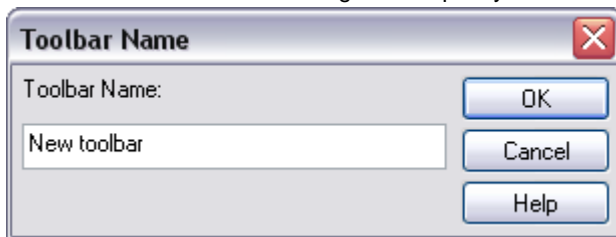


- Select the **Close** button.

To rename a toolbar:


- Open the **Customize** dialogue box.
- Select the toolbar from the list.

- Select the  button.
- In the **Toolbar Name** dialogue box specify the new name and select **OK**:



- Select the **Close** button.
- Note:** You can only rename custom toolbars.

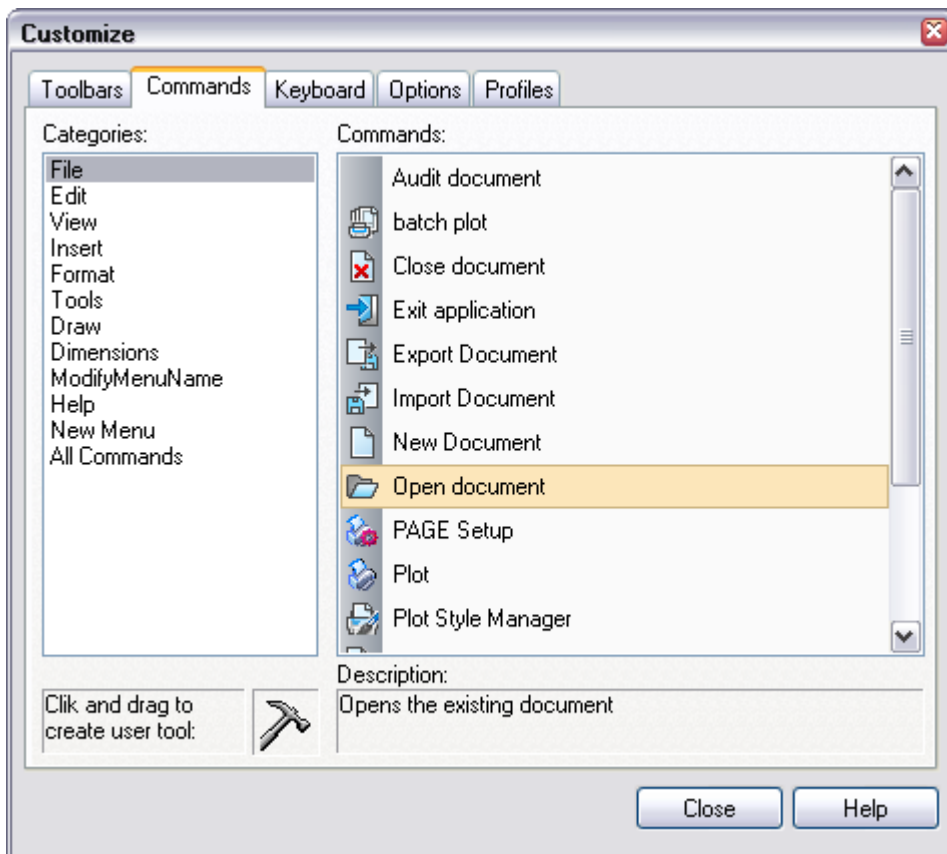
To delete a toolbar:

- Open the **Customize** dialogue box.
- Select the toolbar from the list.
- Select the  button.
- Select the **Close** button.

Note: You can only delete custom toolbars.

Commands tab

This tab allows nanoCAD commands to be added to menus and toolbars.



There are two sections (*Categories* and *Commands*) in the **Commands** tab.

The titles of the main menus are in the *Categories* section.

All the nanoCAD commands are in the **All commands** category of the *Commands* section.

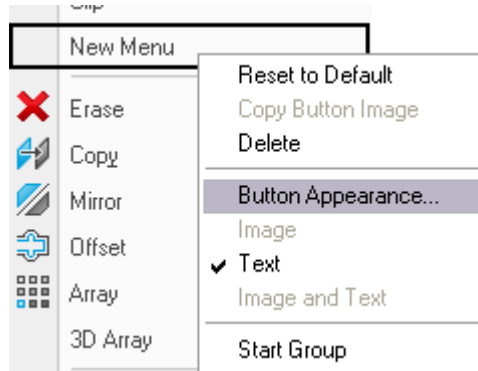
There is short description of the selected command in the *Description* field.

nanoCAD commands can be dragged into the opened menus and toolbars from the dialogue box.

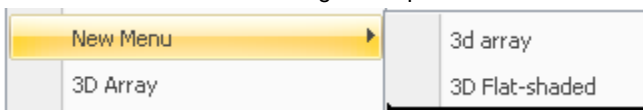
To add new commands to a menu:

- Open the **Customize** dialogue box.
- Select the **New Menu** in the *Categories* section.
- Open the menu to which you want to add new commands.
- From the *Commands* section drag the **New Menu** element to the menu bar.

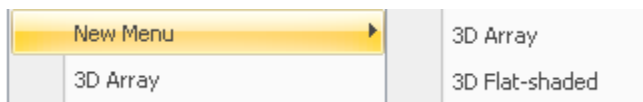
- To rename the inserted menu bar, place the cursor over the **New menu** and press the right button. From the context menu select the **Button Appearance**:



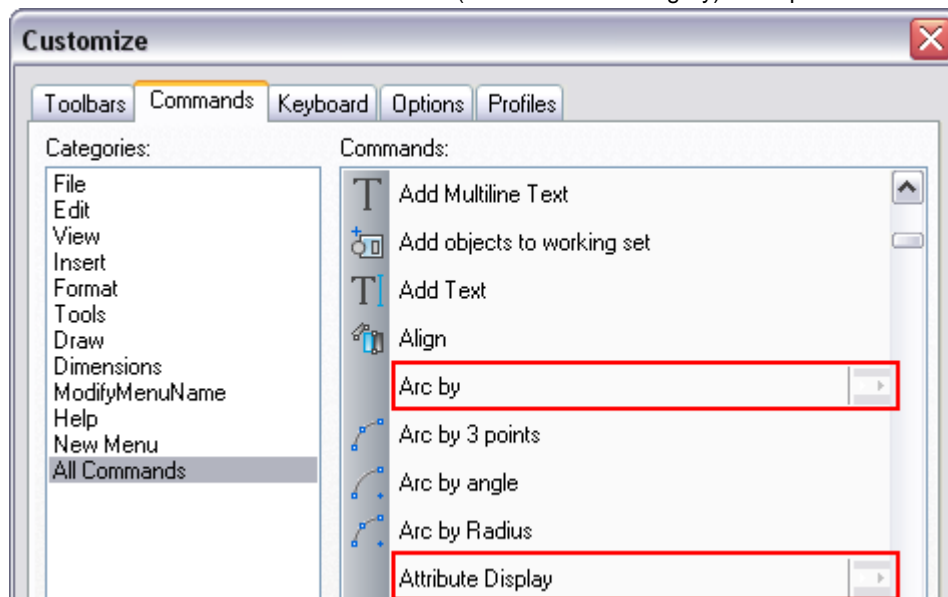
- In the opened **Button Appearance** dialogue box type the new name in the **Text** field and select OK.
- To add a command to a new submenu, click on the inserted menu.
- Select the **All commands** category of the *Categories* section.
 - From the *Commands* drag the required commands to the new menu item:



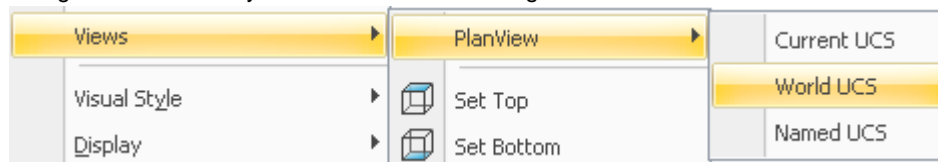
- Close the **Customize** dialogue box.
- A new menu will be created with the commands added



Note: Elements of the *Commands* section (*All commands* category) with special icons are used to create menus:



Using these elements you can create a cascading menu:



To delete a custom menu item:

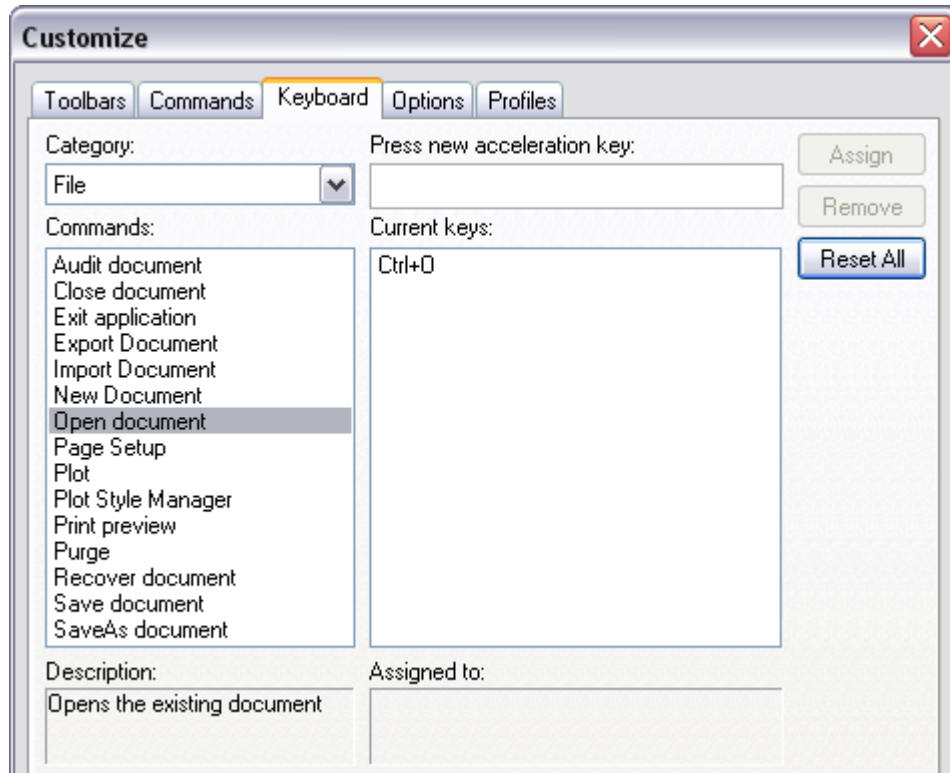
- Open the **Customize** dialogue box.
- Open the menu where you want to delete the item.
- Drag the item outside the menu.

Note: You can delete a menu item and all its submenus.


Keyboard tab

The **Keyboard** tab allows hotkeys to be defined for nanoCAD commands.

Hotkeys are a combination of symbol and command keys: **CTRL**, **SHIFT** and/or **ALT**. For example, **CTRL+O** is the same as **File> Open**, and opens the **Open File** dialogue box; **CTRL+S** – saves a document etc. Hotkey combinations used are shown in the right part of context menus and in the tooltips of toolbar buttons.




To assign hotkeys:

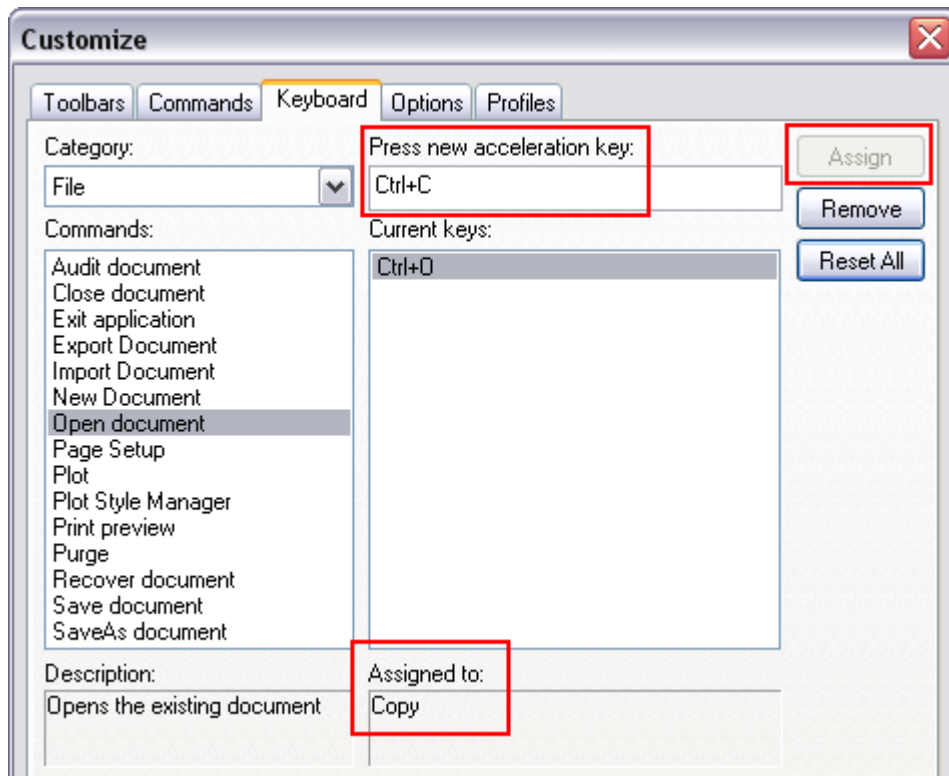
- Select a menu title to display the required list of commands in the *Command* section. The *All commands* category opens a list of all commands in the *Commands* section.
- Choose a command from the *Commands* list. The *Description* field shows the description of the chosen command.
- Place the cursor in the *Press new acceleration key* field and press the hotkeys for this command on the keyboard. The pressed combination is shown in the field.
- Select the  button.

If a combination is set for the first time, in the *Assigned to* section *Unassigned* is shown.

If hotkeys are already set for the selected command, they will be shown in the *Current keys* section.

You can assign several combinations for one command and remove unnecessary combinations by selecting them in the *Current keys* section and selecting the  button.

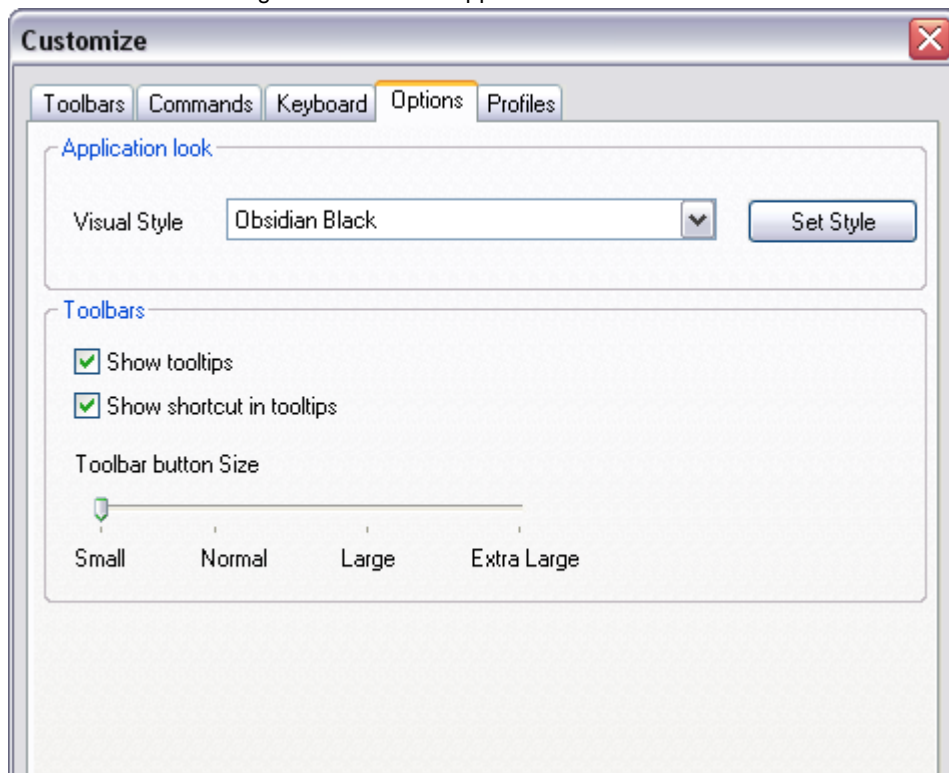
If chosen combination of keys for the selected command is already used for another command, the command having this hotkey combination is shown in the *Assigned to* section and the **Assign** button is blocked:



The **Reset All** button removes all custom hotkeys and restores their original state.

Options tab

The tab is used to change the view of the application and toolbars.



Parameters:

Application look

Visual Style: Selection of program style.
Available styles:
Windows default

Smart 1
Smart 2
Smart 3
Luna Blue
Obsidian Black
Aqua
Silver



The button to apply a selected visual style.

Toolbars

Show tooltips

Switches tooltips on/off when moving the cursor over a button on a toolbar.

Show shortcut in tooltips

Switches on/off shortcuts in tooltips.

Toolbar button size

Use the slider specify the button size on toolbars.

Profiles tab

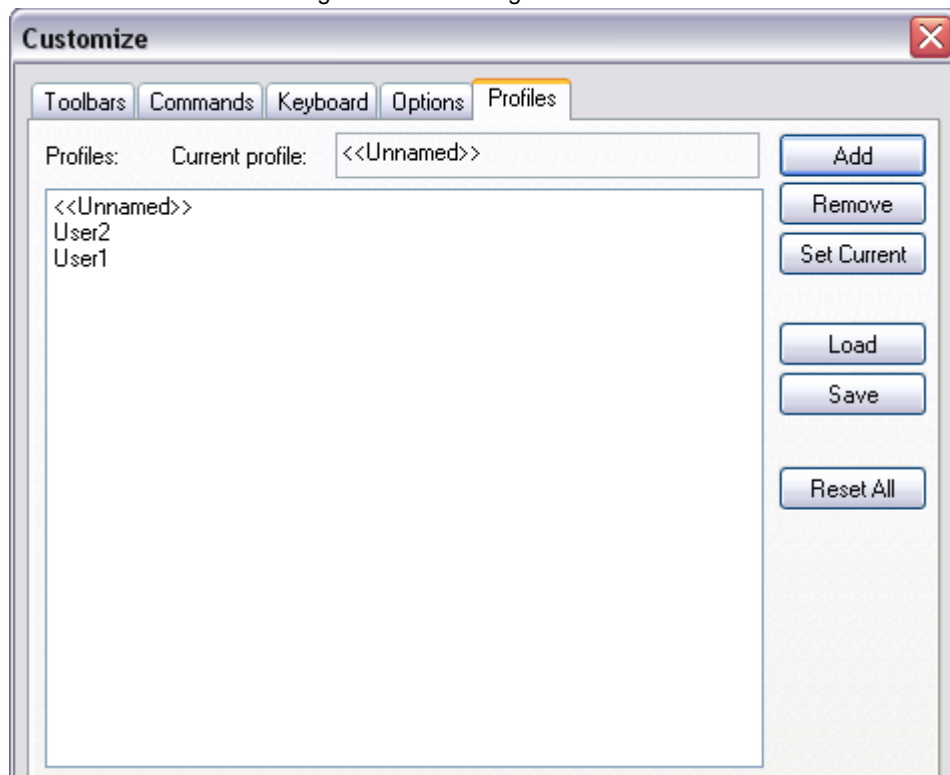
The tab is used to create *profiles*.

A *Profile* is a set of custom settings with a unique name for working environment parameters. By default, nanoCAD saves all current settings in <<Unnamed>> profile.

If several users, using one account to enter the system, work with the program, each of them can create a profile and load it for working. Another example of profile usage is for quickly switching between settings when working with different documents in the current session.

Only one profile can be current. When changing interface settings the current profile is changed automatically.

To use the custom settings of the working environment on other computers, a profile can be saved in a file with a *.wip extension. A WIP-file loaded on the other computer changes only the setting of the current profile, which were saved in this file. Other settings remain unchanged.




The list of all profiles is shown in the *Profiles* section.

The name of the current profile is shown in the *Current profile* field.

A new profile inherits the properties of the current profile.


To set a selected profile as the current one, select the button.

To delete a selected profile from the list select the button.

The  button deletes all profiles and recovers the standard nanoCAD profile.

Note: Before applying the  button, save user profiles or they will be deleted.


To create a new profile:

- In the **Customize** dialogue box, specify the required settings.
- In the **Profile** tab select the  button.
- In the **Profile** dialogue box specify a name for the new profile and select the required checkboxes:




- Select **OK**. The name of the created profile will be shown in the list of profiles.
- Select the **Close** button to close the dialogue box.

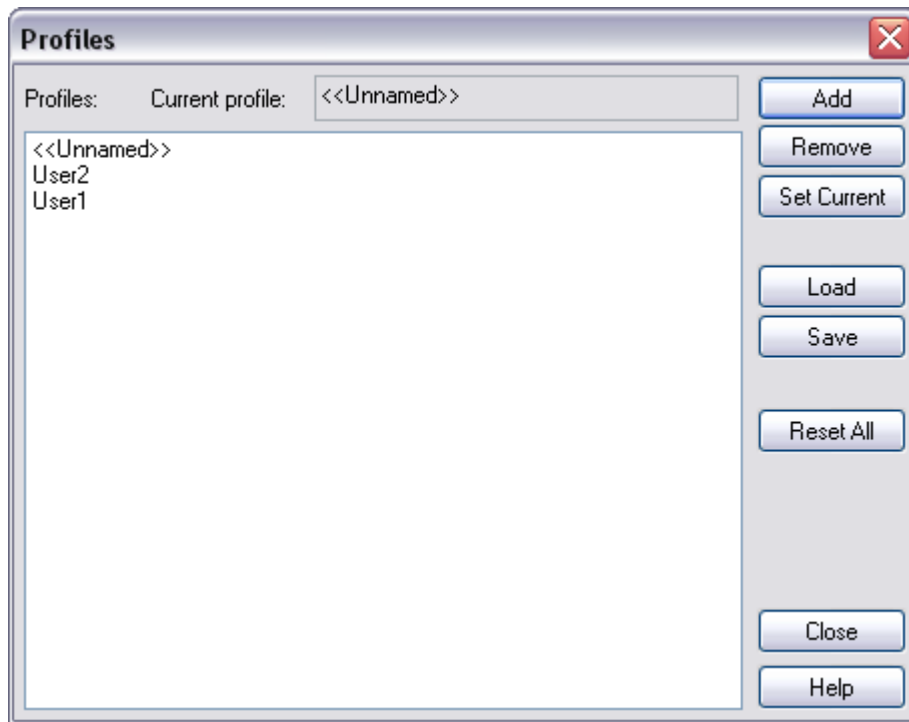
To save a profile:

- Select the profile to save.
- Select the  button.
- In the **Profile** dialogue box specify a name for the new profile and select the required checkboxes.
- Select **OK**.
- In the **Save As** dialogue box specify the name and the folder location to save the profile, after that select the **Save** button.
- Select the **Close** button to close the dialogue box.

To load a profile from other computer:

- Select the  button.
- In the **Open** dialogue box select the nanoCAD profile.
- Select the **Open** button. The file will be added to the profiles list and set as the current one.
- Select the **Close** button to close the dialogue box.

Note: You can open the profile editor from the **Tools** menu: **Customize>Profile Manager**.



The **Profiles** dialogue box is identical to **Profiles** tab of the **Customize** dialogue box.

Design settings

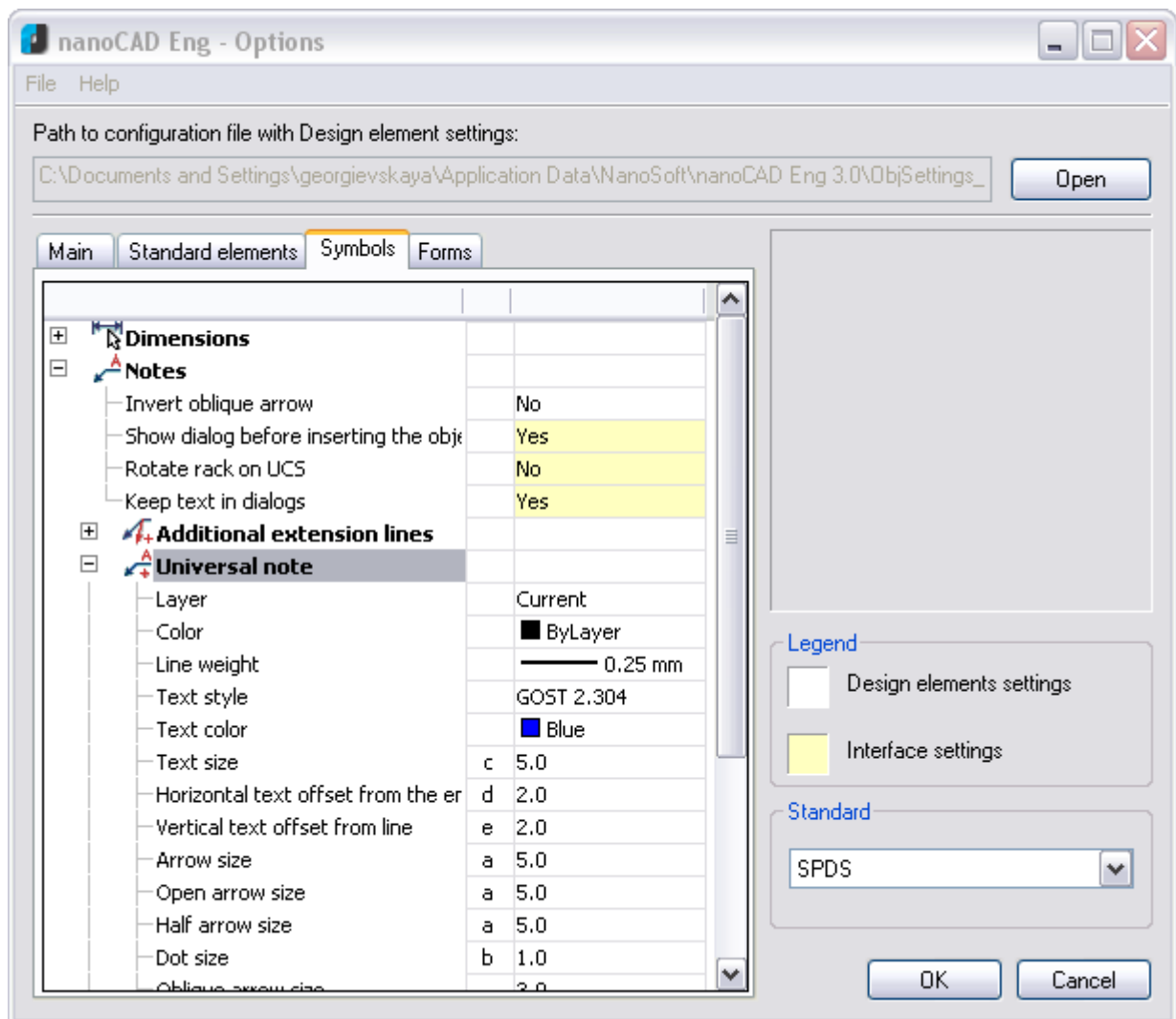


Menu: **Tools – Settings Parameters...**



Command line: **PARAMS**

Interface and parameters of annotations are set in the **Settings** dialogue box in nanoCAD:



The path to the settings file is shown in the top part of the dialogue box.

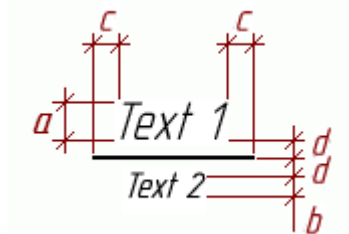
The **Open** button opens a standard dialogue box for the selection and loading of a new settings file.

The main window contains a tab with the settings:

- *Main options,*
- *Standard elements,*
- *Symbols,*
- *Forms.*

The parameters of the settings are shown in a tree form.

There is a graphic illustration of the adjusted parameters in the right part of the dialogue box:



Select a parameter from the drop-down list:

| | | | |
|------------------------------------|---|-----|---|
| Text size | a | 2.5 | ▼ |
| Text color | | 1.8 | |
| Horizontal text offset from the er | b | 2.5 | |
| Vertical text offset from line | c | 3.5 | |
| Arrow size | d | 5 | |
| Dot size | e | 7 | |
| | | 10 | |
| | | 14 | |
| | | 20 | |

or in the dialogue box by pressing the ... button:

| | | |
|------------|------|-------|
| Text color | Blue | ▼ ... |
|------------|------|-------|

The values of some parameters are entered from the keyboard:

| | | |
|-----------------|---|-----|
| Half arrow size | a | 5.0 |
| Dot size | b | 1.0 |

The drop-down list in the *Standard* section is used to change the standard:

| | |
|----------|---|
| Standard | |
| SPDS | ▼ |
| SPDS | |
| GOST | |

It is strongly recommended to match a *.dwt working template with nanoCAD settings.

What can be unified:

- Dimension and text styles. It is recommended to adjust nanoCAD according to *.dwt template settings.
- The global scale of linetypes.
- Layers' names and properties. By default many nanoCAD objects use a "current" layer for insertion. It is recommended to set layers' names from a *.dwt template.
- Layouts' names and their settings.
- Plot styles of objects, if a *.dwt template with named plot styles is used.

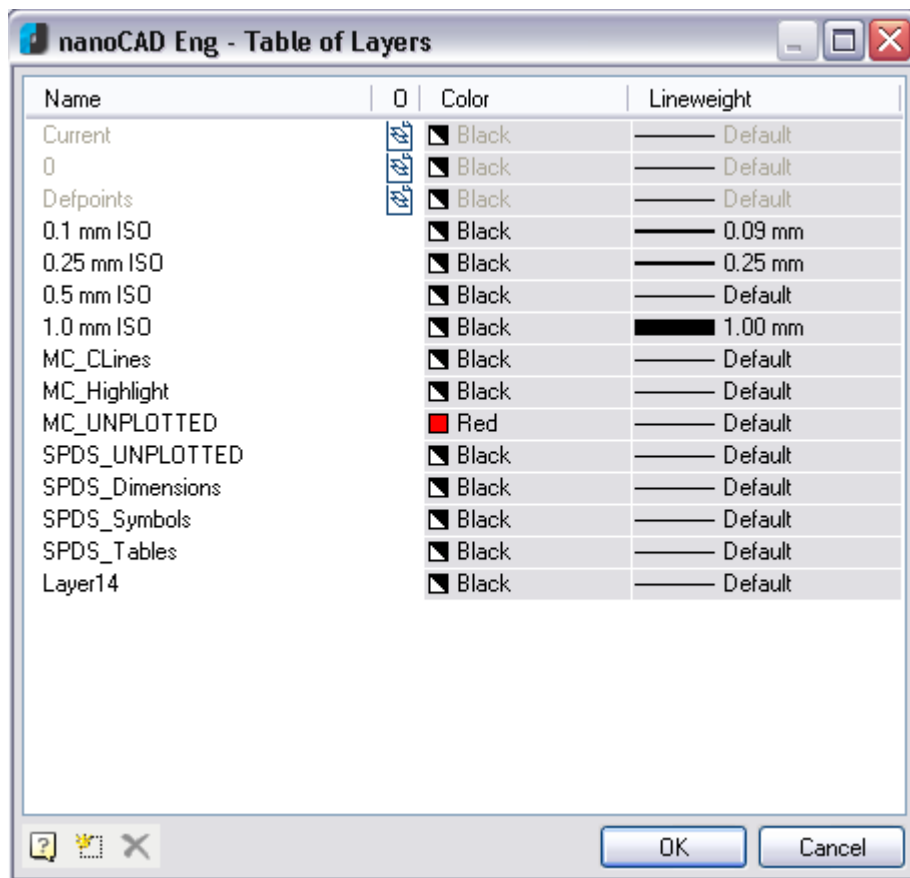
Settings of Layers' profiles


Layers' profiles are used to organise different divisions of the design work within one drawing file. Every user works with their own group of layers.


For example, you have to annotate a drawing with leaders set separately by the compliance supervisor and the technologist.

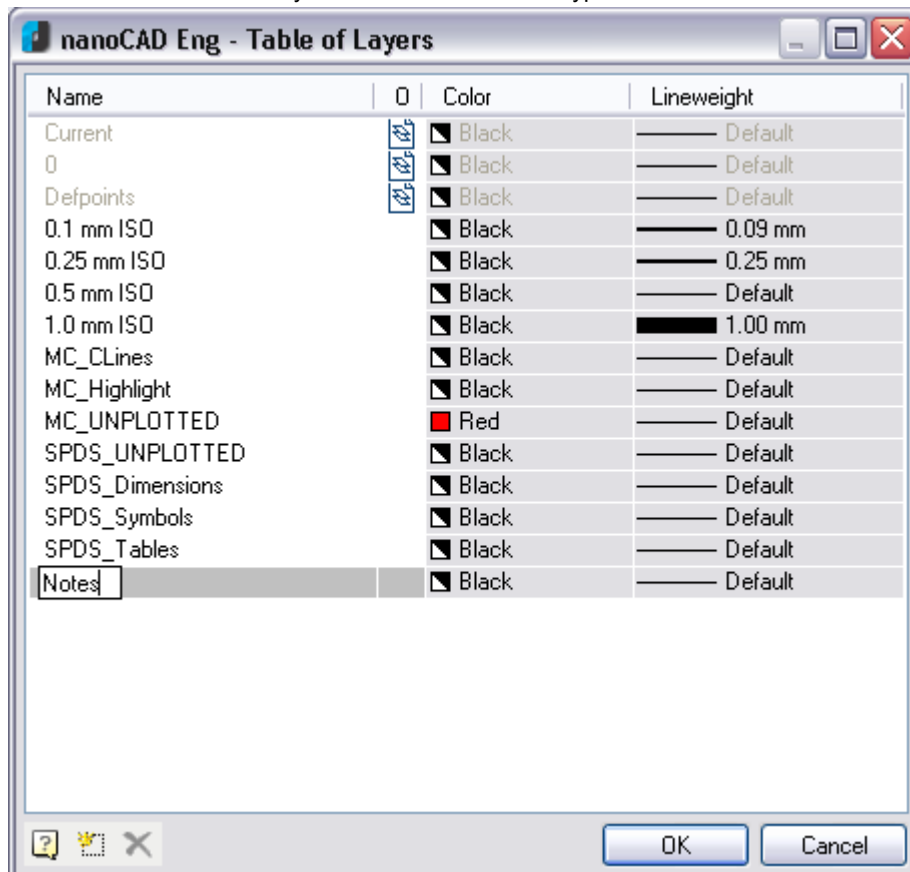
Do:

- Switch to the *Symbols* tab in the **Settings** dialog, select the *Leader note* in the *Notes* section, in the *Layer* field select the <Table of layers>:

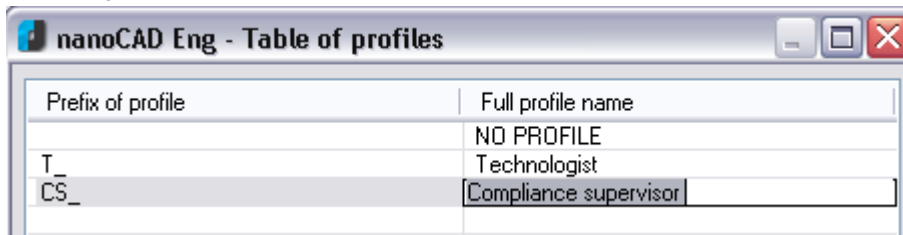


Note: The  button opens the **Table of layers** dialog too.

- Select the  **New layer** button to create a new layer. By default a new layer will have a *LayerN* name, where N – layer's number.
- Click on the created layer's name to rename it. Type a new name – *Notes*:



- Using the same method to create the CS_ profile's prefix :



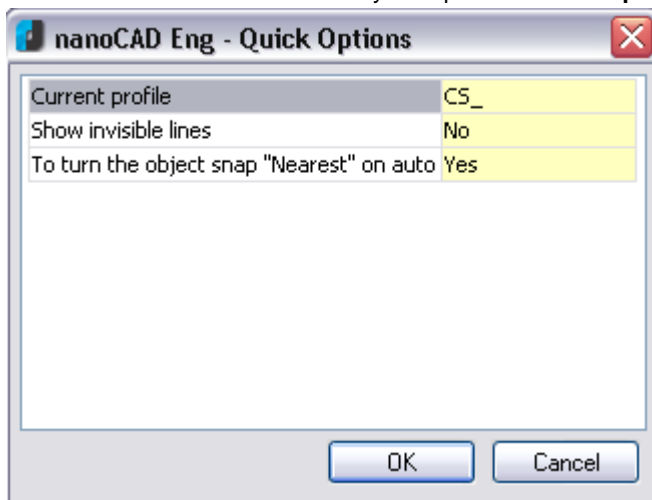
- Select **OK** to close the **Table of profiles** dialogue box.
- Select **OK** in the **Settings** dialogue box.
- In the **Save changes?** Dialogue box select the **Save to config file and current document** parameter and select **OK**:

As a result of the settings, according to which profile is set as current, you can create:

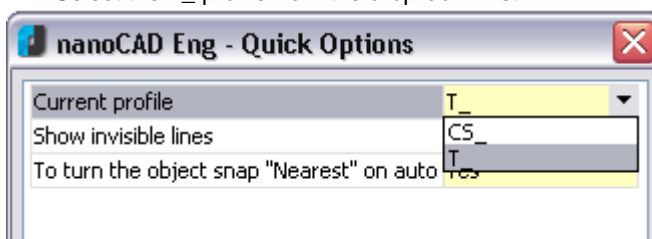
- *Technologist's annotation on the T_Notes;*
- *Compliance supervisor's annotation on the CS_Notes;*
- *Leader note on the Notes layer.*

To create technologist's annotation:

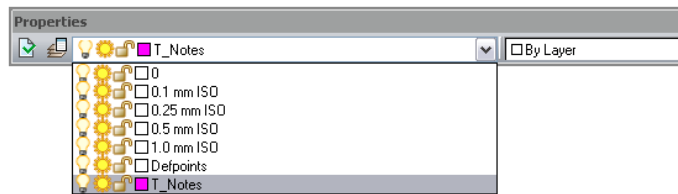
- Select **CTRL+SHIFT+Q** hotkeys to open the **Quick Options** dialogue box:



- Select the T_ profile from the drop-down list:



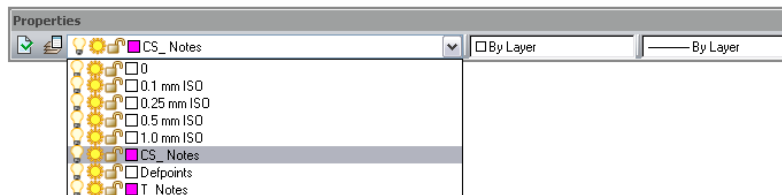
- Select **OK**.
- Create an annotation using the **Notes>Universal notes** command from the **Draw** menu. When the first annotation is being created, a new *T_Notes* layer is created on the *Notes* layer base:



Technologist's annotation

To create the compliance supervisor's annotation:

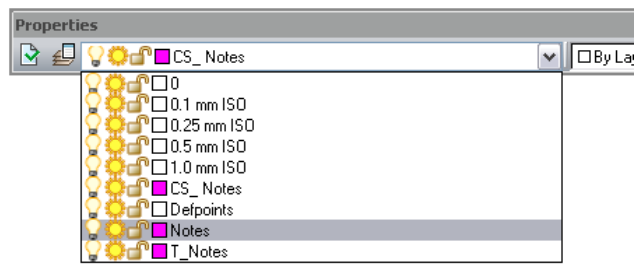
- Select **CTRL+SHIFT+Q** hotkeys to open the **Quick Options** dialogue box.
- Select the **CS_** profile from the drop-down list.
- Select **OK**.
- Create an annotation using the **Notes>Universal notes** command from the **Draw** menu. When the first annotation is being created, a new **CS_Notes** layer is created on the **Notes** layer base:



Compliance supervisor's annotation

To create a universal note:

- Open the **Settings** dialogue box.
- Open the **Table of profiles** dialogue box.
- Select **NO PROFILE**.
- Twice select **OK** to close the dialogue boxes.
- Create a universal note. When the first annotation is being created, a new **Notes** layer is created:



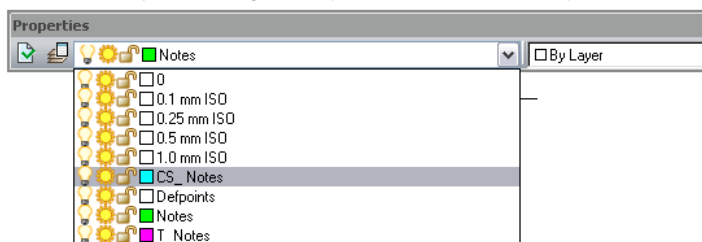
Universal note



To enhance visualisation, you can set different colours for the *Notes*, *CS_Notes* and *T_Notes* layers.

Do:

- Switch to the *Symbols* tab in the **Settings** dialogue box and in the *Notes – Universal note* section select *By layer* in the *Color* and *Text color* fields.
- Select **OK** in the **Settings** dialogue box.
- In the **Save changes?** Dialogue box select the **Save to config file and current document** parameter and select **OK**.
- In the **Layers** dialog specify the colours for the layers.



Compliance supervisor's annotation



Universal note



Technologist's annotation



Note: Hotkeys for the *Quick options* dialogue box are specified in the *Hot keys* section in the *Main options* tab of the **Settings** dialogue box:

| Hot keys | | |
|-----------------------------|--|------------------|
| Call dialog <Quick Options> | | CTRL + SHIFT + Q |
| Call Notify Window | | CTRL + SHIFT + W |

Saving and transferring settings to another computer

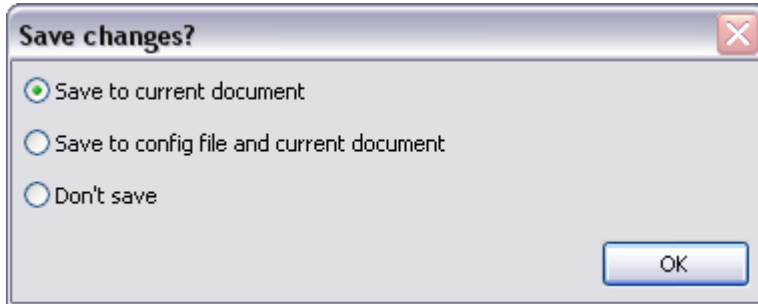
There are two types of settings in nanoCAD – *interface settings* (highlighted yellow) and *design elements settings*:



Interface settings are set by the user and influence the application's work.

Design elements settings are written in a separate file. These settings are read when a new document is created and they define how the drawing created with nanoCAD tools will look.

If the design elements settings are changed, the following dialogue box is shown after selecting the **OK** button:



Parameters:

- | | |
|---|---|
| Save to current document | Saves the design elements settings in the current document. When a new document is created, the settings will be the same as the settings set in a configuration file; the setting will only be applied to the current document. |
| Save to config file and current document | The new design elements settings will be applied to all new documents by default and to the newly created design elements in the current document. To save settings in this file you must have the rights to write to the settings folder. |
| Don't save | Refuse to save the changes to the design element settings. |

This separate saving of *design elements settings* helps to standardise the drawing preparations procedures in an organization: you just have to set a common settings file on a network. Defining who has rights to write to the common file can be done using the operating system tools.

Settings files for users

spot.icf - nanoCAD interface settings file

ObjSettings_spot.cfg – settings file for nanoCAD design elements

are stored in the folder *C:\Documents and Settings\User_name\Application Data\Nanosoft\nanoCAD*.

To transfer settings to another computer, you should copy them into the corresponding folders and specify the path to them in the **Settings** dialog.

Drawing units



Menu: **Format – Units...**



Command line: **UNITS, UN**

The command opens the **Drawing Units** dialogue box, where you can define the format and accuracy of the linear and angle units.

By default, the base direction to measure angles is to the right of the initial point (East). Angles should be measured in a counterclockwise direction to give positive numbers.

Drawing Units

Length
 Type: **Decimal**
 Precision: **0.0000**
☐ Display in "input" format

Angle
 Type: **Decimal Degrees**
 Precision: **0**
 Base Angle: **East**
☐ Clockwise

Insertion scale
 Units to scale inserted content:
Millimeters

Sample output:
 125.5000,254.0039,0.0000
 5.5000<45

Lighting
 Units for specifying the intensity of light:

OK Cancel Help

Parameters:

Linear

Type: Current format of linear units.
 Available formats in the drop-down list:

Architectural
Decimal
Fractional
Engineering
Scientific

Precision: Accuracy of current linear units.

Angle

Type: Current format of angle units.
 Available formats in the drop-down list:

Deg/Min/SEC
Grads
Decimal degrees
Radians
Surveyor's units

Precision: Accuracy of current angle units.

Base Angle: Direction of base angle.
 Available formats in the drop-down list:

East
North
West
South

Pick angle – by specifying two points in the graphic area.

Clockwise To change positive direction to measure angles to clockwise.

Insertion scale

Units to scale inserted content: Units to measure blocks and external references, inserted in a document. If a block or inserted document was created with units which do not coincide with the units set in this section, the block or inserted document will be scaled to the specified units. Scale is defined by the ratio of units in an inserted document and a current document.
If the **Undefined** option is selected, the insertion is performed without scaling.

Sample output Preview of current linear and angle units set in the dialogue box.

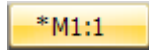
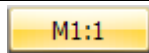
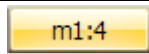
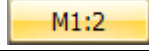
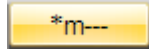
Symbol scale and measurement scale

There are two types of scale: symbol scale and measurement scale. The values for any of these scales can be set for any object, group of objects or document. New objects inherit the scale values from the document.

Note: The list of scale values corresponds to the Unified system for design documentation GOST 2.302-68 "Scales".

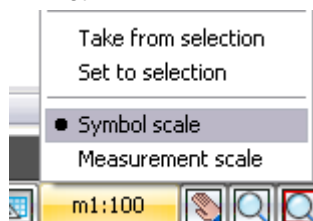
Specifying scale

To set scale, use the scale button on the status bar.

| | |
|---|---|
|  | This button displays the scale of the selected objects. If objects are selected, an asterisk symbol is displayed before the scale value (scale 1:1 means that 1mm of object equals to 1 unit of drawing). |
|  | If there are no objects selected, the button displays the document scale but the asterisk is absent. |
|   | If the symbol scale value is displayed, the M character is in lower case. If the measurement scale value is displayed, the M character is in upper case |
|  | If the selected object does not support the selected scale, or selected objects have different scales, the scale value is not displayed. |

To specify scale:

- Select objects. To change the scale of all objects in the drawing, select all objects. If no objects are selected, a new scale will be set only for new objects that are created.
- Select the scale button in the status bar.
- Select the scale type in the menu (**Symbol scale** or **Measurement scale**) and specify the scale value from the list.

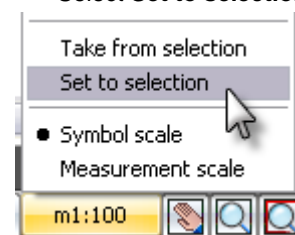


You can set the scale for the selected objects or take the scale value from the selected object.

To apply the scale to the selected objects or to take the scale from the selected object:

To set the current scale of the document to selected objects:

- Select the scale button.
- In the menu that opens, select the scale type (**Symbol scale** or **Measurement scale**).
- Select **Set to selection**.



- Select the objects on the drawing whose scale you want to change.

To copy the scale from a selected object:

- Select scale button.
- In the menu that opens, select the scale type (**Symbol scale** or **Measurement scale**).
- Select **Take from selection**.
- Select the object in the drawing whose scale you want to copy.

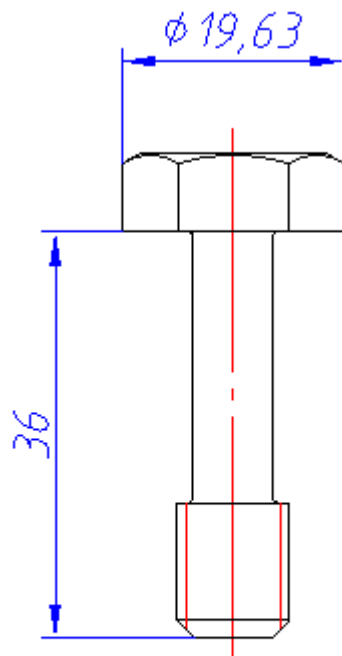
Symbol scale

Symbol scale is a settings parameter which decreases or increases the annotation objects in the drawing. This scale type is used for adjusting the display of annotation if it is too small or too big; for example, to change the size of an object's arrows and text. This scale has no influence on the size of other objects.

Changing symbol scale affects:

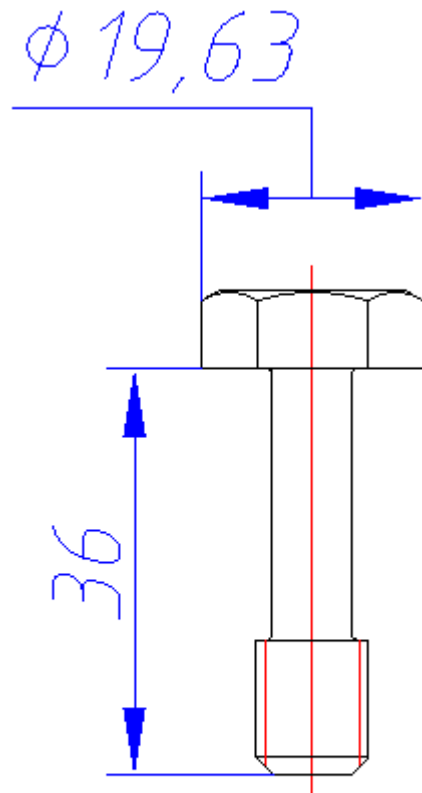
- *Annotation objects.*
- *Linetypes with gaps and text.*
- *Dimensions and special text.*

Symbol scale for objects is 1:1



M1:1

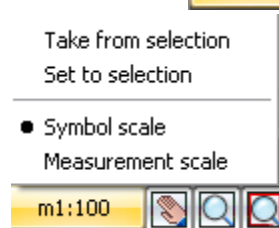
Symbol scale for objects is 1:2



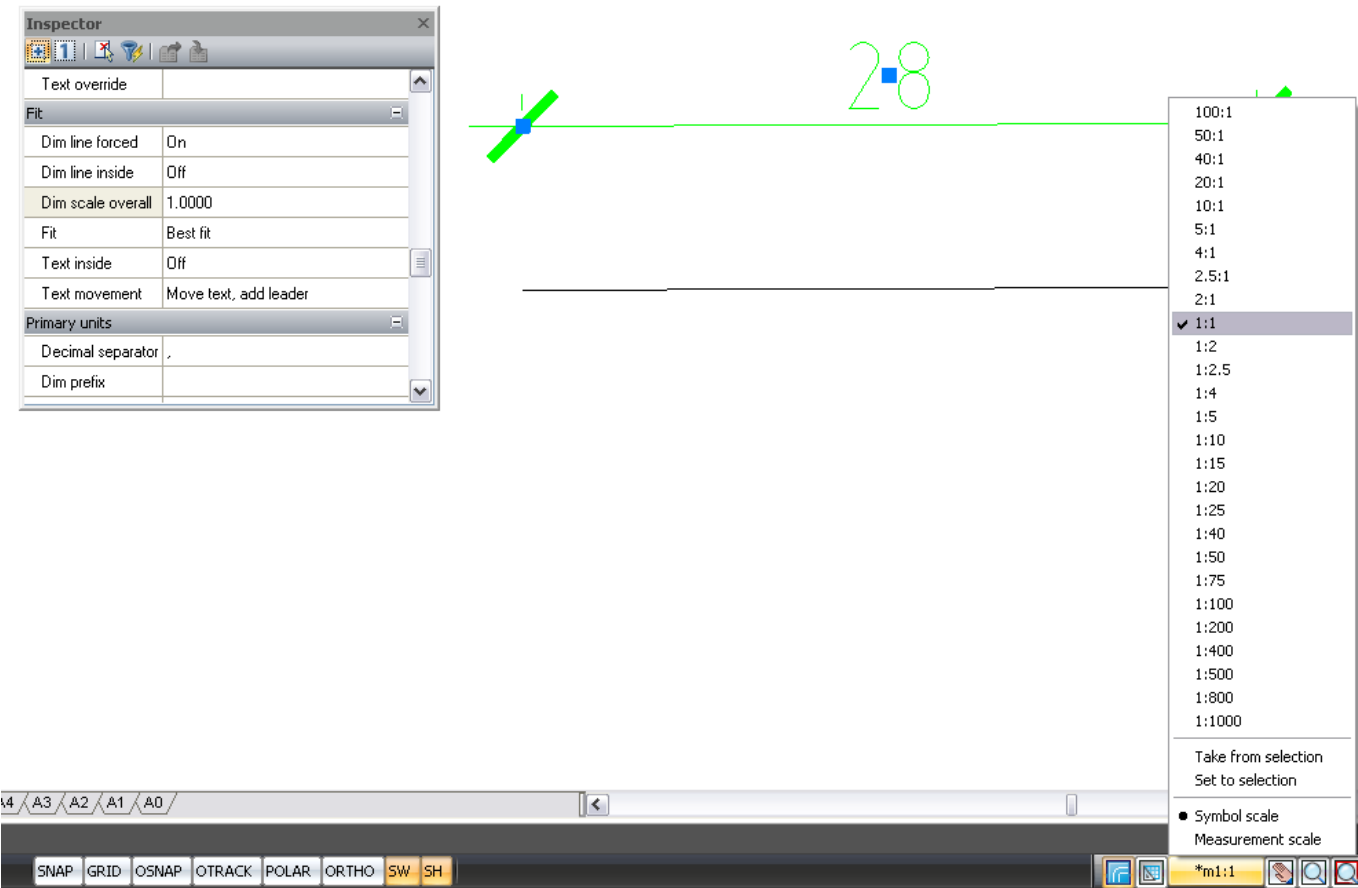
M1:2

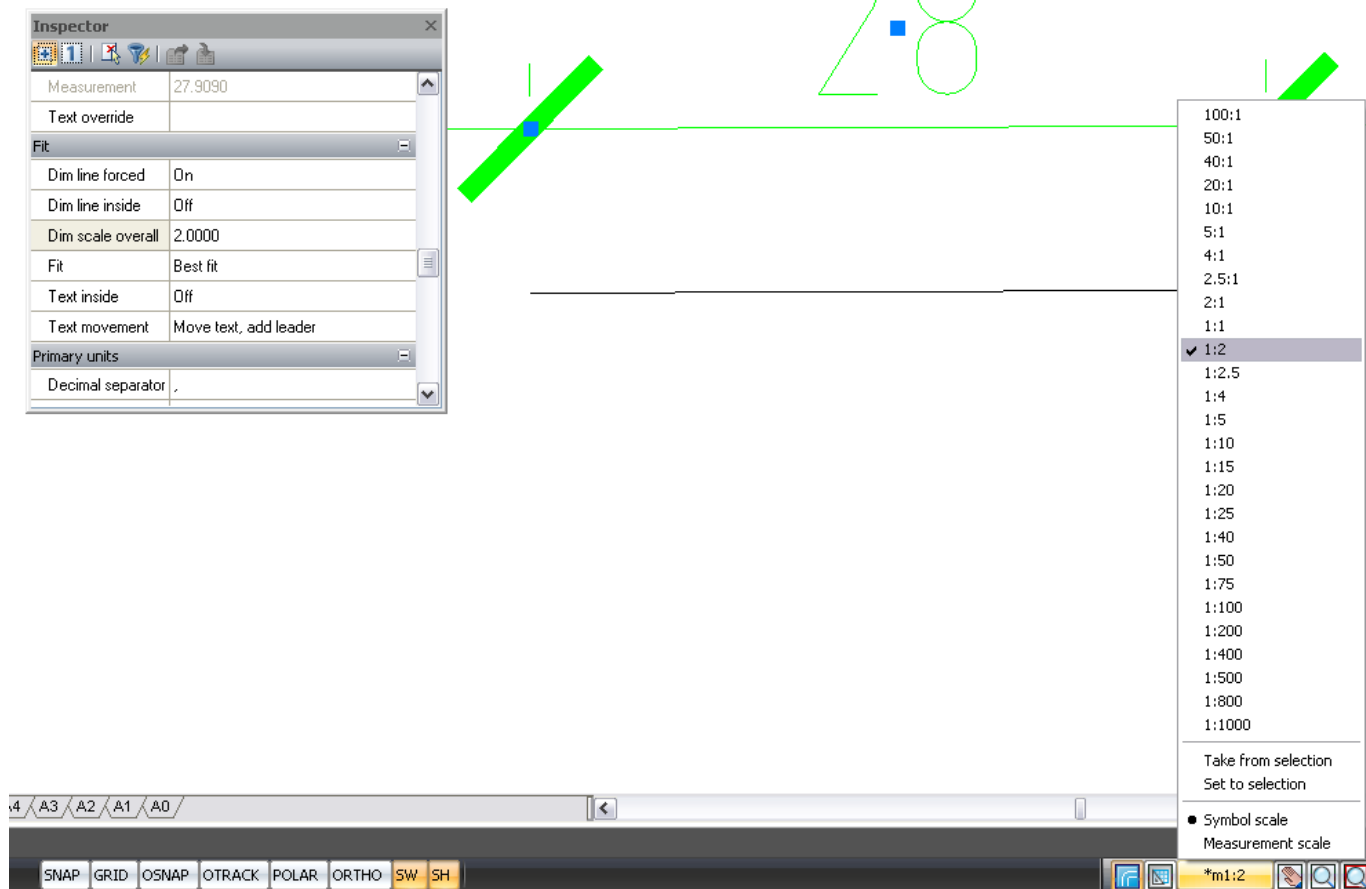
To set symbol scale:

To set and change scales, use the scale button in the status bar. If the value is set to symbol scale, the **m** character is in lower case: **m1:4**



Example of changing the symbol scale of a dimension object:





Measurement scale

This scale influences objects, but the size of the annotation of the objects stays unchanged. Linear dimensions are also scaled.

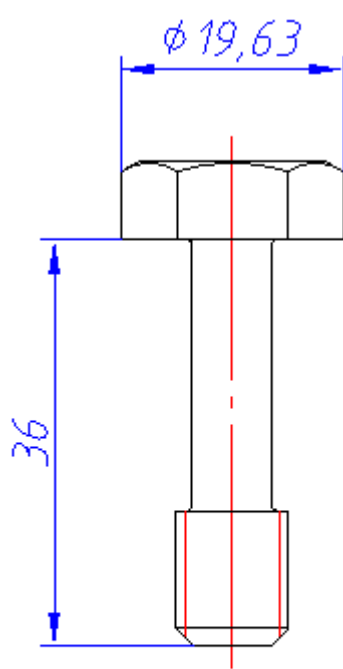
Measurement scale can be set for the following objects:

- *dimension objects*; measurement scale influences the dimension text.
- *special object from vertical applications* can have influences on the measurement scale of an object size in the drawing.

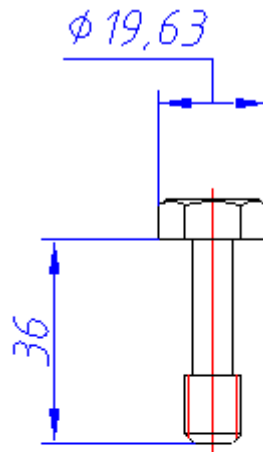
Measurement scale for all objects is
1:1

Measurement scale for all objects is
1: 2

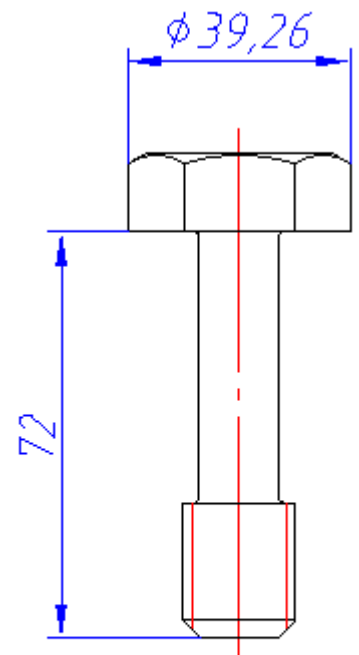
Measurement scale for dimension
objects is 1:2 and for object is 1:1



M1:1



M1:2



M1:2

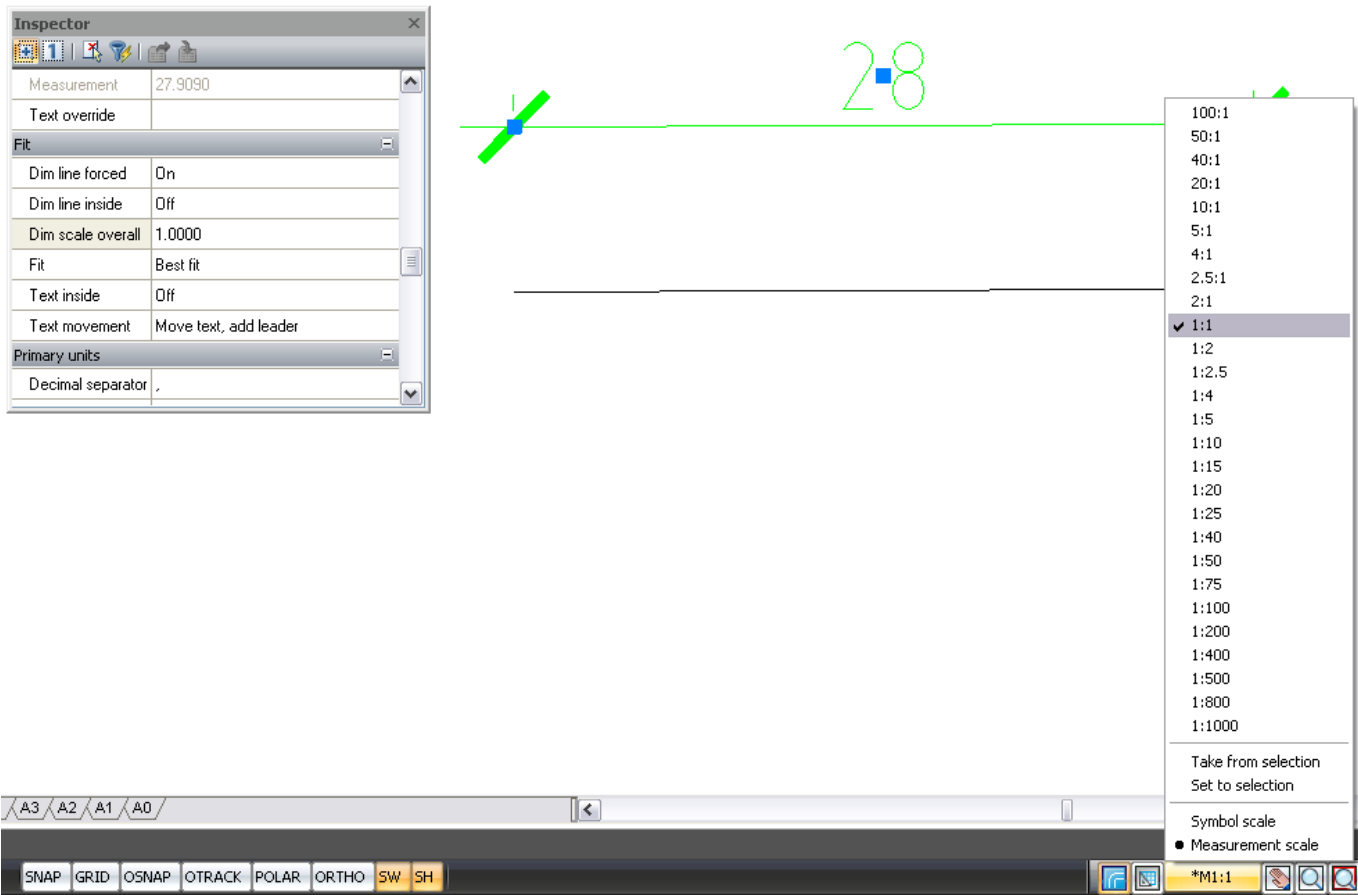
Use this scale if the drawing has a scale that differs from 1:1 scale.

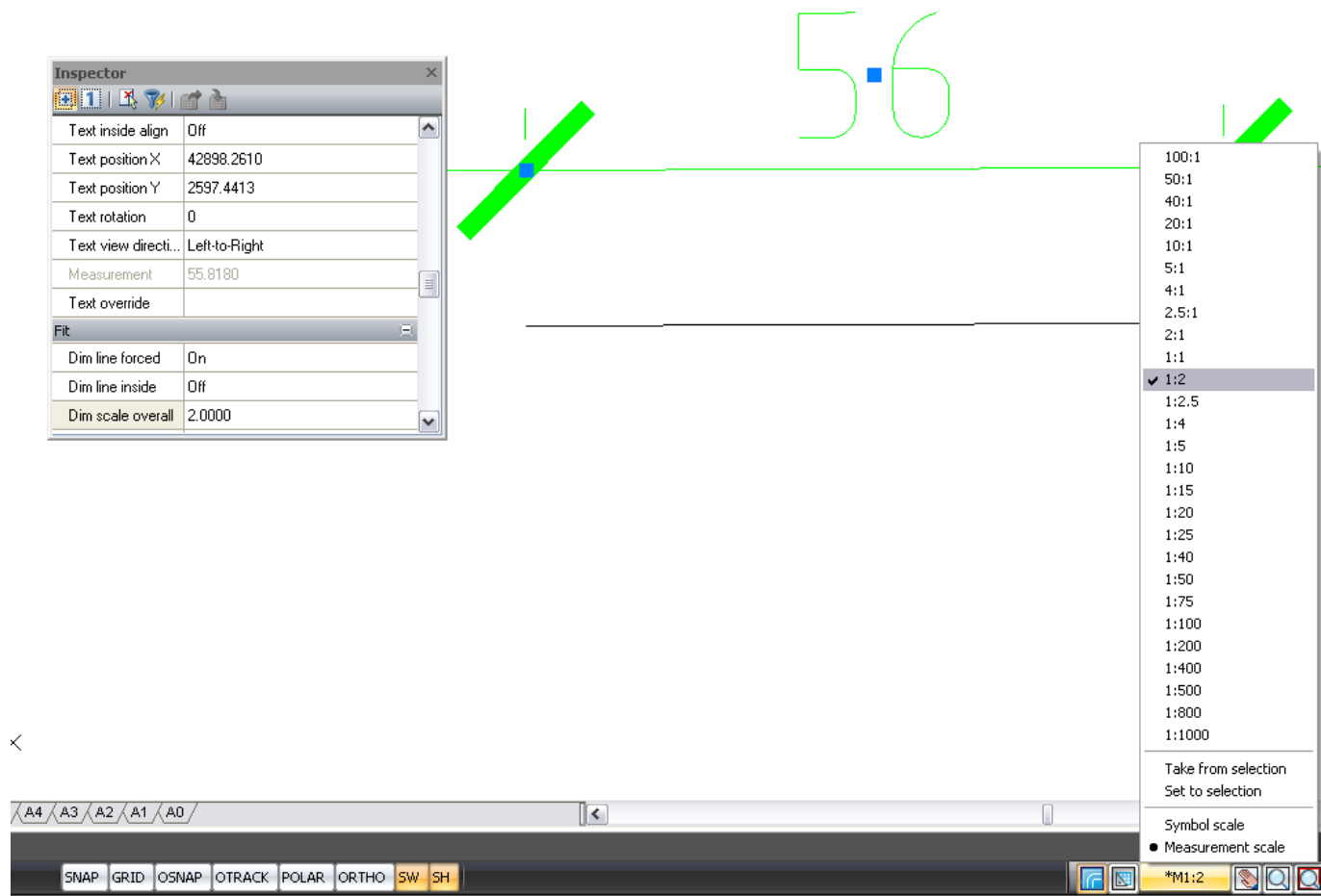
To set measurement scale:

To set and change scales, use the scale button in the status bar. If the measurement scale value is set, the **M** character is in upper case:

M1:1

Example of changing the measurement scale:



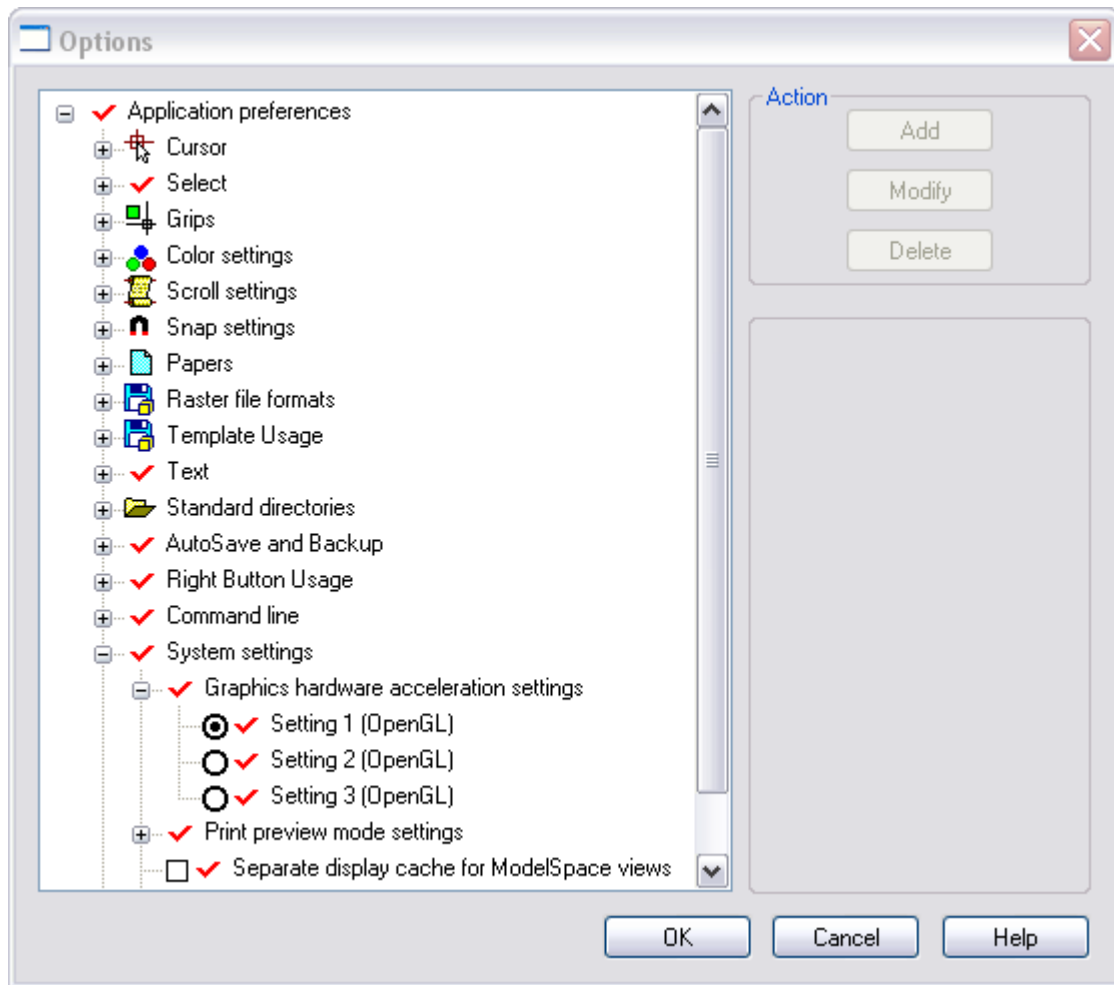


Solving problems

1. The program graphic subsystem might work slowly or become unstable with some video adapters and driver versions. For example, the cursor in the empty drawing moves unevenly or isn't shown.

In this case, try the following:

- Save all changes, close all drawings that you are editing.
- Open the **Options** dialogue box (**Tools> Options**).
- In the dialogue box that opens, select **System settings> Graphics hardware acceleration settings**:



- Try switching between settings 1, 2 and 3. After every attempt, press **OK** in the dialogue box, open a new document and check the cursor speed. The program saves the selected setting until complete reinstallation.
- 2. If an image is a large size (the values of the cursor's coordinates are 3000000 or higher), you sometimes need to switch on the following parameters to correct the graphics display:

- ☒ **Separate display cache for ModelSpace views**
- ☒ **Separate display cache for PaperSpace views**

Coordinate systems

The position of every drawing point is defined by its coordinates. In the command prompt for a point position you can specify it on the screen or type the coordinates in the command line.

Specifying coordinates

Coordinates can be specified in the *Cartesian coordinate system* and *polar coordinate system*. *Cartesian* and *polar* coordinates can be *relative* and *absolute*.

Cartesian coordinates

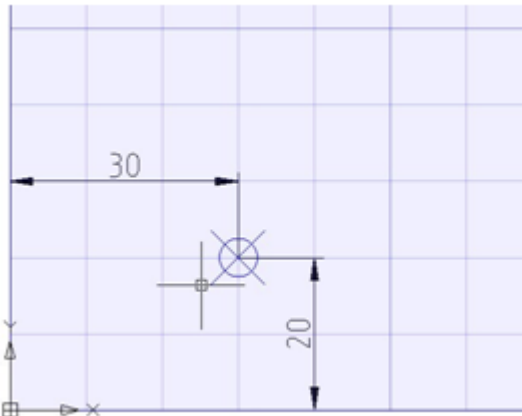
Cartesian coordinate system is defined by three perpendicular axes: X, Y and Z.

The *origin* of the coordinate system is the point of intersection of the three axes and has the coordinates: (0,0,0).

If you work in the plane, the Z coordinate is always 0, you have to specify X and Y. The X value is specified horizontally and Y value vertically. Positive coordinates are set to the right and above the origin, and negative to the left and below.

When working in three dimensions, you have to set the Z coordinates. By default, the Z axis is set perpendicular to the XY plane from the viewpoint of the observer. The positive coordinates are set above the plane and negative below.

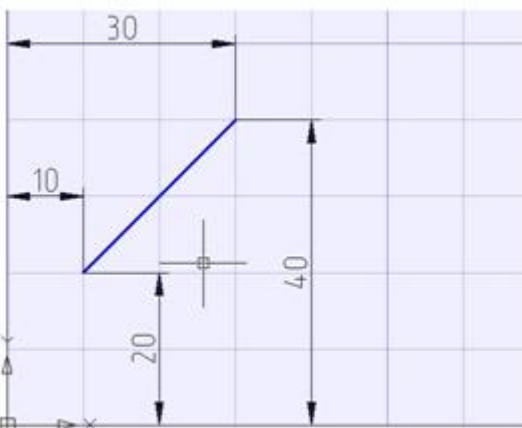
For example, the coordinates (30,20) specify the point set 30 units in on the X axis and 20 units in on the Y axis.



Absolute coordinates are specified from the origin of the coordinate system. Absolute are used if the precise X and Y coordinates are known values.

Example: The point with coordinates X=10 and Y=20 is the start and the point with coordinates X=30, Y=40 is the end of the line. To create the line, enter in the command line:

```
Command:          LINE
Specify first point: 10,20
Specify next point: 30,40
```



Relative coordinates are used when the distance from the previous point is a known value.

To specify relative coordinates, enter the @ symbol before their values. Values specified after the @ symbol are distances along the X and Y axes from the previous point.

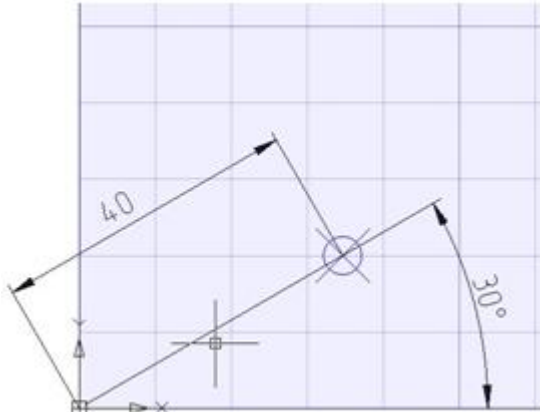
To create a line from the previous example using the *relative coordinates* enter in the command line:

```
Command:      LINE
Specify first   10,20
point:
Specify next    @20,20
point:
```

Polar coordinates

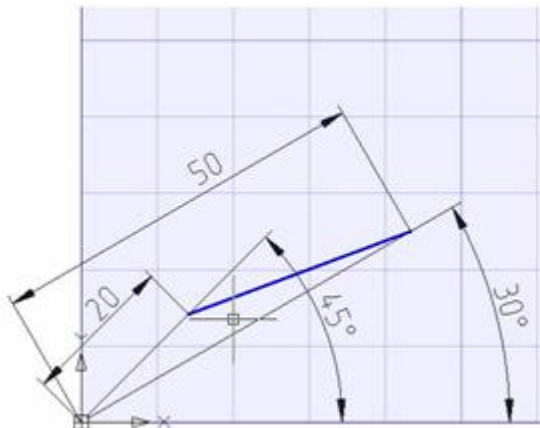
In the *Polar coordinate system*, the *absolute coordinates* of a point are set by the distance from the origin and an angle between the polar axis and a line lying through the point and origin. The angle is set in degrees counterclockwise.

For example, the coordinates 40<30 specify a point on the plane, setting the distance as 40 units from the origin and with a 30° angle from the X axis.



Example: The start point of the line is set as 20 units from the origin and at a 45° angle from the X axis; the end point is set as 50 units from the origin and 30° from the X axis. Enter in the command line:

```
Command:      LINE
Specify first   20<45
point:
Specify next    50<30
point:
```

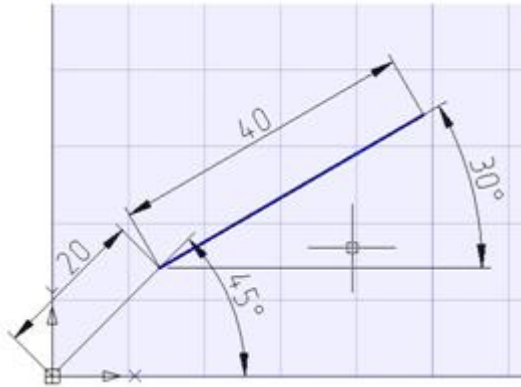


In the *relative polar coordinates* the distance to the point is set not from the origin, but from the previous point. The angle is specified from the polar axis to the line connecting the previous point and the defining point.

The @ symbol is used to specify *relative coordinates*.

Example: The start point of line is set as 20 units from the origin and at a 45° angle from the X axis; the end point is set as 40 units from the previous point and 30° from the polar axis. Enter in the command line:

```
Command:      LINE
Specify first   20<45
point:
Specify next    @40<30
point:
```



Specifying points with “direction – distance”

The direct distance is set instead of entering coordinates; it is very useful for quickly entering the lengths of lines.

Using direct distance specifying, move the cursor in the desired direction and enter the length value in the command line at the command prompt: `Specify next point:.` If ortho mode is switched on, it is very useful for drawing perpendicular lines.

This method can be used in all commands, except commands where just a value needed, for example **Array**, **Divide** etc.

User coordinate system

nanoCAD uses two coordinate systems: *world coordinate system* and *user coordinate system*.

Only one coordinate system is active at any time; this is called *current*.

World coordinate system is a base coordinate system and cannot be redefined (X axis is set horizontally, Y axis vertically and Z is perpendicular to XY plane). The main difference of the *world coordinate system* from the *user coordinate system* is its rigidity and that it can be the only one (for every model space and layout).

The usage of the *user coordinate system* has almost no restrictions; it can be placed at any point of the space and with any angle to the *world coordinate system*. UCS can be moved and rotated to specify points on the three dimensional and rotated views. Node points and base directions, defined by the **SNAP**, **GRID** and **ORTHO** modes are rotated with UCS.

Changing UCS position

Commands for changing the UCS position set a new coordinate system, the so-called *current coordinate system*.

The *current coordinate system* inherits the parameters of the previous coordinate system, only the required values are changed.

World coordinate system

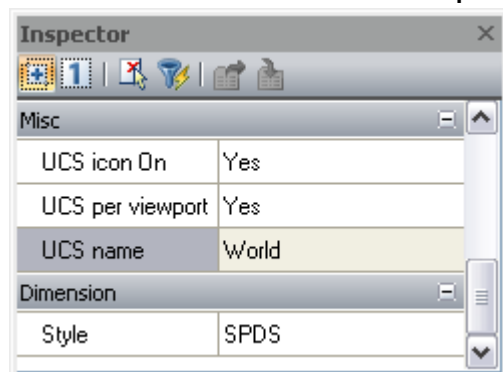


Menu: **Tools – Coordinate system > World UCS**

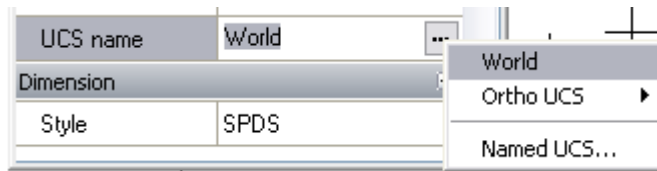
The command sets the parameters of the world coordinate system for the current user coordinate system.

To set a coordinate system quickly:

- Click in the *UCS name* field in the **Inspector** window:



- Click the button, specify *World*.



Aligning UCS to an object



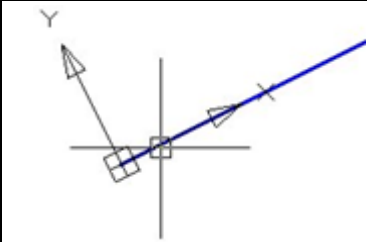
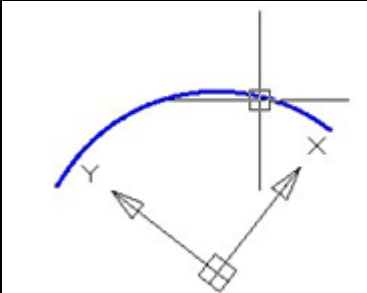
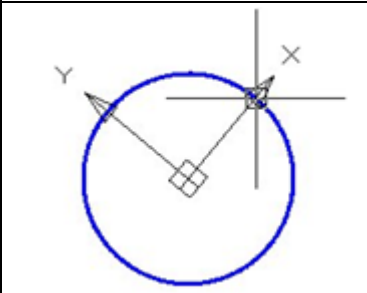
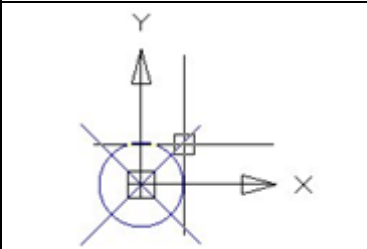
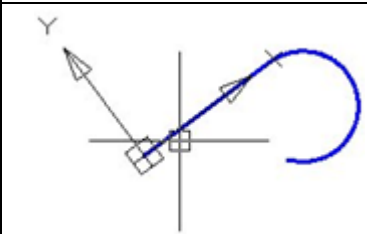
Menu: **Tools – Coordinate system >  Object**

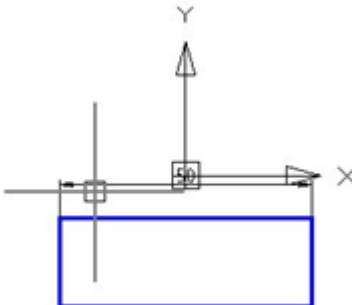
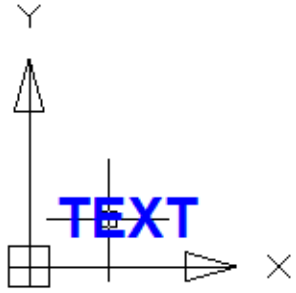


Command line: **SETUCSBYOBJECT**

The command specifies the new position of the coordinate system origin for the current UCS; the direction of the axes is specified according to the geometry of the current object. The extrusion direction of the selected object specifies the positive direction of the Z axis for the new UCS.

Rules to create UCS, aligned to an object:

| | | |
|----------|---|--|
| Line |  | The origin of the new UCS is set at the line end which is closest to the selection point. The X axis is used to place the line in the XZ plane. The Y coordinate of the second end of the line is zero in the new UCS. |
| Arc |  | The origin of the new UCS is the centre of the arc . The X axis is set at the end of the arc which is closest to the selection point. |
| Circle |  | The origin of the new UCS is the centre of the circle . The X axis is set at the selection point. |
| Point |  | The origin of the new UCS is the selection point. |
| Polyline |  | The origin of the new UCS is the start point of the polyline. The X axis is set at the start point and the nearest polyline vertex. |

| | | |
|--|---|--|
| Dimension |  | The origin of the new UCS is in the middle of the dimension text. The new X axis is set parallel to the X axis of the UCS used to specify the dimension. |
| Text block insertion, attribute definition |  | The origin of the new UCS is the insertion point of the object and the direction of the X axis is set by the angle of object rotation around the direction of extrusion. The object used to set the new UCS has a zero rotation angle in this UCS. |

Command prompt:

Select object to align UCS: Select an object.

New origin for coordinates



Menu: **Tools – Coordinate system > Point**



Command line: **SETUCSBYPOINT**

The command specifies the new origin for the current UCS at the specified point.

Command prompt:

Specify origin of UCS [] Enter the coordinates for the UCS origin or specify them on the screen.
<World>:

New origin for coordinates and rotation angle



Menu: **Tools – Coordinate system > Point and angle**



Command line: **SETUCSBYPOINTANGLE**

The command specifies a new origin for the current UCS and the rotation angle of its axes.

Command prompt:

Specify origin of UCS [] Enter the coordinates for the UCS origin or specify them on the screen.
<World>:

Specify point on X axis or Specify the rotation angle on the screen or enter in the command line.
<Accept>:

Changing the UCS position from the command line



Command line: **UCS**

Command options:

Named

Saves and restores frequently used UCS orientations by name.

Restore

Replaces the current UCS with one from the list of named UCSs.

Save

Saves the current UCS with a specified name.

Delete

Deletes the UCS from the list of named UCSs.

?

Shows the list of named UCSs.

Object

Sets the origin and the direction of the UCS axes according to the geometry of the existing

object.

View

Specifies a new coordinate system within the XY plane, set perpendicular to the direction of sight and parallel to the viewport's plane. The position of the origin is not changed. The X axis is set horizontally and the Y axis is set vertically.

World

Matches the current UCS to the World coordinate system (restores the World coordinate system).

X/Y/Z

Rotates the current UCS around the selected axis.

ZAxis

Specifies the UCS using the positive direction of the Z axis: the origin is placed at the first specified point; the positive direction of the Z axis is set through the second specified point.

Command prompts:

Specify origin of UCS or
[Named/Object/View/World/X/Y/Z/ZAxis]
<World>:

Specify point on X-axis or <Accept>:
Specify point on XY plane or <Accept>:

Select UCS or option.

Specify a point on the positive ray of the X axis.

Specify a point on the positive ray of the Y axis in the XY plane.

UCS icon



Menu: **View – Display > UCS icon**

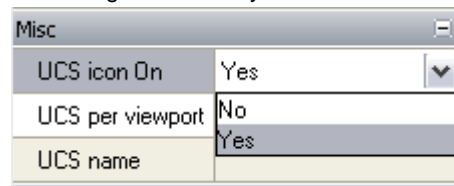


Command line: **UCSICON**

Manages the visibility and position of the UCS icon.

There are **On**, **Off** and **Origin** commands available in the **View – Display > UCS icon**.

To manage the visibility of the UCS icon use the **Inspector** dialogue box:



The full list of options available to manage the UCS icon is available in the command line.

Command options:

ON Shows the UCS icon.

OFF Hides the UCS icon.

All Parameters of visibility for all viewports. If this option is not used, the parameters of the UCS icon are specified only for the current viewport.

On Shows the UCS icon for all active viewports.

Off Hides the UCS icon for all active viewports.

Noorigin Shows the UCS icon in the left corner of the screen regardless of the origin for all active viewports.

Origin Shows the UCS icon at the origin (0,0,0) of the current UCS for all active viewports. If the origin of the UCS is outside the visible part of a drawing, the icon is shown in the left corner of the screen.

Noorigin Shows the UCS icon in the left corner of the screen regardless to origin of UCS.

Origin Shows the UCS icon at the origin (0,0,0) of the current UCS. If the origin of the UCS is outside the visible part of a drawing, the icon is shown in the left corner of the screen.

Command prompt:

Entry an option or [**On**/**Off**/**All**/**Noorigin**/**Origin**] <"On">: *Select the required option.*

Named UCS



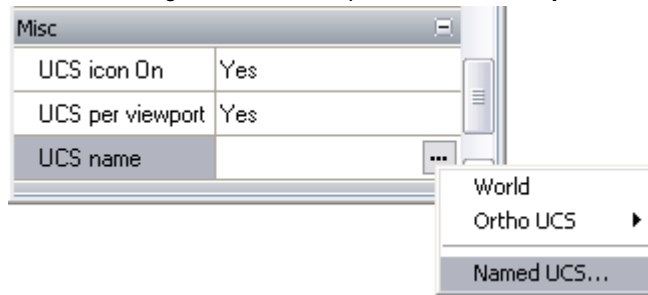
Menu: **Tools –  Named UCS...**



Command line: **UCSMAN, UC**

The command opens the **UCS** dialogue box, where you can select from the specified UCS, the parameters and UCS icon modes for viewports.


The **UCS** dialogue box can be opened from the **Inspector** dialog:

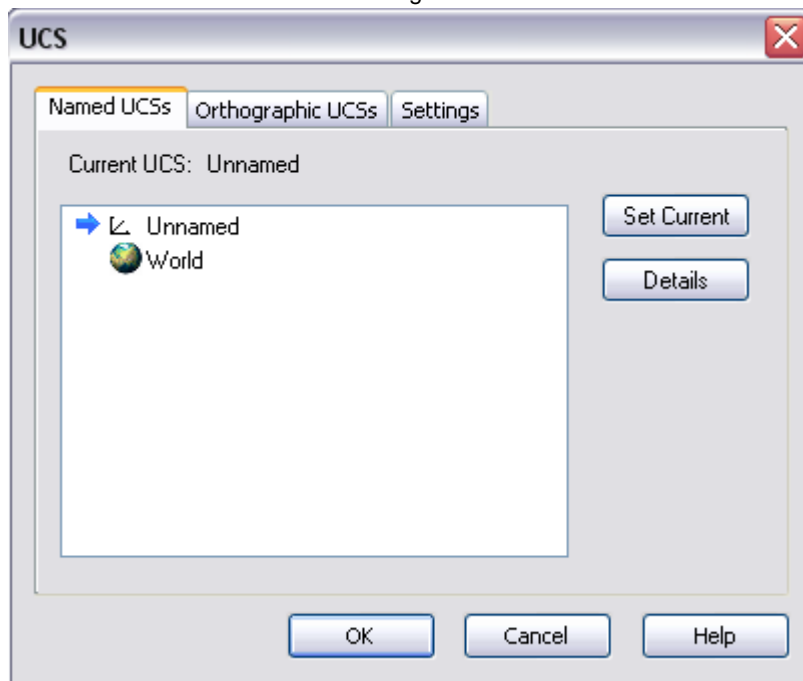


Named UCSs tab

This tab contains the list of coordinate systems specified in the current drawing.

If the UCS is not saved or named, it is shown as **Unnamed** in the list.

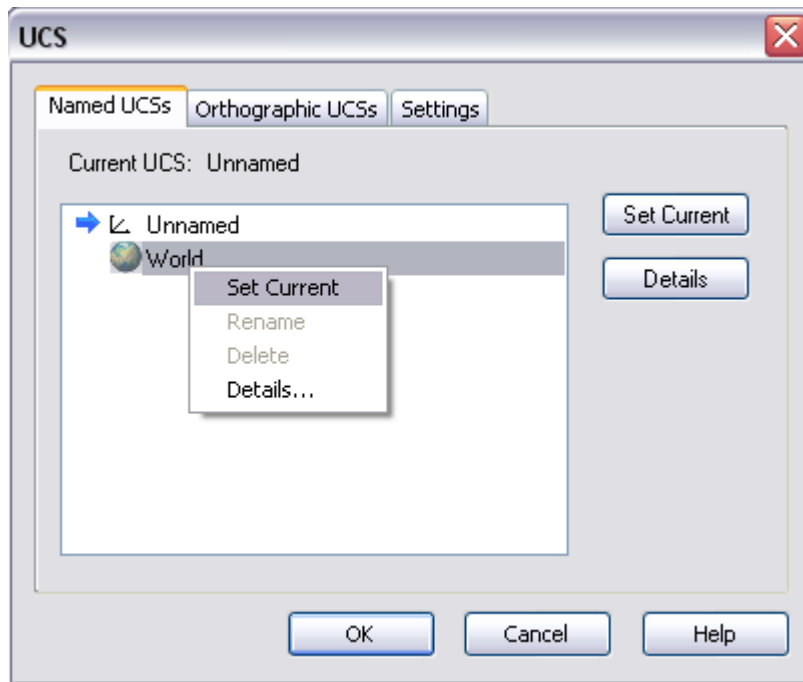
The current UCS is marked with  sign.



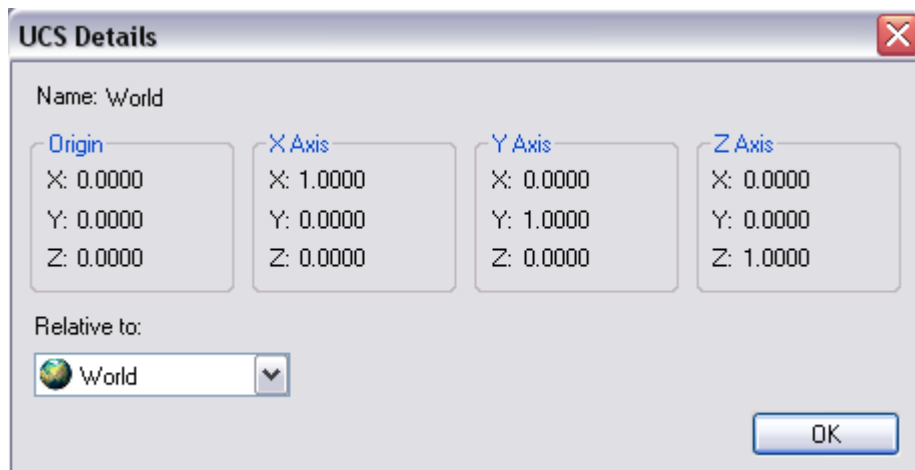
To set a UCS as current:

- Select the UCS from the list,
- Select the **Set Current** button.

You can set the UCS as current by double clicking on the UCS name or select **Set Current** from the context menu:



The **Details** button (or command from the context menu) opens the **UCS Details** dialogue box with information about the coordinates of the selected UCS:



To delete a UCS:

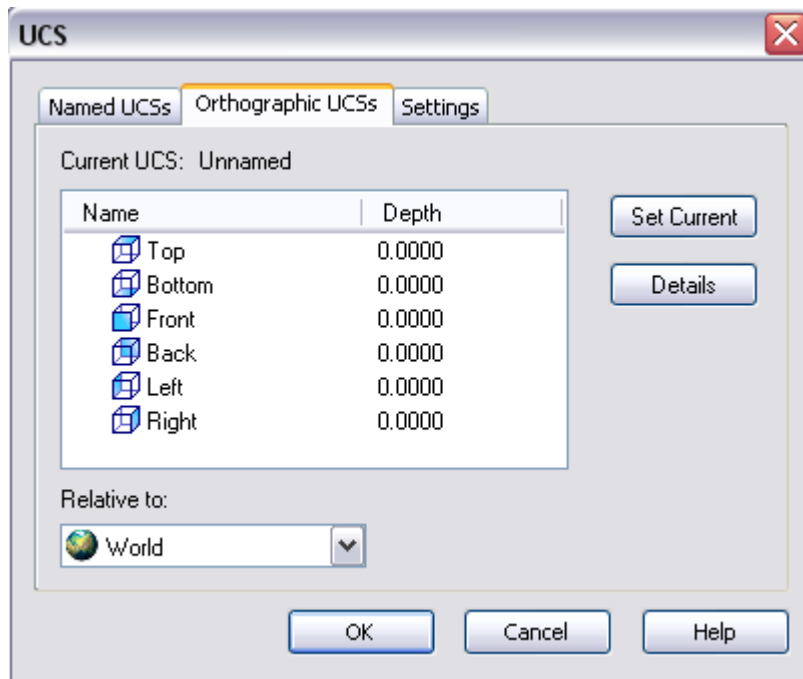
- Select the UCS from the list,
- Open the context menu,
- Select **Delete**.

To rename a UCS:

- Select the UCS from the list,
- Open the context menu,
- Select **Rename**,
- Enter the new name.

Orthographic UCSs tab

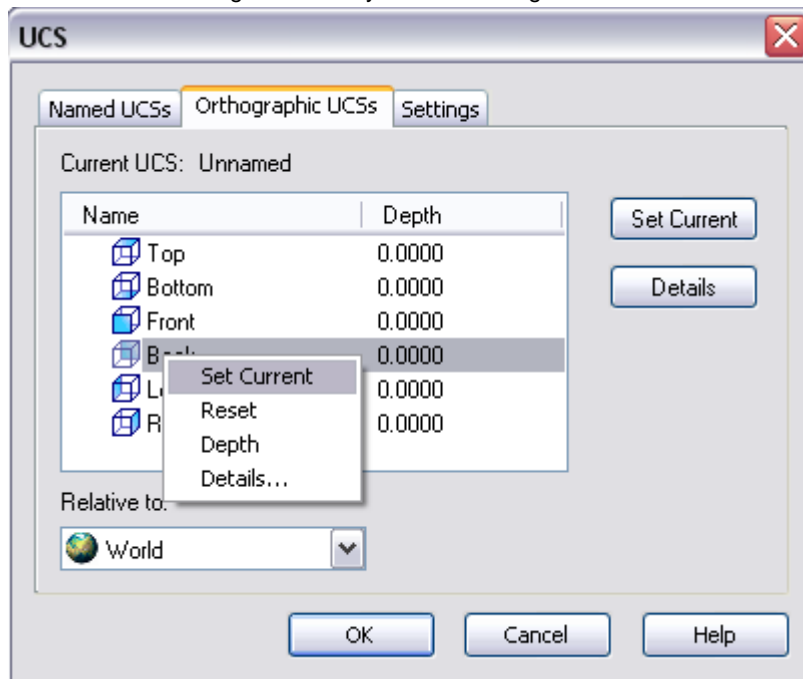
This tab contains six orthogonal coordinate systems which can be set for a UCS selected in the **Relative to** drop-down list. All named UCS existing in the current drawing are shown in the **Relative to** drop-down list.



To set an orthogonal UCS:

- Select the UCS from the list,
- Select the **Set Current** button.

You can set an orthogonal UCS by double clicking on the UCS name or select **Set Current** from the context menu:



Options of the context menu:

Set Current Sets the orthogonal UCS as current one.

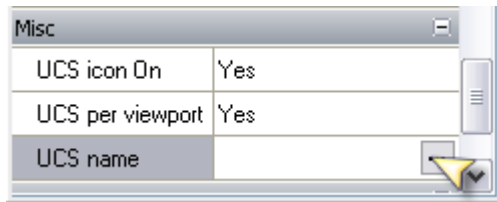
Reset Restores the origin of the selected orthogonal coordinate system (the origin has (0,0,0) coordinates of the base coordinate system).


Depth Sets the direction between the XY plane of the orthogonal UCS and the parallel plane, set through the origin of the base coordinate system.
The parallel plane can coincide with XY, YZ or XZ planes of the base coordinate system.

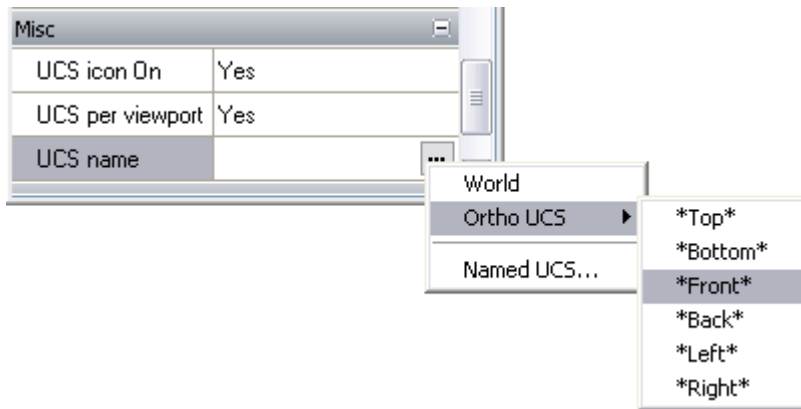
Details Opens the **UCS Details** dialogue box with information about the coordinates of the selected orthogonal UCS.

To set an orthogonal UCS:

- Click in the *UCS name* field in the **Inspector** window:

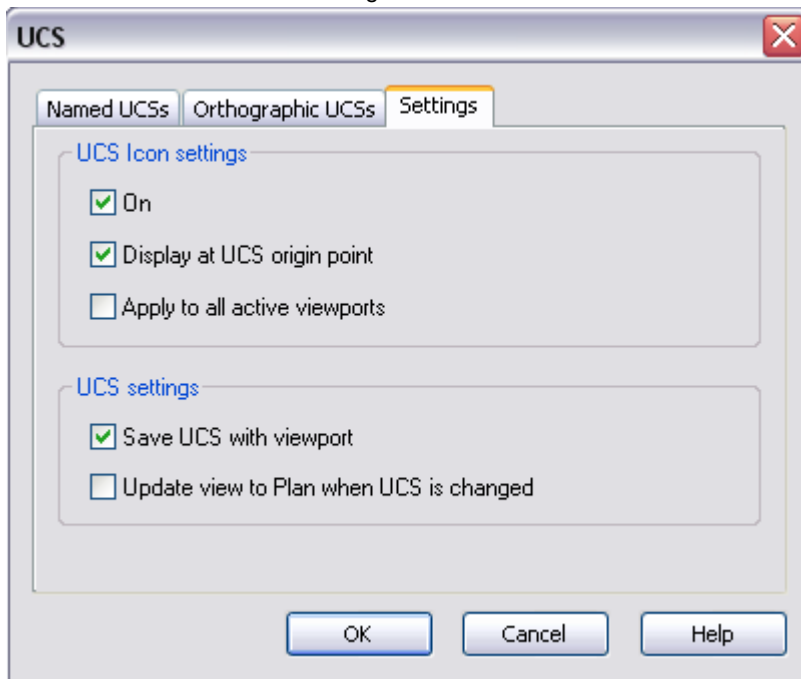


- Click the  button, select UCS from the list:



Settings tab

The tab is used to show and change the UCS icon modes and the UCS modes saved with the viewport:



Parameters:

UCS icon setting

On Displays the UCS icon in the current viewport.

Display at UCS origin point Displays the UCS icon in the current viewport at the origin. If the UCS origin is outside the viewport and the parameter is switched off, the UCS icon is specified in the left corner of the viewport.

Apply for all active viewports Applies the UCS icon modes to all the active viewports of the current drawing.

UCS settings

Save UCS with viewport

Saves the UCS mode with the viewport. If the parameter is switched off, the UCS of the current viewport is used for the specified viewport.

Update view to Plan when UCS is changed

Restores the view in plan when the coordinate system is changed in the current viewport.

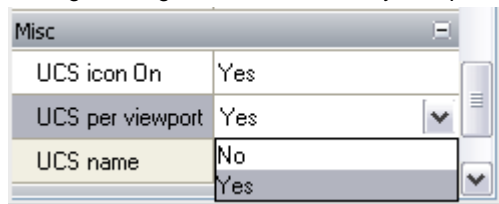
UCS for viewports

Different model views can be shown for different viewports. For example, a configuration of three viewports can have the top view in one viewport, the front view in other and the right side view in the third viewport. You can set and save the UCS for every viewport.

If the **Yes** parameter is set in the **UCS per viewport** field in the **Inspector** dialogue box, the UCS of the viewport is saved before switching to another viewport. When you switch back to the viewport, the saved UCS restores.

If the **No** parameter is set, the UCS of the viewport always coincides with the UCS of the current active viewport.

Manage saving of the UCS for every viewport in the **Inspector** dialogue box:



Precision tools

nanoCAD, as other CAD systems, allows precise geometric creations (up to 14 decimals).

Using precision tools allows:

- speeding up of implementations during the project,
- eliminating errors and inaccuracies in the dimensioning,
- eliminating errors in the measurements of angles, length and distances, which were not dimensioned in the drawing,
- avoiding problems occurring with hatch, due to open contour,
- reducing time spent on the control programs for NC machines (because it is not necessary to edit inaccurate geometry of the parts) etc.

nanoCAD provides precise geometric creations in two ways :

- precise coordinates of object points and
- using precision modes.

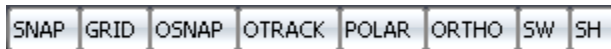
Precision modes allow:

- placing points on the rectangular grid (**SNAP**, **GRID**);
- using object snaps to snap to characteristic points on the objects, for example to the endpoints of a line or to circle centre (**OSNAP**);
- the use of tracking lines to place a created object in relation to other objects (**OTRACK**);
- snap to specified angles and define distances using polar tracking (**POLAR**);
- create and replace objects parallel or perpendicular to coordinate axes (**ORTHO**).

Precision modes

Precise tools are controlled in the context menu by the buttons and in the **Drafting Settings** dialogue box (**Tools> Drafting settings**).

Buttons to switch between precise modes are in the status bar.



The orange colour of a button shows that the mode is switched on.

All or several modes can be switched on at once.

Three ways to switch on/off the modes:

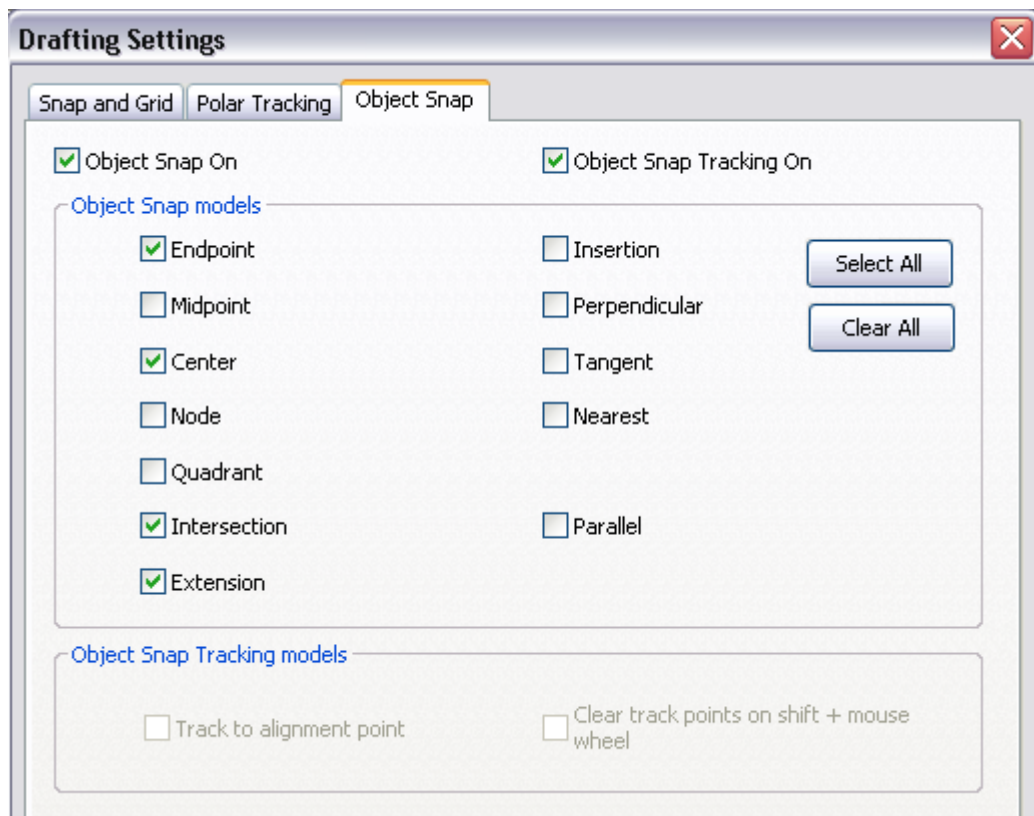
- Click on the button.
- From the context menu of the button select **On** or **Off**.
- In the **Drafting Settings** dialogue box (**Tools> Drafting settings**) select or deselect the checkboxes: **Snap on (F9)**, **Grid on (F7)**, **Polar Tracking on (F10)**, **Object Snap on (F3)**, **Object Snap Tracking on (F11)**.

To open the context menu with the commands to control modes:

- Place the cursor in the status bar,
- Press the right button,
- Select/deselect the required mode.

Note: To cancel the context menu of the OSNAP select the **Exit Menu** command (other context menus are closed after the parameter is selected).

The **Drafting Settings** dialogue box consists of three tabs: **Snap and Grid**, **Polar Tracking** and **Object Snap**:



Snap and Grid mode



Menu: **Tools – Drafting settings... > Snap and Grid tab**



Status bar: **SNAP** and **GRID**



Hotkeys: **F9 and F7**

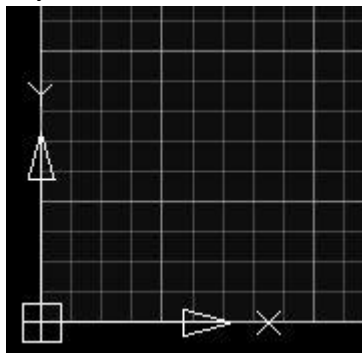


Command line: **DDRMODES, DSETTINGS, SE**

Grid is an ordered sequence of points, which with **SNAP** mode allows specifying restrictions of the cursor movement to define accurate coordinates. You can switch the grid on/off at any time when you are working with a drawing. Changing Grid spacing does not affect any drawing objects.

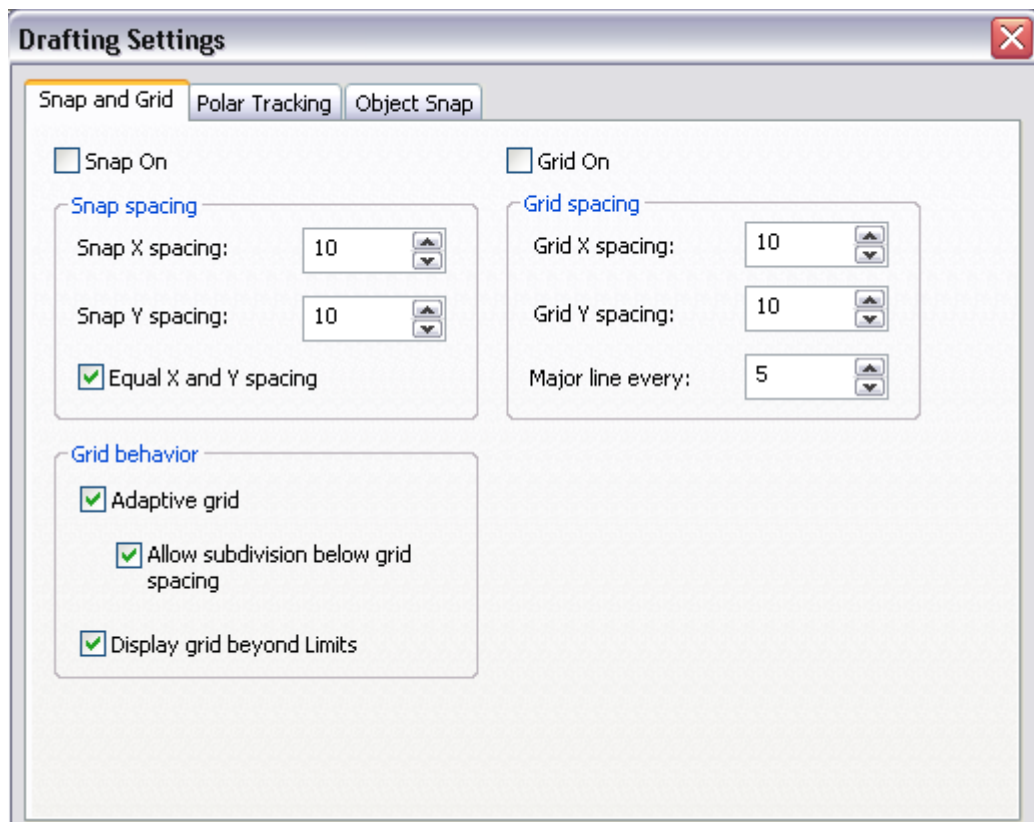
If **SNAP** mode is switched on, the cursor jumps from node to node with the specified snap spacing. Snap spacing and grid spacing can be different but very often their values are the same. Grid can have high spacing and snap spacing can be small to give a user the capability to specify points of high precision. For example, you can set grid spacing to 10 units, and snap spacing to 1 unit. Grid and snap spacing can be different along the X and Y axes.

Major line – additional lines are also highlighted:



Adaptive grid – grid display depends on the drawing scale. When zooming in, additional grid lines appear; when zooming out, they disappear. The frequency of these lines is determined by the frequency of the main grid lines. A grid showing drawing *limits*, allows visualization of the drawing dimensions to place base elements on the initial stages.

The parameters of the **SNAP** and **GRID** modes are specified in the **Snap and Grid** tab of the **Drafting settings** dialogue box or in the context menus.



Tab parameters:

Snap On (F9) Switches snap mode on/off.

Snap spacing

Snap X spacing: Specifies the X spacing.

Snap Y spacing: Specifies the Y spacing.

Equal X and Y spacing Sets equal spacing between X and Y.

Grid spacing

Grid On (F7) Switches display of the grid on/off.

Grid X spacing: Specifies the X grid spacing.

Grid Y spacing: Specifies the Y grid spacing.

Major line every: Specifies the number of cells between major lines.

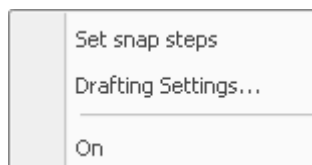
Grid behavior

Adaptive grid Switches adaptive grid mode on/off.

Allow subdivision below grid spacing Switches division of the grid spacing on/off.

Display grid beyond limits Switches display of the grid beyond specified limits on/off.

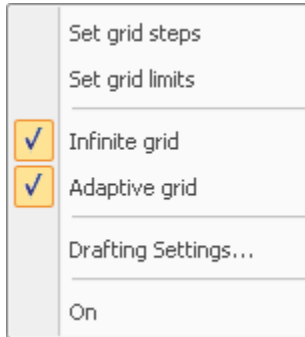
The context menu of the **SNAP** button:



Parameters:

- Set snap steps** Specifies the X and Y spacings in the command line.
- Drafting settings...** Opens the **Drafting settings** dialogue box.
- On/Off** Switches snap mode on/off.

The context menu of the **GRID** button:



Parameters:


- Set grid steps** Specifies the X and Y spacing in the command line.
- Set grid limits** Specifies the limits of the grid display.
- Infinite grid** Switches off restrictions for the grid display.
- Adaptive grid** Switches adaptive grid mode on/off.
- Drafting Settings...** Opens the **Drafting settings** dialogue box.
- On/Off** Switches display of the grid on/off.

Drawing limits



Menu: **Format – Drawing limits**



Status bar: Context menu of  – **Set grid limits**



Command line: **LIMITS**

The command sets the limits for the current drawing in Model Space and in Paper Space.

Drawing limits are specified by the coordinates of two opposite corners of the rectangular area – bottom left corner and top right corner.

Command prompts:

Specify lower left corner <0.0000,0.0000>: Specify coordinates or pick a point on the screen.

Specify upper right corner
<59400.0000,42000.0000>: Specify coordinates or pick a point on the screen.

Polar Tracking mode



Menu: **Tools – Drafting settings... > Polar tracking** tab



Status bar: the  button



Hotkeys: **F10**



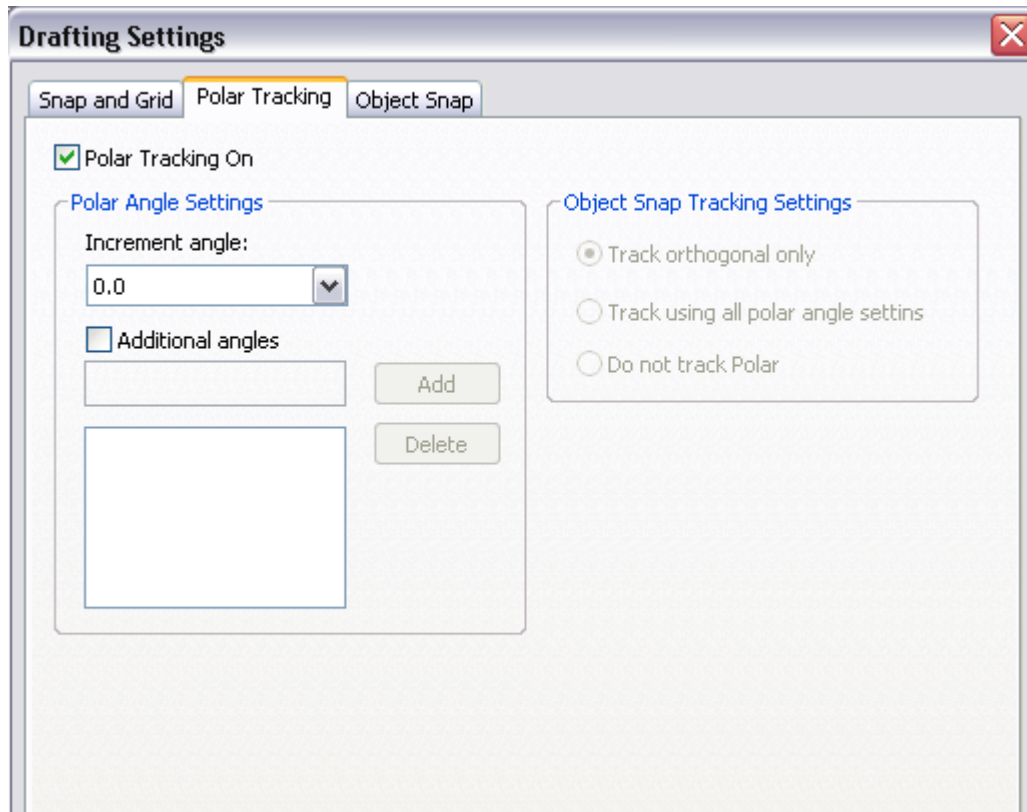
Command line: **DDRMODES, DSETTINGS, SE**

Polar tracking mode helps to specify a point, set at a specified distance and angle from the last selected point. The dotted tracking line with a tooltip shows the distance from the last specified point and current angle value.



Using polar tracking mode you can create geometric creations with any specified interval. For example, if an angle of 30° was selected, tracking lines will be displayed when the direction from this point to the cursor's pickbox is a multiple 30° (e.g. 60° , 90° , 120° and so on).

Specify the increment of the polar angle in the **Polar tracking** tab of the **Drafting Setting** dialogue box or from the context menu of the **POLAR** button.



Parameters:

Polar tracking on (F10) Switches polar tracking mode on/off.

Polar Angle Settings

Increment angle: Specifies the increment angles used for polar tracking lines.

Additional angles Switches on/off the mode for specifying any additional angles (which do not depend on the increment angle)

Object Snap Tracking Settings

Track orthogonal only In **OSNAP** mode tracking lines are only drawn horizontally and vertically.

Track using all polar angle settings Switches on the mode for applying the parameters of polar tracking to object tracking. In this mode the cursor is moved from the point of object snap using aligning angles.

Do not track Polar Switches off the polar angles tracking mode .

To specify additional angles:

- Enter an angle value,
- Select the **Add** button.

To delete an additional angle:

- Select the angle in the section,
- Select the **Delete** button.

The context menu of the **POLAR** button contains the available increment angles:






| | |
|-------------------------------------|-------|
| <input checked="" type="checkbox"/> | 90.00 |
| <input type="checkbox"/> | 45.00 |
| <input type="checkbox"/> | 30.00 |
| <input type="checkbox"/> | 22.50 |
| <input type="checkbox"/> | 18.00 |
| <input type="checkbox"/> | 15.00 |
| <input type="checkbox"/> | 10.00 |
| <input type="checkbox"/> | 5.00 |
| <hr/> | |
| Set custom angle | |
| <hr/> | |
| Off | |

Parameter:

Set custom angle Specifies a new angle of polar snap direction in the command line.

On/Off Switches the polar angles tracking mode on/off.

Object snap mode

-  Menu: **Tools – Drafting settings... Object snap** tab
-  Status bar: the  button
-  Hotkeys: **F3**
-  Command line: **DDRMODES, DSETTINGS, SE**

Object snap is the main and the quickest way to specify an object's characteristic points without knowing their coordinates.

Characteristic points are:

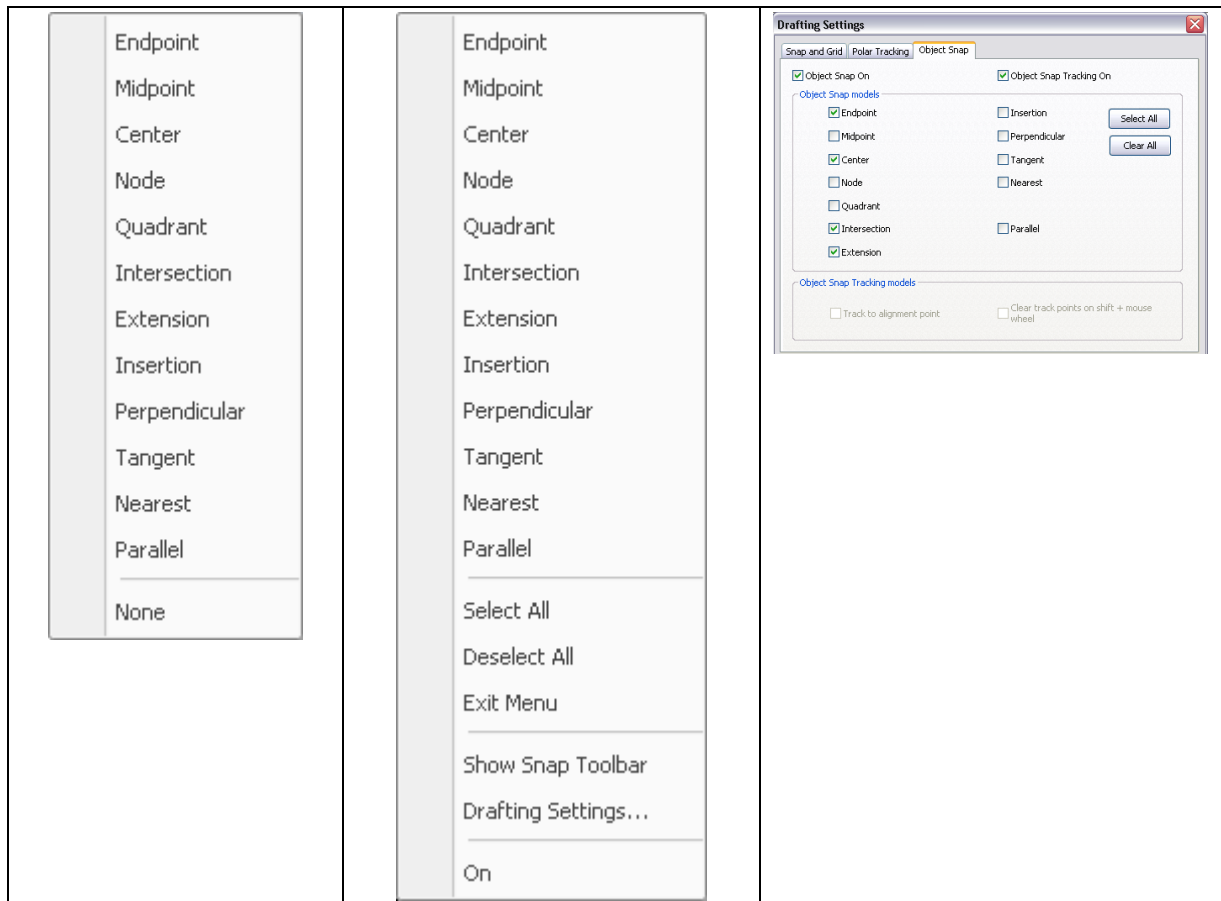
- End points and the middle of a line,
- Centre of a circle and its intersection points with centre lines (quadrant),
- Endpoints, centre and middle of an arc,
- Insertion point of block or text
- Etc.

The mechanism of object snap allows one of the characteristic points of the existing object to be specified as the coordinates for the point of a new object.

Object snap can be used when you need to set a point in the command line.

To use object snap:

| | | |
|--|--|--|
| 1. Switch on <i>one-time object snap</i> during one of the creation or modifying commands, open the context menu of <i>one-time object snap</i> with CTRL (or SHIFT) button pressed and select the required mode: | 2. Switch on <i>temporarily object snap</i> , open the context menu of the OSNAP button in the status bar and select the required mode: | 3. Switch on <i>temporarily object snap</i> , in the Drafting Settings dialogue box select the Object Snap On checkbox and select the required mode: |
|--|--|--|



You can switch on one or several modes of object snap (except the context menu of *one-time object snap*, only one mode can be selected there).

The **Select All** and **Deselect All** are applied to all parameters of the snap.

One-time object snap is used to replace *temporarily object snap* and stays active until the current snapping is finished. When a snap point is selected, *one-time object snap* switches off. For example, when creating geometric objects consisting of lines, as a temporarily snap you can specify **Endpoint**, **Midpoint**, **Perpendicular**, and **Intersection**. If you want to snap one of the lines to a circle, specify *one-time object snap* **Center**, **Quadrant** or **Tangent**.

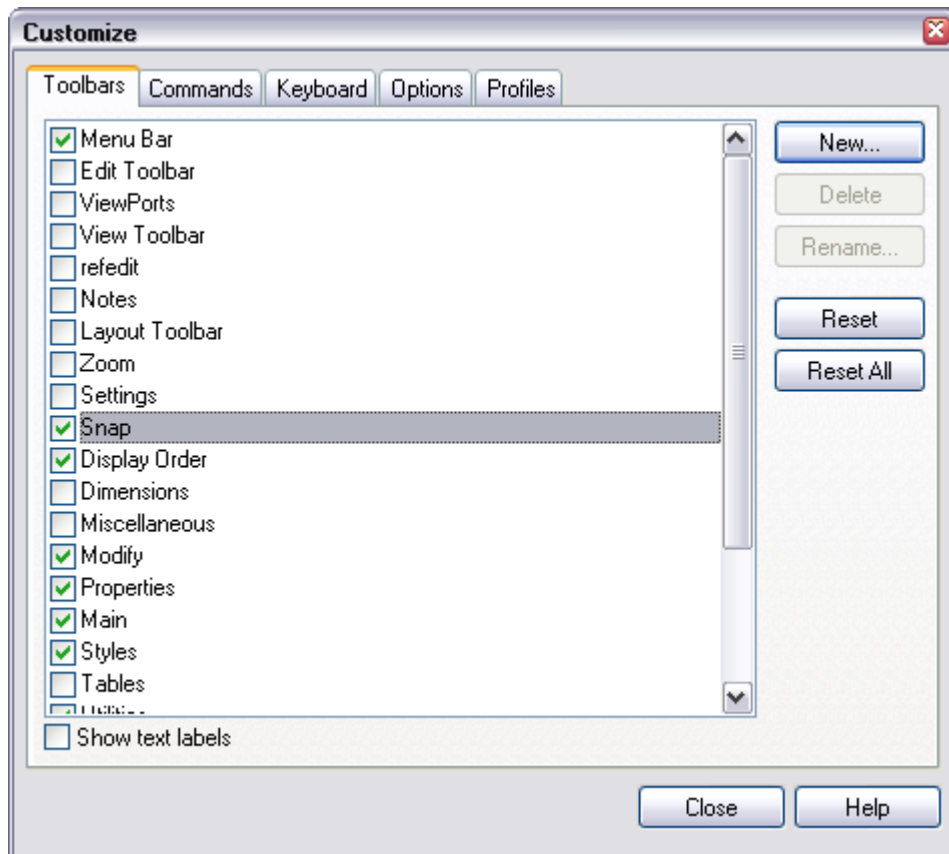
The context menu of *one-time object snap* has an additional option – **None**, which switches off all modes of object snap before finishing the point selection operation. **None** is used for one time application (during one operation), that is why it is absent in the **Object Snap** tab in the **Drafting Setting** and in the context menu of **OSNAP** button.

None is used when you cannot select a point in the drawing due to temporarily snap modes being used. After the point is specified, the temporarily snap mode starts again.

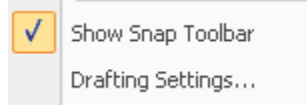
You can switch the object snap mode on/off in the **Snap** toolbar:



To display the **Snap** toolbar, select the **Snap** checkbox in the **Customize** dialogue box (**Tools** menu – **Customize > Interface**):



To open the **Snap** toolbar select the **Show Snap Toolbar** option in the context menu of the **OSNAP** button:



If an object snap mode is switched on, the marker and a tooltip about the available object snap types for the object are shown when moving the cursor over the object.

If several or all types of object snap are switched on as a *temporarily snap*, and if a snap to some characteristic point is available in the cursor's current position (for example, if some objects are placed close to each other), the highest *priority snap* is used.

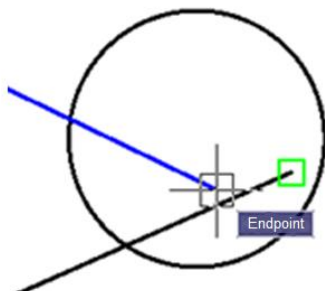
The *priority of snaps* is specified in the **Object Snap** tab in the **Drafting Setting**, the highest priority has **Endpoint**, the lowest – **Parallel**. Any *one-time snap* has a higher priority than any *temporarily* has.

Object Snap types:



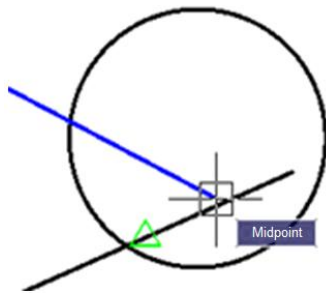
Endpoint

Snap to the object's endpoints (lines, arcs etc.).



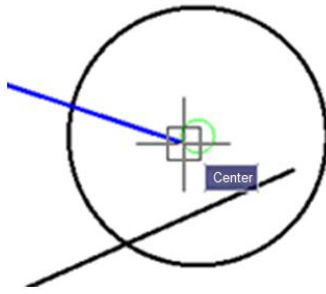
Midpoint

Snap to the middle of objects (lines, arcs etc.).



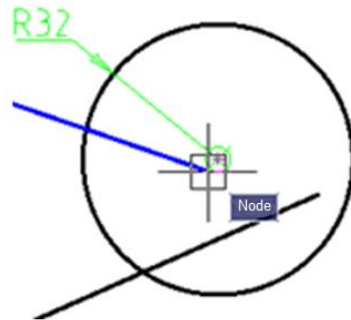
Center

Snap to the centre of an arc, circle or ellipse.



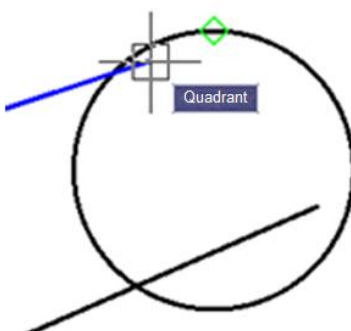
Node

Snap to a **Point** of the object, and for specifying point of dimension or start point of dimension text.



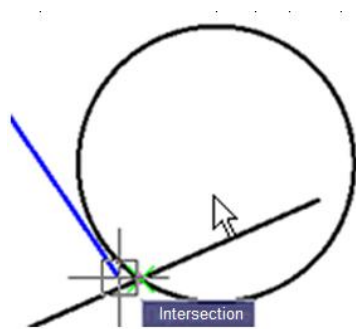
Quadrant

Snap to the nearest quadrant (the point located at an angle of 0, 90, 180 or 270 degrees from centre) of arc, circle or ellipse.



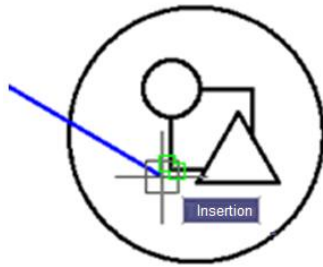
Intersection

Snap to the intersections of objects (lines, circles, arcs etc.).



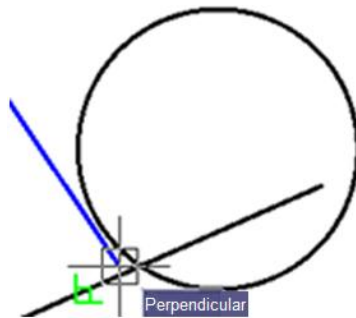
Intersection

Snap to an intersection point of text, block, shape or attribute.



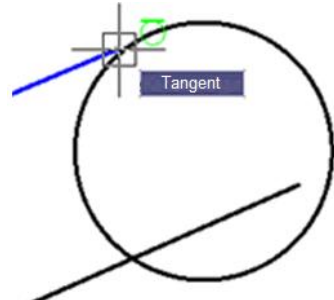
Perpendicular

Snap to the point of the object lying perpendicular to another object or to its imaginary extension.



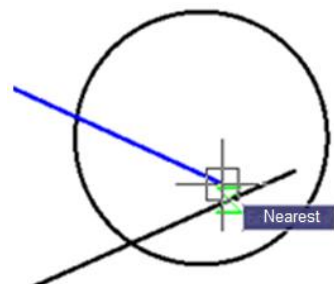
Tangent

Snap to the point on an arc or circle belonging to the tangent of another object.



Nearest

Snap to the point of the object located closest to the cursor position.

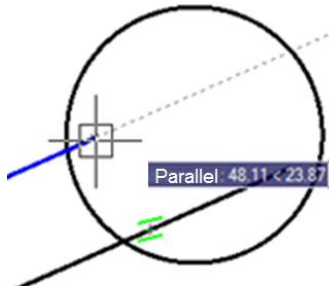




Parallel

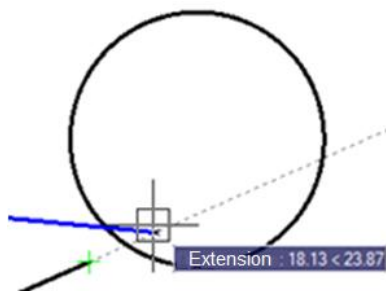
Snap to an existing linear segment for the creation of a parallel linear segment of other object.

After specifying the first point of a linear segment, you need to place cursor's pickbox over a linear segment of the existing object and slowly move the cursor to the expected position of a parallel segment of a new object. The symbol of a parallel snap on the existing object and parallel rubber line to this object means that you can specify the second point of the linear segment at any required place in the rubber line.



Extension

Creates a temporarily auxiliary line which is an extension of an object and the cursor goes over its end point.



Object Snap tracking mode



Menu: **Tools – Drafting settings... Object snap** tab



Status bar: the **OTRACK** button



Hotkeys: **F11**

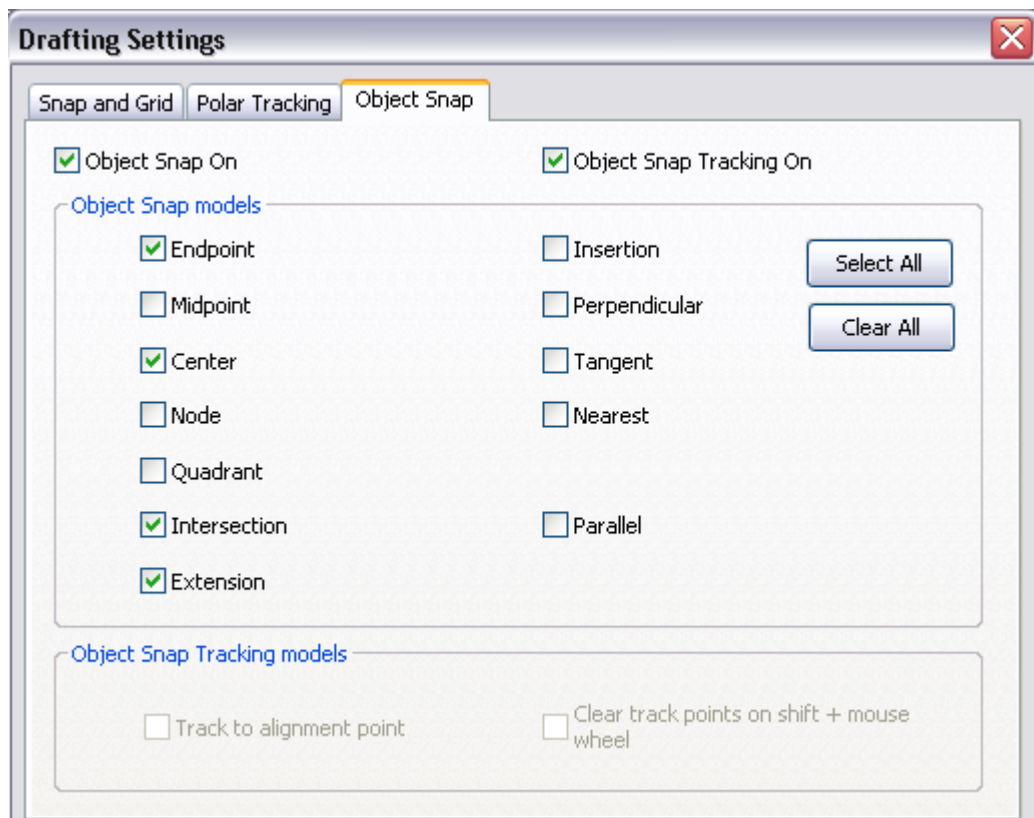


Command line: **DDRMODES, DSETTINGS, SE**

The object snap tracking mode allows the placing of created objects in specific relation to other created objects. When object snap tracking mode is switched on, nanoCAD temporarily displays dotted trajectories of different types (tracking lines) for accurate positioning of objects.

Object tracking modes are used together with object snap and works when the cursor's pickbox is near a probable point of object snap (the pickbox's size determines the zone of tracking lines activation). Object tracking expands and adds to the capabilities of object snap.

The parameters of object tracking are specified in the **Object Snap** tab in the **Drafting Setting** dialogue box (**Tools menu – Drafting Settings**) or in the context menu of the **OTRACK** button.



Parameters:

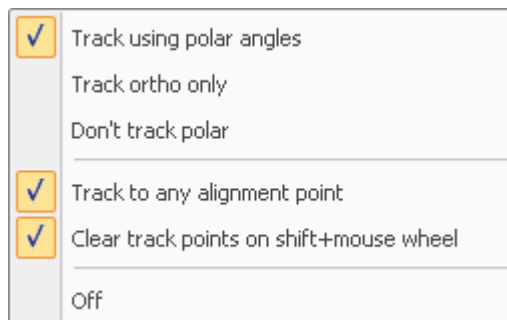
Object Snap Tracking On (F11) Switches object snap tracking mode on/off.

Object Snap Tracking models

Track to alignment point Switches on/off the display of tracking lines to an object's characteristic points.

Clear track points on "shift + mouse wheel" Switches on/off the erasing mode of the snap point marker on a tracking line using **SHIFT** and the mouse wheel.

The context menu of the **OTRACK** button:



Parameters:

Track using polar angles Switches using object tracking together with polar tracking mode on/off. In this case, the tracking lines for all angles specified in the **Polar Tracking** tab in the **Drafting Settings** dialogue box are displayed.

Track ortho only Switches the display of tracking lines only in ortho directions on/off.

Don't track polar Switches off polar tracking mode.

Track to any alignment point Switches on/off the display of tracking lines to specific points.

Clear track points on SHIFT + mouse wheel Switches on/off the erasing mode for the snap point marker on a tracking line.

On/Off

Switches object tracking mode on/off.

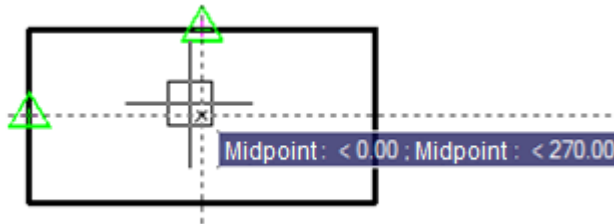
To use object tracking mode:

- Switch on the **OTRACK** and **OSNAP** modes in the status bar.
- Use the cursor to capture the required characteristic objects' points to set tracking lines. Place the cursor near a point to capture it. A captured point is marked with **+** symbol. To deactivate a point already captured, move the cursor over it again.
- Dotted tracking lines, going through one or several captured points and the cursor's pickbox, will appear when you move the cursor within the drawing. You can snap to points on those lines or to the intersection points. The more types of object snap that are switched on, the more points that will be available for capture.

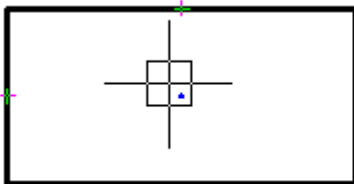
To zoom and pan, the **+** marker is removed from captured points.

Example: Circle in the centre of a rectangle.

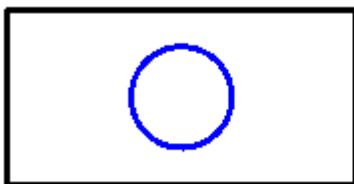
1. Defining the centre of rectangle:



2. Specifying the circle centre:



3. Specifying the circle radius:



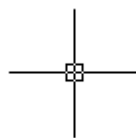
Display of snap elements



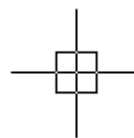
Menu: **Tools – Options...**

The cursor's pickbox is enlarged automatically in nanoCAD snap modes (creation and edition of primitives, dimensioning etc.) for your convenience:

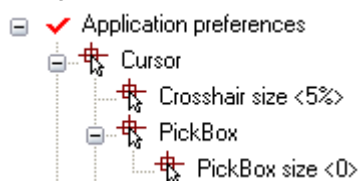
Common cursor's pickbox size



Enlarged cursor's pickbox size



The common cursor's pickbox size is specified in the *Cursor – Pickbox – Pickbox size* section of the **Options** dialogue box (**Tools** menu – **Options**):



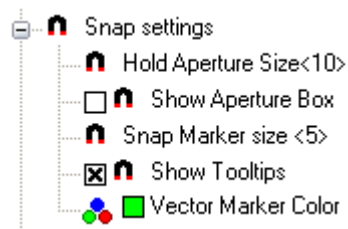
Parameters:

Crosshair size

Sets the size of the Crosshair as a percentage of the display size.

Pickbox size Pickbox size in pixels.

The enlarged cursor's pickbox is specified in the **Options** dialog in the *Snap settings – Hold Aperture Size*:



Parameters:

| | |
|-----------------------------------|---|
| Hold Aperture Size <10> | The size of the cursor frame in snap mode. |
| Show Aperture Box | Switches the aperture box on/off in the snap mode. |
| Snap Marker size <5> | Snap marker size. |
| Show Tooltips | Turns on the display of the snap name. |
| Vector Marker Color | The colour of the snap marker when snapping to a vector object. |

ORTHO mode



Status bar: the  button



Hotkey: **F8**

With orthogonalisation mode set on, vector lines and rubber lines for editing can only be drawn in directions parallel to the coordinate axis.

If an angle of a coordinate system is changed, the corresponding ortho mode angle is also changed.

The **ORTHO** mode has higher priority than the polar tracking mode.

Navigation in the document

View is a combination of the size, position and orientation of a drawing fragment on the screen.

There are different tools and methods allowing you to orient in the document when you edit it. You can zoom or pan the working area for visual control of changes in the document; you can save a selected view for further display or printing; you can separate the working area of a document into several non-overlapping *viewports* to display different drawing fragments at the same time.

Display modes

The main methods to change the document display on the screen are *zooming* and *panning*.

Zoom command enlarges the view of a drawing segment for better detailing or decreases it to display more of the drawing. Absolute sizes are not changed during zooming.

Pan command allows you to pan the drawing without decreasing or enlarging it.

Zoom and *Pan* commands are available in the **View – Zoom** menu or on the **Zoom** toolbar. For your convenience, most frequently used commands are on the **Main** toolbar and in the status bar.

Note: You can zoom using the mouse wheel. Moving the mouse with the wheel pressed and held allows panning.

Pan




Menu: **View – Zoom >**  **Pan**



Toolbar: **Main –** 





Status bar – 



Command line: **PAN, VIEWPAN**

Allows you to move a document in the window with the mouse.

After starting the command, the cursor has the  shape.

With the left button pressed, the cursor has the  shape.

To exit from pan mode, press the **ESC** or **ENTER** buttons.

Zoom




Menu: **View – Zoom >**  **Zoom Dynamic**



Toolbar: **Main –** 




Status bar – 



Command line: **ZOOMD**

Turns on the mode in which you can increase an object's size on the screen by moving the mouse up and decrease it by moving it down.

After starting the command from the View menu or Main toolbar, the cursor has the  shape.

When you release the button the zoom stops; you can move the cursor to another position and press the button again to continue zooming in a new position.

To exit from the zoom mode, press the **ESC** or **ENTER** buttons.

If you start the command in the command line, the following options are available:

Specify corner of window or

[All/Center/Dynamic/Extents/Previous/Scale/Rscale/Window/Object/] <real time>:

Zoom 1:1



Menu: **View – Zoom >**  **Zoom 1:1**





Command line: **ZOOM1**



An image is scaled so that one pixel on the screen corresponds to one image point.


Zoom All



Menu: **View – Zoom >**  **Zoom All**

 Toolbar: **Main –** 

 Status bar – 



 Hotkeys: **ALT+0**


 Command line: **ZOOMALL**

Displays the whole document.

Zoom Selected

 Menu: **View – Zoom >**  **Zoom Selected**



 Toolbar: **Main –** 



 Command line: **FITSEL**


Displays the selected objects.

Zoom Window

 Menu: **View – Zoom >**  **Zoom Window**



 Toolbar: **Main –** 


 Status bar – 

 Command line: **ZOOMW**

Selects an area on the screen.



Zoom In


 Menu **View – Zoom >**  **Zoom In**

 Command line: **ZOOMIN**

Doubles the scale.

Zoom Out

 Menu: **View – Zoom >**  **Zoom Out**

 Command line: **ZOOMOUT**

Decreases the scale by half.


Order of objects

When editing a document, the objects are shown in the order that they were created. Some objects can be overlapped or obscure each other. To correct the situation, the order of the objects (their display on the screen) can be changed, one object can be placed in front of another. Commands to change the order of objects are shown in the **Tools – Display Order** and on the **Display Order** toolbar.

Bring to Front

 Menu: **Tools – Display Order >**  **Bring to Front**

 Toolbar **Display Order –** 


 Command line: **DRAWORDER1**

Forces the selected object to be displayed in front of all other objects.

Send to Back

 Menu: **Tools – Display Order >**  **Send to Back**

 Toolbar: **Display Order –** 

 Command line: **DRAWORDER2**

Forces the selected object to be displayed behind all other objects.

Bring Forward

 Menu: **Tools – Display Order >**  **Bring Forward**



Toolbar: **Display Order** – 



Command line: **DRAWORDER3**

Forces the selected object to be displayed in front of the object located in front of it.

Send Backward



Menu: **Tools – Display Order >  Send Backward**



Toolbar: **Display Order** – 



Command line: **DRAWORDER4**

Forces the selected object to be displayed behind the object located beneath.

Viewports of Model Space

Model space can be separated into several rectangular *non-overlapping* areas called *viewports*.

Non-overlapping areas fill in model space and cannot be placed over each other. When starting a project, usually one viewport, filling in all the model space, is used. This viewport can be separated into several viewports and different fragments of the drawing or model views can be displayed on every viewport at the same time. The changes made in one viewport are also shown in the whole drawing (in other viewports). You can switch between viewports at any time, even whilst a command is being performed. To switch to another viewport, click at any point on the screen. The cursor is displayed in the current viewport, –the common arrowhead is displayed in all the others.

For every viewport you can specify the display scale, pan the viewport image independently of the other viewports, specify UCS and display the modes of grid and snap usage. You can save the setting parameters of any viewports to use them again and restore an image of any viewport.

Only one *non-overlapping viewport* can be printed.

Configuration of *non-overlapping viewports* can be different:

1 Viewport



Menu: **View – ViewPort >  1 Viewport**



Command line: **VIEWPORT_SINGLE**

In Model Space: restores the configuration to one viewport (the view is taken from the last active window).

In Paper Space: creates one viewport.

2 Viewports



Menu: **View – ViewPort >  2 Viewports**



Command line: **SPLITVIEWPORT_VERTICAL**

Creates two identical viewports.

3 Viewports



Menu: **View – ViewPort >  3 Viewports**



Command line: **SPLITVIEWPORT_3**

Creates three viewports.

After starting the command, there is a prompt in the command line:

Enter an option [Horizontal/Vertical/Left/Right/Top/Bottom] <Right>

Select the option for the required viewport configuration.

4 Viewports



Menu: **View – ViewPort >  4 Viewports**



Command line: **SPLITVIEWPORT_4**

Creates four viewports.

Named viewports



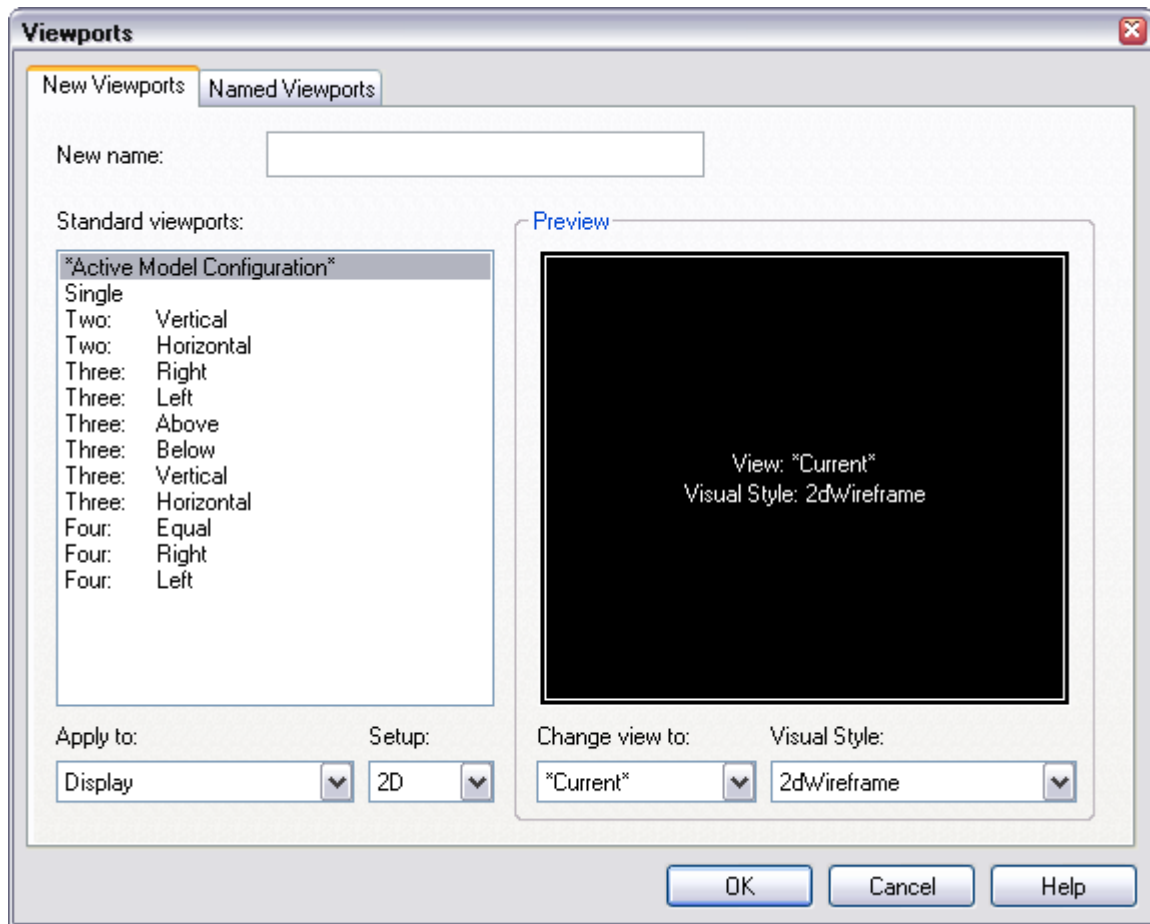
Menu: **View – ViewPort >  Named views**



Command line: **VPORTS**

The command opens the **Viewports** dialogue box.

On the **New Viewports** tab you can create the required configuration of viewports on the standard base and save it for further usage:



Parameters:

New name: Name of the saved configuration of viewports.

Standard viewports: List of standard configurations of viewports.

Apply to:

Display Applies the selected configuration of viewports to the whole Model Space.

Current viewport Applies the selected configuration of viewports to the current viewport.

Setup:

2D Sets the selected configuration as the current viewport for all viewports.

3D Sets the selected configuration of standard model views for all viewports.

Preview Preview of the selected configuration of viewports.
The current viewport is shown with a double frame.

Change view to: Changes the view of the common viewport.
There are existing named views of the drawing in the drop-down list (there are additional standard model views for 3D mode).

Visual style: Changes the visual style of the selected viewport.
There available styles in the drop-down list are:

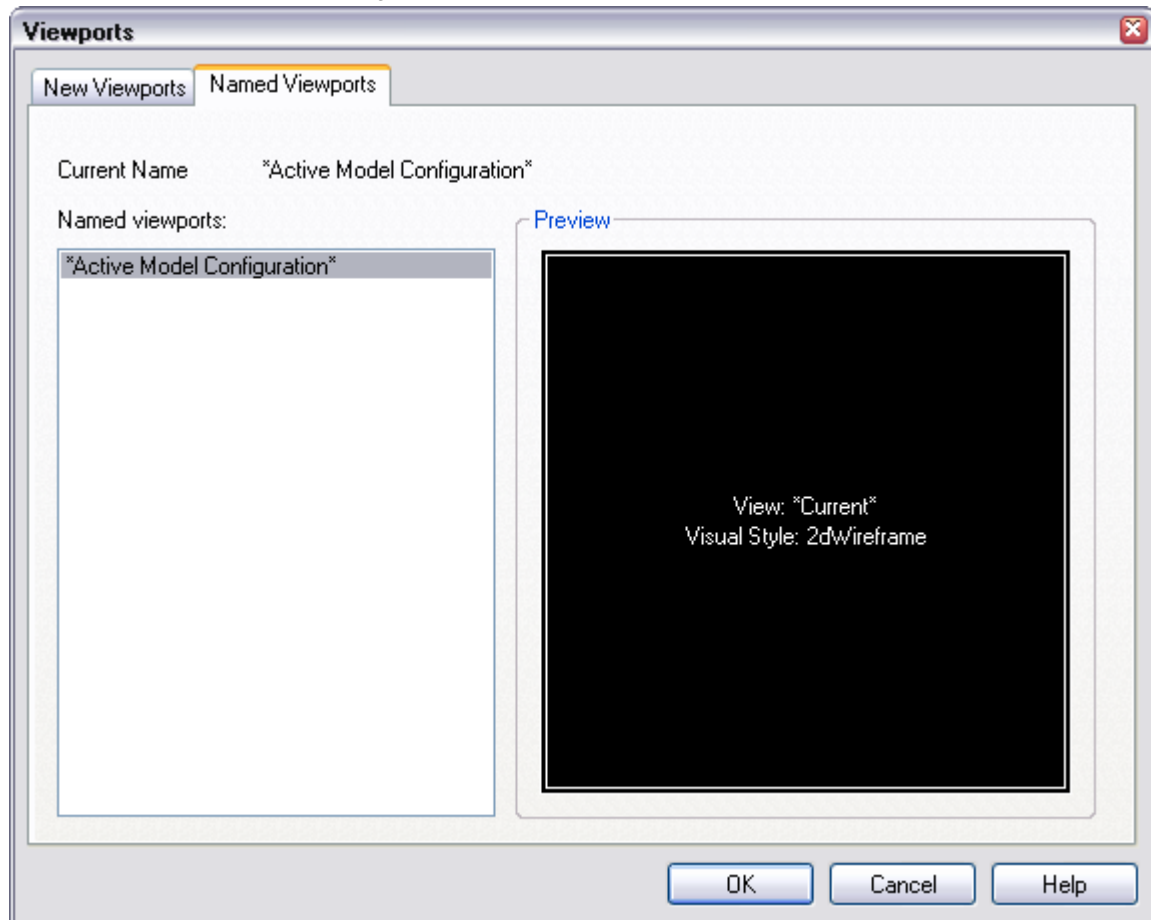
Current
2D Wireframe
3D Hidden
3D Hidden
Conceptual
Realistic

To change a view or a visual style for a viewport:

- Double click to select the viewport in the **Preview** window (selected object is shown with double frame).
- Select the required view or visual style from the drop-down list.

If a name was specified in during the creation of the configuration of the viewports, the configuration will be saved as a *named configuration*. A *named configuration of viewports* can be used without preset.

The list of created and saved configurations is shown on the **Named Viewports** tab:



To create named configurations of viewports:

- Select a standard configuration in the **New Viewports** tab.
- In the *New name* enter a configuration name.
- Select **OK**.

The name of the created configuration is shown in the *Named viewports* section of the **Named Viewports** tab when the **Viewports** dialog is opened next time.

The **Save Configuration** command from the **View>Viewports** menu allows a name to be specified in the command line for the current configuration of viewports.

To restore a named configuration of viewports:

- Select the required configuration in the *Named viewports* list (after selection, a scheme of viewports will be selected in the *Preview* window).
 - Select **OK**.
- Or:
- Start the **Restore Configuration** command (the **View>Viewports**).
 - In the command line, type the configuration name as an answer to the prompt `Enter viewport configuration name:` (the list of available configurations is shown in the command line's protocol).
 - Press **ENTER** to finish the command.

To delete a named configuration of viewports:

- Start the **Delete Configuration** command (the **View>Viewports**).
- In the command line, type the configuration name as an answer to the prompt `Enter viewport configuration name:` (the list of available configurations is shown in the command line's protocol).
- Press **ENTER** to finish the command.

Image regeneration


Sometimes, when you are working with a document after the command is finished, some visual elements stay on the drawing; for example, markers of characteristic points. To delete such visual elements, use forced *regeneration* (specified manually) or *redrawing* of a drawing.

Two commands in nanoCAD are used for it: **Regen** and **Redraw** (**View** menu).


Redrawing of a drawing is faster than regeneration because all the drawing objects' coordinates are recalculated (converting values with floating points from the drawing database to integer display coordinates) during regeneration.

Regeneration



Menu: **View** –  **Regen**



Status bar: 



Command line: **RE, REGEN**

The **Regen** command is used for a forced update of a current drawing.

Redrawing



Menu: **View** – **Redraw**



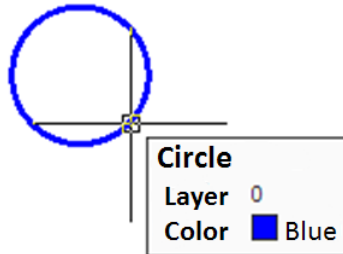
Command line: **REDRAW**

The **Redraw** command is used for a forced update of a display.

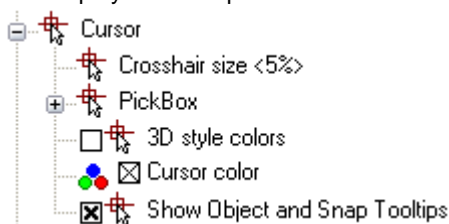
Objects' properties

In nanoCAD, graphic drawing objects have properties, such as colour, type and weight (width) of lines, transparency and plot style, which can be changed.

When placing the cursor over any object on the screen, a tooltip with the object's name, name or number of its colour and layer where the object is placed, appears:



The display of a tooltip can be switched off in the *Cursor* section of the **Options** dialogue box (**Tools – Options**):



Set different properties for objects for the document's clarity. When creating new objects, their properties are inherited from the description of the layer where they were created.

Inspector



Menu: **Modify – Properties**



Toolbar: **Main – Properties**



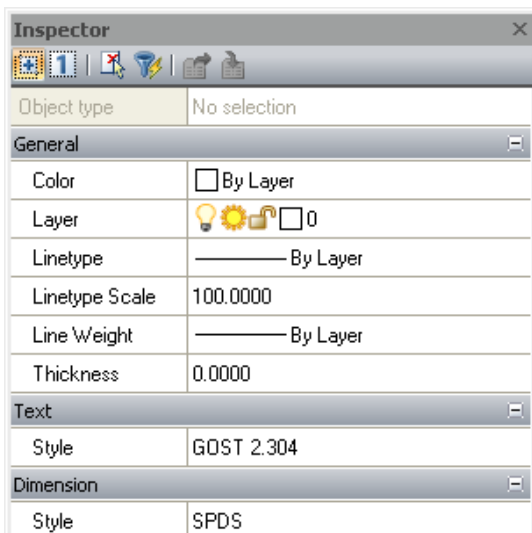
Hotkeys: **CTRL+1**



Command line: **INSP, INSPECTOR, PROPERTIES**

The **Inspector** window is used to display information about selected objects, to change objects' properties, to specify a selection mode and call up selection commands.

The list of properties is separated into groups. Manage the visibility of properties of any group by using the **+** and **-** buttons in the group name. The **+** button shows the properties for the hidden group. When this button is selected the properties list of the group is shown and the button switches to the **-** button.



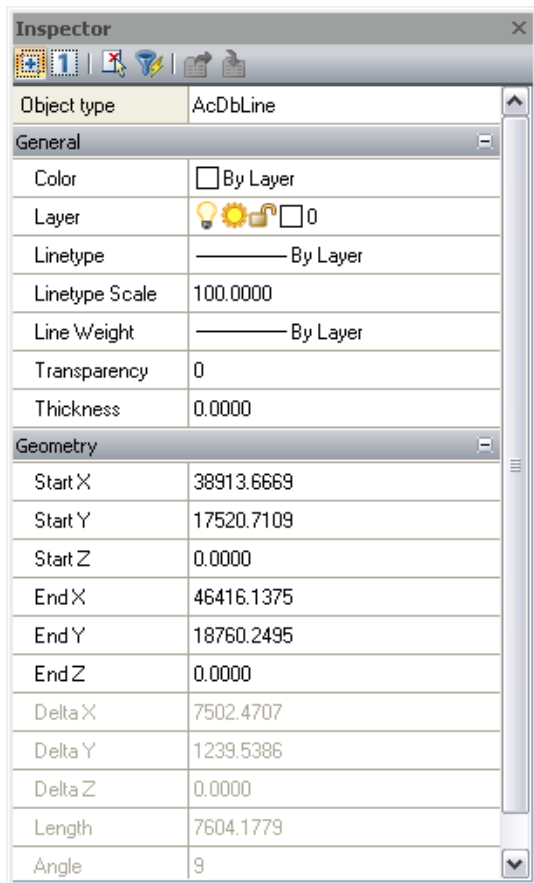
The different properties of objects are displayed in the left column of the **Inspector** window, their values in the right column.

Information in the **Inspector** depends on the current command and different parameters of the selected objects are shown.

If no object is selected, **No selection** is shown in the **Object type** field:



Current settings parameters for the properties of created objects of the document are shown in the **General** group. For example, the **Line Weight** has a «1.00» value, so new lines, arcs and circles will be created with this weight.



If several objects of one type are selected, their type and number (in brackets) are displayed in the left column:

| | |
|-------------|----------------|
| Object type | AcDbCircle (2) |
|-------------|----------------|

If several objects of different types are selected, All (2) is displayed in the **Objects type** field:

| | |
|-------------|----------------|
| Object type | AcDbEntity (2) |
|-------------|----------------|

The **General** group contains information about an object's properties: colour, layer, line type and etc.

The **Geometry** group displays information about the geometric parameters of an object and its position in the document.

Properties that can be changed are shown in black in the left column.

You can specify new values for these properties in the corresponding fields. Values outside of the limits are not used automatically.

The grey colour is used to display information about properties which cannot be changed and for properties depending on other properties.

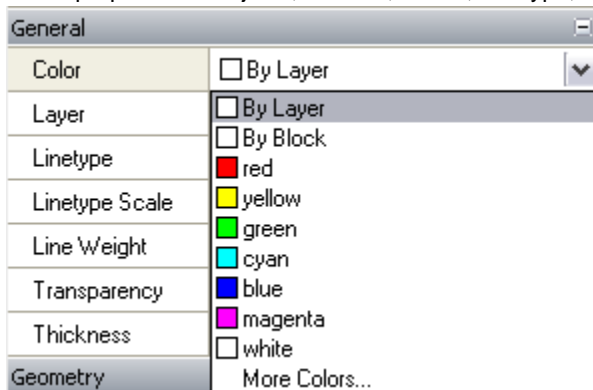
If several objects are selected, only properties common to all the objects are displayed.

If any property does not have a value (the field in the column is empty); two or more objects having the property among selected objects, but the values of the property are different, for example centre coordinates for two non-concentric circles:

| | |
|----------|--|
| Center X | |
| Center Y | |
| Center Z | |

The value, entered in such a field, is one for this property for all objects selected, if it can be applied.

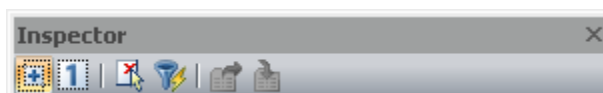
Some properties of objects, such as, colour, line type, and line weight can be selected from the drop-down list:



To change the properties of objects in the Inspector window:

- Select one or several objects.
- Click in the left column of the property that you want to change.
- Select the required value in the drop-down list to the right of the column or type a new value.
- To apply a typed property value to the objects, press **ENTER**. Values, selected from a list, are applied to the selected objects immediately without pressing the **ENTER** button.
- To deselect objects, click in the drawing area or press **ESC**.

There are mode and selection commands buttons in the top part of the **Inspector** dialog:



Creation of custom properties windows

The fields of the **Inspector window** display information about selected objects which can be dragged onto an existing or created toolbar, creating a *user properties window*:

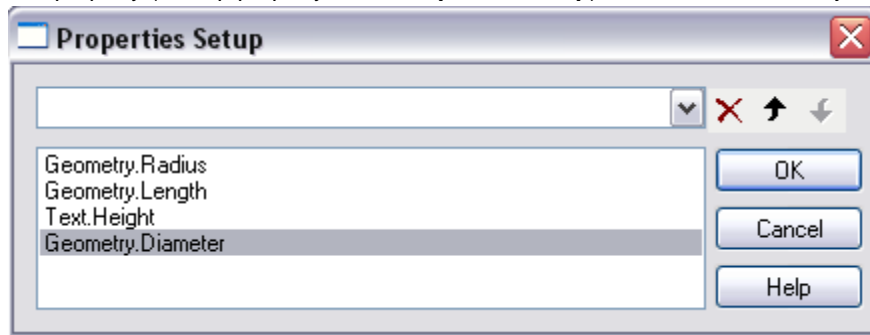


After dragging, only the right column with the values from the **Inspector window** is shown in the toolbar.



The value of a property is shown in the *user properties window* after selection of object, the property of which is shown in the window. The window remains empty if an object is not selected or if the selected object does not have properties contained in the window. If several objects are selected, the window is empty unless similar objects are selected, for example circles of one diameter.


One *user properties window* can contain several fields from the **Inspector window**.

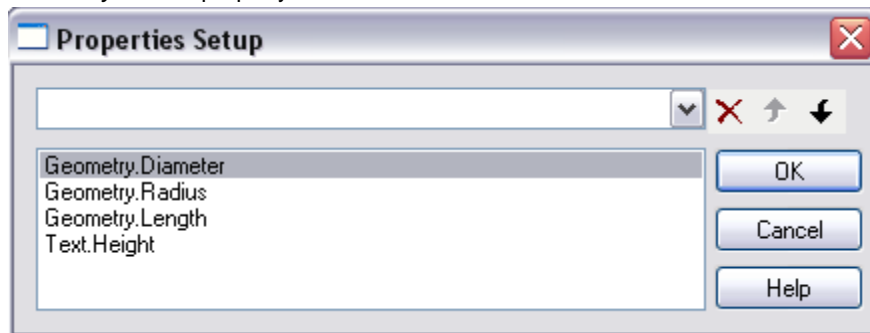
If the user window contains several properties of one properties group or several properties of one object type, only one property (the top property in the **Properties Setup**) is shown when the object is selected:




In this case, when a circle is selected the radius value will be shown; if a line is selected, its length value will be shown; if a single line of text is selected, its height will be shown.

The  and  buttons are used to move the selected property in the list.

To display the *Geometry.Diameter* property when a circle is selected, use the  to move this property above the *Geometry.Radius* property:



The  button deletes the selected property from the list.

To display two or more properties from one group or one object type, create several windows on the toolbar.

As an example, we will show how to create a toolbar with 3 *user properties windows*, the first displays the circle diameter and text height, the second – circumference and text rotation and the third – circle area and text oblique.

To create user properties windows:

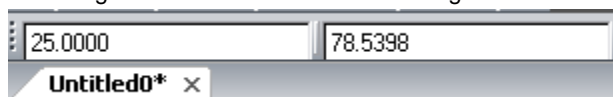
- Create a new toolbar:



- Select a created circle.
- Place the cursor over the *Diameter* field in the **Inspector window**.
- Drag the *Diameter* field, using the left mouse button and **Alt** button, onto the created toolbar:



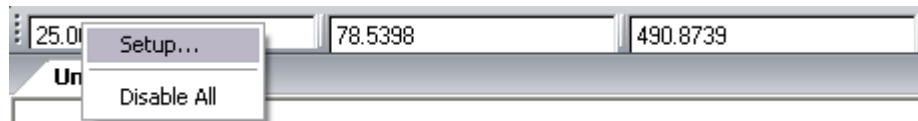
- Drag the *Circumference* field to the right of the first field on the toolbar in the same way:



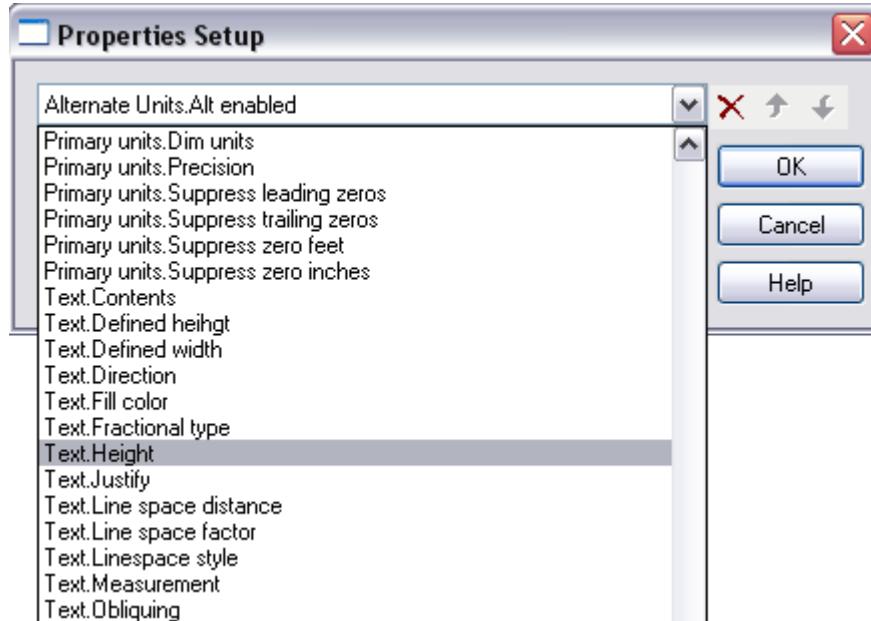
Drag the *Area* field to the right of the second field on the toolbar:



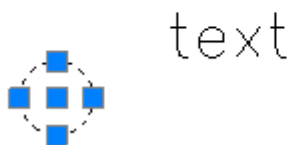
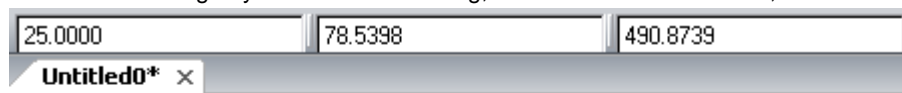
- Press **ESC** to deselect the circle.
- From the context menu of the first field select **Setup**:



- In the drop down list of the dialogue box that appears, select the *Text.Height* property:



- Select **OK** to close the dialog.
- From the context menu of the second field select *Text.Rotation*.
- Select **OK** to close the dialog.
- For the third field select the *Text.Obliquing*.
- Select **OK** to close the dialog.
- After selecting any circle on the drawing, the values for its diameter, circumference and area are shown:



- After selecting any single line of text on the drawing, the values of its diameter, circumference and area are shown:

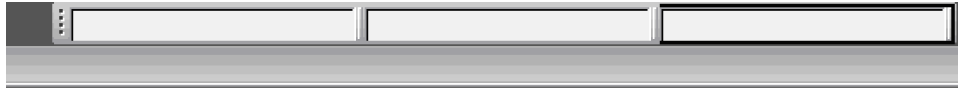


To delete a field from a new toolbar:

- Place the cursor over the field:



- When the field is selected, press **Alt** and drag the field into the drawing area:



Note: The **Disable All** switches off the display of properties' values not only in the user windows, but in the **Properties** and **Styles** standard toolbars.

Distributing object by layers

The main advantage of CAD is organising a document by layers. Layers are used to separate, sort and edit drawing objects.

To explain the function of layers we will use an example with tracing paper. Layers in the drawing are like transparent tracing papers. One tracing paper (or layer) contains the floor plan, the second – water supply system, the third – heating, the fourth – electric power supply etc. By combining different layers (tracing papers), you can create the required sets of design documentation.

Placing different groups of objects on separate layers helps to order and simplify many of the operations of drawing management. For better organisation and operation of data, drawing elements of one type are placed on the same layer. For example, auxiliary lines are placed on a separate layer to facilitate and accelerate their removal. Dimensions, text objects, hatches etc. can be placed on separate layers.

Every newly created object contains by default the **0** layer, which cannot be deleted or renamed. The drawing must contain at least one layer because any graphic object must be placed on a layer (it is often said that an object belongs to a layer).

It is not recommended to create all drawing objects on the **0** layer. It is recommended to create a new layer for the correct organisation of graphic objects.

When the first dimension is set, the **Defpoints** utility layer is created; this is where the control points of dimensions are set. The display view of these points is not changed with the **DDPTYPE** command. The **Defpoints** layer is never printable.

When an object is created, its properties are taken from the layer where it is placed (if **By layer** is selected).

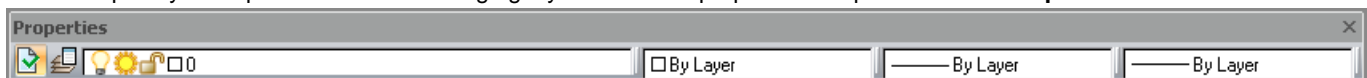
If you specify a particular colour, type and line weight instead of the **By layer** parameter in the **Color**, **Line type** and **Line Weight** drop down lists in the **Inspector** window, these values will be applied to all newly created objects on the layer.

To create a new object you do not need to specify its properties every time. You switch to the required layer and create the object.

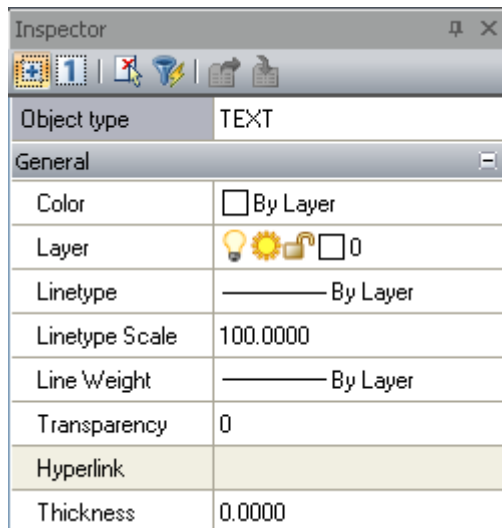
Distributing objects by layers helps to edit objects' properties separately. By blocking separate layers you can prevent editing of objects placed on these layers if you do not want accidental changes. Layers can be switched on/off, visible/invisible. Every layer can be made printable or unprintable.

An active layer, selected for working with, is called *current*. By default, newly created objects are placed on the *current layer*.

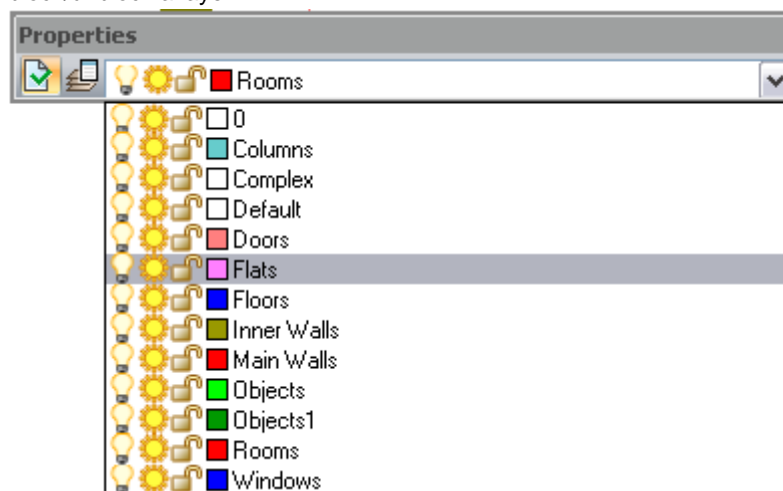
Some frequently used parameters for managing layers and their properties are placed on the **Properties** toolbar:



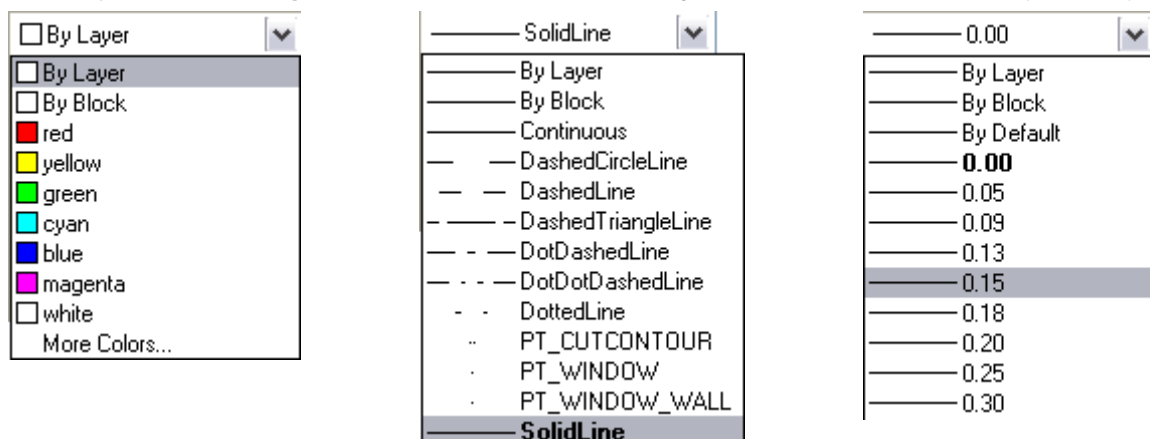
And in the **General** section of the **Inspector** window:



The **Layer** drop down list allows reassigning of the *current* layer, and also switch on/off, freeze/unfreeze and block/unblock a layer:



Color, **Line type** and **Line weight** drop down lists are used to change the properties of a selected layer quickly:



The **Layers** button on the **Properties** toolbar opens the **Layers** dialogue box where you can create, delete, rename and do other operations with layers.

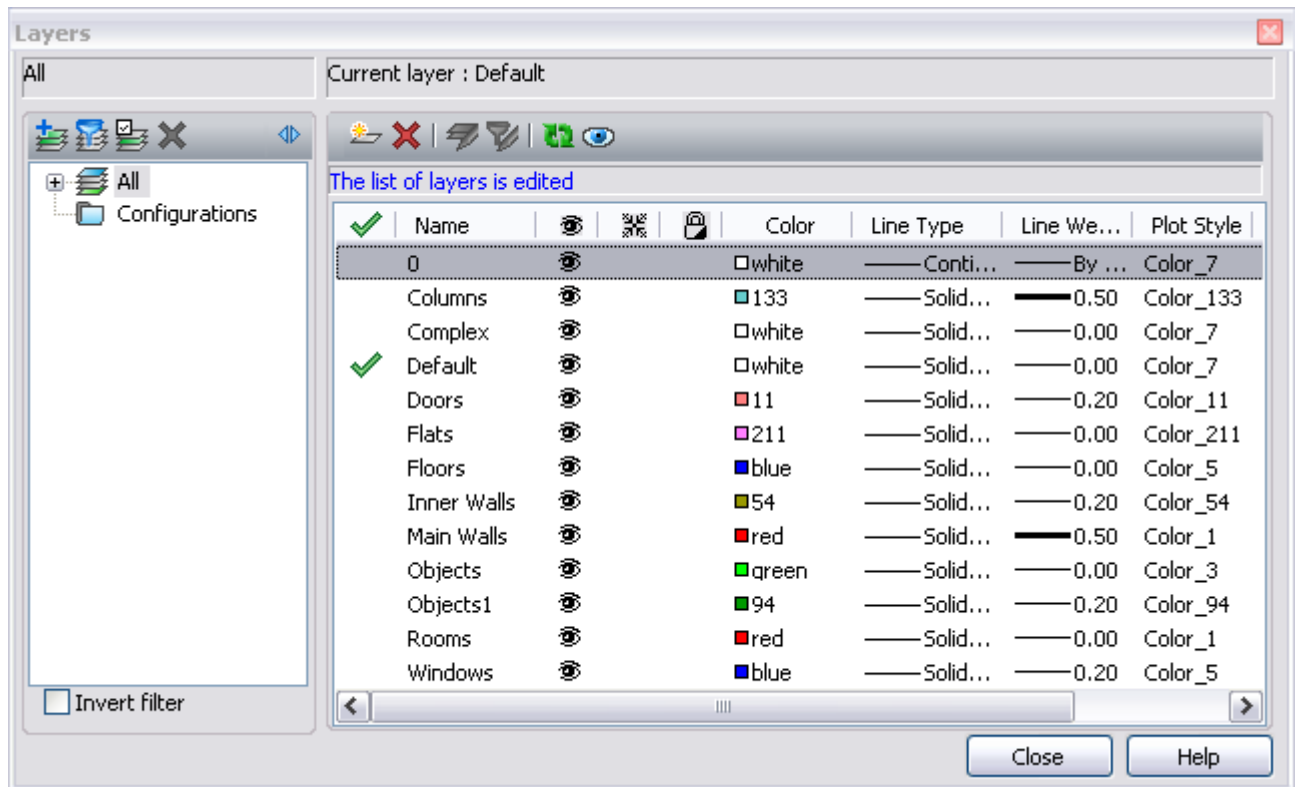
Layers

Menu: **Format** – **Layers...**

Toolbar: **Properties** –

Command line: **LAYER, LAYERS, LO**

The **Layers** dialogue box is used to manage layers and their properties:



There is a window in the left part of the dialogue box with a *category tree*. In general, a *category tree* consists of the following elements:

- All,
- All Used Layers,
- External reference,
- Filters,
- Groups,
- Configurations.

In the right part of the dialogue box is the list of layers for the selected element of the category tree.

Below the dialogue box title:

All->All Used Layers – the selected element from the category tree is shown,

Current layer : Default – the current layer is shown.

Parameters:

Tree categories window



Add group

The button adds groups of layers.



Add filter

The button adds a filter.



Add configuration

The button adds configurations of layers.



Delete

The button deletes the tree's elements.



Close/Open tree

The button folds/unfolds windows of the categories tree.

Remark:

Short description of the layers' configuration. The parameter is displayed above the **Invert filter** when you select the layers' configuration in the tree.

Invert filter

The Inversion mode for the display of layers in the layers' list.

List of edited layers

Buttons



Add



Delete



**Show all layers
(for group)**



Edit property



Refresh



**Single layer view
mode**

Buttons



Icon of the current layer.

Name

Column displays the layer's name.



Column displays the icon for visibility of layers.



Column displays the icon for freezing of layers .



Column displays the icon for blocking a layer.

Color

Column displays an icon for the layer's colour.

Line Type

Column displays the layer's line type.

Line Weight

Column displays the layer's line weight.

Plot Style

Column displays the layer's plot style.



Column displays the icon permitting printing.

Description

Short information about the layer.

Editing of a layer's parameters

To select layers in the dialogue box

Click on a layer to select it.

You can edit the parameters of several selected layers at once.

With **SHIFT** pressed, select the first and the last layer in the list. These layers and any layers between them will be selected.

With **CTRL** pressed, you can select individual layers from the list.

To sort layers by property

A list of layers can be sorted by any parameter. To sort a list of layers, click on the parameter column title.

For example, click on the **Color** title and all layers in the list are sorted by colour. Clicking on the title again reverses the order of the sorted layers:

| ✓ | Name | | | | Color |
|---|-------------|--|--|--|-------|
| | Doors | | | | 11 |
| | Columns | | | | 133 |
| | Flats | | | | 211 |
| | Inner Walls | | | | 54 |
| | Objects1 | | | | 94 |
| | Floors | | | | blue |
| | Windows | | | | blue |
| | Objects | | | | green |
| | Main Walls | | | | red |
| | Rooms | | | | red |
| | 0 | | | | white |
| ✓ | Complex | | | | white |
| ✓ | Default | | | | white |

| ✓ | Name | | | | Color |
|---|-------------|--|--|--|-------|
| | 0 | | | | white |
| ✓ | Complex | | | | white |
| | Default | | | | white |
| | Main Walls | | | | red |
| | Rooms | | | | red |
| | Objects | | | | green |
| | Floors | | | | blue |
| | Windows | | | | blue |
| | Objects1 | | | | 94 |
| | Inner Walls | | | | 54 |
| | Flats | | | | 211 |
| | Columns | | | | 133 |
| | Doors | | | | 11 |

Specifying the colour, type and width of lines in the layer

In *Layers* it is possible to specify all the properties that can be inherited by the objects created on it. Ensure the *By layer* value is set in the corresponding boxes of the object properties.

Colour, type and weight of lines can be specified for several layers at once:

- Select a layer to change its properties.
- Click on the required parameter in the column: *Colour*, *Lineweight* or *Linetype*.
- Select the value from the list.

Creation of a new layer

A newly created layer has default properties. After a new layer is created, you can change its properties.

To create a layer:

- Select button **Add**.

A new layer with the name *New Layer (N)* appears in the list. The default name for the new layer can be changed.

To rename a layer:

- Select layer in the dialogue box.
- Select *Name* in the **Layers** dialogue box
- Enter the new name.
- Select **ENTER**.

Removing a layer

Only layers not used in the document can be deleted. The current layer, even if it is not used cannot be deleted.

To delete a layer:

- Select layer in the dialogue box.
- Select the **Delete** button.

You can delete several unused layers at once.

To delete several layers:

- Select **All Used Layers** in the categories tree.
- Switch on the **Invert filter** **Invert filter** parameter.

Select all sorted unused layers and select the **Delete** button or **Delete** command from the context menu.

To make a layer active:

- Select a layer in the *Layers* dialogue box.
- Click in the column icon of the selected layer. The icon means that a layer is current.

To control layer visibility

Objects with visibility switched off are not displayed on layers and are unprintable, but are used in regeneration of a drawing. When layers are switched on/off, regeneration is not performed. It is recommended to switch on/off layers when you need to do it frequently and not for a long time. In other cases it is better to freeze them.

Visible layers are marked with icon. Absence of the icon means that visibility of the layer is switched off.


To switch on/off visibility of a layer:

- Select one or several layers in the list.

- Click in the column icon of the selected layer.

Freezing a layer

Objects placed on frozen layers are not displayed on the screen and are unprintable and not used in regeneration of the drawing. You can freeze unused layers to make display and regeneration operations faster. But freezing a layer causes regeneration of the drawing, which takes time. Freeze layers if you only do it occasionally or you want to freeze the layers for a long time. In other cases it is better to switch them off.

Frozen layers are marked with  icon.

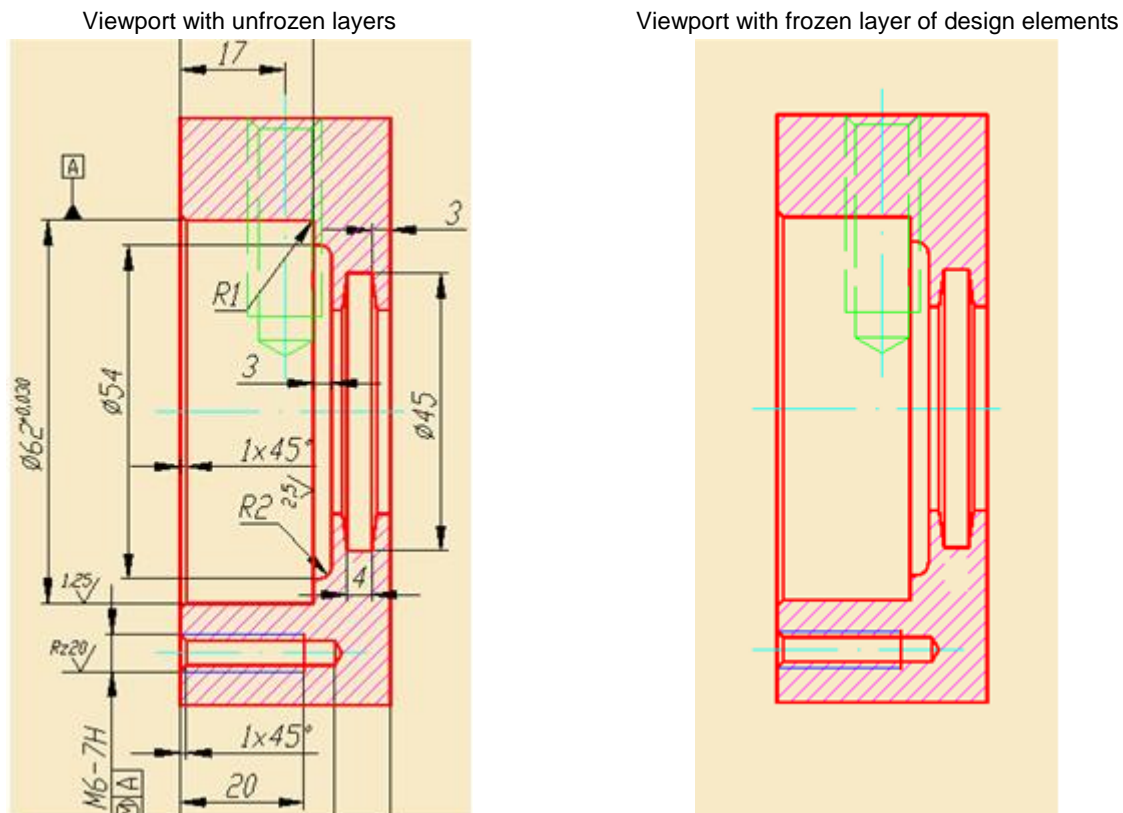
You can freeze and unfreeze several layers at once. The current layer cannot be frozen.

Locking or unlocking a layer to edit it:

- Select one or several layers in the list.
- Click on the column with the lock icon for the selected layer.

Freezing layers in the viewports of a layout

Layers can be frozen in separate viewports of a layout. You can get different displays of the same objects in different viewports without creating additional geometry; for example, you create two viewports for the same object and freeze the layer with design elements in the second viewport:




To freeze a layer in a viewport:

- Make the selected viewport active by double clicking on its frame.
- Open the **Layers** dialogue box.
- Freeze the selected layer.
- Close the **Layers** dialogue box.

The layer will be frozen only in this viewport and is visible in other viewports. It will not be printed only in this viewport.

Blocking a layer from changes

If a layer is unblocked, you can create a new object on it. Objects created before blocking are visible and they can be selected to view their properties, but they cannot be edited. You can snap to the object on a blocked layer with an object snap. You can change the colour, line type, weight type, make printable or unprintable on a blocked layer.


Blocked layers are marked with  icon.

You can block or unblock several layers at once.

To block/unblock layer:

- Select one or several layers in the list.
- Click on the block column of the selected layer.

Controlling layer printability

The  icon means that objects on the layer can be printed. Object on the layers without this icon will not be printed. You can make several layers printable or unprintable at once.

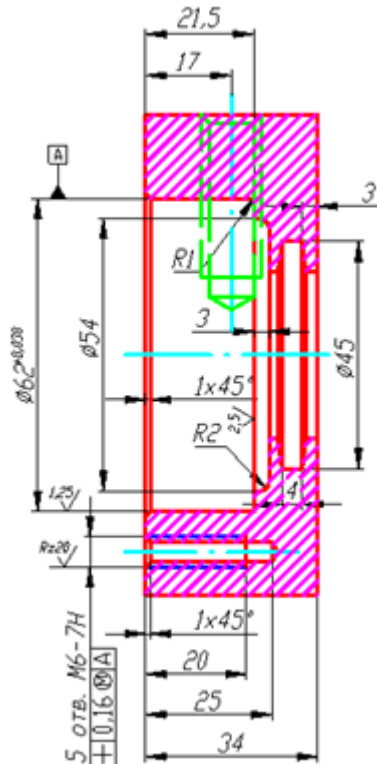
To make layers printable or unprintable:

- Select one or several layers in the list.
- Click on the printability column of the selected layer.

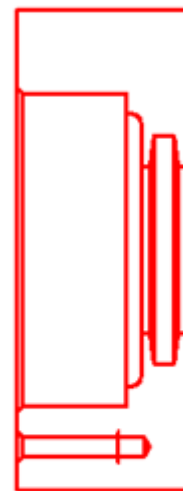
Single layer view mode

There is a mode to view only the selected layer in nanoCAD. Visibility of all layers in this mode, except the one selected, is switched off automatically; visibility is restored when you close this mode. This mode is a quick alternative to switching off the visibility of all layers to display content of one layer; it is very convenient if there are a lot of layers in the drawing.




All layer view mode



Single layer view mode



To switch on single layer view mode:

- Select the  **Single layer view mode** button, the  icons disappear for all layers.
- Select a layer by clicking (in the  column). Visibility of all layers, except the one selected, is temporarily switched off.

With the **Layers** dialogue box opened you can zoom and pan the drawing using the corresponding commands.

To restore visibility of all the layers, switch off the  **Single layer view mode** button or close the **Layers** dialogue box.

This mode is a quick alternative to switching off the visibility of all layers to edit the content of one layer. After selecting the mode, select any command and the **Layers** dialogue box closes and the selected layer will be displayed. To restore the visibility of all layers after editing, open the **Layers** dialogue box and the visibility of all layers is restored automatically.

Creating and editing group of layers





The **Layers** dialogue box allows merging of layers into groups. Grouping of layers is recommended when a document contains a lot of layers and you often need to change the settings of several layers; for example, to quickly block/unblock them or to control their printability.

One layer can belong to different groups.

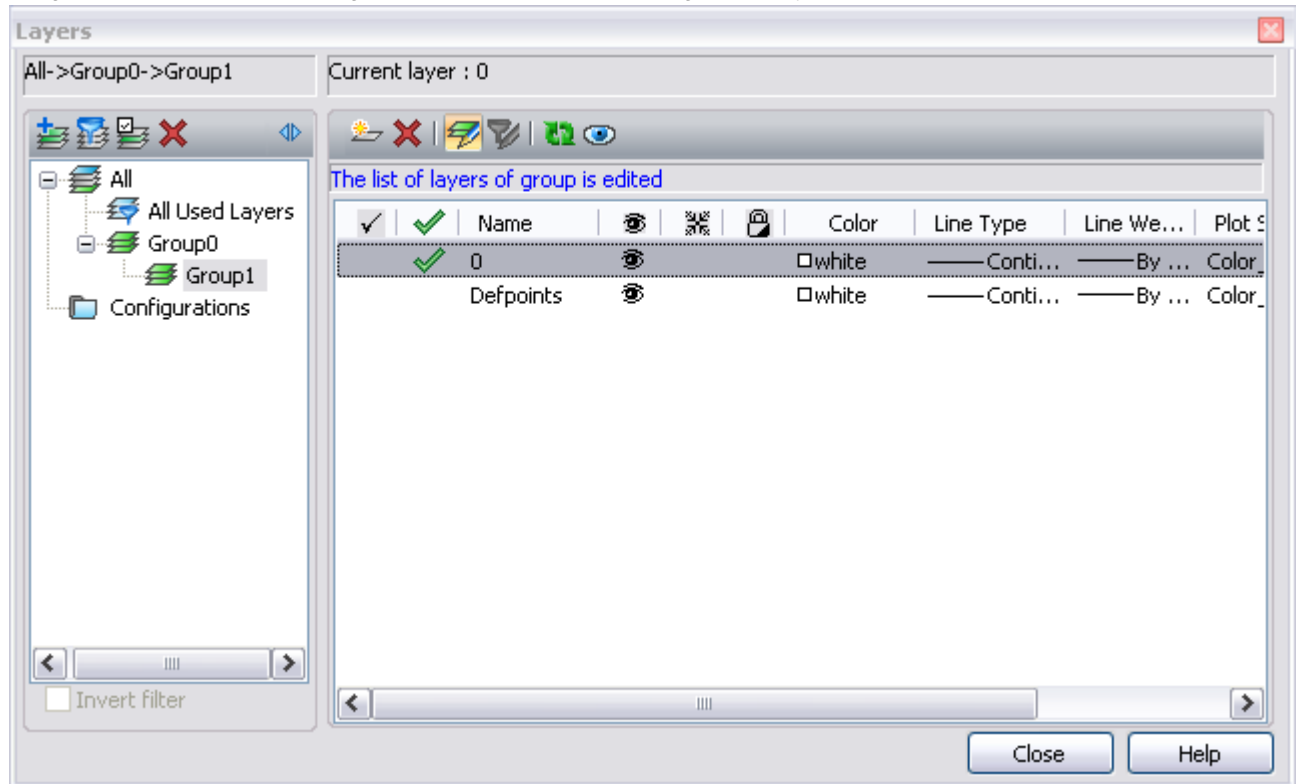
Creation of a new group of layers

To create a group of layers for all the document:

- Select **All** in the categories tree.

- Select the button  **Add group** and the button  **Show all layers (for group)**. To the left of the current layer icon column, the column  for selecting layers for the groups is shown.
- Enter a group name (the default name is Group...N, where N – is the number of the created group).
- Select layers to make a group.
- Select the button  **Show all layers (for group)** to create the group.

If a group is selected in the categories tree instead of **All**, a subgroup of layers will be created:





When adding new layers to a subgroup they are automatically added to all parent groups.

You cannot create a subgroup of layers for a filter.

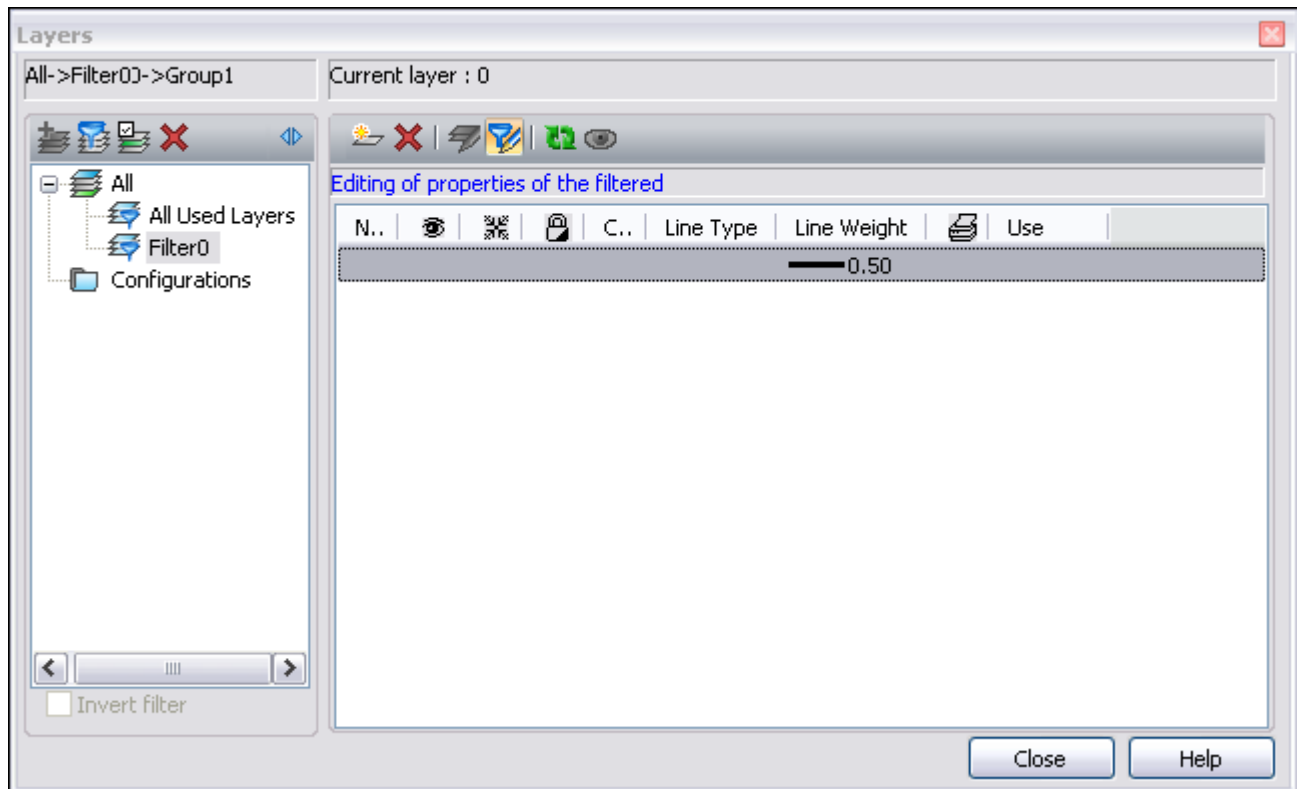
Editing a group of layers

To add or exclude layers from a group:

- Select the group in the categories tree.
- Select the **Show all layers (for group)**  button.
- In the  column for selecting layers for groups, select the layers to form the group. Layers of the parent group are always selected.

Creating and editing filters

A filter forms a list of layers that satisfy certain selection criteria. Selection in the filter is performed by one or several parameters of the layers; for example, selection of layers having line weight 0.50 which can be printed.



Creation of a new filter

To create a layers' filter for the whole document:

- Select **All** in the categories tree.
- Select the **Add filter** button. The **Edit property** button is automatically selected.
- Enter a filter name (a filter has a default name FilterN, where N is the number of the created filter).
- In the **Editing of properties of the filter** window, specify the selection criteria for the layers in the filter.
- Criteria, specified in one row are linked with the logical "AND".
- Criteria, specified in different rows are linked with the logical "OR".
- The **Add** and **Delete** buttons are used to add or delete rows of selection criteria.
- Select the **Edit property** button to finish filter creation.

A layer's name is often used as a selection criterion. Names can be specified as templates.

It is possible to use the following symbols in the template of a layer's name:





| Symbol in a template | | Corresponding symbols in a layer's name |
|----------------------|--------------------------|---|
| * | Asterisk | Any sequence of symbols. |
| ? | Question mark | Any symbol. |
| # | Hash | Any figure. |
| @ | At sign | Any letter. |
| . | Point | Any symbol except a letter or figure. |
| ~ | Tilde | Any sequence of symbols except the sequence after the tilde. |
| [] | Square brackets | Any symbol except the symbols in brackets. |
| [~] | Tilde in square brackets | Any symbol except the symbols in brackets. |
| [-] | Dash in square brackets | Any symbol from the range starting with the symbol to the left of the dash and ending with the symbol to the right of the dash. |
| ` | Backtick character | Backtick character shows that the following symbol is an ordinary symbol and not |

| | | |
|--|--|----------|
| | | special. |
|--|--|----------|











If you select a group instead of the **All** element, a filter is created for the group.

Editing layers' filter

To add to or exclude layers from a filter:

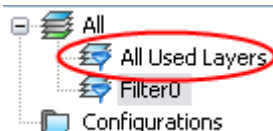
- Select the **All** in the categories tree (to create a filter for all layers in the document) or a group (to create a filter only for the group).
- Select the  **Edit property** button.
- In the **Editing of properties of the filter** window specify the selection criteria for layers in the filter.
- Criteria, specified in one row are linked with the logical "AND".
- Criteria, specified in different rows are linked with the logical "OR".
- The  **Add** and  **Delete** buttons are used to add or delete rows of selection criteria.
- Select the  **Edit property** button to finish filter creation.

Criteria symbols to select layers for a filter:

| | |
|---|---------------------------------|
|  | Visible layer. |
|  | Invisible layer. |
|  | Frozen layer. |
|  | Unfrozen layer. |
|  | Blocked layer. |
|  | Unblocked layer. |
|  | Layer available for printing. |
|  | Layer unavailable for printing. |
|  | Used layer. |
|  | Unused layer. |

"All used layers" filter

By default, all nanoCAD documents contain a layer's filter **All used layers**:

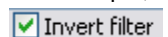


The filter shows all layers used in the current document. A filter does not show unused layers in a document. You cannot delete this filter.

"Invert layers" checkbox

The **Invert filter** parameter below the categories tree can be used to display all the layers, which are not in the group, filter or configuration.

For example, to display all unused layers in a document, select the **All used layers** and select the **Invert filter**.



When you deselect the **Invert filter** the list of layers recovers.

It is often needed to display layers of a particular category.

You can use the **Invert filter** for this:


- Select a group or a filter in the categories tree.
- Select the **Invert filter**.
- Select all layers in the list and make them invisible.

Configurations of layers

Configurations of layers are backup copies of layers' parameters.

Configurations are placed in the separate **Configurations** folder in the categories tree.

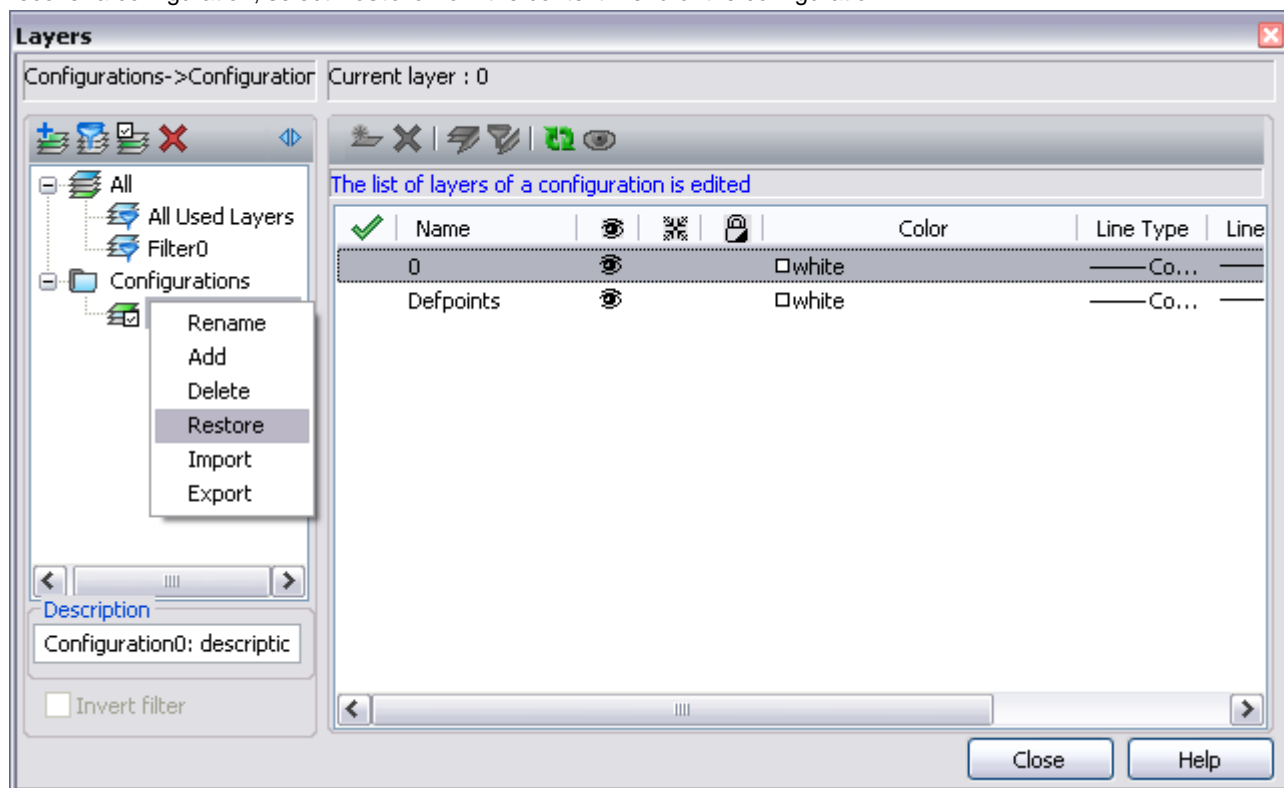
A configuration is created for all layers in the document.

To create a configuration, select the  **Add configuration** button. A new configuration has a default name – Configuration N, where N is the number of created configurations.

A created configuration can be edited, e.g. to change the parameters of its layers.

Configurations can be restored, renamed and deleted.

A recovery of a configuration places a copy of the parameters of the layer with the same name into the document; it means recovering the parameters of the layers and returning them to the moment the configuration was created. To recover a configuration, select **Restore** from the context menu of the configuration:



A configuration can be exported to a LAS-file and imported from a LAS-file or *.dwg file (**Import** and **Export** commands of the context menu).

Line types



Menu: **Format – Styles...**



Command line: **LINETYPE**

Different line types have different purposes; for example, to create axes you use one type of line, another to create invisible lines –etc. Using different line types to create different objects improves visual perception of graphic data and makes working with drawings more efficient.

Line type is a repeated sequence of lines, dots and spaces along a line or curve. Complex types of lines can contain built-in shapes which are stored in the (*.SHX) shape file. Line types are stored in files with a *.LIN extension. Every type has its own name; sequence of dashes, points, relative lengths of dashes and spaces and other characteristics are specified in a line's description. One LIN-file can contain many line types.

There is a GOST 2.303-68.lin file, containing descriptions of GOST line types, in nanoCAD. After installation of the program, you can find this file in *c:\Documents and Settings\All Users\Application Data\Nanosoft\nanoCAD 2.5\SHX*. There is a GOST 2.303-68.shx file, containing shapes, used to describe line types in the GOST 2.303-68.lin file, in this folder.

nanoCAD supports line types created for AutoCAD.

Users can create their own line types by adding their descriptions to an existing LIN-file or creating their own new files. Edit an existing file or create a new file in any text editor or text processor.

There is a built in **Line type editor** to create user line types and edit existing ones.

Before using a line type, you have to load it into the drawing. Loaded line types can be renamed when you are working with the drawing. Renaming a line type only changes its description in the current drawing; the name of the line type stays the same in the LIN-file.

Unused line types can be deleted in the **Linetype Manager** with the **Purge** command (**File** menu – **Drawing Utilities**).

Note: By layer, **By block** and **Solid line** types cannot be renamed or deleted.

All newly created objects inherit the line type currently set in the **Linetype** row of the **Inspector** window.

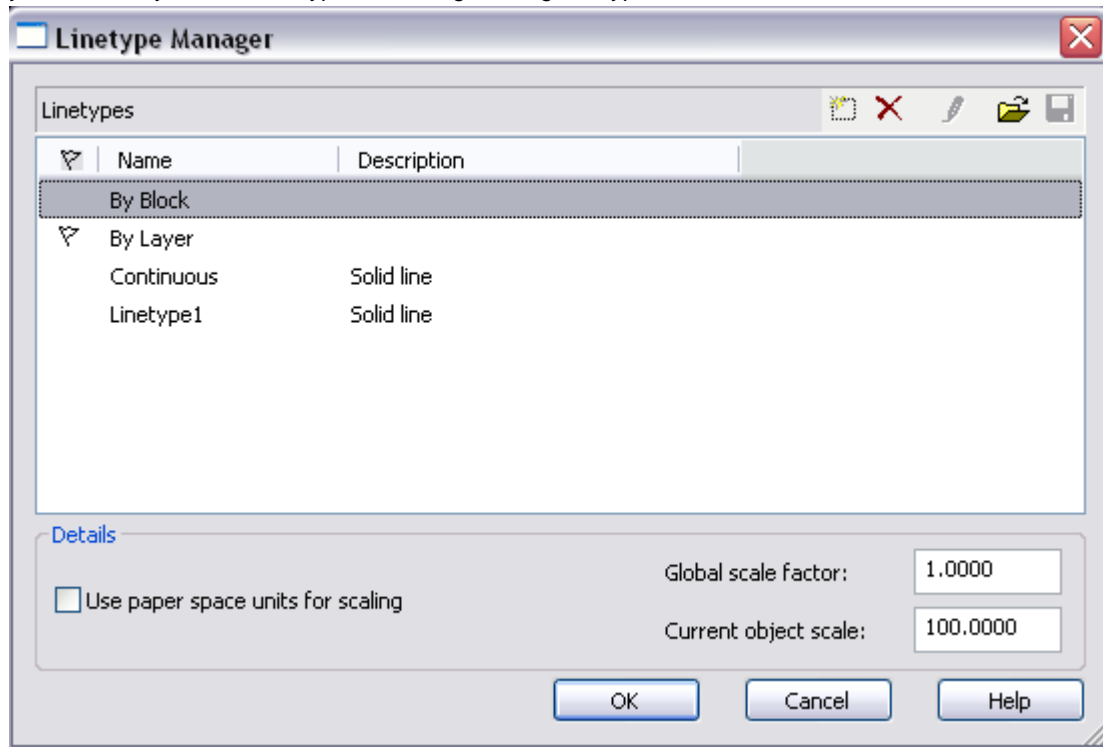
Note: The **Linetype** drop-down list only contains loaded line types, shown in the **Linetype Manager**.

If a current line type is specified By layer, new objects will have the line types defined for this layer.

Line types can be assigned not only to layers, but also to objects. To change an object's line type, replace it on the layer with another line type, change the line type of the layer where it is placed or specify another line type especially for the object.

Line types are saved with the document in the *.dwg file. They can be saved in a (*.dwt) file template to transfer to another computer.






Information about all line types in the document is shown in the **Linetype Manager**. **Linetype Manager** allows loading, deleting and renaming of line types. There is a button to open the built-in **Edit Linetype** dialogue box which allows you to create your own line types or editing existing line types.



Parameters:

Line types Shows the list of line types loaded in the current document.

Buttons

-  **Add linetype** Creation of a new style of line type using a selected type from the list as a base.
-  **Delete linetype** Removal of the selected line type from the current document.
-  **Edit linetype** Opens the **Linetype Editor**.
-  **Open linetype** Imports line types into the current document.
-  **Save linetype** Saves the selected line types into a file with *.LIN extension.

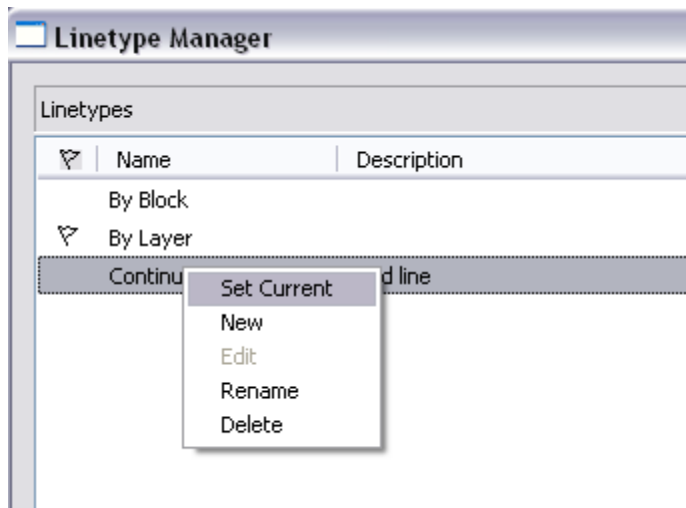
Details

Use paper space units for scaling Sets the same scale of line types for paper space and model space. This option is useful if you use several viewports at once.


Global scale factor: Sets a global scale factor for all line types.

Current object scale: Sets the scale factor for line types for newly created objects. The scale is a multiplication of the global scale and the current scale.


Note: There is a context menu in the **Linetype Manager**:




To set a line type as current:

- Select the required line type in the dialogue box.
- Click in the left field of the selected line type. The  tag appears, the line type is set as current.


To import line types:

- Select the  **Load line type**.
- In the **Open Line Type Import** dialogue box, specify the path to load the line type file (*.LIN), containing the descriptions of the line types.
- In the **Load/Reset Line Types** dialogue box, select the line types for import. To select several line types, use the **SHIFT** or **CTRL** buttons.
- Select **OK** to start loading the selected line types.
- Select **OK** to close the **Linetype Manger** dialogue box.

To export line types:

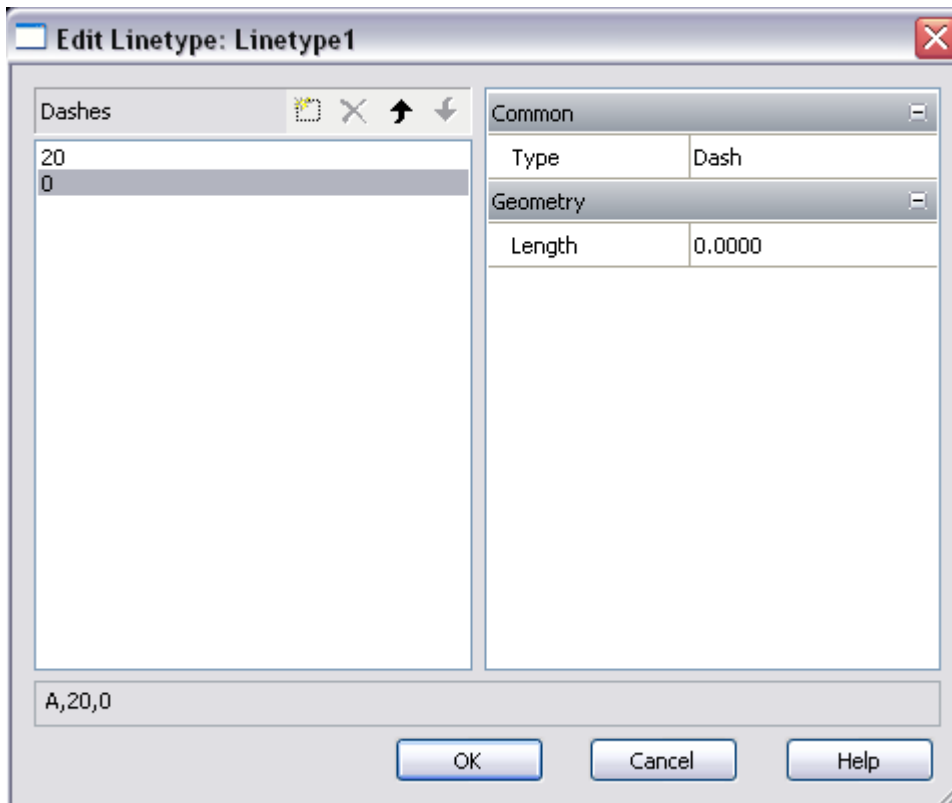
- In the dialogue box, select one or several line types to export (you can use **SHIFT** and **CTRL** buttons).
- Select the  **Save linetype** button.
- In the **Save Line Type Import** select the folder and specify a name.
- Select the **Save** button.
- Select **OK** to close the **Linetype Manger** dialogue box.

To delete a line type:

- In the dialogue box, select one or several line types to delete (you can use **SHIFT** and **CTRL** buttons).
- Select the  **Delete** button.
- Select **OK** to close the **Linetype Manger** dialogue box.

Line type Editor

In the **Linetype Manager** you can edit the selected line type or created a new one using the built-in **Linetype Editor**:



Parameters:

Description of line types

Description of line types is shown.

Buttons



Add dash

Adds a new element to a line.



Delete dash

Deletes an element from a line.



Move dash up

Moves one element up.



Move dash down

Moves one element down.



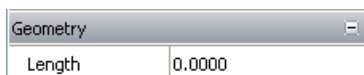
Elements are selected in the **Common** section.

There are the following options in the drop-down list:

Dash

Text

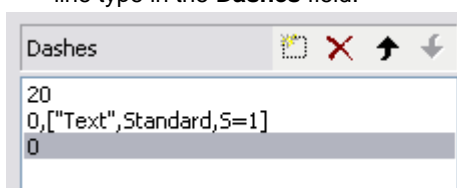
Shape



An element's value is shown in the **Geometry** section.

To edit a line type:


- Select a line type in the **Linetype Manager**.
- Select the  **Edit linetype** button to start the built-in **Linetype Editor**. There is a description of the selected line type in the **Dashes** field:



- Select an element and enter its new value in the **Length** field of the **Geometry** section.

Note 1: When you specify a positive value, a dash is drawn, when you specify a negative value, a space is drawn, when a value is zero a point is drawn.

Note 2: The length of the first element must be greater than or equal to zero, i.e. the dash or point must be drawn first.

- To display a new value, press **ENTER** or just click.
- To add a new element, select the  **Add dash** button. A new element is added to the end of the list and a zero value is assigned to it by default.
- Specify a new value for an added element in the **Length** field of the **Geometry** section. To display a new value press **ENTER**.
- Select **OK** to close the **Edit Linetype** dialogue box.
- Select **OK** to close the **Linetype Manger** dialogue box.

To add text to a description of a line type:

- In the **Type** field select the **Text** option:

| | |
|--------|-------|
| Common | |
| Type | Dash |
| | Dash |
| | Text |
| | Shape |

- In the **Style** field of the **Text** section, select a text style from the drop-down list. In the **Text** field enter symbols, specify parameters in the **Geometry** section:

| | |
|-------------------|----------|
| Common | |
| Type | Text |
| Text | |
| Style | Standard |
| Text | Text |
| Geometry | |
| Length | 0.0000 |
| Scale | 1.0000 |
| Rotation | 0 |
| Absolute Rotation | No |
| Offset X | 0.0000 |
| Offset Y | 0.0000 |

- Select **OK** to close the **Edit Linetype** dialogue box.
- Select **OK** to close the **Linetype Manger** dialogue box.

To add a shape to a description of a line type:

- In the **Type** field select the **Shape** option:

| | |
|----------|-------|
| Common | |
| Type | Dash |
| Geometry | Dash |
| | Text |
| Length | Shape |

- В разделе **Форма** в поле **Файл** выбрать из раскрывающегося списка файл с формами, в поле **Имя** выбрать из раскрывающегося списка имя формы, в разделе **Геометрия** задать необходимые параметры:

This needs changing to English



| Общие | |
|-----------------|--------------|
| Тип | Форма |
| Форма | |
| Файл | CS_Gost23... |
| Имя | LINE FEED |
| Геометрия | |
| Длина | 20.0000 |
| Масштаб | 1.0000 |
| Угол поворота | 0 |
| Абсолютный п... | Нет |
| Смещение по X | 0.0000 |
| Смещение по Y | 0.0000 |

This needs changing to English

- Select **OK** to close the **Edit Linetype** dialogue box.
- Select **OK** to close the **Linetype Manger** dialogue box.

Creation of a new line type




A new line type is created on the base of an existing one:

- In the **Linetype Manager** select a line type.
- Select the  **New linetype** button. The new row has a default name – **Linetype1** appears.
- Click **Linetype1** and enter a new name.
- In the **Description** field enter information about the line type.
- Select the  **Edit linetype** button.
- Change the parameters in the **Edit Linetype** dialogue box.
- Select **OK** to close the **Edit Linetype** dialogue box.
- Select **OK** to close the **Linetype Manger** dialogue box.

Managing the transparency of objects

Objects in nanoCAD have transparency properties. The transparency is used to reduce the visibility of objects or drawing fragments on the screen to increase the readability of the drawing.

The **Transparency** parameter is displayed in the **Inspector** dialogue box (**General** section) when one or several objects are selected and can have values from 0 to 100:

| Inspector | |
|----------------|---|
| Object type | CIRCLE |
| General | |
| Color | <input type="checkbox"/> By Layer |
| Layer |    0 |
| Linetype | ——— By Layer |
| Linetype Scale | 100.0000 |
| Line Weight | ——— By Layer |
| Transparency | 70 |
| Hyperlink | |
| Thickness | 0.0000 |



If a value is 100, the selected object becomes invisible. Totally opaque objects have values of 0. The transparency of objects is not shown when printing or previewing the document.

To set the transparency for an object:

- Select an object.
- In the *Transparency* field of the *General* section in the **Inspector** dialogue box, specify the required value.

Geometric objects

Base elements to create a drawing are graphic primitives – nanoCAD objects, which are whole objects and cannot be separated with the **Explode** command.

Auxiliary lines

Lines, infinite in one or both directions, are called rays and lines. Their lines can be used as auxiliary lines to create objects. It is recommended that auxiliary lines are created on a separate layer, which can be frozen or made unprintable. Auxiliary lines can be edited: copy, replace, rotate etc.

Ray



Menu: **Draw –**  **Ray**



Toolbar: **Draw –** 



Command line: **RAY**

This command creates a ray – a semi-infinite construction line.

Command prompts:

Specify start point: *Specify start point.*

Specify through point: *Specify second point.*

Specify through point: *Specify next point or press **ESC** to finish the command.*

Construction Line



Menu: **Draw –**  **Construction Line**



Toolbar: **Draw –** 



Command line: **XL, XLINE**

This command creates an infinite construction line.

Command options:

Hor Creation of infinite line through the specified point and parallel X axis.

Vert Creation of infinite line through the specified point and parallel Y axis.

Command prompts:


Specify start point or [Hor/Vert]: *Specify start point.*

Specify through point: *Specify second point.*

Linear objects

Point



Menu: **Draw – Point >**  **Single Point**



Toolbar: **Draw –** 



Hotkeys: **CTRL+ALT+T**



Command line: **PO, POINT**

The command allows the creation of point objects which can be shown as a common point or a special symbol. Points are used to mark or as reference points for object snap. A point is shown as a special symbol because a point object of the default style is hard to see on the screen.

The **Point Styles** dialogue box (**Format** menu) allows changing of the size of a point object and its style.

Note: Changing the size and/or style of point objects affect not only new objects, but also created objects.

Command prompts:

Specify point: Specify the point coordinates or click on the drawing.

Specify point: Specify the next point or press **ESC** to finish the command.

Size and style of points

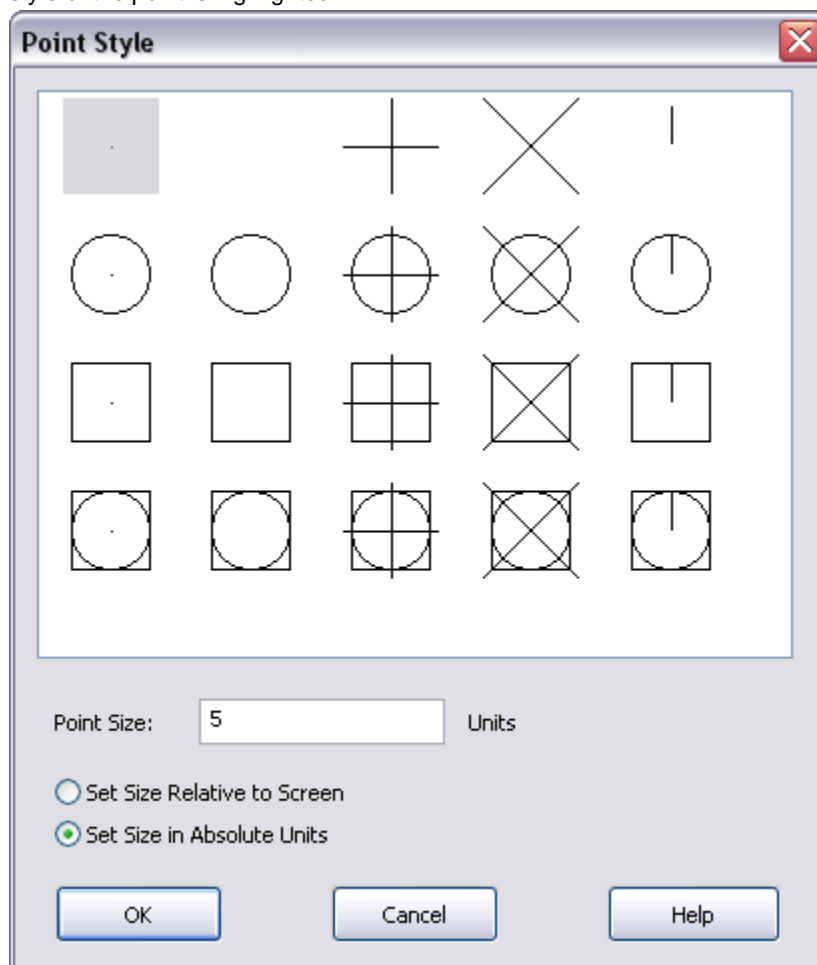


Menu: **Format** –  **Points Styles...**



Command line: **DDPTYPE**

The **Point Style** dialogue box is used to display and change the current style and size of a point. The current display style of the point is highlighted.



Parameters:

Point Size: Specify point size.

Set Size Relative to Screen Shows point size as a percentage of the screen size (the point size is not changed if the screen is zoomed).

Set Size in Absolute units The point size is changed if the screen is zoomed.

To change the current size and style of a point:

- Select the style;
- Specify the point size in the **Point Size** dialog.

Two points



Menu: **Draw** –  **Two points**



Toolbar: **Draw** – 



Hotkeys: **CTRL+ALT+L**



Command line: **L, LINE**

The two point segment is one of the base primitives and is often used for drawing. Segments can be drawn separately and merged into zigzag lines.

Command options:

Undo Deletes the last created segment.

Close Creates the segment, connecting the last point of the last segment with the start point of the first segment. This option is available after the creation of the second segment.



Command prompts:

Specify first point:

Specify point 1.

Specify next point:

Specify point 2.

Specify next point [**Undo**]:

*Specify next point or select **Undo** to cancel previous action.*

Specify next point [**Undo/Close**]:

*Specify next point or select **Close** to create closed contour.*

Specify next point [**Undo/Close**]:

*Specify next point or press **ESC** to finish the command.*

Polyline



Menu: **Draw** –  **Polyline**



Toolbar: **Draw** – 



Hotkeys: **CTRL+ALT+P**



Command line: **PL, PLINE**

A polyline can contain linear and arc segments and their combinations and is a graphic primitive. Arc segments of a polyline are created by two points, or by angle, centre and tangent or arc radius.

Command options of linear segments:

Arc Arc segment mode.

Close Closes a polyline by linear segment.

Halfwidth Halfwidth of polyline segment.

Length Length of segment, created as continuation of the previous segment and in the same direction.

Undo Deletes the last created segment.

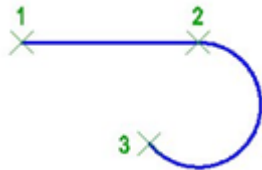
Width Width of following segments.

Command options of arc segments:

Angle Centre angle (positive value – arc is created counterclockwise; negative value – arc is created clockwise).

Center Centre of arc.

| | |
|------------------|--|
| <u>Close</u> | Closes a polyline with an arc segment. |
| <u>Direction</u> | Direction of tangent. |
| <u>Halfwidth</u> | Halfwidth of polyline segment. |
| <u>Line</u> | Linear segment mode. |
| <u>Radius</u> | Arc radius. |
| <u>Second pt</u> | Second point of arc in three point creation. |
| <u>Undo</u> | Deletes the last point. |
| <u>Width</u> | Width of following segments. |



Command prompts:

Specify start point:

Specify next point or

[Arc/Halfwidth/Length/Undo/Width]:

Specify next point or

[Arc/Close/Halfwidth/Length/Undo/Width]:

Specify endpoint of arc or

[Angle/Center/Close/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:

Specify point 1.

Specify point 2.

Select Arc to create an arc segment.

Specify endpoint 3 of arc. Press **ENTER** to finish command.

Polygon



Menu: **Draw** –  **Closed Polygon**



Toolbar: **Draw** – 



Command line: **POL, POLYGON**

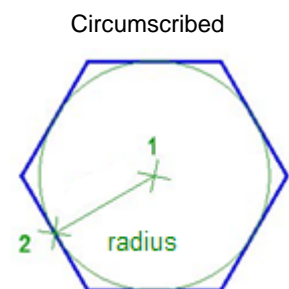
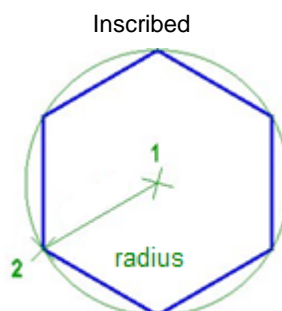
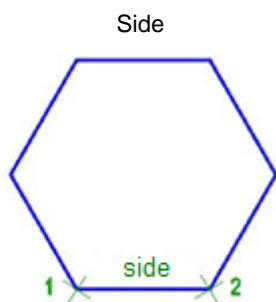
The command creates polygons with similar sides from closed polylines. The rotation angle of the polygon is specified from the horizontal axis before point 2.

Command options:

Edge Creation of a polygon by edge.

Inscribed Creation of an inscribed polygon.

Circumscribed Creation of a circumscribed polygon.



Command prompts when creating a polygon by side:

Enter number of sides <6>:

Specify center of polygon or [Edge]:

Enter number of sides for polygon.

Select Edge.

Specify first endpoint of edge:

Specify second endpoint of edge:

Command prompts when creating a polygon by radius:

Enter number of sides <6>:

Specify center of polygon or [Edge]:

Extend <"Inscribed"> or
[Inscribed/Circumscribed]:

Specify radius of circle:

Specify point 1.

Specify point 2.

Enter number of polygon's sides.

Specify centre point 1.

Select option.

Specify radius (point 2).

Rectangle

Rectangle by two points



Menu: **Draw – Rectangle by >**  **Two points**



Toolbar: **Draw –** 



Hotkeys: **CTRL+ALT+R**



Command line: **REC,RECT,RECTANG,RECTANGLE**

The command creates a rectangle from two opposite points.

Command options:

Chamfer Fillet sizes for rectangle's corner.

Elevation Current elevation.

Fillet Fillet of rectangle's corners.

Thickness Current three-dimensional height.

Width Width of polyline to create a rectangle.

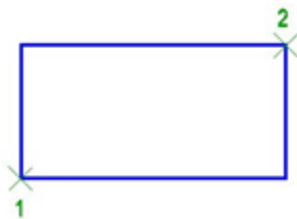
Area Area of rectangle.

Length Length of rectangle.

Width Width of rectangle.

Dimension Height and length of rectangle.

Rotation Rotation angle of rectangle.



Command prompts:

Specify first corner point or
[**Chamfer**/**Elevation**/**Fillet**/**Thickness**/**Width**]:

Specify other corner point or
[**Area**/**Dimensions**/**Rotation**]:

Specify first corner (point 1).

Specify second corner (point 2).

Rectangle by three points



Menu: **Draw – Rectangle by >**  **Three points**

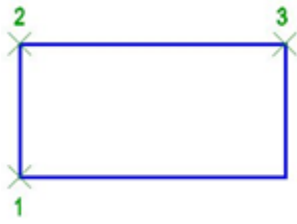


Toolbar: **Draw –** 



Command line: **FRAME**

The command creates a rectangle by angle and two sides.



Command prompts:

Specify first point of rectangle:

Specify second point of rectangle:

Specify third point of rectangle:

Specify rectangle's angle (point 1).

Specify length of first side (point 2).

Specify length of second side (point 3).

Curved objects

Arc

An arc can be created with different methods. There are 11 ways to create an arc in nanoCAD.

Arc by three points



Menu: **Draw – Arc >**



3 points



Toolbar: **Draw –**



Hotkeys: **CTRL+ALT+A**



Command line: **A, ARC**

Command options:

Center Arc centre.

Angle Arc angle.

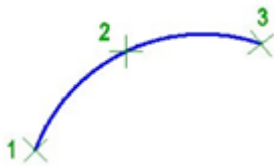
chordLength Chord length.

End End point of arc.

Angle Arc angle.

Direction Direction of tangent from the start point of the arc.

Radius Radius.



Command prompts:

Specify start point of arc or **[Center]**:

Specify second point of arc or **[Center/End]**:

Specify end point of arc:

Specify start point 1.

Specify second point 2.

Specify end point 3.

Arc by centre, start and angle



Menu: **Draw – Arc >**



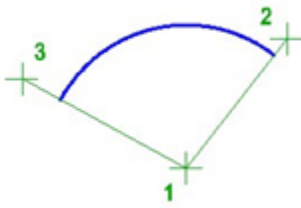
Center, start, angle



Toolbar: **Draw** –



Command line: **ARC2**



Command prompts:

Specify center
point of arc:

Specify arc centre (point 1).

Specify start point
of arc:

Specify start point 2.

Specify included
angle:

Specify angle (point 3).

Arc by centre, angles and radius



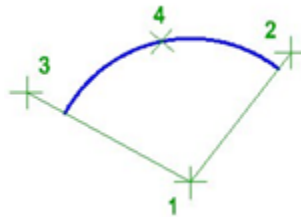
Menu: **Draw** – **Arc** >  **Center, angles, radius**



Toolbar: **Draw** –



Command line: **ARC3**



Command prompts:

Specify center
point of arc:

Specify centre of arc (point 1).

Specify start angle
of arc:

Specify start angle (point 2).

Specify end angle
of arc:

Specify end angle (point 3).

Specify radius of
arc:

Specify radius (point 4).

Arc by centre, start and end



Menu: **Draw** – **Arc** >  **3 points**



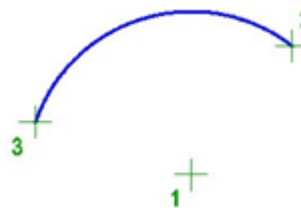
Toolbar: **Draw** –



Hotkeys: **CTRL+ALT+A**



Command line: **A, ARC**



Command prompts:

Specify start point of arc or [Center]: Select Center.

Specify center point of arc:
Specify start point of arc:
Specify end point of arc or
[Angle/chordLength]:

Specify arc centre (point 1).
Specify start point 2.
Specify end point 3.

Arc by centre, start and chord length



Menu: Draw – Arc >  3 points



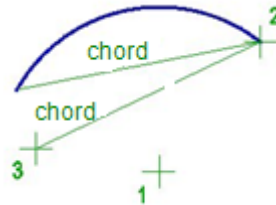
Toolbar: Draw – 



Hotkeys: CTRL+ALT+A



Command line: A, ARC



Command prompts:

Specify start point of arc or [Center]:
Specify center point of arc:
Specify start point of arc:
Specify end point of arc or
[Angle/chordLength]:
Specify length of chord:

Select Center.
Specify arc centre (point 1).
Specify start point 2.
Select chordLength.
Specify chord length (point 3).

Arc by start, centre and end



Menu: Draw – Arc >  3 points



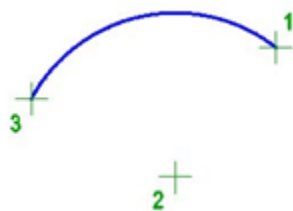
Toolbar: Draw – 



Hotkeys: CTRL+ALT+A



Command line: A, ARC



Command prompts:

Specify start point of arc or [Center]:
Specify second point of arc or
[Center/End]:
Specify center point of arc:
Specify end point of arc or
[Angle/chordLength]:

Specify start point 1.
Select Center.
Specify arc centre (point 2).
Specify end point 3.

Arc by start, centre and angle



Menu: Draw – Arc >  3 points



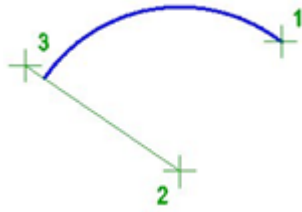
Toolbar: Draw – 



Hotkeys: CTRL+ALT+A



Command line: **A, ARC**



Command prompts:

Specify start point of arc or [Center]:

Specify start point 1.

Specify second point of arc or [Center/End]:

Select Center.

Specify center point of arc:

Specify arc centre (point 2).

Specify end point of arc or [Angle/chordLength]:

Select Angle.

Specify included angle:

Specify centre angle (point 3).

Arc by start, centre and chord length



Menu: **Draw – Arc >**  **3 points**



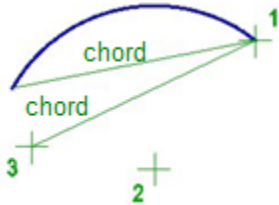
Toolbar: **Draw –** 



Hotkeys: **CTRL+ALT+A**



Command line: **A, ARC**



Command prompts:

Specify start point of arc or [Center]::

Specify start point 1.

Specify second point of arc or [Center/End]:

Select Center.

Specify center point of arc:

Specify arc centre (point 2).

Specify end point of arc or [Angle/chordLength]:

Select chordLength.

Specify length of chord:

Specify chord length (point 3).

Arc by start, end and angle



Menu: **Draw – Arc >**  **3 points**



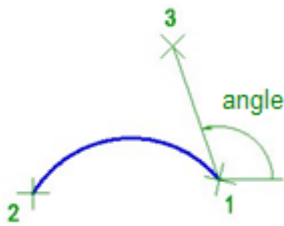
Toolbar: **Draw –** 



Hotkeys: **CTRL+ALT+A**



Command line: **A, ARC**



Command prompts:

Specify start point of arc or [Center]:

Specify second point of arc or [Center/End]:

Specify end point of arc:

Specify center of arc or [Angle/Direction/Radius]:

Specify included angle:

Specify start point 1.

Select End.

Specify end point 2.

Select Angle.


Specify centre angle (point 3).

Arc by start, end and direction



Menu: Draw – Arc >  3 points



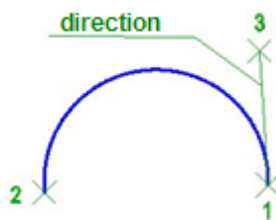
Toolbar: Draw – 



Hotkeys: CTRL+ALT+A



Command line: A, ARC



Command prompts:

Specify start point of arc or [Center]:

Specify second point of arc or [Center/End]:

Specify end point of arc:

Specify center of arc or [Angle/Direction/Radius]:

Specify tangent direction for the start point of arc:

Specify start point 1.

Select End.

Specify end point 2.

Select Direction.

Specify tangent direction (point 3).

Arc by start, end and radius



Menu: Draw – Arc >  3 points



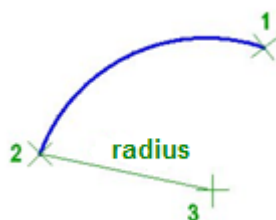
Toolbar: Draw – 



Hotkeys: CTRL+ALT+A



Command line: A, ARC




Command prompts:

Specify start point of arc or [Center]: Specify start point **1**.
 Specify second point of arc or [Center/End]: Select End.
 Specify end point of arc: Specify end point **2**.
 Specify center of arc or [Angle/Direction/Radius]: Select Radius.
 Specify radius of arc: Specify arc radius (point **3**).

Circle

Circle by centre and radius



Menu: **Draw – Circle >**  **Center, radius**



Toolbar: **Draw –** 



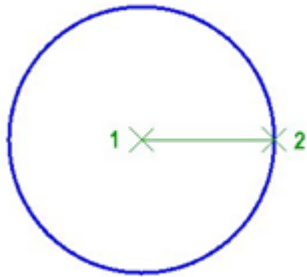
Hotkeys: **CTRL+ALT+C**



Command line: **C, CIRCLE**

Command options:

- 3P** Circle by three points.
- 2P** Circle by diameter.
- TTR** Circle by two tangents and radius.
- Diameter** Circle by centre and diameter.




Command prompts:

Specify center point of circle or [3P/2P/TTR]: Specify centre (point **1**).
 Specify radius: Specify radius of circle (point **2**).

Circle by centre and diameter



Menu: **Draw – Circle >**  **Center, radius**



Toolbar: **Draw –** 



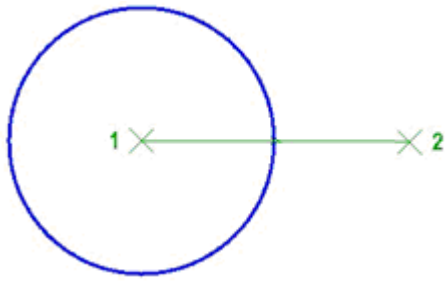
Hotkeys: **CTRL+ALT+C**



Command line: **C, CIRCLE**

Command options:

- 3P** Circle by three points.
- 2P** Circle by diameter.
- TTR** Circle by two tangents and radius.
- Diameter** Circle by centre and diameter.



Command prompts:

| | |
|---|-----------------------------|
| Specify center point of circle or [<u>3P</u> / <u>2P</u> / <u>TTR</u>]: | Specify centre (point 1). |
| Specify radius or [<u>Diameter</u>]: | Select <u>Diameter</u> . |
| Specify diameter: | Specify diameter (point 2). |

Circle by diameter



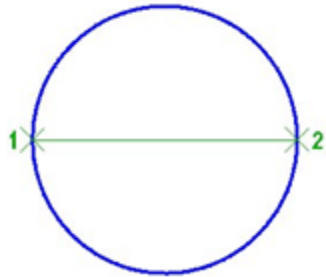
Menu: **Draw – Circle** >  **Diameter**



Toolbar: **Draw** – 



Command line: **CIRCLE2**



Command prompts:

| | |
|---|---|
| Specify first end point for circle's diameter: | Specify first point 1 for circle diameter |
| Specify second end point for circle's diameter: | Specify end point 2 for circle diameter |

Circle by three points



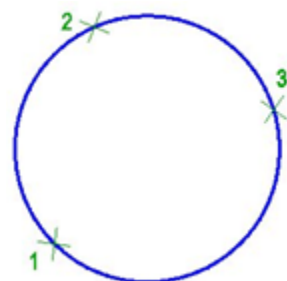
Menu: **Draw – Circle** >  **3 points**



Toolbar: **Draw** – 



Command line: **CIRCLE3**




Command prompts:

| | |
|---------------------------------|-----------------------|
| Specify first point on circle: | Specify first point. |
| Specify second point on circle: | Specify second point. |
| Specify third point on circle: | Specify third point. |

circle:

Circle by two tangents and radius



Menu: **Draw – Circle >**  **Center, radius**



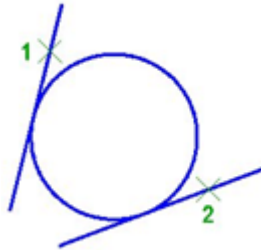
Toolbar: **Draw –** 



Hotkeys: **CTRL+ALT+C**



Command line: **C, CIRCLE**



Command prompts:

Specify center point of circle or
[**3P**/**2P**/**TTR**]:

Specify a point on first tangent
object:

Specify a point on second tangent
object:

Specify radius:

Select the **TTR**.


Select a first tangent object (point **1**).

Select a second tangent object (point **2**).

Specify radius of circle.

Spline



Menu: **Draw –**  **Spline**



Toolbar: **Draw –** 



Command line: **SPL, SPLINE**

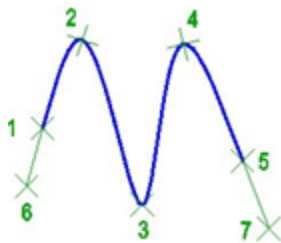
A spline is a smooth curve passing through a given set of points. Example of spline usage: line breaks of objects, horizontals in GIS.

Command options:

Object Converts a polyline, smoothed by the **Spline** command, to an equivalent spline.

Close Closes a spline.

Fit tolerance The maximum distance from a spline curve to any points defining it.



Command prompts:

Specify first point or [**Object**]:

Specify second point:

Specify next point or [**Close**/**Fit tolerance**]
<start tangent>

Specify next point or [**Close**/**Fit tolerance**]
<start tangent>

Specify a start point **1**.

Specify second point **2**.

Specify **3, 4, 5** etc. points.

Press **ENTER** to finish specifying points.

Specify start tangent:

Specify a tangent **6** for the start spline point.

Specify end tangent:

Specify a tangent **7** for the end spline point.

Ellipse

Ellipse by centre and radiuses



Menu: **Draw – Ellipse By >**  **Center and Radiuses**



Toolbar: **Draw –** 



Command line: **ELLIPSE1**

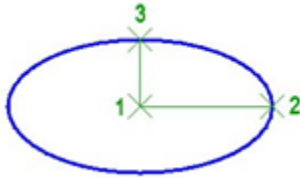
The command creates an ellipse by centre and radiuses.

Command options:

Arc Elliptic arc mode

Center Central point of ellipse.

Rotation Relation between ellipse axes by rotation around first axis.



Command prompts:

Specify center point of ellipse:

Specify centre point for ellipse (point **1**).

Specify endpoint of axis:

Specify first radius (point **2**).

Specify distance to other axis or
[Rotation]:

Specify second radius (point **3**).

Ellipse by diameter and radius



Menu: **Draw – Ellipse By >**  **Diameter and Radius**

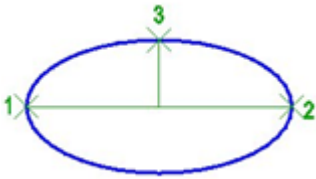


Toolbar: **Draw –** 



Command line: **ELLIPSE2**

The command creates an ellipse by diameter and radius.



Command prompts:

Specify axis endpoint of ellipse:

Specify point **1**.

Specify other axis endpoint of
ellipse:

Specify diameter of ellipse (point **2**).

Specify distance to other axis or
[Rotation]:

Specify radius of ellipse (point **3**).

Elliptic Arc



Menu: **Draw – Ellipse By >**  **Elliptic Arc**



Toolbar: **Draw –** 



Command line: **ELLIPTICARC**

Command options:

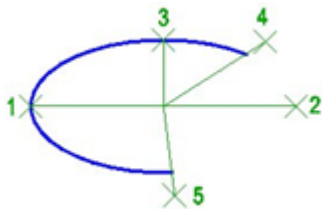
Center Centre of elliptic arc.

Rotation Relation between ellipse axes by rotation around the first axis

Parameter Select *Parameter* when specifying the start and end points of the elliptic arc to create an elliptic arc according to the parameter vector: $p(u) = c + a \cdot \cos(u) + b \cdot \sin(u)$, where c – ellipse centre; a – major axis; b – minor axis.

Inner angle Inner angle of the elliptic arc; specified relative to the origin angle.

Angle Switch between **Parameter** and **Angle** modes.



Command prompts:

Specify axis endpoint of elliptical arc or [Center]:

Specify other axis endpoint of elliptical arc:

Specify distance to other axis or [Rotation]:

Specify start angle or [Parameter]:

Specify end angle or [Parameter/Inner angle]:

Specify point 1.

Specify length of ellipse axis (point 2).

Specify length of another axis (point 3).

Specify start angle (point 4).

Specify end angle (point 5).

Editing objects

To edit objects and their parameters you have to select them on the drawing. It is possible to edit them using grips and their properties in the **Inspector** window.

By double clicking on objects, the editing command starts or the **Inspector** dialogue box appears. According to the object's type, the editing command opens an editing dialogue box (dimensions, notes) or offers to edit parameters in the command line (polylines, spline).

Selection of objects

To edit objects you have to select them.

There are different ways to select objects in nanoCAD.

One of the main ways to select objects is by using the cursor. Objects can be selected separately or in groups:

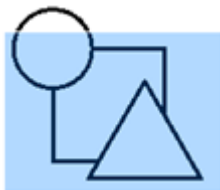
- **Selection of separate objects** is performed with the pickbox.
- **Selection of a group of objects** is performed using a rectangular selection area.

The order in which the corners of the area are specified is important when you select a group:

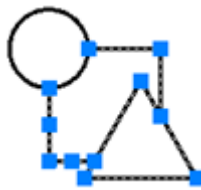
- If the corners of the area are specified from left to right (selection with *window*), only objects that are entirely inside the window are selected.
- If the corners of the area are specified from right to left (selection with *crossing window*), objects are selected that are entirely inside the window and also those that are crossed by the window.

For better visual perception a *window* is displayed with a solid line and the area inside it is highlighted in blue; a *crossing window* is displayed with a dotted line and the area inside it is highlighted in green.

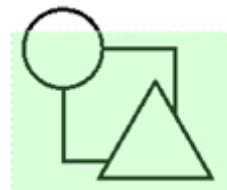
Selection with window



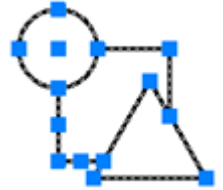
Result





Selection with *crossing window*



Result



It is important which selection mode is selected in the **Inspector window** when you select objects:

- If the multiple selection mode is switched on (the button ) , all newly selected objects are added to the set. Objects selected in this mode are excluded from the set if you press **Shift**.
- If single selection mode is switched on (the button ) , only newly selected objects are added to the set. Objects selected in this mode are added to the set if you press **Shift**.

Selection of objects using the command line

For many editing commands, you can launch additional options for object selection in the command line.

- Start an editing command.
- To the command prompt about object selection enter? (question mark):

Select objects: ?

- Press **ENTER**:

Select an option or

[Window/Last/Crossing/Box/All/Fence/Wpolygon/Cpolygon/Group/Add/Remove/AUTO]:

- Select the required option.

Command options:

Window Selection of objects that are entirely inside the rectangular area, specified by two points from left to right.

Last Selection of the last created object.
Objects cannot be on a frozen layer.

Crossing Selection of objects, crossed by the rectangular area or inside the rectangular area, specified by two points from right to left.

Box Selection option, with choice of Window or Crossing options.
If points in the rectangular area are specified from right to left, the selection of objects is equal to the

Crossing option.

If points in the rectangular area are specified from left to right, the selection of objects is equal to the Window option.

All All objects are selected, excluding objects belonging to blocked or frozen layers.

Fence Selection of objects crossed by the selection line.

WPolygon Selection of objects which are entirely inside the polygonal area.
A polygon can have any shape, but there must not be any self-intersections.
The segment of a polygon created last is the closing segment.

CPolygon Selection of objects which are entirely inside the polygonal area or are crossed by this area.
A polygon can have any shape, but there must not be any self-intersections.
The segment of a polygon created last is the closing segment.

Group Selection of objects in the specified group.

Add Adds objects to the current set using any selection method . Automatic mode and adding mode are used by default (Auto and Add modes).

Remove Deletes objects in the current set using any selection method .
You can exclude objects by pressing the SHIFT button in the adding mode (Add mode) or automatic mode (Auto mode).

AUTO Selection of objects with pickbox.
When an empty area is selected by the pickbox, the Box option is switched on and the first point of the window or crossing window is specified.
Automatic mode and adding mode are used by default (Auto and Add modes).

Question mark (?) can be entered not only to the prompt `select objects:`, but as an answer to any prompt to select objects containing a hint . For example:

`Select objects to trim or [Fence/Crossing/Project/Edge/ERase/Back] :`,
Etc.

While the command is being performed, you can launch a prompt with selection options many times.

Selection of all objects



Menu: **Edit – Select all**



Hotkeys: **CTRL+A**



Command line: **SELECTALL**

The command allows the selection of all objects in the current drawing, except objects on frozen layers.

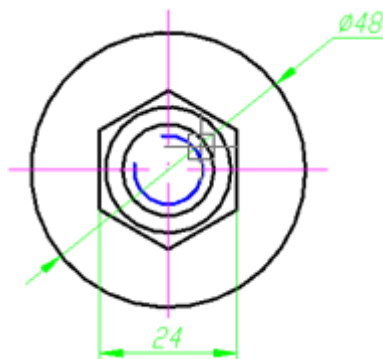
The command is available from the context menu.

To cancel the selection, press **ESC**.

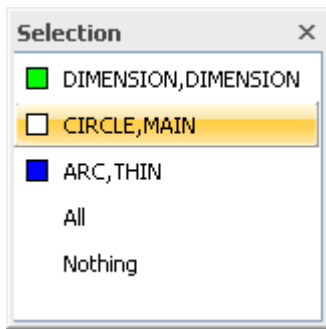
Selection of superimposed objects

Selection of objects which are very close to each other is not difficult in nanoCAD.

If several objects are in cursor's pickbox:



the dialogue box automatically opens showing the list:



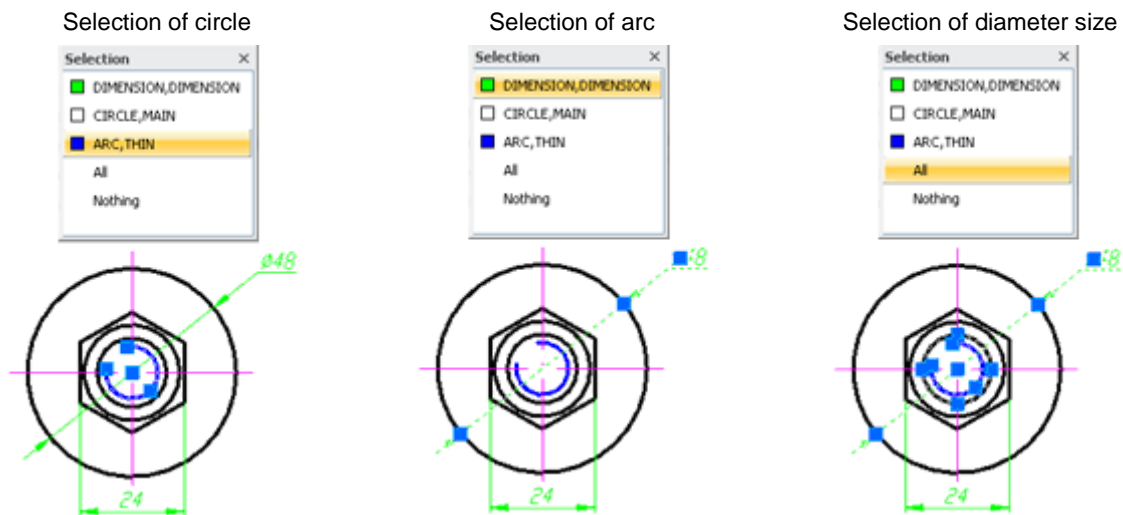
Parameters:

All Selection of all the objects in the dialogue box list.

Nothing No objects in the dialogue box list are selected.
Pressing **ESC** is also cancels the selection and closes the dialogue box.

Placing the cursor over any object in the list highlights it on the screen. Clicking on any object in the dialogue box selects it. The dialogue box closes after an object is selected.

Examples of different object selection:



The **Show selection dialog** command from the **View** menu switches on/off display of the **Selection** dialogue box.

Selection of objects using the Inspector



Menu: **Modify** – **Properties...**



Toolbar: **Main** –



Hotkeys: **CTRL+1**



Command line: **INSP, INSPECTOR, PROPERTIES**

There are buttons for modes and commands to select objects in the top part of the **Inspector** dialog:



Mode and commands to select objects

Modes of object selection



Multiple selection

Mode of multiple object selection; all the selected objects are added to the previous selection set.

ESC cancels the selection .

Objects selected in this mode are excluded from the set if you press **Shift**.



Single selection

Mode of single object selection; all newly selected objects are added to the selection set.

Previously selected objects are removed from the set.

Objects selected in this mode are added to the set if you press **Shift**.

Commands of object selection



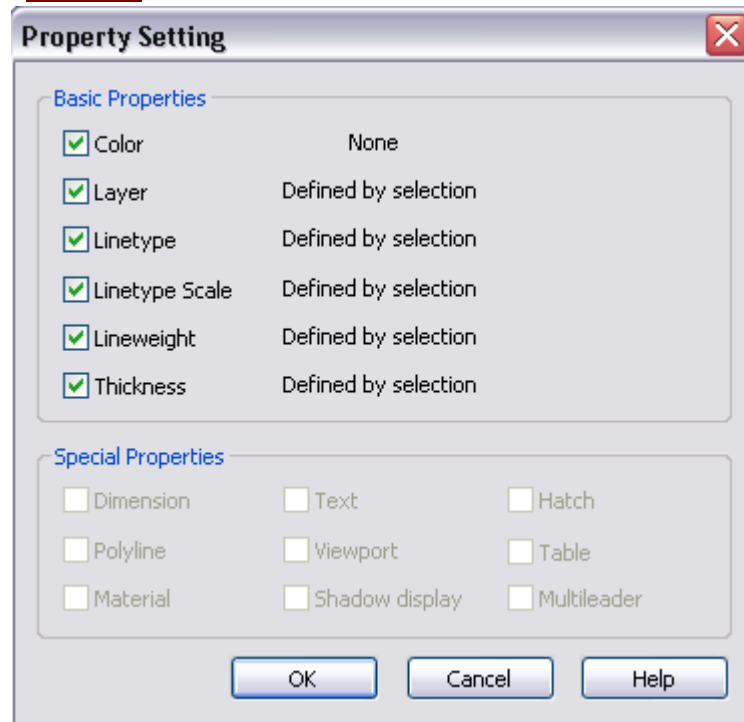
Select

Selection of objects using the cursor.



Select similar objects

Only objects, whose properties correspond to the specified template object are selected. In the **Property Settings** dialogue box you can specify the properties for object selection. The dialogue box opens if you select **Settings** in the command prompt: *Select patterns or [Settings]* :



Note 1: If no property is selected, objects of the same type as the template object will be selected.

Note 2: If one property is selected (for example **Color**), objects of the same type and colour as template object will be selected.

Note 3: If several properties are selected (for example, **Color**, **Layer** and **Lineweight**) objects are selected:

- Objects of the same type and colour as the template object,
- Objects of the same type and layer as the template object,
- Objects of the same type and lineweight as the template object,



Remove from selection

Excludes the specified objects from the selection set.



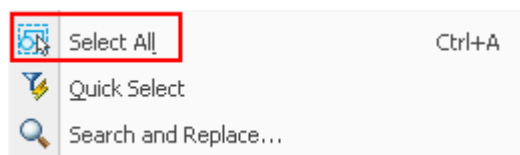
Leave in selection

Leaves only the specified objects in the selection set.

To exclude objects from a selection set

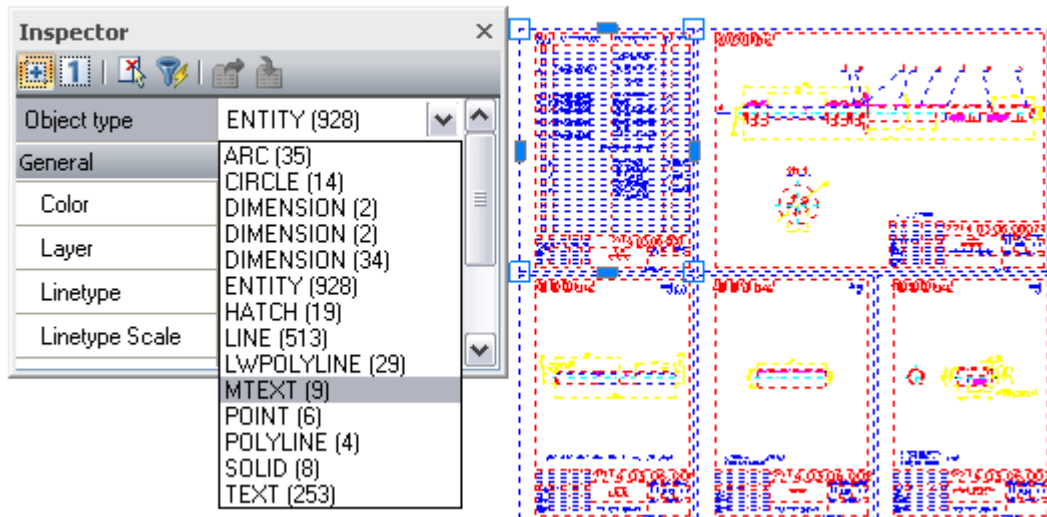
To exclude objects from a selection set:



- Select the **Select all** in the context menu or from the **Edit** menu:



All objects in the drawing will be selected.


- Select the object type that you want to exclude from the set in the drop down list in the **Object type** of the **Inspector** window:



- Select the  **Remove from selection**. All objects of the specified type will be deleted from the selection set.
- Select another type of object in the drop-down list.
- Select the  **Remove from selection**.
- Selection and removal of objects can be repeated until only the required objects remain.

To leave objects in a selection set

To leave only specified objects in a selection set:

- Select **Select all** in the context menu or from the **Edit** menu. All the objects in the drawing will be selected.
- Select the object type which you want to exclude from the set in the drop down list in the **Object type** of the **Inspector** window.
- Select the  **Leave in selection**. All objects, except the specified type, will be deleted from the selection set..

Quick selection of objects



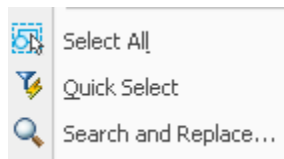
Menu: **Edit** –  **Quick Select...**



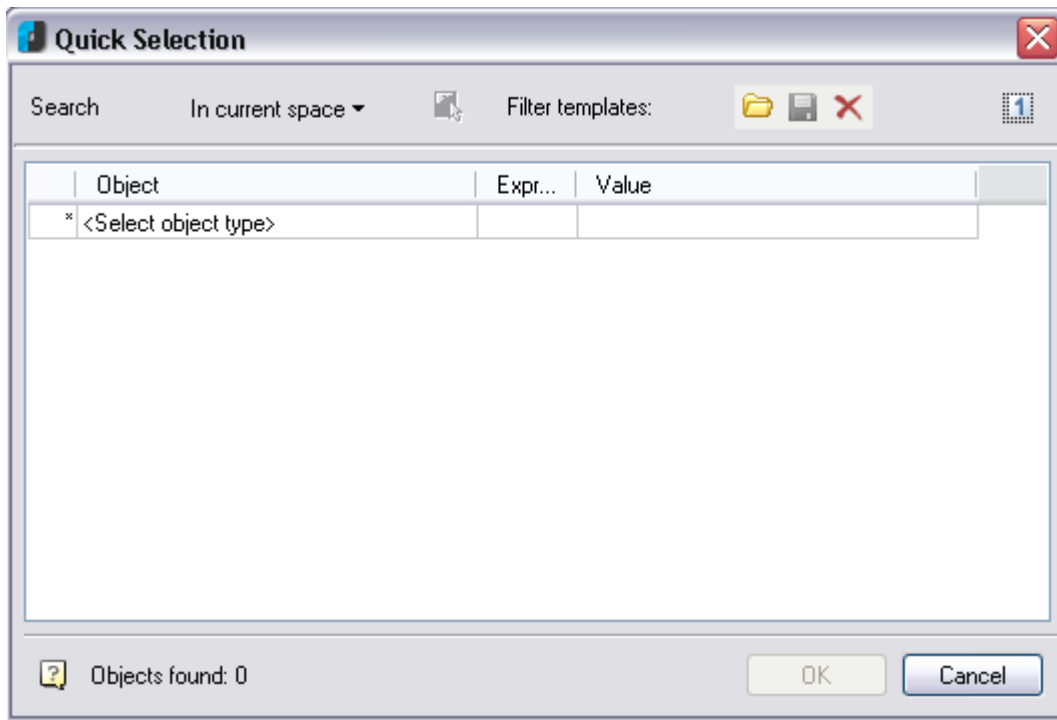
Command line: **QS**

Using the **Quick select** command you can select inserted objects using specified conditions.

The **Quick Select** command can be launched from the context menu:

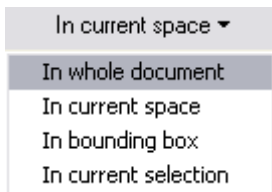


In the **Quick Selection** dialogue box you can adjust the search conditions:



Parameters:

Search There are selection areas in the drop-down list:



Available options:


In whole document – search in the whole document.

In current space – search in the current space.

In bounding box – search in the selected rectangular area in the drawing.

In current selection – search among selected objects in the drawing.

If **In bounding box** or **In current selection** is selected, the **Quick Selection** dialogue box temporarily closes to allow the rectangular area to be specified on the screen or for objects to be selected.

The  **Set selection** button becomes available after the area is specified or the objects are selected; using this button you can select a new rectangular area or change the current set of objects.



This button allows found objects to be added to selected objects on the drawing. According to which method is selected, the view of a button can be changed.

The following methods are available:



Result set to new selection – Objects corresponding to the specified search conditions are selected in the drawing, other objects are deselected.



Result add to selection – Found objects will be added to the selected objects on the drawing.



Result subtract from selection – If objects were selected, they become deselected.

Filter templates:

Using a set of tools you can select objects using set and saved conditions.



Load preset



Save preset



Remove preset

To add a condition for object search:

- Click in the *<Select object type>* field and select the required object type from the list:

| Object |
|--------|
| * Text |
| Text |
| Block |
| -None- |

To reject the selection, click on *None*.

- There is a list of parameters of all objects belonging to this type and to the specified selection area:

| Object | Expr... | Value |
|------------------------|---------|----------------------|
| Block | | Selected: 1 (from 1) |
| Rotation | | |
| Name | | |
| Layer | | |
| * <Select object type> | | |

Objects found: 1

OK Cancel

- Select the value of the required search parameter in the *Value* column:

| Object | ExprT... | Value |
|------------------------|----------|---------------------------|
| Block | | Selected: 1 (from 1) |
| Rotation | | |
| Name | | |
| Layer | | 1 |
| * <Select object type> | | Get from object -None- |

- Select a logic search condition in the *ExprType* column:

| Object | ExprT... | Value |
|------------------------|----------|----------------------|
| Block | | Selected: 1 (from 1) |
| Rotation | | |
| Name | = | 1 |
| Layer | | |
| * <Select object type> | | |

Available logic conditions for search:

| | |
|----|----------------|
| = | equals |
| != | does not equal |
| < | less |
| > | more |
| >= | not less |

| | |
|---------------|--|
| <= | not more |
| Like | Search of substring. Searches for objects whose parameter value contains the substring specified in the <i>ExprType</i> column. |
| Unlike | Search of substring. Searches for objects whose parameter value does not contain the substring specified in the <i>ExprType</i> column. |

Note: for every object type you can set numerous search conditions using one or several parameters.

- To add another type of object, click on the *<Select object type>* bottom field:

| Object | ExprT... | Value |
|--------------|----------|-----------------------------|
| Block | | Selected: 1 (from 1) |
| Rotation | | |
| Name | = | 1 |
| Layer | | |
| * Text | | |
| Text | | |
| Block | | |
| -None- | | |

The number of objects satisfying the search conditions is displayed in the bottom part of the dialogue box: **Object found: 3.**

Ways to edit objects

You can edit objects (change their properties) in different ways:

- Editing with the **Styles** toolbar; when a text or dimension style for the selected objects is changed.
- Using the **Main** toolbar; when a layer, colour, type or lineweight for previously selected objects is changed.
- Using the **Inspector** window.
- Editing of text objects (search and replace) using the **Search and Replace**.
- Using the **Copy object properties** command.
- Using the Windows clipboard.
- Editing with *grips*; the selected objects are highlighted (object lines become dotted) and marked with special markers, small blue rectangles, placed at the characteristic points of the objects. The markers of grips can be triangles, circles, rhombs etc. The colour of markers can be light blue or green.
- Editing using editing commands; the editing command starts (for example, **Copy** from the **Edit** menu) and objects are selected. For many commands you can first select objects and then start the editing command.
- Editing by double clicking on an object and an editing command relevant to the object's type starts. It can be the **Inspector** dialogue box, if it was closed. The editing command can open a dialogue box to edit the object's properties (dimensions, notes, tables etc.) or offer to edit in the command line (for example, polylines and splines).

Copy of objects properties



Menu: **Modify** –  **Copy object properties**



Toolbar: **Main** – 



Command line: **MATCHPROP**

This command is used to copy a part of the selected object properties or all of them and assign them to one or several objects.

You can copy colour, layer, linetype, lineweight and other properties.

Command options:

Settings Opens the **Property Settings** dialog.

Command prompts:

Select source objects:

Select a source object.

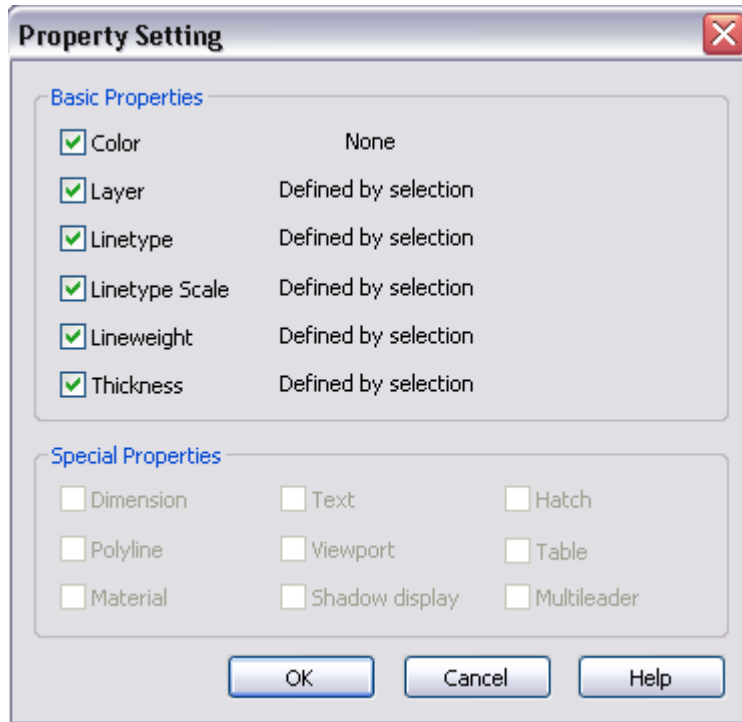
Select destination object(s) or
[**Settings**]:

Select object(s) to assign the copied properties to or select
the **Settings**.

Select destination object(s) or
[**Settings**]:

Select object(s) or pres **ENTER** to finish the command.

The **Settings** option opens the **Property Settings** dialogue box where you can specify the copied properties from the source object:



Parameters:

Basic properties

| | |
|-----------------------|---|
| Color | Switches on/off the colour copying mode . |
| Layer | Switches on/off the layer copying mode . |
| Linetype | Switches on/off the linetype copying mode . |
| Linetype Scale | Switches on/off the linetype scale copying mode . |
| Lineweight | Switches on/off the lineweight copying mode . |
| Thickness | Switches on/off the height copying mode . |

Special properties


| | |
|-----------------------|--|
| Dimension | Switches on/off the dimension style copying mode . |
| Polyline | Switches on/off the width and type of polyline mode . |
| Material | Switches on/off the material copying mode . |
| Text | Switches on/off the text style copying mode. |
| Viewport | Switches on/off the viewport properties copying mode . |
| Shadow display | Switches on/off the shadow display copying mode . |
| Hatch | Switches on/off the hatch copying mode . |
| Table | Switches on/off the table style copying mode . |
| Multileader | Switches on/off the leader note style copying mode . |

Copying and insertion of objects using clipboard

The clipboard can be used to copy a drawing or part of it from one document to another. Data copied to the clipboard can be inserted into other applications, for example into open documents in MS Office.
nanoCAD commands using the clipboard are in the **Edit** menu.

Cut



Menu: **Edit** –  **Cut**



Toolbar: **Main** – 



Hotkeys: **CTRL+X**




Command line: **CUT, CUTCLIP**

This command deletes the selected objects from the document and places them on the clipboard.

Copy



Menu: **Edit** –  **Copy**



Toolbar **Main** – 



Hotkeys: **CTRL+C**



Command line: **COPYCLIP**

This command copies the selected objects and places a copy of them on the clipboard.

Copy With Base Point



Menu: **Edit** –  **Copy With Base Point**



Hotkeys: **CTRL+SHIFT+C**




Command line: **COPYBASE**

This command copies the selected objects and places a copy of them on the clipboard. When copying, a base point is specified and it is used to insert a copy of the objects into the document.

Paste



Menu: **Edit** –  **Paste**



Toolbar: **Main** – 



Hotkeys: **CTRL+V**



Command line: **PASTE, PASTECLIP**

This command inserts the contents of the clipboard into the document.

Paste as block



Menu: **Edit** –  **Paste as Block**



Hotkeys: **CTRL+SHIFT+V**



Command line: **PASTEBLOCK**

This command inserts the contents of the clipboard as a block into the document.

Paste to Original Coordinates



Menu: **Edit** –  **Paste to original coordinates**



Command line: **PASTEORIGIN**

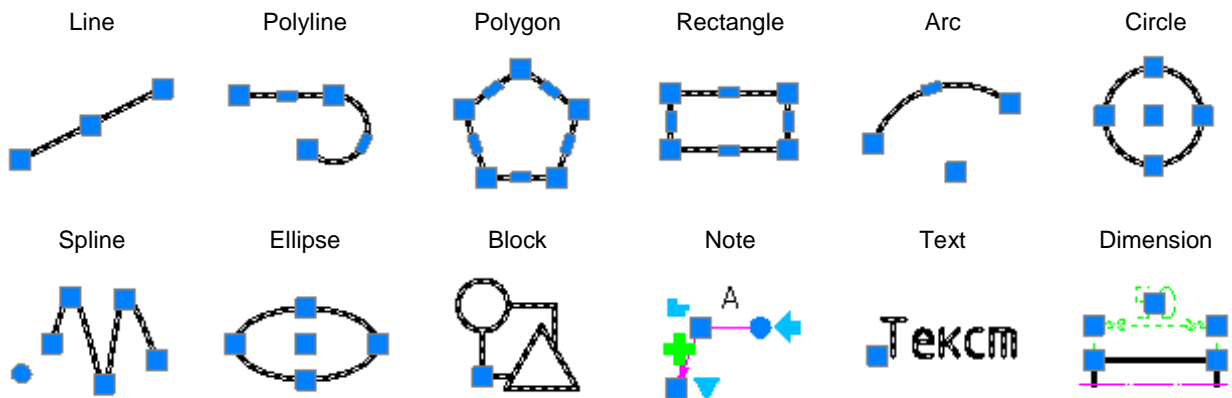
This command inserts objects from the clipboard into the document with the same coordinates that they had in the initial document.

Editing objects using ordinary grips

Editing commands can be applied to objects previously selected and marked with grips. Available parameters can be changed in the **Inspector** dialogue box. Sometimes, editing with grips is the quickest and most effective method to change the shape and location of objects because actions with grips are performed with the cursor, so you do not need to use menus and toolbars.

You can use grips to move, copy, stretch or scale objects.

The number and location of grips depends on the selected object's type; for example, a line segment has 3 grips. Two of them located at the endpoints and one in the middle, 5 grips are displayed for circle, four of them are located in the quadrants and one in the centre etc.:



All newly selected objects are added to the current selection set of objects. To delete objects from a set, press and hold the **SHIFT** button during selection of objects.

To switch to the editing mode with grips, click on any grip after the objects are selected. The selected grip becomes *active* and changes its colour to red. This grip is used to perform editing operations and is called the *base*. According to which grip is selected as the *base*, the object can be copied, moved, stretched and scaled by moving the cursor. The selected base grip is snapped to the cursor's crosshair and moved with it until you click to specify a new grip position.

Any grip in the drawing can be specified as the base if you select the **Base** option in the command line after the base point is selected:

Specify stretch point or [**Base**/**Copy**/**Undo**/**Exit**] :

Options available in the command line for editing with grips:

Base Specify a point as the base.

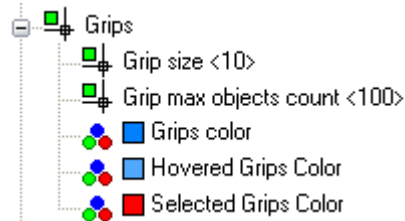
Copy Switch to copying mode.

Undo Cancels the last operation.

Exit Exits from the editing with grips mode. The display mode of the selected objects is not cancelled.

You can make several grips *active* if the **SHIFT** button is pressed during grips selection. Editing with grips mode is switched on after one of these grips is selected as the *base* (select it without **SHIFT** button).

You can change grips in the **Grips** section of the **Options** dialog:



Editing objects using multifunctional grips

Grips that can change editing mode in rotation are called *multifunctional*. Rotation of editing modes is carried out for the *active* (selected) grip using the **CTRL** button.

Objects which have multifunctional grips:

- *line segment*,
- *arc*,
- *spline*,
- *polyline*,
- *hatch*,
- *viewport*.

To edit objects with multifunctional grips:

- Select an object.
- Activate the multifunctional grip.
- Press **CTRL** to select the editing variant.

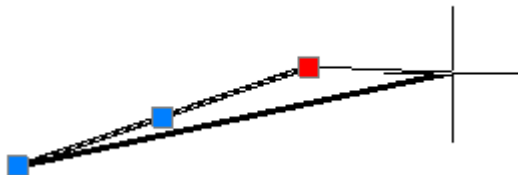
- Move the cursor for dynamic display of an object's properties.
- Click to confirm the change.

Segment editing

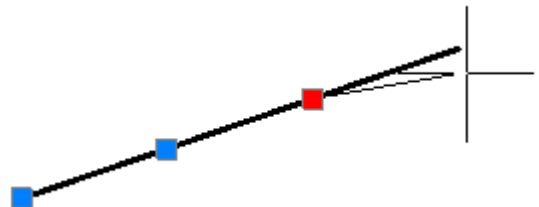
A line segment has multifunctional grips at the endpoint. Two editing modes are available

- *Normal mode*: the length of the segment changes when you move the grip. In general, not only the segment length is changed, but also its orientation.
- *Change of length*: only the length of the segment changes when you move the grip. A new position for the endpoint is provided by the projection of the specified point along the segment's imaginary extension. Segment orientation is not changed.

Normal mode



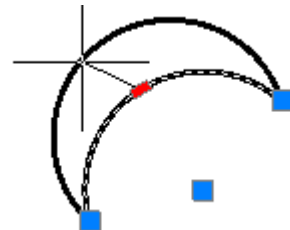
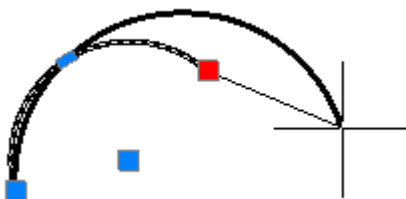
Change of length



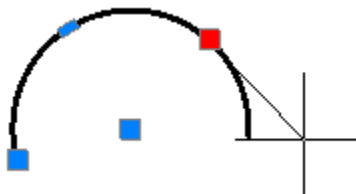
Arc editing

Multifunctional grips are at the endpoints and in the middle of the arc. Three editing modes are available:

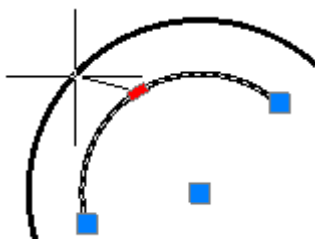
- *Normal mode*: when you hold and move a grip on the end or middle point, the length and radius are changed:



- *Change of arc length*: when you hold and move a grip on the end point, only the length is changed:



- *Change of arc radius and length*, when you hold and move the grip on the middle point, an arc is created similar to the source arc:



Spline editing

Spline has two editing modes with grips:

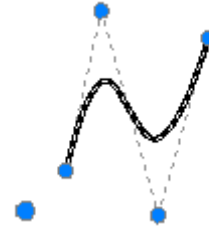
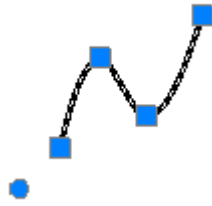
Editing of characteristic points (rectangular grips): allows the shape of a small part of the spline to be changed. .

Editing of control vertexes (round grips ape): allows the general shape of a spine to be changed. .

To switch between modes, click on a round grip located near the spline and having a bigger diameter than the grips of the control vertexes.

Grips of characteristic points





Grips of control vertexes



Polyline editing

Polyline has multifunctional grips at the end of the segments and in the middles of the segments. Rotational editing modes, accessed with the **CTRL** button, depend on the location of the grips (end point or middle point) and segment type (linear or arc).

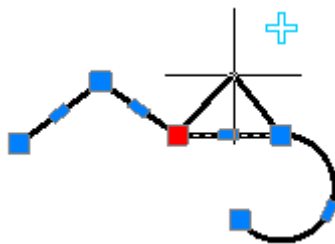
For improved functionality, additional symbols are displayed beside the polyline shape and near the cursor. Which symbols are shown depends on the editing mode selected:

-  Adds a vertex
-  Deletes a vertex.
-  Transforms a linear segment to an arc.
-  Transforms an arc segment to linear.

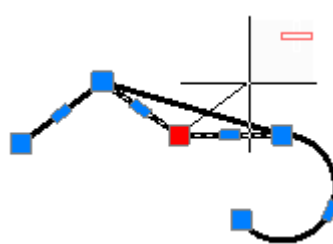
Absence of a symbol means that the normal editing mode is current - stretching by moving a segment or stretching a vertex (depends on selected grip).

Variants of polyline editing with grips in vertexes

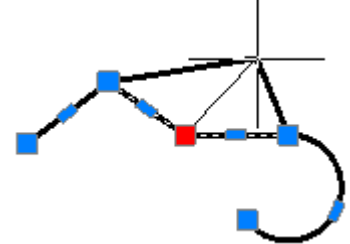
Adding vertex



Deleting vertex

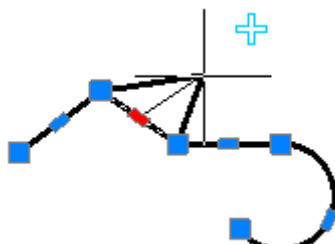


Stretching with vertex

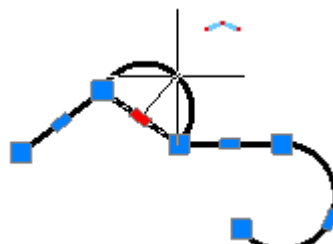


Variants of polyline editing with grips in the middle of a linear segment:

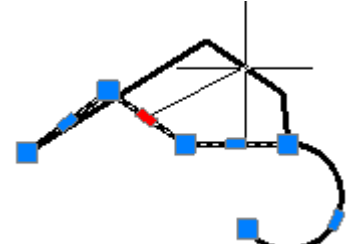
Adding vertex



Transforming to arc segment

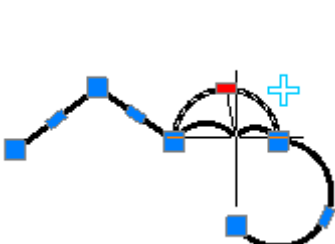


Stretching

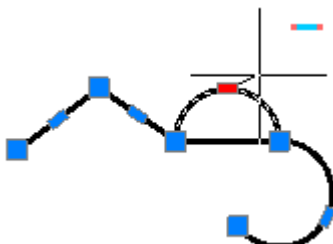


Variants of polyline editing with grips in the middle of an arc segment:

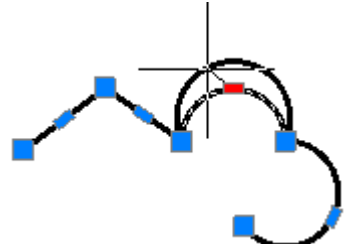
Adding vertex



Transforming to linear segment



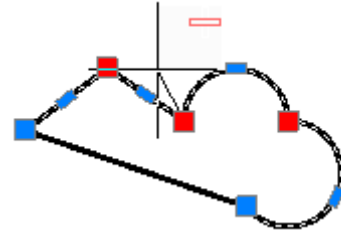
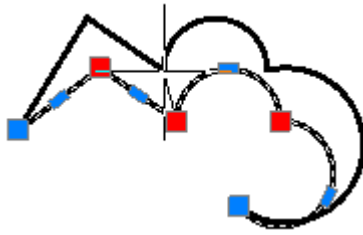
Stretching



A multifunctional mode for editing polylines can be applied to several grips located in vertexes. Two editing variants are available: normal stretching of the polyline and deleting of selected vertexes. Instead of deleted vertexes, a linear segment is drawn, even if arc segments are between the deleted vertexes.

Stretching vertexes

Deleting vertexes



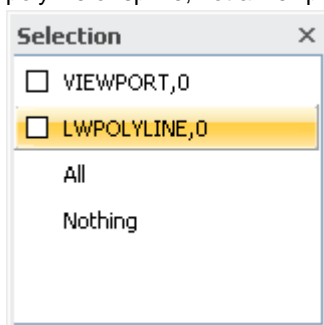
To edit several vertexes of a polyline using multifunctional grips:

- Press **SHIFT**.
- Select the required grips with **SHIFT** pressed.
- When you finish selecting grips, release **SHIFT**.
- Select a base grip.
- Select an editing mode with **CTRL**.
- Move the cursor for dynamic display of the polyline shape.
- Click to fix changes.

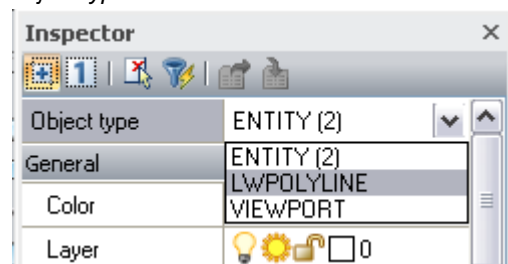
Editing viewports of paper space

You can edit any viewports of paper space using multifunctional grips where closed polylines and splines are used as boundaries. The process of editing the boundaries of the viewport is the same as editing of a spline or polyline with multifunctional grips.

As viewports consist of two objects (viewport and display border), to edit them in the Selection dialogue box select a polyline or spline, not a viewport:



If display of the **Selection** dialogue box is switched off, select a polyline or spline in the **Inspector** window in the **Object type** field:



Select a viewport with a window or crossing window, if you select with the pickbox in the **Object type** field in the **Inspector** window, **Viewport** is displayed by default.

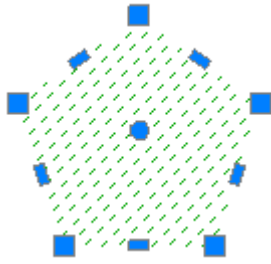
Editing hatch and fill

In nanoCAD, using multifunctional grips you can edit the shape of:

- *associative hatches, which use closed polylines and splines as linked contours;*
- *non-associative hatches.*

Changing the shape of an *associative* hatch using multifunctional grips and a linked contour does not differ from the editing of a polyline or spline using multifunctional grips.

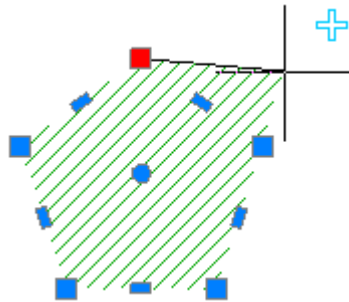
If you select *non-associative* hatch, the same multifunctional grips are displayed as in a polyline, except the round grips are used to move the hatch:



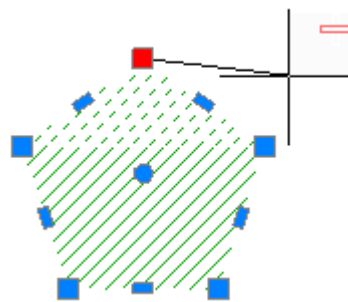
The process of editing a non-associative hatch shape is the same as editing of a polyline with multifunctional grips.

Variants of non-associative hatch shape editing with grips in a boundary vertex:

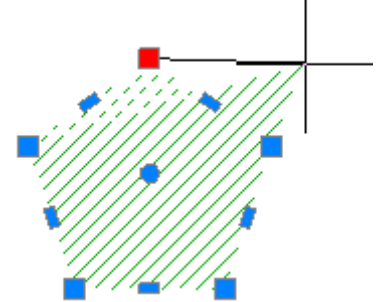
Adding of boundary vertex



Deleting of boundary vertex

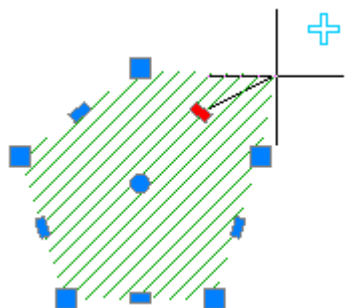


Stretching boundary by top

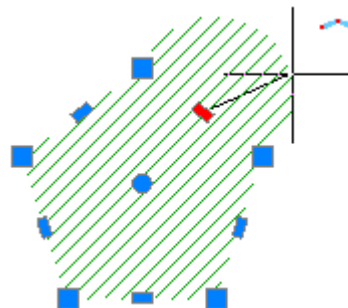


Variants of non-associative hatch shape editing with a grip in the middle of a boundary linear segment:

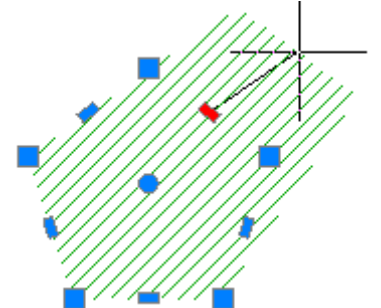
Adding of boundary vertex



Transforming of linear segment of boundary into arc segment

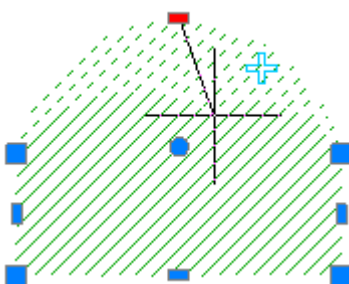


Stretching

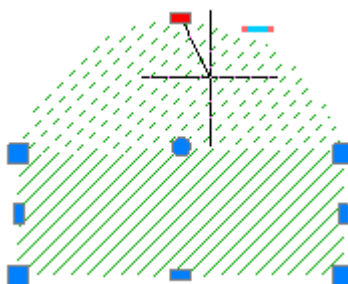


Variants of non-associative hatch shape editing with a grip in the middle of a boundary arc segment:

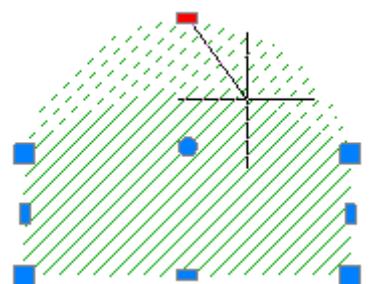
Adding of boundary vertex



Transforming of arc segment of boundary into linear segment



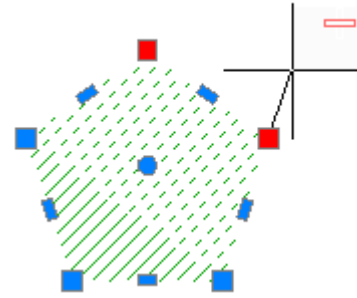
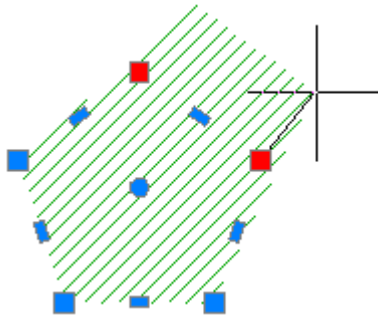
Stretching



For non-associative hatch, a multifunctional editing mode can be applied to several grips located in the boundary vertices. Two variants of editing are available: normal stretching of boundary and deleting of selected vertices. Instead of deleted vertices a linear segment is drawn, even if arc segments were between deleted vertices.

Stretching of boundary vertices

Deleting boundary vertices



To edit several vertexes of non-associative hatch area using multifunctional grips:

- Press **SHIFT**.
- Select the required grips with **SHIFT** pressed.
- When you have selected the grips, release **SHIFT**.
- Select a base grip.
- Select an editing mode with **CTRL**.
- Move the cursor for dynamic display of the hatch shape.
- Click to fix changes.

Commands to edit geometric objects

Lengthen objects



Menu: **Modify** –  **Lengthen objects**



Command line: **LEN, LENGTHEN**

Using the **Lengthen** command you can change the central angle of arcs and the length of lines, arcs and open polylines. The length or angle of objects is changed on the side closest to the specified point. The length or angle of an object can be changed several times whilst one command is being performed. If a value is positive, the arc is lengthened, if a value is negative – the arc is shortened.

Command options:

DElta The value of the object's length is changed.

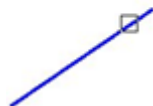
Angle The value of the object's centre angle is changed.

Percent The object's length as a percentage of the source length.

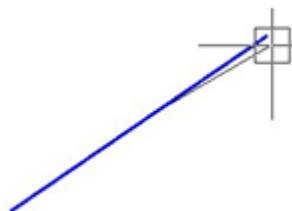
Total The total length or centre angle.

DYnamic Dynamic change of the selected object's length by replacing the endpoint closest to the selection point with the fixed position of another endpoint.

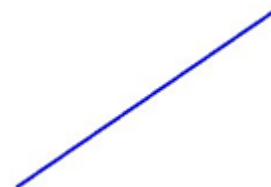
Selection of object



Specifying a new length



Result



Command prompts:

Select objects or [**DElta**/**Percent**/**Total**/**DYnamic**]:

Select an object to change:

Select an object to change:


Select the **DYnamic**.

Specify an object.

Specify an object or press **ESC** to finish the command.

Trim vectors



Menu: **Modify** –  **Trim vectors**



Toolbar: **Modify** – 



Command line: **TR, TRIM**

The **Trim vectors** command allows the trimming of vector objects by their boundaries (or cutting edges), specified for one or several objects. Objects not intersecting with cutting edges can be cut at the point of their imaginary intersection. One object can be a cutting edge and a cut object.

Command options:

Fence

Selection of objects by a crossing line which can consist of several segments.

Crossing

Selection of objects with a crossing frame.

Project

Mode for cutting objects by intersection of their projections with the boundary in 3D space.

None

Only objects that are crossed by the specified boundary in 3D space are selected .

Ucs

Defines the projections of objects in the XY plane of the current UCS and cuts objects which are not intersected by the boundary in 3D space.

View

Defines the projections of objects in the direction of the specified view and cuts objects which are not intersected by the edge.

Edge

Specifies the cutting mode of objects by imaginary extension of the edge.

No extend

Switches off the cutting of objects by imaginary extension of the edge.

Extend

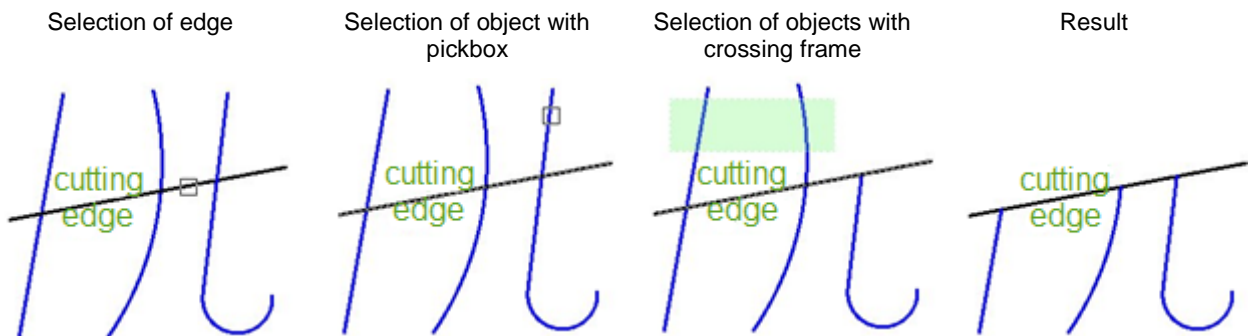
Switches on the cutting of objects by imaginary extension of the edge.

eRase

Deletes the selected objects.

Back

Cancels the last change executed during performance of the command.



Command prompts:

Select objects:

Select object:

Select object to trim or
[Fence/Crossing/Project/Edge/eRase/Back] :

Select object to trim or
[Fence/Crossing/Project/Edge/eRase/Back] :

Select an object which will be used as the cutting object.

Select the next object or press **ENTER** to select the objects to be cut.

Select an object.

Select the next object or press **ENTER** to finish the command.

Smart trim



Menu: **Modify** –  **Smart trim**



Toolbar: **Modify** – 



Command line: **SMARTTRIM**

The **Smart trim** command differs from the **Trim vectors** in automatically selecting all vector objects in the document as potential cutting edges. Selected objects are cut to the nearest cutting edges.

Command options:

| | |
|------------------|---|
| <u>Fence</u> | Selection of objects by a crossing line which can consist of several segments. |
| <u>Crossing</u> | Selection of objects with a crossing frame. |
| <u>Project</u> | Mode of cutting objects by intersection of their projections with the boundary in 3D space. |
| <u>None</u> | Only objects that are crossed by the specified boundary in 3D space are selected . |
| <u>Ucs</u> | Defines the projections of objects in the XY plane of the current UCS and cuts objects which are not intersected by the boundary in 3D space. |
| <u>View</u> | Defines the projections of objects in the direction of the specified view and cuts objects which are not intersected by the edge. |
| <u>Edge</u> | Specifies the cutting mode of objects by imaginary extension of the edge. |
| <u>No extend</u> | Switches off the cutting of objects by imaginary extension of the edge. |
| <u>Extend</u> | Switches on the cutting of objects by imaginary extension of the edge. |
| <u>eRase</u> | Deletes the selected objects. |
| <u>Back</u> | Cancels the last change executed during performance of the command. |

Command prompts:

| | |
|--|--|
| Select object to trim or [<u>Fence</u> / <u>Crossing</u> / <u>Project</u> / <u>Edge</u> / <u>eRase</u> / <u>Back</u>] : | Select objects to be cut. |
| Select object to trim or [<u>Fence</u> / <u>Crossing</u> / <u>Project</u> / <u>Edge</u> / <u>eRase</u> / <u>Back</u>] : | Select <u>eRase</u> . |
| Select objects to erase: | Select objects and press ENTER to delete them.. |
| Select object to trim or [<u>Fence</u> / <u>Crossing</u> / <u>Project</u> / <u>Edge</u> / <u>eRase</u> / <u>Back</u>] : | Select <u>Back</u> to cancel deleting. |
| Select object to trim or [<u>Fence</u> / <u>Crossing</u> / <u>Project</u> / <u>Edge</u> / <u>eRase</u> / <u>Back</u>] : | Press ESC to finish the command. |

Extend vectors



Menu: **Modify –**  **Extend vectors**



Toolbar: **Modify –** 



Command line: **EX, EXTEND**

The **Extend** command extends open vector objects to their real or imaginary crossing with other objects, called *boundaries* or *boundary edges*. Extended objects are selected by specifying an extended part.

When several boundaries are specified, the object is extended to the nearest one. The object can be selected again to extend it to the next boundary edge.

Command options:

| | |
|-----------------|--|
| <u>Fence</u> | Selection of objects by a crossing line which can consist of several segments. |
| <u>Crossing</u> | Selection of objects with a crossing frame. |
| <u>Project</u> | Mode of extending objects by intersection of their projections with the boundary in 3D space. |
| <u>None</u> | Only objects, crossed with the specified boundary in 3D space. |
| <u>Ucs</u> | Defines the projections of objects in the XY plane of the current UCS and extends objects which are not intersected by the boundary in 3D space. |
| <u>View</u> | Defines the projections of objects in the direction of the specified view and extends objects which are not intersected by the edge. |
| <u>Edge</u> | Specifies the extending mode of objects by imaginary extension of the edge. |

No extend

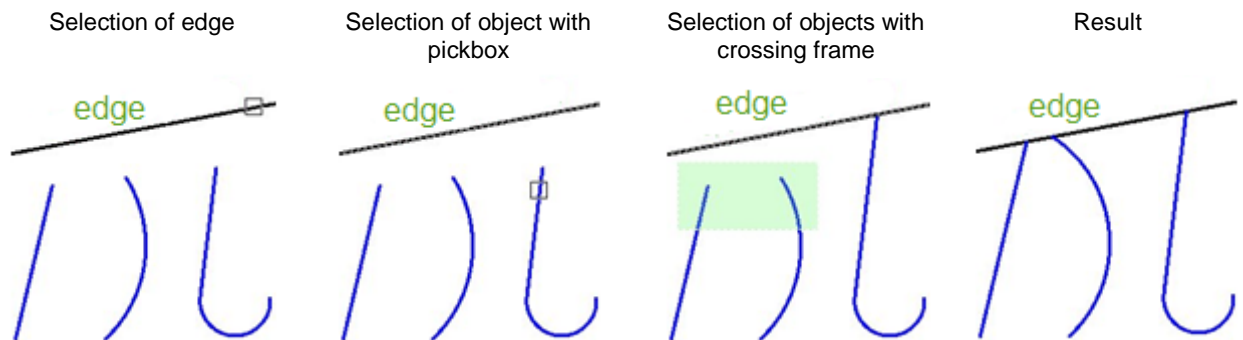
Switches off the extending of objects by imaginary extension of the edge.

Extend

Switches on the extending of objects by imaginary extension of the edge.

Back

Cancels the last change executed during performance of the command.



Command prompts:

Select objects:

Select objects:

Select object to extend or

[Fence/Crossing/Project/Edge/Back]:

Select object to extend or

[Fence/Crossing/Project/Edge/Back]:

Select the object that is to be used as the edge.

Select next object or press **ENTER** to start selecting objects to extend.

Select an object.

Select Crossing and select other objects. Press **ESC** to finish the command.

Break vectors



Menu: **Modify** –  **Break vectors**



Toolbar: **Modify** – 



Command line: **BREAK**

The **Break vectors** command breaks lines, arcs, circles, polylines, ellipses, splines and other vector objects into two parts. A break can be performed by deletion of a part of it (breaking of the object at two points) or without deletion (breaking of the object at one point).

To break an object at one point, at the command prompt to enter the second point, enter @0,0:

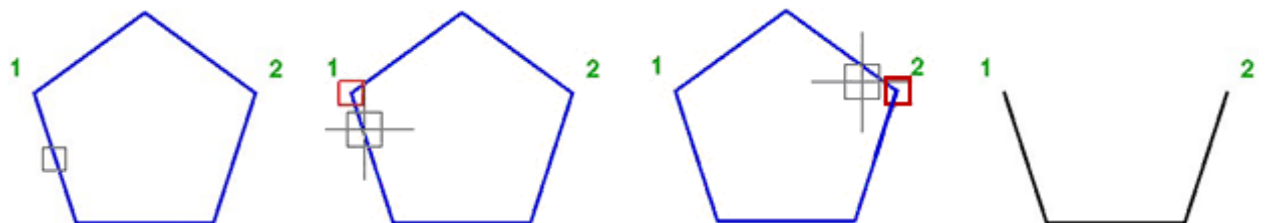
Specify second break point or [First point]: @0,0

Command options:

First point Switches to the mode of first point selection.

Break of object in two points

The part of the object between the specified points will be deleted. By default, the selection point of the object is the first point of the break. To specify another point as the first point, select the **First point** option.



Command prompts:

Select objects:

Specify second break point or [First point]:

Specify first break point:

Specify second break point:

Break of object in one point

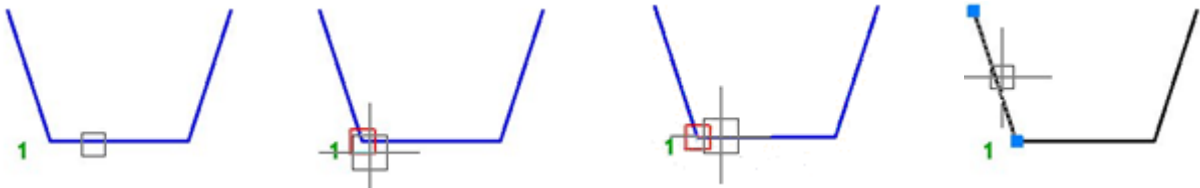
Select an object.

Select First point.

Specify point 1.

Specify point 2.

Specify the same point as the first and the second points of the break.



Command prompts:

Select objects:

Specify second break point or [First point]:

Specify first break point:

Specify second break point:

Select an object.

Select First point.

Specify point 1.

Specify again point 1.

Break vectors at point



Menu: **Modify – Break vectors at point**



Toolbar: **Modify –**



Command line: **VCBREAKATPOINTCMD**

The **Break vectors at point** command breaks lines, arcs, circles, polylines, ellipses, splines and other vector objects into two parts at one point (without deleting part of the object).

Command prompts:

Select objects:

Specify break point:

Select object.

Specify a point.

Reverse



Menu: **Modify –**



Reverse



Command line: **REVERSE**

This command is used to change the order of the vertexes of segments, polylines and splines.

For example, a segment with 0,0 coordinates for the start point and 100,100 for the end point will have 100,100 coordinates for the start point and 0, 0 for the end point after the direction is changed.

Command options:

Exit Finishes the command.

Command prompts:

Select object to flip or
[Exit]

Select an object.

Select object to flip or
[Exit]

Select the next object or press **ESC** to finish the command.

Join objects



Menu: **Modify –**



Join objects



Toolbar: **Modify –**



Command line: **JOIN**

The **Join objects** command joins separate segments of objects into one object. The command can be applied to segments, arcs and open polylines. The command allows the creation of circles from arcs (**C**lose option).

Arcs are joined in the counterclockwise direction from the source arc.

Joined objects are in one plane.

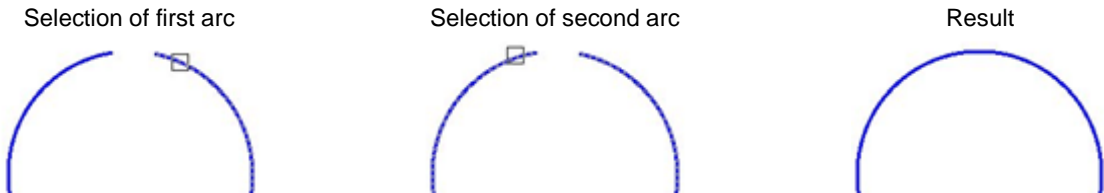
Additional restrictions for joined objects types:

| | |
|----------------------|---|
| Line segments | Must lie on one infinite line. There can be gaps between segments. |
| Polylines | Lines, polylines and arcs can be joined to a polyline. |

| | |
|----------------------|--|
| | Objects must lie in one plane. Parallel to the XY plane of the UCS. There must be gaps between objects. |
| Arcs | Objects must lie in one imaginary circle. There can be gaps between arcs. |
| Elliptic arcs | Objects must lie in one imaginary ellipse. There can be gaps between arcs. |

Command options:

Close it Transforms an arc to a circle.





Command prompts:


Select source object:
Select arcs to join to source or [Close it]:
Select arcs to join to source or [Close it]:

Specify the first arc.
Specify the second arc.
Press **ENTER** to finish the command.

Editing a polyline

 Menu: **Modify – Object>**  **Polyline**

 Toolbar: **Modify object –** 

 Command line: **PE, PEDIT**

Double clicking on a polyline starts the editing mode.

Using the **Polyline** command you can edit polylines and also convert elementary objects, consisting of arcs and lines, into polylines.

Command options:

| | |
|--------------------|---|
| <u>Multiple</u> | Selection of several objects mode. |
| <u>Close</u> | Closes (draws a segment from start to end vertex) a polyline. |
| <u>Open</u> | Opens a polyline (deletes a segment from start to end vertex). |
| <u>Edit vertex</u> | Switches to the editing of vertexes mode (edited vertex is marked with «X» label). |
| <u>Next</u> | Goes to the next vertex. |
| <u>Previous</u> | Goes to the previous vertex. |
| <u>Break</u> | Breaks a polyline at the selected vertex. |
| <u>Insert</u> | Adds a vertex in the specified place. |
| <u>Move</u> | Changes the position of the selected vertex. |
| <u>Regen</u> | Regenerates a polyline. |
| <u>Straighten</u> | Changes a line segment to an arc segment. |
| <u>Tangent</u> | Specifies the direction of the tangent in the selected vertex for further polyline fitting. |
| <u>Width</u> | Specifies the first and second width of the segment, going after the selected vertex. |
| <u>Exit</u> | Closes the editing of vertexes mode. |
| <u>Join</u> | Joins segments, arcs and polylines into one object – polyline. |
| <u>End</u> | Closes selection of objects. |
| <u>Width</u> | Specifies a new width for all polylines. |
| <u>Fit</u> | Fits the polyline with arcs between polyline vertexes. |
| <u>Spline</u> | Transforms a polyline into a spline which is an approximation of the source polyline between the start and end vertexes and is very similar to the source polyline. |
| <u>Decurve</u> | Returns a polyline to its original condition, cancels the results of the Fit or Spline commands. |
| <u>Ltype gen</u> | Specifies a mode of the specified line type generation. If Off option is selected, generation of line type starts from dash and end dash in very vertex. |
| <u>Undo</u> | Cancels all actions to the original polyline. |

Command prompts:



Select polyline or [Multiple]:
Select polyline or [Multiple]:
Enter an option [Close/Edit
vertex/Join/Width/Fit/Spline/Decurve/Ltype
gen/Undo]:

Select Multiple.


Select objects.

Select the required option. Press **ENTER** to finish the command.

Editing a spline

 Menu: **Modify – Object >**  **Spline**

 Toolbar: **Modify Object –** 

 Command line: **SPE, SPLINEDIT**

Double clicking on a spline starts the editing mode.

This command allows changing of a spline's shape by adding, deleting or replacing characteristic points, changing the direction of tangents at the start and end points of the spline. Using this command you can close or open a spline, change a tolerance value. A tolerance defines the maximum distance from the real spline to any of its characteristic points. The smaller the value a tolerance has, the closer the spline is to the characteristic points.

Command options:

| | |
|--------------------------|---|
| <u>Fit data</u> | Switches to the editing of spline data mode, including tolerance values. |
| <u>Add</u> | Adds characteristic points to the spline. |
| <u>Close</u> | Closes an open spline with fitting at a join point. |
| <u>Open</u> | Opens a closed spline. |
| <u>Delete</u> | Deletes characteristic points from the spline and rebuilds the spline. |
| <u>Move</u> | Changes the position of the previous point. |
| <u>Next</u> | Selects the next point. |
| <u>Previous</u> | Selects the previous point. |
| <u>Select point</u> | Selects the control point. |
| <u>eXit</u> | Returns to the previous prompt in the command line. |
| <u>Purge</u> | Deletes the spline data from the document database. |
| <u>Tangents</u> | Edits the start and end points of the spline. |
| <u>System default</u> | Sets tangents at the spline ends by default. |
| <u>tolerance</u> | Specifies a new tolerance value . |
| <u>eXit</u> | Returns to the main prompt in the command line. |
| <u>Close</u> | Closes an open spline with fitting at a join point. |
| <u>Open</u> | Opens a closed spline. |
| <u>Move vertex</u> | Changes the position of control vertexes and deletion of characteristic points. |
| <u>Next</u> | Selects the next point. |
| <u>Previous</u> | Selects the previous point. |
| <u>Select point</u> | Selects the control point. |
| <u>eXit</u> | Returns to the previous prompt in the command line. |
| <u>Refine</u> | Changes the spline's shape |
| <u>Add control point</u> | Adds a control point near the point, selected on spline, between two control points. |
| <u>Elevate order</u> | Increases the degree of the spline (increases the number of control points). |
| <u>Weight</u> | Changes the weight factors at the control points of the spline (the greater the weight factor is, the closer the spline is to the control point). |
| <u>Next</u> | Selects the next point. |
| <u>Previous</u> | Selects the previous point. |
| <u>Select point</u> | Selects the control point |
| <u>eXit</u> | Returns to the previous prompt in the command line. |
| <u>eXit</u> | Returns to the main prompt in the command line. |
| <u>rEverse</u> | Changes the spline direction to the opposite. |
| <u>Exit</u> | Closes the spline editing mode . |

Command prompts:

Select spline:

Select a spline.








Enter an option [Fit data/Close/Move vertex/

Refine/rEverse/Undo/eXit] :

Select the required option. To finish editing, select **Exit** option.

Commands to edit objects

Erase

-  Menu: **Modify** –  **Erase**
-  Toolbar: **Modify** – 
-  Toolbar: **Main** – 
-  Command line: **E, ERASE**

This command deletes objects from the document.

The **Erase** command is also available in the context menu.






Selected objects can be deleted by pressing the **DEL** button on the keyboard.

Command prompts:

Select objects: *Select objects.*

Select objects: *Continue selection of objects or press **Enter** to finish the command.*

Copy

-  Menu: **Modify** –  **Copy**
-  Toolbar: **Modify** – 
-  Command line: **CO, COPY, CP**

This command can copy selected objects once or several times.

The **Displacement** option allows copying of objects by specifying a relative distance with coordinates. The coordinates define a displacement value for the objects being copied.

Command options:

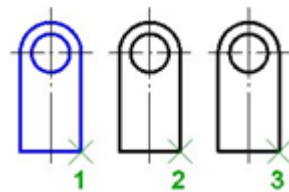
Displacement Specifies a relative distance and direction with coordinates.

Mode Changes the copy mode.

Single Single copy of objects.

Multiple Multiple copies of objects.

Exit Finishes the command in the multiple copy of objects mode.



Command prompts:

Select objects:

Specify base point or [Displacement/Mode]
<Displacement>:

Specify second point or <use first point as displacement>:

Specify second point or [Exit] <Exit> :

*Select objects. Press **ENTER** when the selection is finished.*

*Specify **1** base point.*

*Specify **2** second point.*

*Specify **3** end point. Press **ENTER** to finish the command.*

Mirror

-  Menu: **Modify** –  **Mirror**



Toolbar: **Modify** – 



Command line: **MI, MIRROR**

This command is used to create mirror copies of objects along the specified axis.

Using this command you can quickly create symmetrical objects, you just create one part of the object and the other part of the object is drawn by the command.

Mirrored texts, attributes and their definitions have a mirror view. For a normal view of text, set 0 value (default value) for the **MIRRTEXT** system variable

Command options:

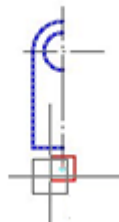
Yes Deletes source objects.

No Source objects are not deleted.

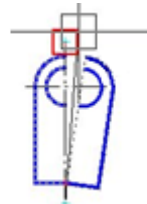
Selection of objects by frame



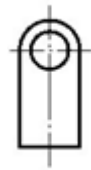
Specifying the first point



Specifying the second point



Result



Command prompts:

Select objects:

Select objects. Press **ENTER** when the selection is finished.

Specify first point of mirror line:

Specify the first point.

Specify second point of mirror line:


Specify the second point.

Erase source objects? [**Yes**/**No**] <N>:

Select the **No** option.

Offset



Menu: **Modify** –  **Offset**



Toolbar: **Modify** – 



Command line: **O, OFFSET**

This command allows the creation of a new object similar to the selected one and is placed at the specified distance from it.

Similarly, arcs and circles have diameters smaller or larger than the source objects, according to which side the offset was made.

If the **Multiple** option is selected, all similar objects are created with the current offset.

Command options:

Erase Erases the source objects after creating similar ones.

Yes Deletes the source objects.

No Source objects are not deleted.

Layer Defines the layer of the source objects.

Source Creates similar objects on the source layer.

Current Creates similar objects on the current layer.

Multiple Switches on the mode for multiple creation of similar objects.

Exit Finishes the command.

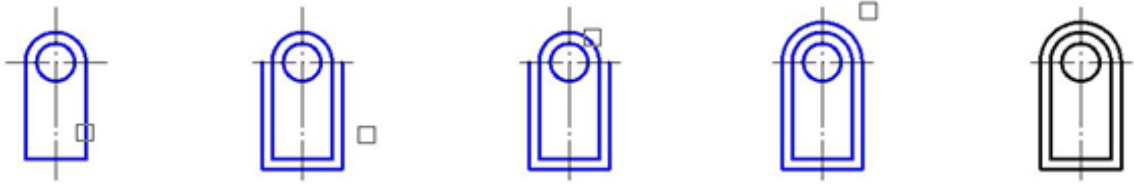
Selection of object

Offset

Selection of object

Offset

Result



Command prompts:

Specify offset distance or [**Erase/Layer**]:

Specify distance.

Select objects to offset or [**Exit**]:

Select an object.

Specify point on side to offset or [**Exit/Multiple**] <Exit>:

Specify a point.

Select objects to offset or [**Exit**]:

Select an object.

Specify point on side to offset or [**Exit/Multiple**] <Exit>:

Specify a point.

Select objects to offset or [**Exit**]:

Select the **Exit** option to finish the command.

Array



Menu: **Modify** –  **Array**




Toolbar: **Modify** – 



Command line: **AR, ARRAY**

This command is used to create two-dimension arrays of the selected objects by placing their copies on the specified circle (polar array) or in the nodes of the specified rectangular grid (rectangular array).

Array command opens the **Array** dialogue box.

There is the  **Select objects** command in the top corner of the dialogue box; it temporarily closes the dialogue box whilst selecting the source objects. There is a line showing the number of selected objects - **Objects selected** below the button.

There is a preview window below the line.



Rectangular array



Array


☒ Rectangular Array ☐ Polar Array

Rows: Columns:

Offset distance and direction

Row offset:  

Column offset:  

Angle of array: 

By default, if the row offset is negative, rows are added downward. If the column offset is negative, columns are added downward.

Select objects
0 objects selected

OK
Cancel
Help

Parameters:

Rectangular array Switches on the rectangular array mode.

Rows: Number of rows.

Columns: Number of columns.

[Offset distance and direction](#)

Row offset: Distance between rows.

Column offset: Distance between columns.

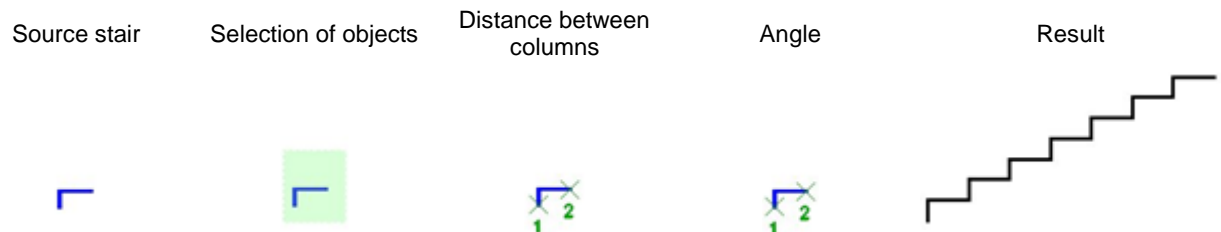
Angle of array: Field to enter an angle of array.



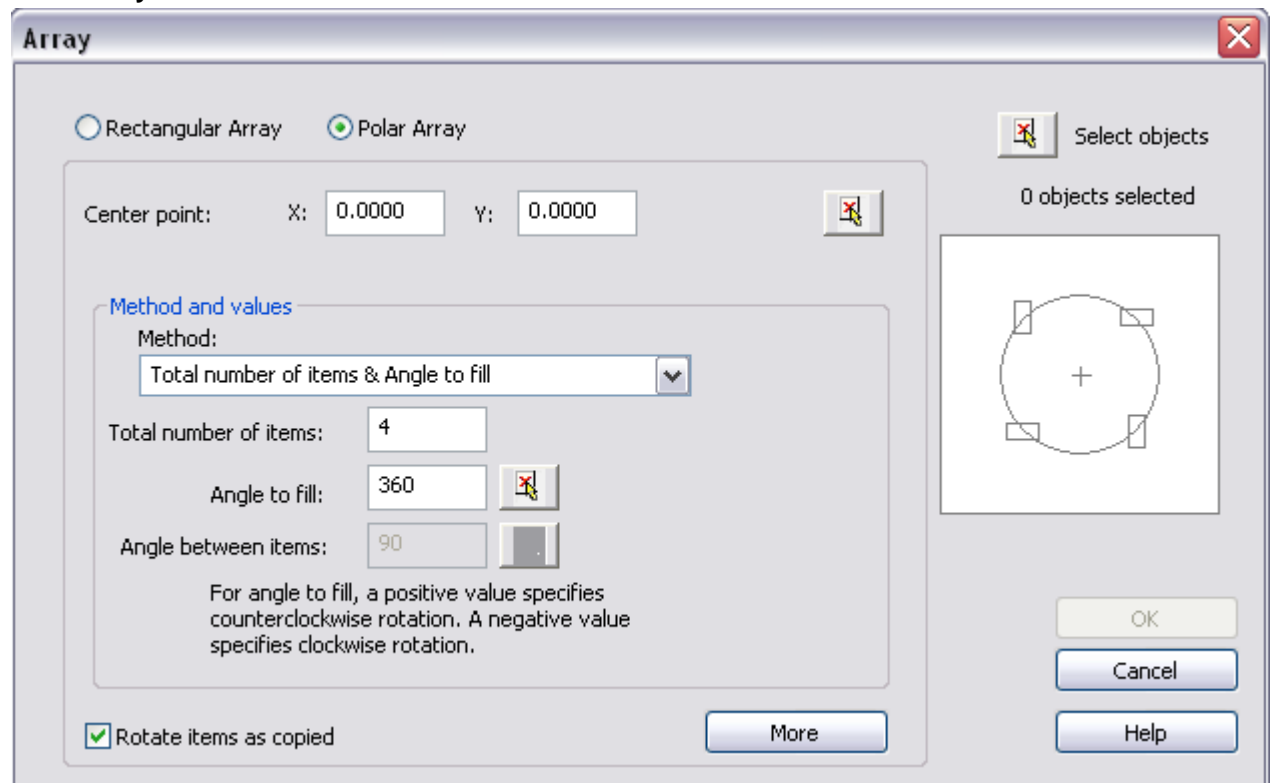
Button temporarily closes the dialogue box to specify the distance between rows and columns and also angle on the screen.

Example of creating stairs using rectangular array:

- Create one more stair;
- Select created objects;
- Specify number of rows - **1**;
- Specify number of columns - **7**;
- Specify distance between columns (specify point **1**, after that point **2**);
- Specify angle (specify point **1**, after that point **2**);




Polar array



The **More** button opens an additional section of the dialogue box to specify the base point of the X,Y axes.

Offset distance and direction

☐ Set to object's default

Base point: X: 0.0000 Y: 0.0000 

Parameters:

Polar array Switches on the polar array mode.

Center point: X: Y: Fields to enter the X, Y coordinates of the array centre.



Button temporarily closes the dialogue box to specify the centre of the array on the screen.

Method and values

Method: A drop-down list to select the method of array creation.

Available options in the drop-down list:

Total number of items & Angle to fill

Total number of items & Angle between items

Angle to fill & Angle between items

Total number of items: Number of elements (with source object).

Angle to fill: Angle of array fill.



Button temporarily closes the dialogue box to specify the fill angle on the screen.

Angle between items: Angle between neighbouring array items.



Button temporarily closes the dialogue box to specify the angle between neighbouring array items.

Rotate items as copied Switches on/off the mode for rotating the elements in the array.

More/Less This button opens/closes an additional part of the dialogue box.

Offset distance and direction

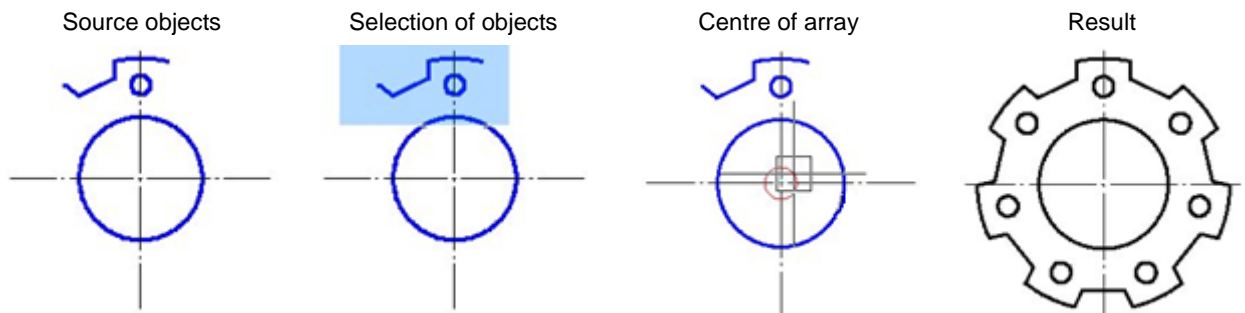
Set to object's default Switches on/off the mode for specifying the base point.

Base point: X: Y: Fields to enter the X,Y coordinates of the base point.



Button temporarily closes the dialogue box to specify the base point on the screen.

Example of creating polar array:



Move



Menu: **Modify** –  **Move**



Toolbar: **Modify** – 



Hotkeys: **CTRL+D**



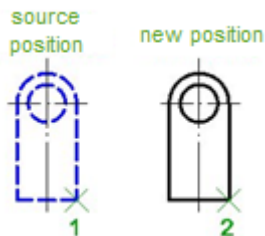
Command line: **M, MOVE**

This command is used to move the selected objects to a specified distance and in a specified direction.

The **Displacement** option allows moving of objects by specifying a relative distance with coordinates. Coordinates define a value for the displacement of objects.

Command options:

Displacement Specifies relative distance and direction using coordinates.



Command prompts:

Select objects:

Specify base point or [**Displacement**]:

Specify second point or <use first point as displacement>:


Select objects. Press **ENTER** when the selection is finished.

Specify a **1** base point.

Specify a **2** second point.

Rotate



Menu: **Modify** –  **Rotate**



Toolbar: **Modify** – 



Hotkeys: **CTRL+E**



Command line: **RO, ROTATE**

The command rotates the selected objects to a specified angle around the specified point.

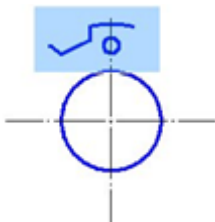
Command options:

Copy Rotates a copy of the selected object.

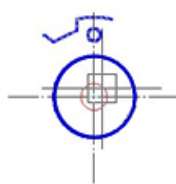
Reference Specifies the angle from the reference line.

Points Specifies the angle from the reference line to the line specified by two points.

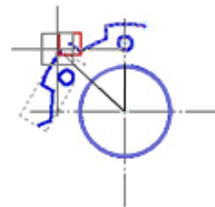
Selection of objects



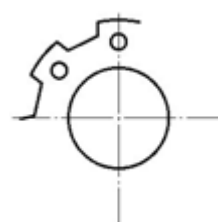
Specifying a reference line by two points



Specifying rotation angle



Result



Command prompts:

Select objects

Specify base point:

Specify rotation angle or [**Copy/Reference**]:

Specify rotation angle or [**Copy/Reference**]:

Select objects. Press **ENTER** when the selection is finished.

Specify a base point.

Select the **Copy** option.

Select the **Reference** option.

Specify the reference angle:
Specify second point:
Specify the new angle or [Points]:

Specify a first point of reference.
Specify a second point of reference.
Specify the angle.

Scale



Menu: **Modify** – 1:2 **Scale**



Toolbar: **Modify** – 1:2



Command line: **SC**, **SCALE**

This command is used to decrease or increase the size of the selected objects whilst retaining their proportions (the scale factors along the X and Y axes are identical). If the scale factor is more than 1, the objects are increased in size; if it is less than 1, they are decreased.

Command options:

Copy

Scaling copy of the selected objects.

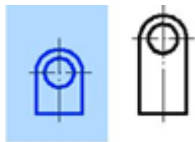
Reference

Scaling the selected objects towards the reference line whose length is a single scale factor and a new length for the reference line.

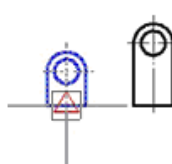
Points

Scaling the selected objects towards the reference line whose length is a single scale factor and a new length for the reference line, specified by two points.

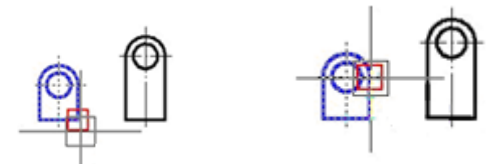
Selection of objects



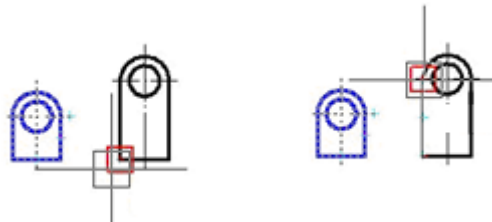
Specifying the base point



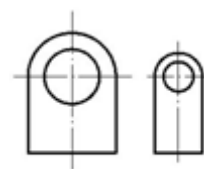
Specifying the reference line



Specifying a new length for the reference line



Result



Command prompts:

Select objects:

Select objects. Press **ENTER** when the selection is finished.

Specify base point:

Specify the base point.

Specify scale factor or
[Copy/Reference]:

Select the Reference option.

Specify reference length:

Specify the first point.

Specify second point:

Specify the second point.

Specify the new length or
[Points]:

Select the Points option.

Specify first point:

Specify the first point.

Specify second point:

Specify the second point.

Stretch



Menu: **Modify** – Stretch



Toolbar: **Modify** – Stretch



Command line: **S**, **STRETCH**

This command allows stretching or moving of objects. Objects, intersecting with a secant frame or polygon are stretched. Objects inside a frame or polygon are moved by the **Stretch** commands as they moved by the **Move** command.

Lines, arcs and segments are stretched only by moving their end points inside the secant frame or polygon. The position of endpoints outside the secant frame or polygon remains the same.

Other primitives are moved or not, according to whether or not their characteristic points are inside the secant frame (polygon).

Characteristic points are the centre of a circle, the insertion point of a block, the leftmost point of a base line for text and to define attributes (it does not depend on the aligning type used for creation).

If the insertion point of a block is moved by the **Stretch** command, all its attributes are also moved.

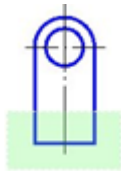
During the preliminary selection of objects, only objects selected with the ordinary or secant frame (polygon) are stretched with **Stretch** command.

The **Displacement** option allows stretching or moving of objects by specifying a relative distance with coordinates. Coordinates define a value for the stretching or offset of objects.

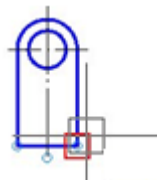
Command options:

Displacement Specifies the relative distance and direction using coordinates.

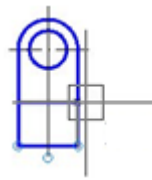
Selection of objects by secant frame



Specifying base point



Specifying a new point



Result



Command prompts:

Select objects:

Specify base point or [**Displacement**] <Displacement>:

Specify second point or <use first point as displacement>:

Select objects. Press **ENTER** when the selection is finished.

Specify the base point.

Specify the second point.

Align



Menu: **Modify – Align**



Command line: **ALIGN**

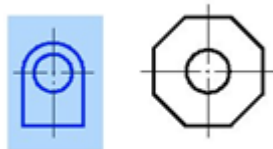
This command moves and rotates an object to align it with another object; scaling is also possible.

Command options:

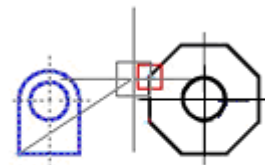
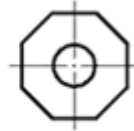
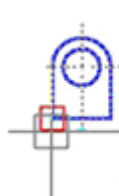
Yes Scales object by aligning points.

No Does not scale object by aligning points

Selection of objects



Specifies first point for aligning objects

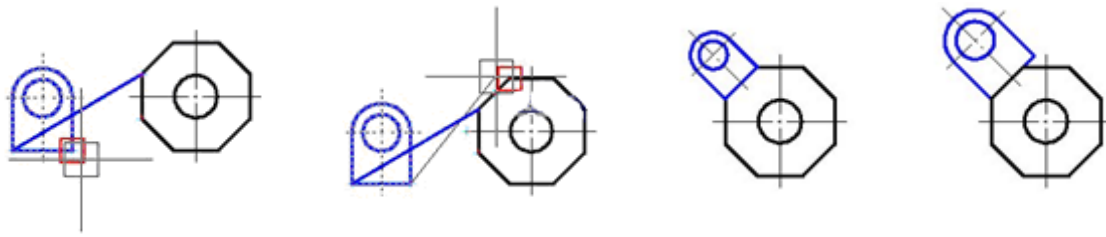


Specifies second point for aligning objects

Result

With scaling

Without scaling



Command prompts:

Select objects:

Select objects for aligning. Press **ENTER** when objects are selected.

Specify first source point:

Specify the first point on the object to be aligned.

Specify first destination point:

Specify the first point on the destination object.

Specify second source point:

Specify the second point on the object to be aligned.

Specify second destination point:

Specify the second point on the destination object.

Specify third source point:

Press **ENTER**.

Scale objects based on alignment points? <N> or **[Yes/No]**:

Select the option.

Distributing copies

Divide



Menu: **Draw – Point >**  **Divide**



Toolbar: **Draw–** 



Command line: **DIVIDE**

The **Divide** command distributes points or blocks at the same distance from each other along an object's length or perimeter. Distribution along object points or blocks divides the object into segments of a specified length.

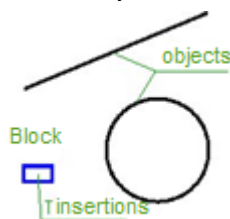
Command options:

Block Switches to block insertion mode.

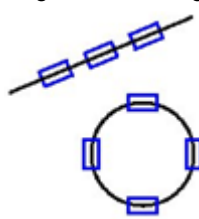
Yes Switches on the object to block aligning mode.

No Switches off the object to block aligning mode.

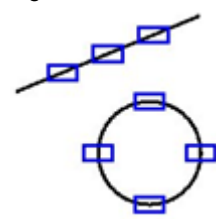
Block and objects to divide



Dividing with block aligning



Dividing without block aligning



Command prompts:

Select objects to divide:

Select an object.

Enter number of segments or **[Block]**:

Select the **Block** option.

Enter name of block to insert:

Specify the block's name and press **ENTER**.

Align block with object? or **[Yes/No]**:

Specify the option.

Enter number of segments:

Specify the number of segments and press **ENTER**.

Chamfer



Menu: **Modify –**  **Chamfer...**



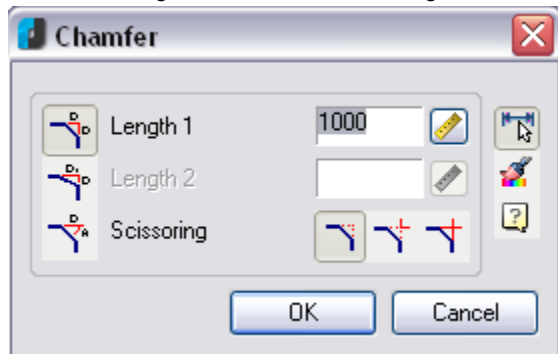
Toolbar: **Modify** –



Command line: **CHA, CHAMFER**

This command is used to create chamfers at the points of intersection of objects, with automatic dimensioning ability. The command allows the creation of several chamfers individually.

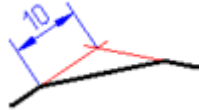
After launching the command, the dialogue box for specifying the chamfer parameters opens:



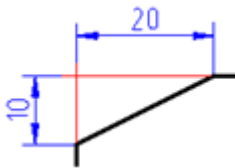
Parameters:



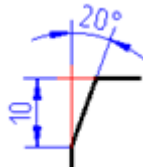
This button switches on the mode for creating a chamfer with similar lengths. In this mode the **Length 2** parameter is unavailable.



This button switches on the mode for creating a chamfer with different lengths.



This button switches on the mode for creating a chamfer by length and angle. In this mode there is a **Corner** parameter instead of the **Length 2** parameter.



Length 1 The first length of the chamfer. This field is used also to specify chamfers with similar lengths.

Length 2 The second length of the chamfer.

Angle Angle of the chamfer.



This button temporarily closes the dialogue box to permit measuring the chamfer length and angle on the drawing. The **Value picker** dialogue box appears to perform measurements.



This button switches on the mode for cutting of full contour lines.



This button switches on the mode for cutting of partial contour lines before their intersection.



This button switches on the mode without lines cutting.

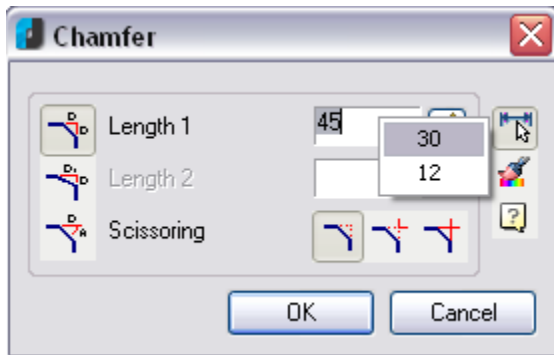


This button switches the automatic dimensioning mode on/off .

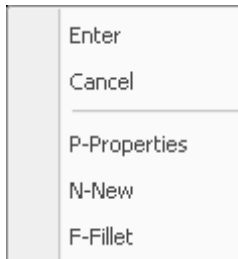


This button temporarily closes the dialogue box to permit copying of properties from created chamfers.

Double clicking or right clicking in the fields to enter values will open the context menu with a list of the recently entered values:



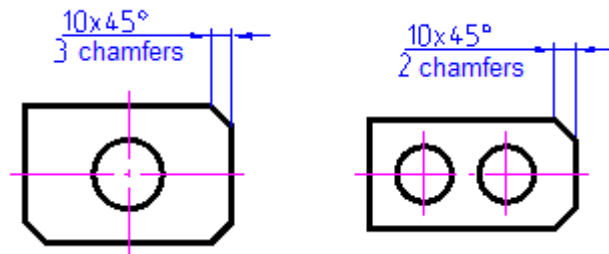
The context menu is available during the process of chamfer creation:



Commands from the context menu:

P-Properties The **Chamfer** dialogue box opens to change the chamfer parameters.


N-New Finishes the creation of one group of chamfers and starts another. The command is applied when you need to create some chamfers with similar dimensions on one object and with the same dimensions on another object:



F-Fillet Switches to the mode for creating fillets. The **Fillet** dialogue box opens to specify the parameters for the fillet.

Fillet



Menu: **Modify** –  **Fillet...**



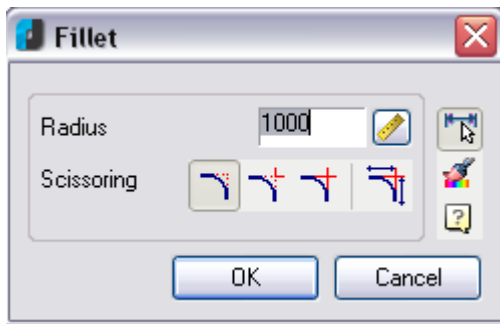
Toolbar: **Modify** – 



Command line: **F, FILLET**

This command is used to create fillets in the intersection points of objects, with automatic dimensioning ability. The command can create fillets individually.

The dialogue box opens after the command is launched:



Parameters:

Radius Radius of fillet.



This button temporarily closes the dialogue box to allow measuring of the fillet radius on the drawing. The **Value picker** dialog appears to perform measurements.



This button switches on the lines cutting mode .



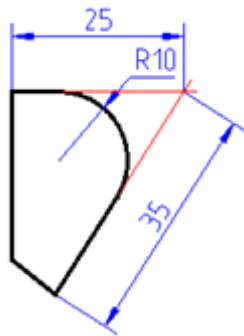
This button switches on the mode for cutting of lines before their intersection.



This button switches on the mode without lines cutting.



This button switches on the additional dimensioning mode .

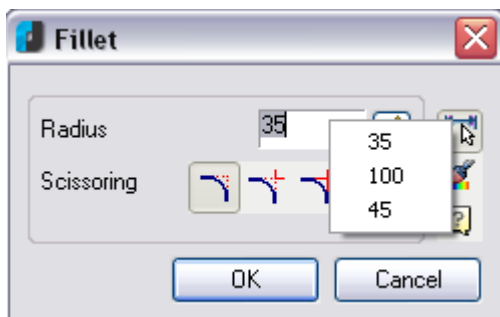


This button switches the automatic dimensioning mode on/off .



This button temporarily closes the dialogue box to allow copying of properties from created chamfers.

Double clicking or right clicking in the fields to enter values will open the context menu with the list of the recently entered values:



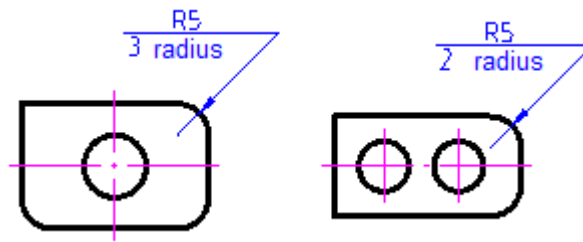
The context menu is available during the process of chamfer creation:



Commands from the context menu:

P-Properties The **Fillet** dialogue box opens to change the chamfer parameters.


N-New Finishes the creation of one group of fillets and starts another. The command is applied when you need to create some fillets with one radius on one object and with the same radius on another object:



C-Chamfer Switches to the mode for creating chamfers. The **Chamfer** dialogue box opens to specify the parameters of the fillet.

Explode



Menu: **Modify** –  **Explode**



Toolbar: **Modify** – 



Command line: **EXPLODE, X**

The **Explode** command divides complex objects, such as polylines, dimensions, hatches and blocks into separate elements. For example, dividing a polyline causes its separation into lines and arcs; an associative dimension divides into a set of simple objects; a block is divided into a set of containing its objects; multiline text is divided into lines.

The **Explode** command is applied only to one nesting level of a complex object, i.e. if it is required to explode a polyline belonging to a block, firstly you have to explode the block and after that the polyline.

Command prompts:

Select objects: *Select an object.*

Select objects: *Select the next object and press **ENTER** to finish the command.*

Explode all objects



Menu: **Modify** –  **Explode all objects**



Toolbar: **Modify** – 



Command line: **EXPLODEALL**

The command explodes all nanoCAD objects (notes, tables etc.) into primitives.

It is recommended not to apply the command, unless necessary, because the exploded objects are not restored. After explosion of objects you cannot apply nanoCAD special functions and editing commands to them.

Creating and editing complex objects

Groups of objects



Menu: **Modify – Group...**



Command line: **GROUP**

To edit a set of objects they can be grouped in the document. You can select the whole group and its elements to edit. New objects can be added to a group and existing objects can be excluded from a group. Objects can belong to several groups at once, and a group can belong to another group. To restore the configuration of source groups grouped into one group, you have to ungroup nested groups.

Groups can be assigned names or use default names. An unnamed group (a group without its own name) has a default name - An , where n is the number of created groups.

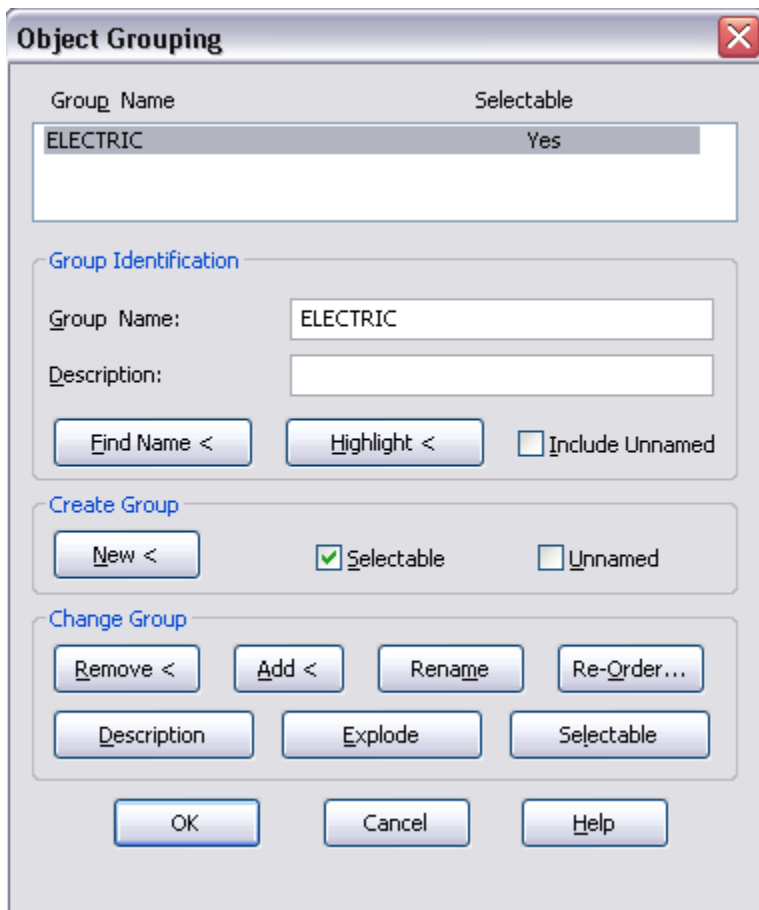
A group name can contain up to 31 symbols and cannot contain spaces; it can contain letters, figures and special symbols (dollar sign (\$), hyphen (-) and the break character (_)).

Groups are saved with a document and can be used in the following working sessions.

Main distinction between groups and blocks:

- Objects in a group can be edited without ungrouping, but to edit objects in a block, the block must be exploded.
- A group cannot be transferred to another document; it can be used only in the document where it was created.

The **Group** command opens the **Object Grouping** dialogue box where there is a list with the names of existing groups in the document and descriptions for every group that is selectable:



The **Object Grouping** dialog box is shown. It has a title bar with a close button. The main area is divided into several sections. At the top, there is a table with two columns: 'Group Name' and 'Selectable'. The first row shows 'ELECTRIC' and 'Yes'. Below this is a section titled 'Group Identification' with fields for 'Group Name' (containing 'ELECTRIC') and 'Description'. There are buttons for 'Find Name <' and 'Highlight <', and a checkbox for 'Include Unnamed'. Below that is a section titled 'Create Group' with a 'New <' button and checkboxes for 'Selectable' (checked) and 'Unnamed'. At the bottom is a section titled 'Change Group' with buttons for 'Remove <', 'Add <', 'Rename', 'Re-Order...', 'Description', 'Explode', and 'Selectable'. At the very bottom are 'OK', 'Cancel', and 'Help' buttons.

Parameters:

Group Identification

Group Name: Specify a group name.

Description: Displays the description of the group (if it was specified during group creation).



This button temporarily closes the dialogue box for selecting an object when you need to clarify its group.



This button temporarily closes the dialogue box for highlighting a group's objects on the screen.

Include Unnamed

Switches on/off the input of unnamed groups in the list in the **Object Grouping** dialogue box.

Create Group



This button temporarily closes the dialogue box for selecting the objects of a group.

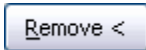
Selectable

Switches on/off the mode of whole group selection if one object is selected.

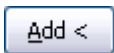
Unnamed

Switches on/off the mode for creating unnamed groups.

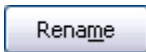
Change Group



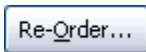
This button temporarily closes the dialogue box to allow selecting objects to delete from the selected group.



This button temporarily closes the dialogue box to allow selecting objects to add them to the selected group.



This button updates a change of name for a group.



This button opens the **Order Group** dialogue box to change the order of objects in a group.



This button updates the changed description for the selected group.




This button deletes the selected group. Objects of the group are not deleted from the document.

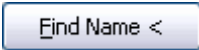


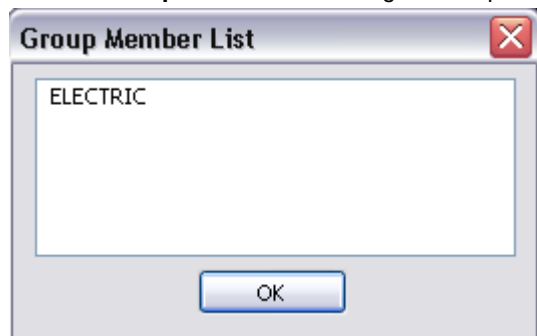
This button changes the selectable property of a group in the document.

To create a group:


- Enter the name for the created group and its description in the **Group Name** and **Description** fields of the *Group Identification* section. A name can contain letters, figures and special symbols, but must not contain spaces and. The **Description** field can be empty.
- Select the  button to temporarily close the dialogue box to select the objects of a group.
- Select the objects, press **ENTER**.
- Select **OK** in the opened dialogue box.

To identify a group:

- To define the groups the object belongs to, select the .
- After the dialogue box temporarily closes, specify an object on the screen.
- The **Group Member List** dialogue box opens with all groups the selected object belongs to:

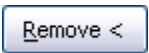



- Select **OK** to get return to the previous dialogue box.
- Select **OK** to close the dialogue box.

The  button temporarily closes the dialogue box to allow highlighting of a group's objects on the screen. The **Object grouping** dialogue box is displayed with the highlighted object:




To delete objects from a group:

- Select a group in the list.
- Deselect the **Selectable** checkbox.
- Select the .
- Select the objects to delete.
- Select the **End** option (in the context menu or command line).
- Select **OK** to close the dialogue box.

Note 1: The description of the group is saved when all the objects are deleted from the group. To delete a group from a document, select the .

Note 2: When a group is deleted from a document, objects of the group are not deleted.

To add objects to a group:

- Select group in the list.
- Select the .
- Select the objects to add.
- Select the **End** option (in the context menu or command line).
- Select **OK** to close the dialogue box.

Note: When adding objects belonging to other groups to a group, the **Selectable** checkbox should be deselected. If it is selected, the groups that the objects belong to will be added to the group.


To rename a group or change a description:

- Select the group.
- Make changes in the **Group Name** and **Description** fields of the *Group Identification* section.
- Select the  or .

The messages like *Group name has been updated* or *Group description has been updated* are shown in the bottom left corner.

- Select **OK** to close the dialogue box.

To change the order of objects in the group:

- Select the group.
- Select the .
- Specify the required parameters in the **Order Group** dialogue box:

Order Group

Group Name
ELECTIC

Description
None

Remove from position (0 - 2):

Enter new position number for the object (0 - 2):

Number of objects (less than 3):

Re-Order Highlight Reverse Order

OK Cancel Help

Parameters:

| | |
|--|--|
| Group Name | List of existing groups. |
| Description | Shows the description of the selected group in the list (if it was specified when the group was created) |
| Remove from position (0-2) | Field to enter the current position (number) of the object in the group. |
| Enter new position number for the object (0-3): | Specifies the new position number of objects in a group. |
| Number of objects (less than 4): | Specifies the number of objects, whose positions are changed. |

Re-Order

This button changes the positions of objects in the group.

Highlight

This button opens the **Object Grouping** dialogue box to display the order of objects in the group

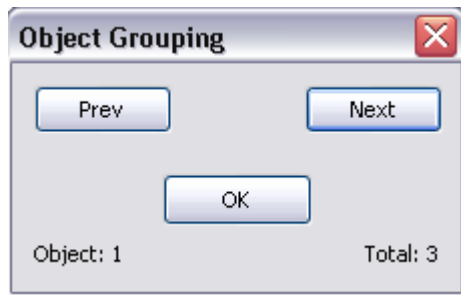
Reverse Order

This button changes the objects into their opposite order.

Note 1: The order of object numbering is the order in which objects were selected when they were added to the group. Numbering starts from 0.

Note 2: You can change the positions of several objects at once.

The  temporarily closes the **Order Group** dialogue box and opens the **Object Grouping** dialogue box:



Parameters:

Prev The button for the previous object selection.

Next The button for the next object selection.

Object: 1 Shows the current position of the object in the group.

Total: 4 Shows the number of objects in the group.

Note: A selected object is highlighted on the screen..

Blocks

A block is a collection of objects that can be associated together to form a single object. A block can be inserted in a drawing as many times as you need.

Blocks are used to create elements of drawings which are used many times, to simplify the drawing process..

Blocks can be used to create user libraries of frequently used parts and components.

Relating all occurrences of a block to the same description of the block in the drawing database allows you to decrease the file size because every new insertion of the block causes the adding of the information about the insertion place, scale factors and rotation angle to the information available.

Reference to the block means the block definition or the insert block.

When you create a block, its definition, which is stored in the blocks table of the document file, is created and is not displayed on the drawing. A block definition can contain other (nested) blocks. The only restriction on nested blocks is that you cannot insert blocks that reference themselves.

During an insert block, the drawing component of the block is created. Any definition can have a set of components or not have any. When you change the block definition, all its components will change automatically.

An existing block can be changed by redefining it. A new block is created with the name of the existing block. After the block description is changed, all occurrences of this block in the document are changed automatically.

When a block is created, objects are placed in the block with their current properties; these values cannot be corrected in future.

For example, if you create a block from red segments, after it is created you cannot influence the colour of these objects', unless you have set these segments' colour to a **By Block** value.

By modifying the properties of a block insertion point, you can change the display of objects inside the block (those with **By Block** value).

If the object coming into the block has a property with a **By Layer** value, that changes the layer properties; it is possible to change the object's properties inside the block.

Users can also set attributes for a block to be created. A block attribute is text included in the block whose value can be re-defined for each block intersection. For example, insert the following multiline text in the block displaying a workstation. Such fields are referred to as the block attributes and the text registered in such fields is called the attribute value.

There are several ways to create and use blocks:

- Group objects and save them as block in the current drawing (**Draw** menu – **Block > Make**);
- Save a block in a separate file to use it in other drawings by specifying in the command line **WBLOCK**;
- Insert any other drawing as incidental to a drawing with blocks (**Insert** menu – **DWG reference...**;
- Create a file with a set of frequently used blocks to use it as a library.

Creation of a block



Menu: **Draw – Block >** **Make...**

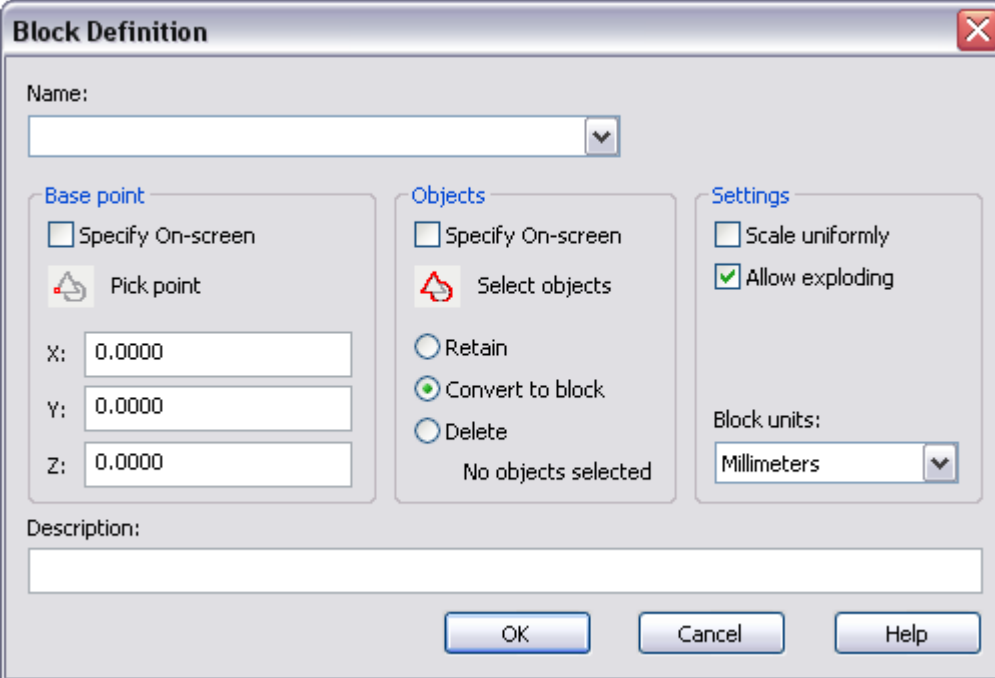


Toolbar: **Draw –** **Create Block...**



Command line: **ACADBLOCKDIALOG, B, BLOCK, BMOD, NEWBL**

The **Block Definition** dialogue box allows the specifying of the parameters to create a new block and redefine a definition for existing block:



The dialog box is titled "Block Definition" and has a close button (X) in the top right corner. It contains several sections: "Name:" with a text field and a drop-down arrow; "Base point" with a checkbox "Specify On-screen", a "Pick point" button, and three text fields for X, Y, and Z coordinates, all currently set to 0.0000; "Objects" with a checkbox "Specify On-screen", a "Select objects" button, and three radio buttons: "Retain", "Convert to block" (which is selected), and "Delete", followed by the text "No objects selected"; "Settings" with a checkbox "Scale uniformly", a checked checkbox "Allow exploding", and a "Block units:" section with a drop-down menu currently set to "Millimeters"; and a "Description:" text field at the bottom. At the very bottom are three buttons: "OK", "Cancel", and "Help".

Parameters:

Name: Type a name for the created block.
To redefine the existing block, choose its name from the drop-down list.

Base point

Specify On Screen At the block insertion in the drawing, this point will be located at the centre of the cursor after the dialogue box is closed.



Pick Point

At the block insertion in the drawing, this point will be located at the centre of the cursor after the dialogue box is closed temporarily.

X: Y: Z: Specify the point coordinates for the block insertion.

Objects

Specify On Screen Select the objects to be entered into the block after the dialogue box is closed.



Select Objects

Select the objects to be entered into the block after the dialogue box is closed temporarily.

Retain Retains the objects selected for the block in the drawing after creation of the block definition.

Convert to block Creates the block entry at the location of the objects selected for the block after creation of the block definition.

Delete Deletes the objects selected for the block from the drawing after creation of the block definition.

Selected objects The information on the quantity of objects selected for the block.

Settings

- Scale uniformly** Prevents the block entries from using a different scale.
- Allow exploding** Defines whether the block entries can break into their components using the **Explode** command.
- Block units:** Specify the insertion units for the block entry.
- Description:** Allows the hidden attribute of text description of the block to be specified.
- Note:** The objects to be added to the block can be selected in advance, before opening the **Block Definition** dialogue box window.

Block attributes

A block's attribute is used to link text information or any other data, called an *attribute value*, with the block.

It is possible to enter an attribute's value, which is saved with the block, when you insert a block with variable attributes. You can assign a different value to an attribute when you insert the same block.

Information, stored in drawing attributes, can be exported in electronic tables or databases for further processing and generation of different documents; for example, specifications or bill for materials. You can link several attributes with different names within one block.

Attributes for a block should be created before block definition.

To create an attribute



Menu: **Draw – Block >**  **Define Attributes...**

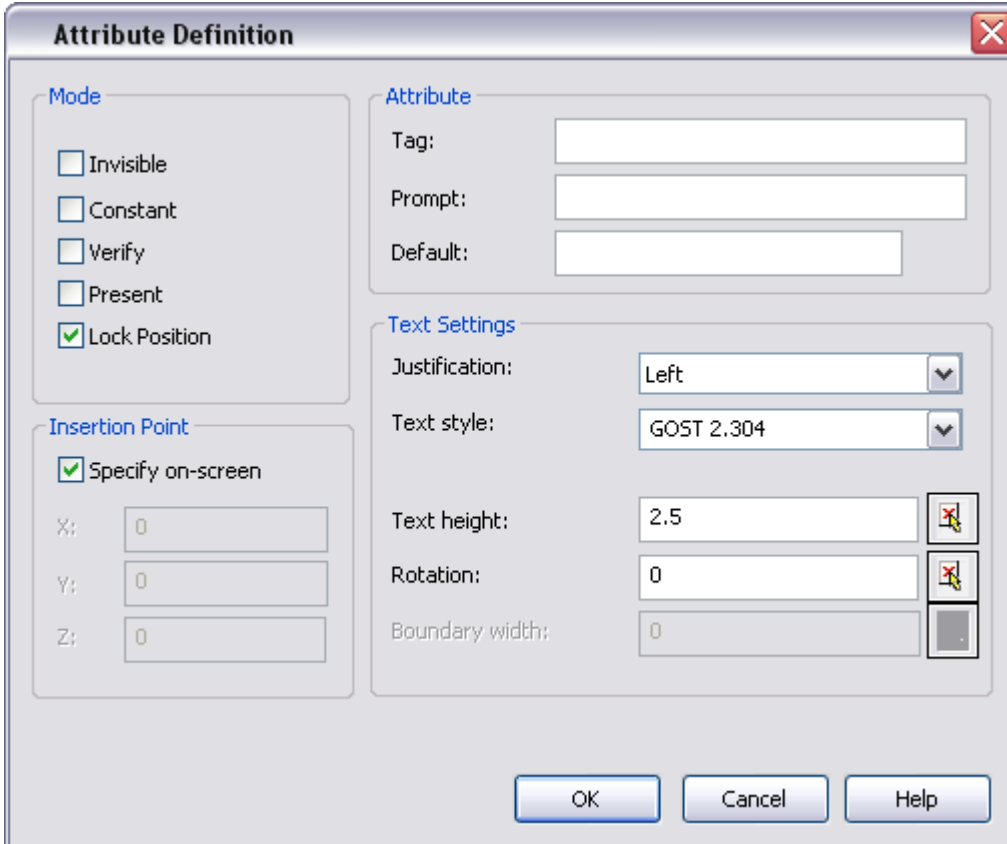


Toolbar: **Draw –**  **Define Attributes ...**






Command line: **ATT, ATTDEF**

The **Attribute Definition** dialogue box is used to specify an attribute's parameters:



The **Attribute Definition** dialog box is used to specify an attribute's parameters. It contains the following sections:

- Mode:**
 - ☐ Invisible
 - ☐ Constant
 - ☐ Verify
 - ☐ Present
 - ☒ Lock Position
- Attribute:**
 - Tag:
 - Prompt:
 - Default:
- Text Settings:**
 - Justification:
 - Text style:
 - Text height: 
 - Rotation: 
 - Boundary width: 
- Insertion Point:**
 - ☒ Specify on-screen
 - X:
 - Y:
 - Z:

Buttons: **OK**, **Cancel**, **Help**

Parameters:

Mode

Invisible The attribute value of the inserted block is not displayed on the screen and not printed.

| | |
|----------------------|--|
| Constant | Sets the fixed attribute value for all block definitions. |
| Verify | Checks the accuracy of the attribute value during a block insertion. |
| Present | Whether to attribute a default value at a block insertion. |
| Lock Position | Locks the position at a block insertion. |

Attribute

| | |
|-----------------|---|
| Tag: | Sets an attribute name. |
| Prompt: | The text containing the current attribute is displayed at the block intersection.. If left empty, the prompt will be the attribute tag. |
| Default: | Value which is applied to the attribute by default |

Insertion point

| | |
|--------------------------|---|
| Specify on screen | Specify the insertion point on the drawing. |
| X: Y: Z: | Coordinates of the insertion point. |

Text Settings

| | |
|-----------------------|-------------------------|
| Justification: | Choice of alignment |
| Text style: | Sets the text style. |
| Text height: | Sets the text height. |
| Rotation: | Sets the text rotation. |



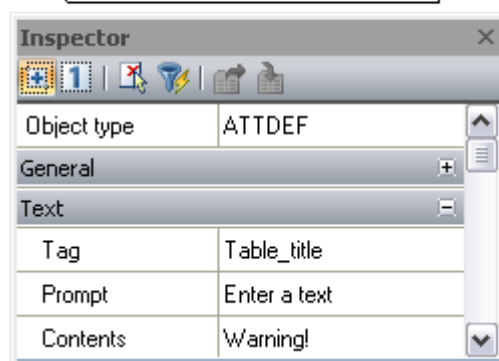
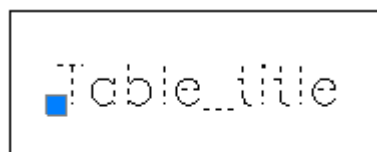
The dialogue box temporarily closes and you can specify height or rotation on the screen.

After an attribute is created, you can add it to a set of objects during the creation of the block. You should select not only geometric objects but also attributes in answer to the command prompt for selection of objects during creation of a block. The order of selection of attributes defines the order of the prompts to enter the values of the attributes during block insertion.

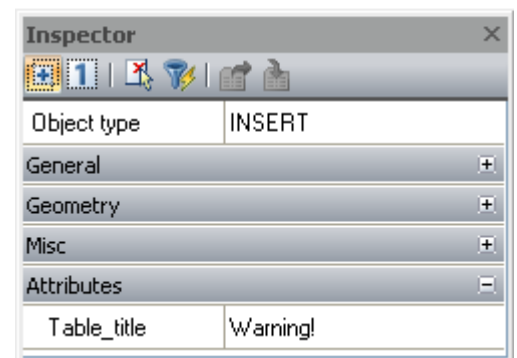
You can link an attribute with a block when redefining a block.

Example of "Table_title" attribute with "Warning!" value:

Before an attribute insertion in the block:

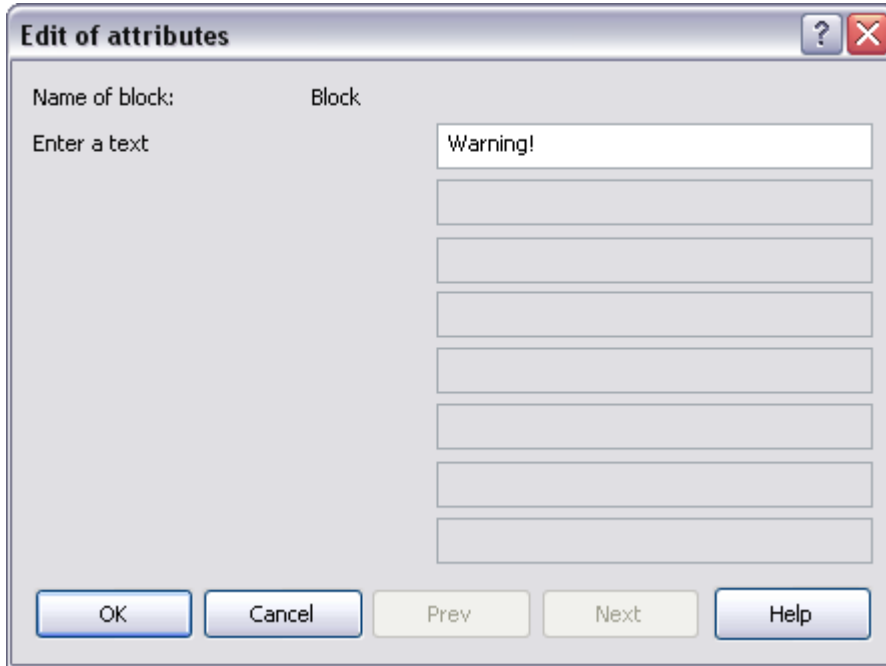


After an attribute insertion in the block:



Attributes of inserted blocks can be edited using the **Attributes block manager** (**Modify** menu – **Object** or **Modify Object** toolbar) and in the **Inspector** dialogue box.


The **Attributes block manager** opens the **Edit of attributes** dialogue box where you can change the values of the block attributes:



The **Edit of attributes** dialog box has a title bar with a question mark and a close button. It contains a label 'Name of block:' followed by the text 'Block'. Below this is a label 'Enter a text' followed by a text input field containing 'Warning!'. There are five more empty text input fields stacked vertically. At the bottom, there are five buttons: 'OK', 'Cancel', 'Prev', 'Next', and 'Help'.

Block Insertion



Menu: **Insert** –  **Block...**



Toolbar: **Draw** –  **Insert Block...**



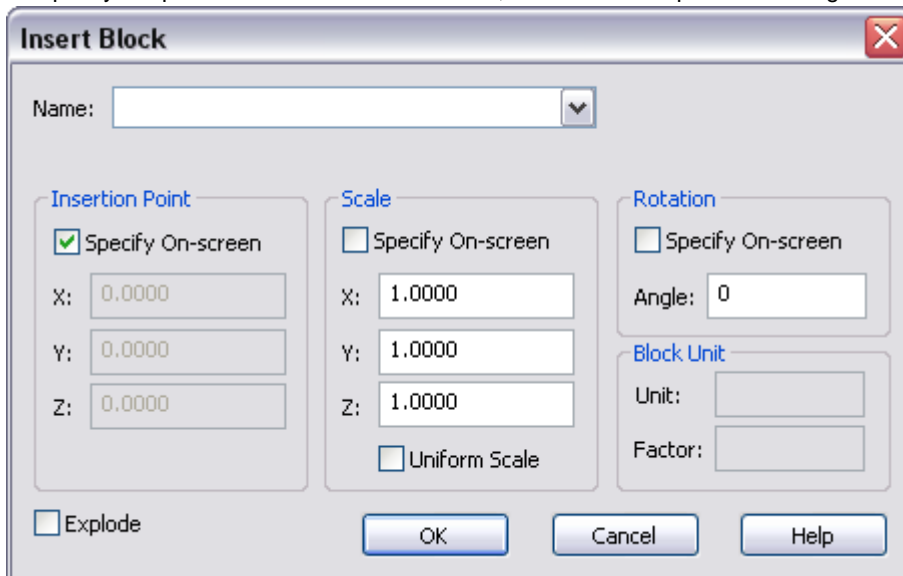
Command line: **I**, **INSBL**, **INSERT**

This command inserts previously defined blocks in the drawing.

If the rotation angle of the block is specified, the centre of rotation is placed at the insertion point. If the rotation angle of the block is specified on the screen (by point), the slope angle of an imaginary line starting from a specified point to the insertion point of the block is set as the rotation angle.

After inserting it in the drawing, a block is processed as a graphic primitive. To be able to work with separate objects of a block, it can be exploded during insertion by selecting the **Explode** checkbox in the **Insert Block** dialogue box, or after insertion by using the **Explode** command from the **Modify** menu.

To specify the parameters of an inserted block, the command opens the dialogue box:



The **Insert Block** dialog box has a title bar with a close button. It contains a 'Name:' label followed by a drop-down menu. Below this are three sections: 'Insertion Point' with a checked 'Specify On-screen' checkbox and input fields for X (0.0000), Y (0.0000), and Z (0.0000); 'Scale' with an unchecked 'Specify On-screen' checkbox and input fields for X (1.0000), Y (1.0000), and Z (1.0000), plus an unchecked 'Uniform Scale' checkbox; and 'Rotation' with an unchecked 'Specify On-screen' checkbox and an 'Angle:' input field (0). Below these is a 'Block Unit' section with 'Unit:' and 'Factor:' input fields. At the bottom left is an unchecked 'Explode' checkbox. At the bottom right are 'OK', 'Cancel', and 'Help' buttons.

Parameters:

Name: Choose the name of the block definition from the drop-down menu. If the list is empty, the current document does not contain any block definitions.

Insertion point

Specify on screen If this checkbox is on, the appropriate parameter is requested from the user after closing the dialogue box window.

X: Y: Z: Coordinates of an insertion point.

Scale

Specify on screen If this checkbox is on, the appropriate parameter is requested from the user after closing the dialogue box window.

X: Y: Z: Scale factors for the X, Y, Z axes.

Uniform Scale The specified scale for the X axis will be applied to the Y and Z axes automatically.

Rotation

Specify on screen If this checkbox is on, the rotation angle is requested from the user after closing the dialogue box window.

Angle: Rotation angle for an inserted block.

Block Unit

Unit: Insertion units, which have been set at the creation of the block definition.


Factor: The scale factor calculated as a ratio of the block units to the drawing units.

Explode Allows insertion of the block split into separate objects.

Note: If you set a negative scale value along any axis, when the block is inserted, the block will be mirrored towards this axis.

Managing blocks in a current drawing

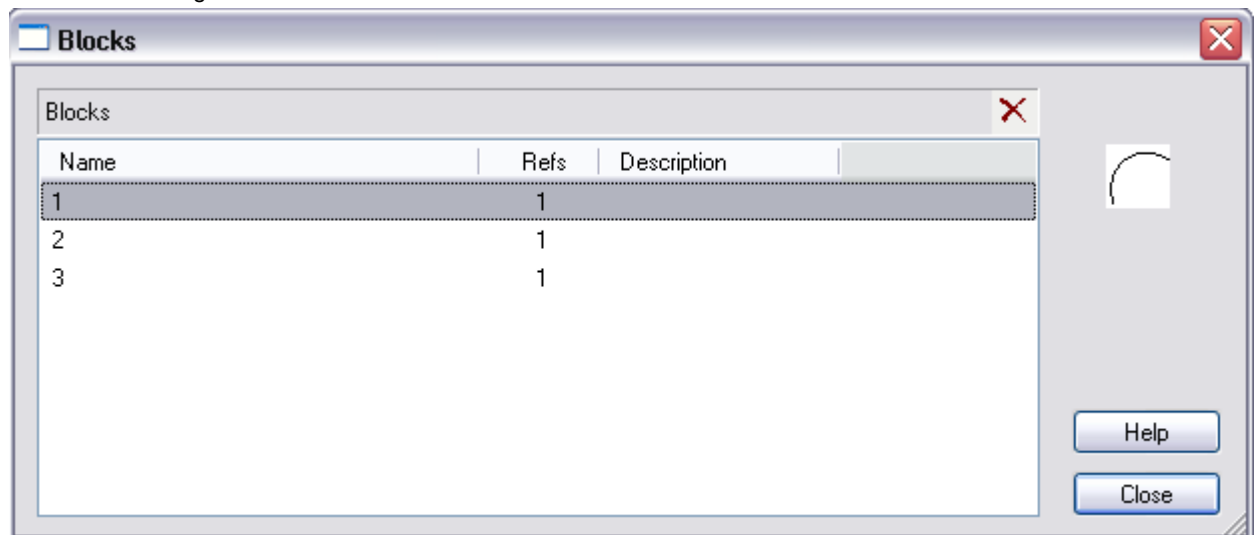


Menu: **Format** –  **Blocks...**



Command line: **BLOCKSCMD**

The **Blocks** dialogue box contains information about all blocks in the document and about their insertion numbers:



Columns:

Name List of blocks in the document.

Refs Information about the number of block insertions in the current document.


Description Information from the **Description** field of the **Block Definition** dialogue box.

To rename a block:

- Click twice on the block's name.

- Enter a new name from the keyboard.

To delete the description of a block:

- Select the block from the list.
- Select the  button or select the **Delete** command from the context menu.
- Select the **Yes** button.
- Select **Close** to close the dialogue box.

Note 1: The **Delete** command (or **Del** button) deletes only the block insertion in the document. To delete the block description, use the **Block Definition** dialog.

Note 2: Some block descriptions are system. They cannot be deleted.

Block editor



Menu: **Tools** –  **Edit block insertion...**




Command line: **REFEDIT**

You can edit block descriptions without closing the current drawing by editing block insertions.

Exploding block insertion

When a block's insertions are exploded, a block is divided into its separate objects.

To explode a block, select the **Explode** command or select the  button on the **Modify** toolbar. If it was prohibited to explode the block when the description was created, such block insertions cannot be exploded.

Saving a block in a separate file

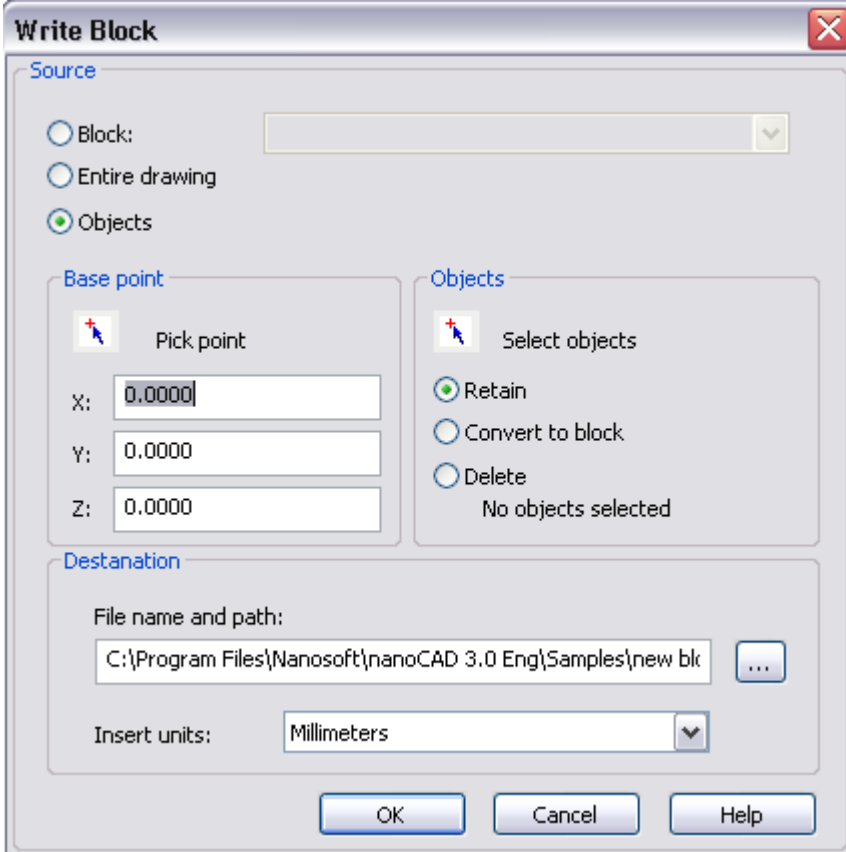


Command line: **WBLOCK, ACADWBLOCKDIALOG, W**



The command allows saving in a separate file:

- A whole drawing with all changes;
- A block in the current document;
- Separate fragments of the drawing.

The **Write Block** dialogue box appears after the **WBLOCK** command is launched:



The **Write Block** dialog box is shown with the following settings:

- Source:**
 - ☐ Block: (empty dropdown)
 - ☐ Entire drawing
 - ☒ Objects
- Base point:**
 -  Pick point
 - X: 0.0000
 - Y: 0.0000
 - Z: 0.0000
- Objects:**
 -  Select objects
 - ☒ Retain
 - ☐ Convert to block
 - ☐ Delete
 - No objects selected
- Destination:**
 - File name and path: C:\Program Files\Nanosoft\nanoCAD 3.0 Eng\Samples\new blk ...
 - Insert units: Millimeters

Buttons: OK, Cancel, Help

Parameters:

Source

| | |
|------------------------|--|
| Block: | Switches on the mode to select a block in the document, from a list, in order to save it. |
| Entire document | Switches on the mode for saving the whole document in a separate file. |
| Objects | Switches on the mode for saving selected objects in the current document in a separate file. |

Base point



Pick point

This button temporarily closes the dialogue box to allow a base point to be specified on the screen.

X: Y: Z: Coordinates of the base point.

Objects



Select objects

This button temporarily closes the dialogue box to allow objects on the screen to be selected.

Retain Switches on the mode to keep the selected objects in the drawing after saving them in a separate file.


Convert to block Switches on the mode for creation of a block insertion instead of selected objects after they were saved in a separate file.

Delete Switches on the mode for deleting the selected objects after they were saved in a separate file.

Objects selected: 18 Information about the number of objects selected for saving.

Destination

File name and path:

The  button opens the **Save Document File** dialogue box to specify the path, format and name for the file.

Insert units Drop-down list for selecting the measurement units of the saved file.

Insert External Reference



Menu: **Insert –**  ***.dwg Reference...**



Toolbar: **Draw –** 



Command line: **XA, XATTACH**

External references allow you to add information from other drawings to the current document. It is possible to insert several external references into one document. Conversely, the same document can be used as an external reference in several other documents. External references can contain inserted external references. When you add an external reference, all external references inserted in it will also be displayed in the current drawing.

External references can only link external documents to the current document. The objects placed in the drawing file with external references are displayed in the current drawing with other objects in this drawing, but they are not added into the drawing. The external reference is a kind of label that indicates the path to an external drawing. When you add the external reference, its objects are not copied into the current drawing or loaded from the external reference file every time you open basic drawing or restart the external reference. Any changes made to the external reference will be also displayed when you open the basic document or restart an external reference.

When you insert an external reference into the drawing, the file size of the current drawing is increased slightly.

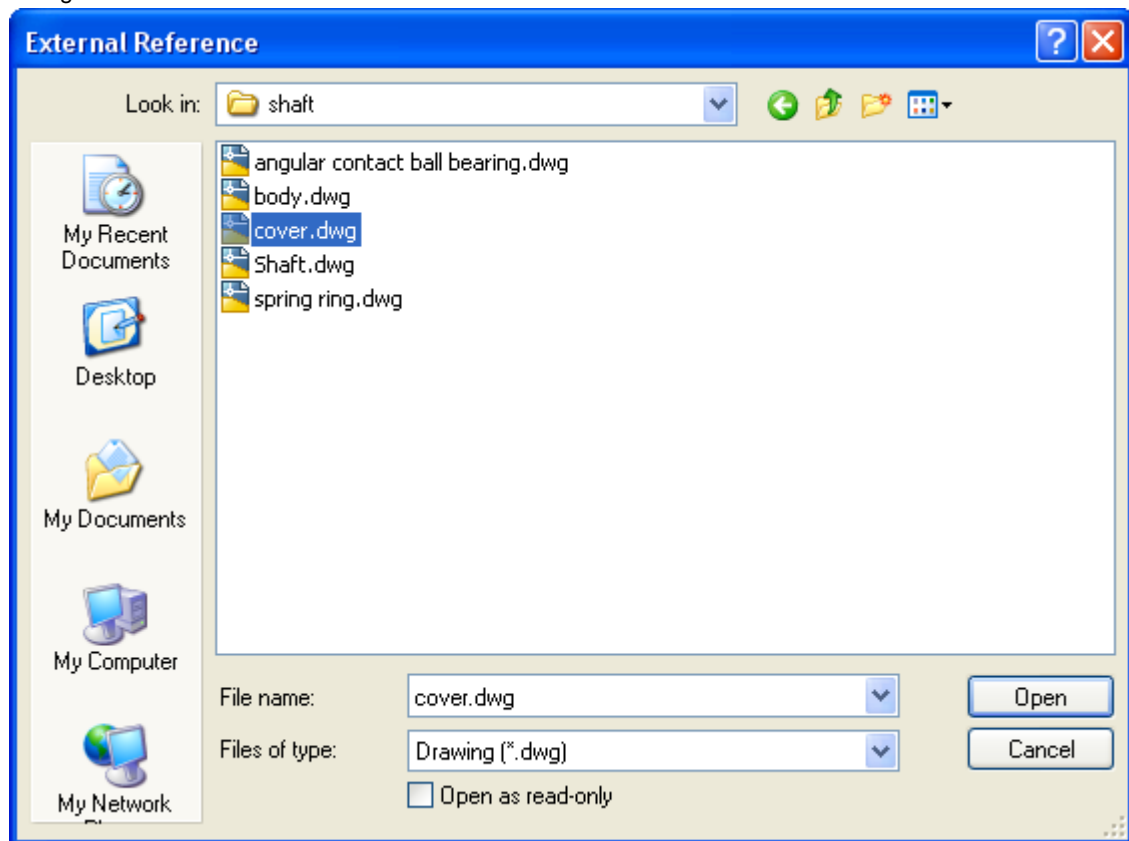
Since the external references are always kept in separate files, then the exchange of drawings should convey not only the basic drawings, but all drawings which are referred to.

You can specify different types of external reference: *attachment* into the drawing and *overlay* on the drawing. When you insert an external reference using an *attachment* type, then all external references inserted into the drawing are added to it. If you choose an *overlay* type when you insert an external reference, then external references inserted into it are ignored. *Overlay* external references are used when the information provided by an external reference in the current drawing is not needed for later use of this drawing as an external reference.

There are three types of folder path information to save with an attached reference:

- *Full (absolute) path* is a fully specified folders hierarchy that locates the file reference. This is the most specific but least flexible option.
- *Relative path* is a partially specified folders hierarchy that is defined relative to the current drawing (the folder in which it is stored). If you choose this type, it is necessary to save the current drawing. For the enclosed reference the relative path specifies a reference location, which can be the current open document. This is the most flexible option, and enables you to move your current drive to a different drive that uses the same folder structure. If the file that is being referenced is located on a different local hard drive or on a network server, the relative path option is not available.
- *No path* – not specify a path to an external reference. Specifying the *No path* option is useful when moving a set of drawings to a different folder hierarchy or to an unknown folder hierarchy.

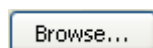
To insert an external reference into the drawing, specify the path and file name in the opened **External Reference** dialogue box:



Select the necessary external reference and click **Open**. The **External Reference** dialogue box will open.

Options:

Name: The list of the names of the external references inserted into the drawing.



Opens the **Open** dialogue box to choose files for the insertion of new references.

Found in: Displays the file path where the external reference is to be found.

Saved path: Displays the saved path of access to the external reference.

Reference Type

Attachment When you attach a drawing file as an external reference, you link that referenced drawing to the current drawing.

Overlay An overlaid external reference is not included when the drawing is itself attached or overlaid as an external reference to another drawing.

Path Type Choose the saved path type to the external reference:

Full path
Relative path
No path

Insertion Point

Specify On-screen Select the box to set the X,Y,Z coordinates values in the command line or specify the position on the screen. The «X» «Y» «Z» fields of this section are inaccessible.

X: Y: Z: Set the X, Y, Z coordinate values in the corresponding fields of the current document.

Scale

Specify On-screen Select the box to set the scale values in the command line or specify the position on the screen. The «X» «Y» «Z» fields of this section are inaccessible.

X: Y: Z: Set the scale values in the corresponding fields of the current document.

Uniform Set the scale factors on the Y and Z axes to the same scale factor as the X axis.

Scale

Rotation

Specify On-screen Select the box to set the rotation angle value in the command line or specify its position on the screen. The *Angle* field of this section is inaccessible.

Angle: Set the angle value on which it is necessary to turn the reference entry in the current document.

Block unit

Unit: Displays the specified parameter value of the *insertion units* for the inserted block.

Factor: Displays the scale factor, which is calculated based on the *block insertion units* parameter value and *document units*.

Edit references



Menu: **Tools – External reference > Edit reference**

Command line: **REFEDIT**

The use of external references allows you to quickly combine multiple drawings in a single document. To make work with external references more convenient, you can edit the references directly in the current drawing to which they are added.

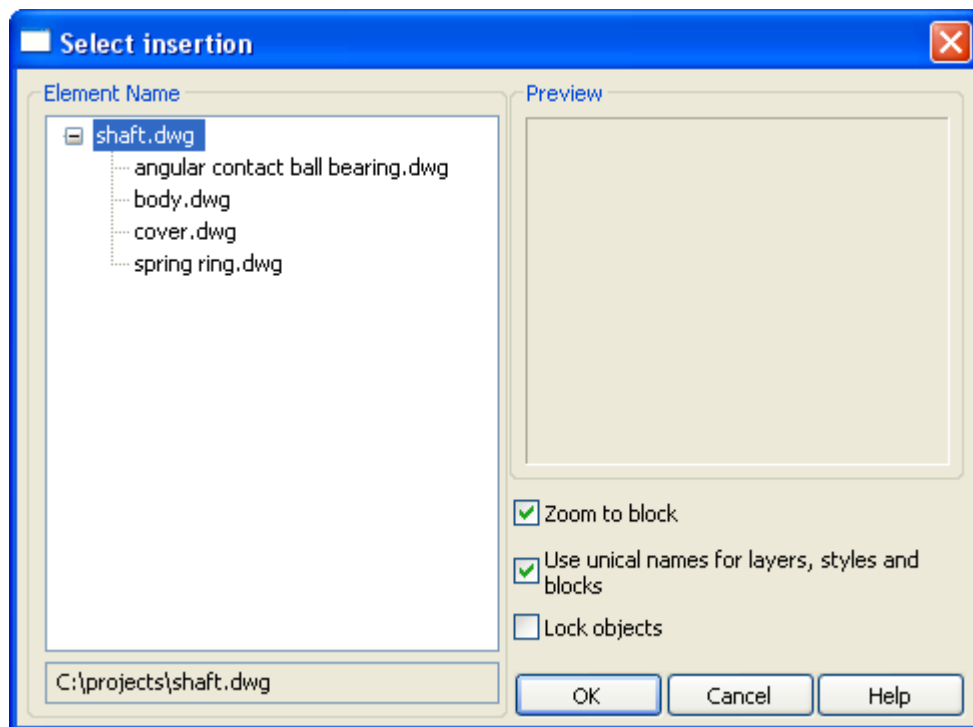
To edit a referenced drawing from within the current drawing, you use the *working set* to identify objects that belong to the external reference or block insertion rather than the current drawing. The working set includes only the objects belonging to the reference selected for editing.

You can add or remove objects from the *working set*. If you create a new object while editing a reference, it is almost always added to the working set automatically. Editing changes in the *working set* can be stored in the source file of the external reference or block insertion.

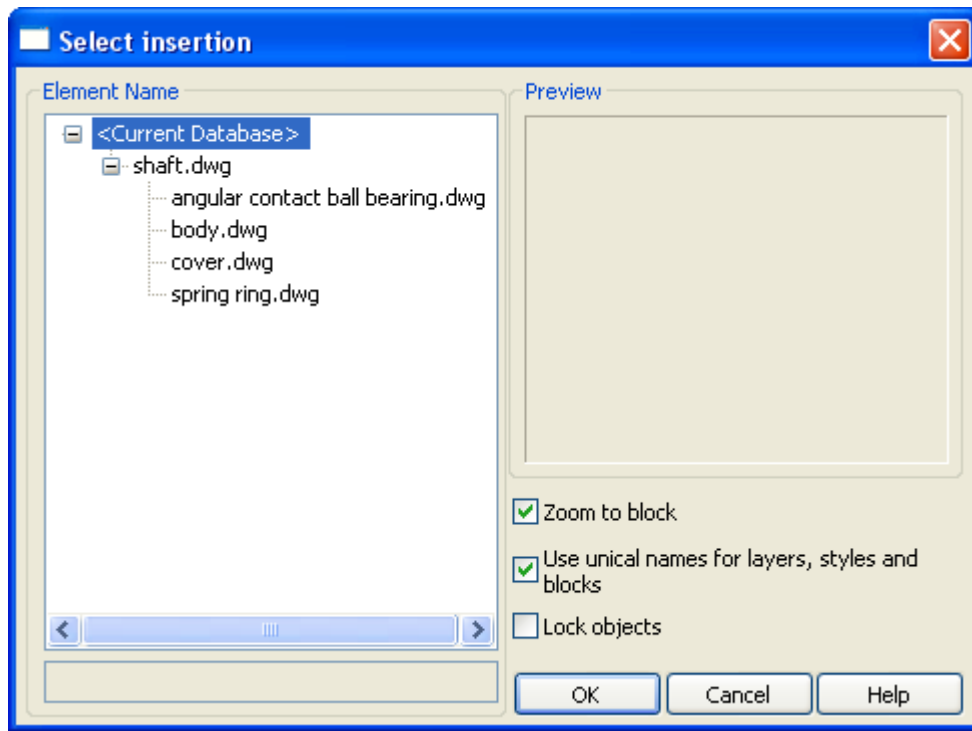
Start the **Edit reference** command. In the command line you can see:

Select reference or block insertion or [All insertions]:

Select the external reference on the drawing and in the **Select insertion** dialogue box that opens, choose the objects to edit:



If you select the All insertions option in the command line, the Select insertion dialogue box opens immediately and in the Element Name section all references and blocks inserted into the drawing will be displayed:



Options:

Zoom to block

Switches on/off the mode to display the selected reference on the screen.

Use unical names for layers, styles and block

Controls whether layers and other named objects extracted from the reference are uniquely altered. If selected, named objects in external references are altered (names are prefixed with \$#\$), similar to the way they are altered when you bind external references. If cleared, the names of layers and other named objects remain the same as in the reference drawing. Named objects that are not altered to make them unique assume the properties of those in the current host drawing that share the same name.

Lock objects

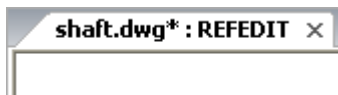
Locks all objects not in the working set. This prevents you from accidentally selecting and editing objects in the host drawing while in a reference editing state. The behaviour of locked objects is similar to objects on a locked layer.

Select the objects for editing and click **OK**. The **refedit** toolbar will appear.



Use the **refedit** toolbar to add or remove objects from a working set and also save and discard external reference editing.

REFEDIT, separated by a colon, is added to the document name in the tab. It means that work with the document occurs in the external reference editing mode.



Add objects to the working set



Menu: **Tools – External reference >**  **Add objects to working set**



Toolbar: **refedit –** 



Command line: **REFSET**

Use this command to transfer the objects from the current drawing to the working set.

Remove objects from the working set



Menu: **Tools – External reference >**  **Remove objects from working set**



Toolbar: **refedit –** 



Command line: **REFSET**

Use this command to remove selected objects from the working set.

Save external reference changes



Menu: **Tools – External reference >**  **Save and close**



Toolbar: **refedit –** 



Command line: **REFCLOSES**

Use this command to save external reference changes and close the **refedit** toolbar. The external reference editing mode will be completed. The **REFEDIT** term, separated by a colon, that was added to the document name in the tab will disappear.

Discard external reference changes



Menu: **Tools – External reference >**  **Discard changes**



Toolbar: **refedit –** 



Command line: **REFCLOSED**

Use this command to discard external reference changes and close the **refedit** toolbar. The external reference editing mode will be completed. The **REFEDIT** term, separated by a colon, that was added to the document name in the tab will disappear.

Insert image



Menu: **Insert –**  **Image from file...**

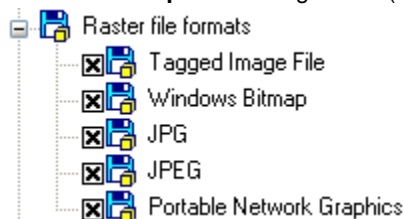


Toolbar: **Draw –** 



Command line: **ROPEN**

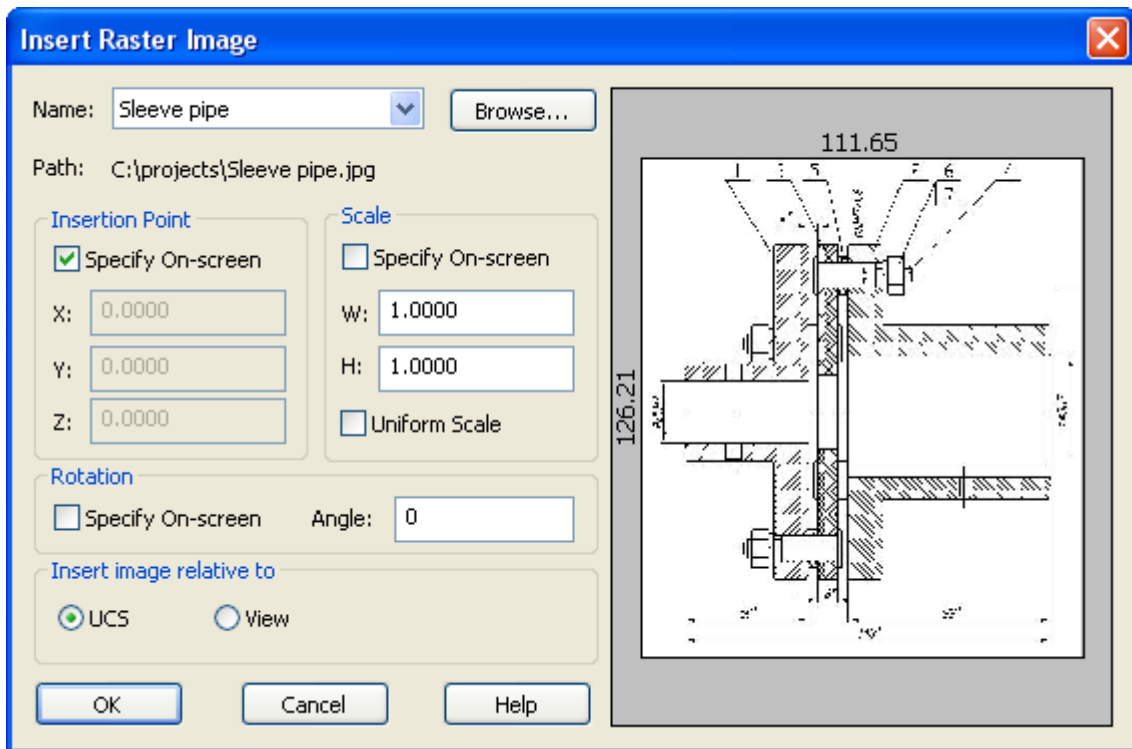
You can insert images into the drawing. The list of supported raster formats is displayed in the **Raster file formats** section of the **Options** dialogue box (the **Tools** menu – **Options**):



Raster images can be referenced and placed in drawing files but, like external references, they are not actually part of the drawing file. The image is linked to the drawing file through a path name. Linked image paths can be changed or removed at any time.

Once you've attached an image, you can reattach it multiple times, treating it as if it were a block. Each insertion has its own clip boundary and its own settings for brightness, contrast, fade, and transparency.

To insert a raster image, specify the necessary options in the opened **Insert Raster Image** dialogue box.



Options:

Name: Includes the list of names of the inserted images.

Browse... Opens the **Insert Raster Image** dialogue box.

Path: Displays the path where the raster image file must be located.

Insert Point

Specify On-screen Select the box to set the X,Y,Z coordinates values in the command line or specify the position on the screen. The «X» «Y» «Z» fields of this section are inaccessible.

X: Y: Z: Enter the X, Y, Z coordinate values for the raster image insertion in the corresponding fields.

Scale

Specify On-screen Select the box to set the scale value of the raster image in the command line or specify the position on the screen. The «X» «Y» «Z» fields of this section are inaccessible.

W: Set the scale factor width.

H: Set the scale factor height.

Uniform Scale Specifies the scale factor for the *Width* or *Height* values. A value specified for *Width* is also reflected in the *Height* value.

Rotation

Specify On-screen Specifies the rotation angle for the inserted image, using the pointing device.

Angle: Sets the rotation angle for the inserted image.

Insert image relative to

UCS Sets the insert image mode relative to the User Coordinate System (UCS).

View Sets the insert image mode relative to the World Coordinate System (WCS).

Image Settings



Menu: **Modify – Object > Image...**



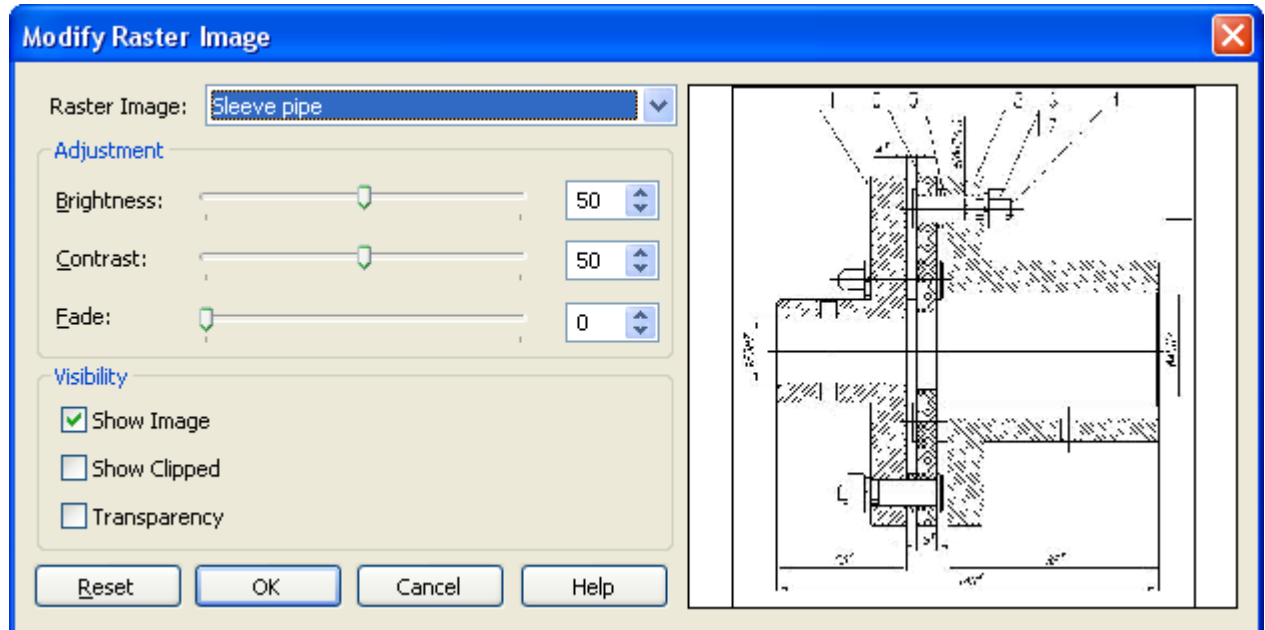
Toolbar: **Modify Object – Image**



Command line: **MODIFYRASTER**

You can adjust the brightness, contrast and fade for the display of the raster image without affecting the original raster image file. The image adjustment is intended for improvement of the display of raster images (adjust contrast to make poor-quality images easier to read) or special effects. Bitonal images cannot be adjusted for brightness, contrast or fade.

To adjust raster image options, use the **Modify Raster Image** dialogue box:



Options:

Raster Image: List of the raster images inserted in the drawing.

Adjustment

Brightness: Controls the brightness of the image.

Contrast: Controls the contrast of the image.

Fade: Controls the fading effect of the image.

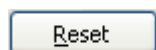
Visibility

Show Image Sets the display of the image on the screen. If the box is not selected, only the frame of the raster image is displayed.

Show Clipped Displays the image area in the show border that has been set.

Transparency Sets the transparency mode of the raster images. When the transparency mode is set to on, it is possible to show the objects of the raster image through transparent pixels. For example, in bitonal images, background pixels are treated as transparent.

You can set the transparency of the raster image in the **Inspector** dialogue box (*General* section).



Resets the values for brightness, contrast and fade to the default settings.

Methods for setting the slider:

- Place the cursor over the slider. When the slider is highlighted, press the left mouse button and keeping it held down, move the cursor left to decrease or right to increase the setting value. Choose the required position for the slider and release the mouse button; the value will be displayed in the box to the right of the scale.
- Place the cursor at the point on scale to which the slider should be moved. Left click to set the slider to the selected point.

- Enter the required value settings in the box to the right of the scale.
- In the box to the right of the scale, click the up arrow to increase or down arrow to decrease the value settings by one.

It is possible to set a new show boundary for the inserted raster image (the **New** command from the **Modify** menu – **Clip**).

The show boundary of the raster image can be transparent.

To change the transparency of the raster image boundary:

- Select the raster image.
- Set the transparency level in the *Transparency* field of the *General* section of the **Inspector** window.

External references control

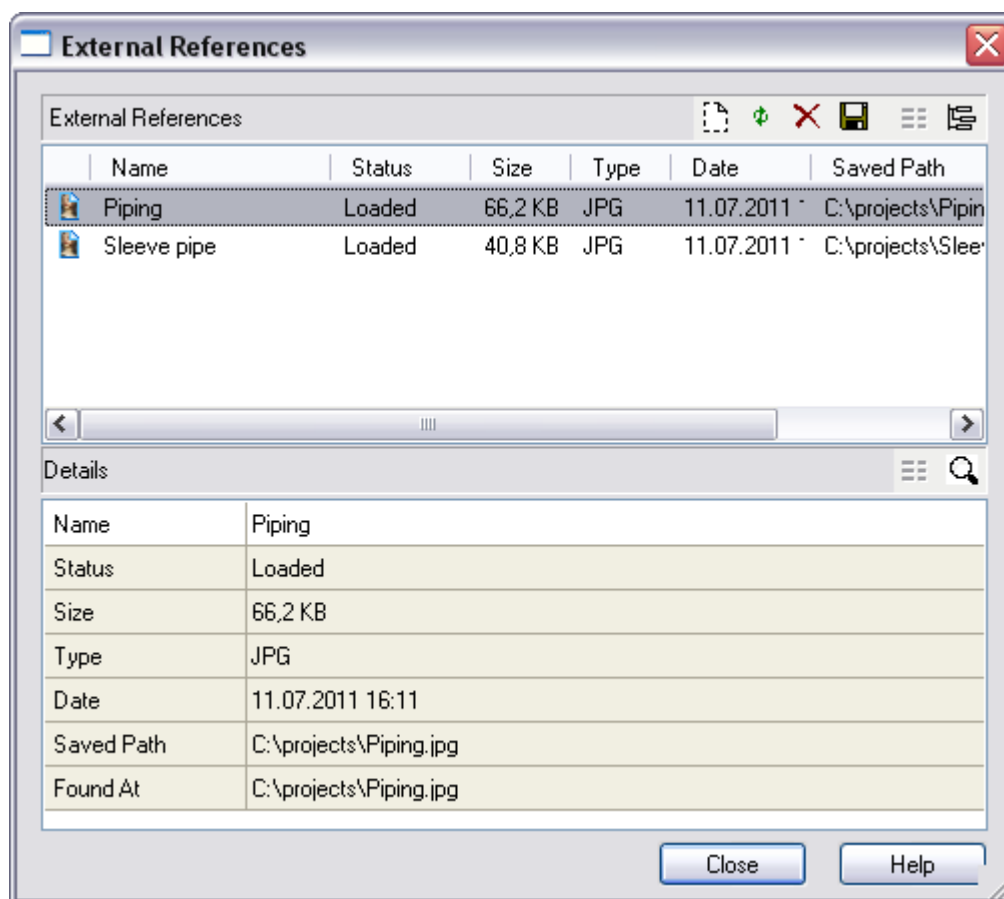


Menu: **Insert – External References...**



Command line: **EXTERNALREFERENCES, IMAGES**



The command opens the **External References** dialogue box. It displays all external references contained in the document and manages these references:





The dialogue box displays information about the references to the inserted drawings as well as for the raster images.

The dialogue box contains the tool buttons and two sections: **External References** and **Details**. Both panes are working in the double data display mode .

The **External References** pane can display information about references in a list structure (the **Show List**

 button) or in a tree structure (the **Show tree**  button). The list view is set by default in the window.







The **Details** pane displays the properties of the references selected in the upper section in the list view (the **Show properties**  button) or displays the content of the selected reference in a preview window (the **Show preview**  button).

The options of the **External References** pane in the list view:

Columns

| | |
|-------------------|---|
| Name | Displays the name of the external reference file. |
| Status | Displays the status of the external reference file: Loaded – the referenced file is attached to the currently drawing. Unloaded – the referenced file is marked to be unloaded from the drawing. Not Found – the referenced file no longer exists in the valid search path. Unresolved – the referenced file cannot be read. Orphaned – the referenced file is attached to another file that has an Unresolved status. |
| Size | Displays the size of the attached referenced file. |
| Type | *.dwg files display the file type of the referenced file: Attachments Overlays Raster images display their file format: TIFF BMP JPG JPEG PNG |
| Date | The date when the referenced file was created or last saved. |
| Saved Path | Displays the path that is saved with the drawing when the referenced file is attached. |

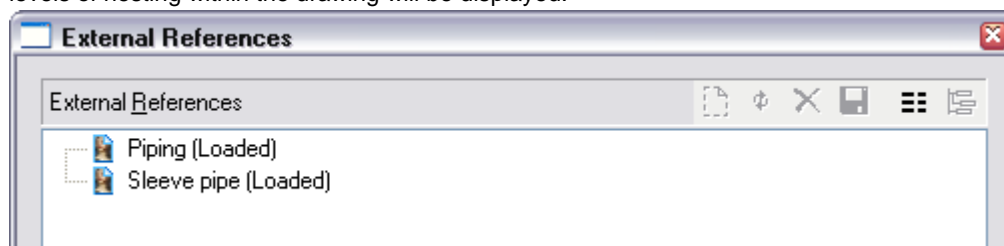
Buttons

| | | |
|---|--------------------|--|
|  | Unload XRef | Unloads the selected file references from the drawing. |
|  | Reload XRef | Reloads the selected file references to the drawing. |
|  | Detach XRef | Detaches the selected file references. |
|  | Save XRef | Changes the selected raster image references. It is possible to change the reference file name, file type and file format. |
|  | Show list | Displays the references information in the list view. |
|  | Show tree | Displays the references information in the tree view. |

When the **External References** pane is set to the list view, it is possible to select several references:

- when the **SHIFT** key is pressed, then all references located between the first and last click will be selected;
- when the **CTRL** key is pressed, you can select any references from the list by clicking.

When the **External References** pane is set to the tree view, a tree structure of all external references along with their levels of nesting within the drawing will be displayed.



It is possible to select only one file reference in the tree view of the **External References** pane.

The options of the **Details** pane in the list view:

Columns

| | |
|-------------|---|
| Name | Displays the name of the external reference file. |
|-------------|---|

| | |
|-------------------|---|
| Status | <p>Displays the status of the external reference file:</p> <p>Loaded – the referenced file is attached to the current drawing.</p> <p>Unloaded – the referenced file is marked to be unloaded from the drawing.</p> <p>Not Found– the referenced file no longer exists in the valid search path.</p> |
| Size | <p>Displays the size of the attached referenced file.</p> <p>The size of the attached referenced file doesn't display when the reference has the status:</p> <p>Unloaded</p> <p>Not Found</p> |
| Type | <p>*.dwg files display the file type of the referenced file:</p> <p>Attachment</p> <p>Overlay</p> <p>Raster images display their file format:</p> <p>TIFF</p> <p>BMP</p> <p>JPG</p> <p>JPEG</p> <p>PNG</p> |
| Date | <p>The date when the referenced file was created or last saved.</p> <p>The date doesn't display when referenced file has the status:</p> <p>Unloaded</p> <p>Not Found</p> |
| Saved Path | <p>Displays the path of the referenced file.</p> <p>The saved path does not necessarily indicate the present file location.</p> |
| Found at | <p>Displays the path where the external reference file is located.</p> <p>This place is the actual file location.</p> <p>The full path and saved path may be different.</p> |

Buttons



Show properties

Displays the properties of the selected referenced file in the list view.



Show preview

Displays contents of the selected referenced file in the preview window.

The **External references** dialogue box displays not only information about inserted references, but also allows you to perform operations with drawing files which are referenced.

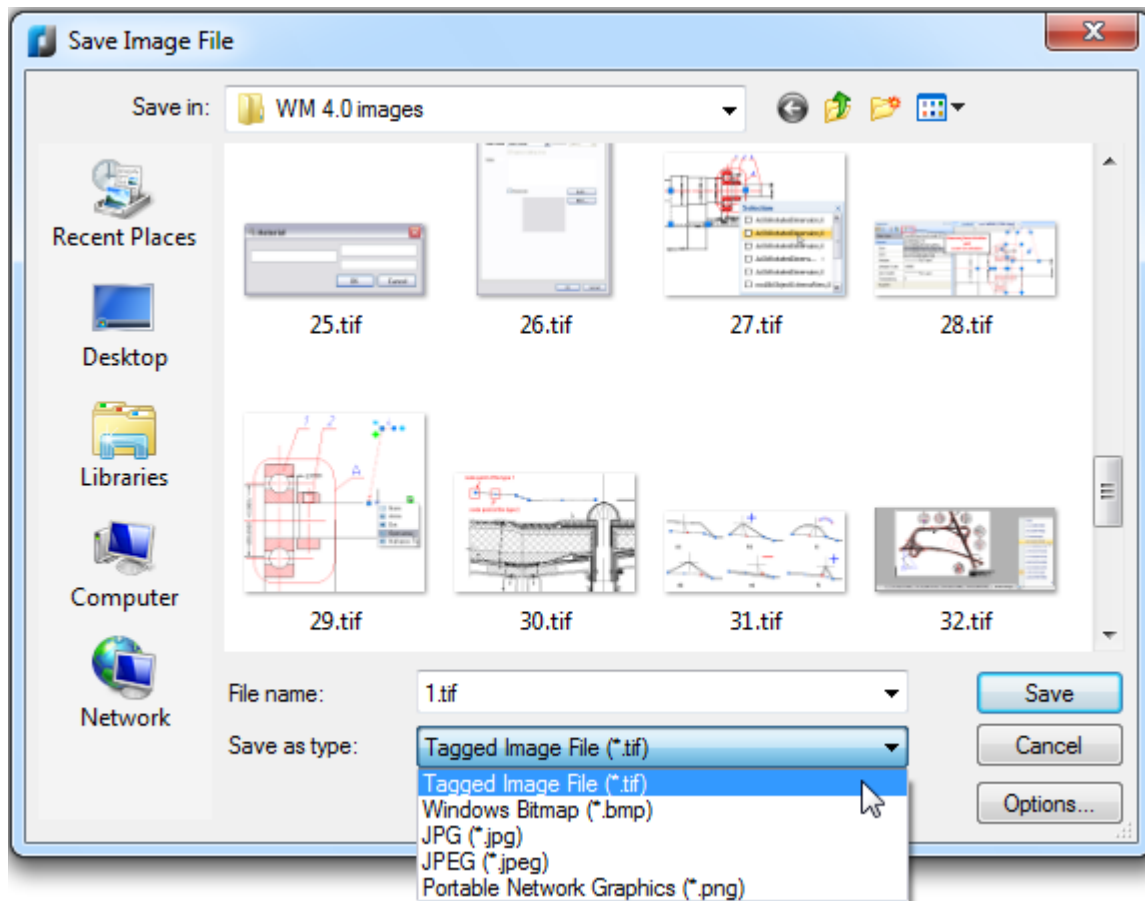
These operations are:

- Load/unload of the external reference in the current drawing.
- Reload of the external reference from the current drawing (without reloading the current drawing).
- Delete the entry of the external reference with all data from the current drawing. If you only delete the reference from the drawing it will not delete the layers of the external reference. To permanently delete an external reference, use the **Detach** command in the **External references** dialogue box.
- Changes the selected raster image references. It is possible to change the referenced file name, file type and file format.

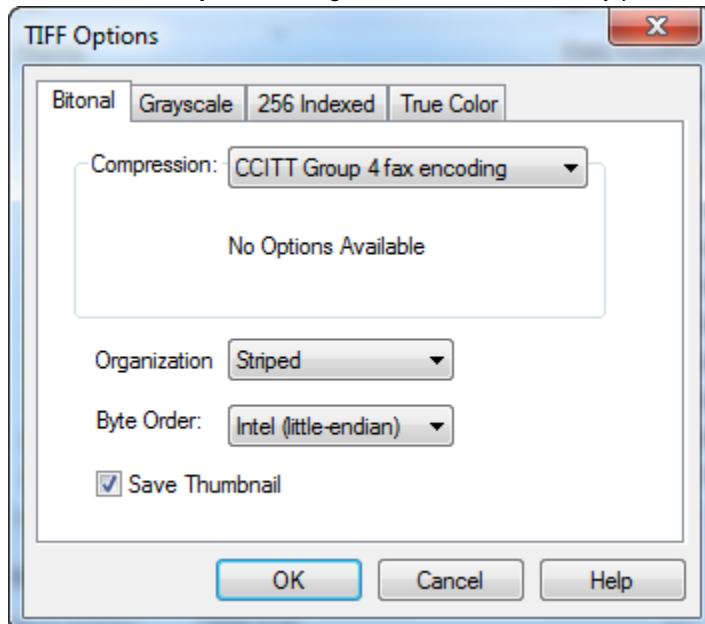
Change of the external reference to the raster image

To change reference to the image:

- In the **External References** dialogue box, select the reference to the image in the list
- Click the **Save** button.
- In the opened **Save Image File** dialogue box, specify the file name and file type:



- Click the **Options** button.
- In the **TIFF Options** dialogue box, set the necessary parameters:



- Click **OK**.
- Click **Save**.

Image settings



Menu: **Insert – External References...**



Command line: **EXTERNALREFERENCES, IMAGES**

nanoCAD allows you to convert the inserted images into other supported raster file formats. The list of supported raster file formats is shown in the **Options** dialogue box (the **Tools** menu – the **Options** command).

For the **TIFF** format, you can also change settings such as colour mode, compression, organisation, byte order; for

JPG and JPEG formats – Image Quality.

To save an image to another format or to change the format parameters:

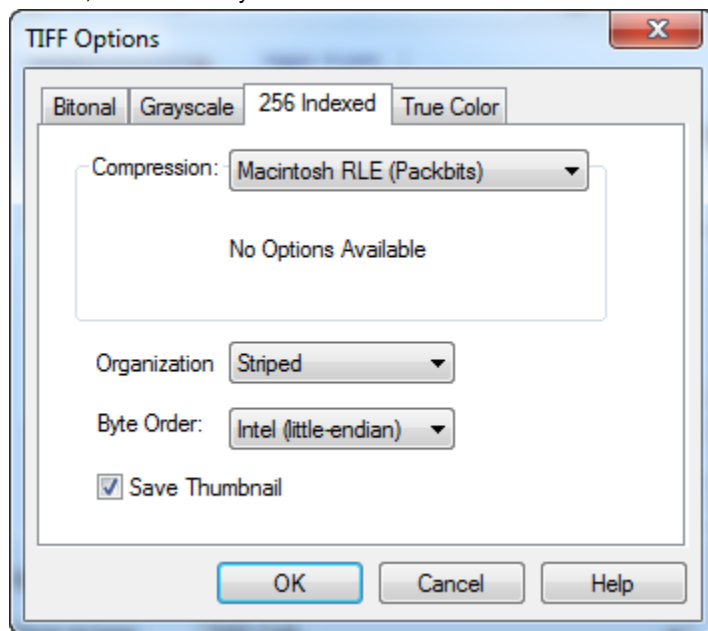
- In the **External References** dialogue box select raster references and click **Save**.
- In the opened **Save Image File** dialogue box, type the new file name, select the file type from the drop-down list and click the **Options** button.
- Depending on the selected file type, the **TIFF Options** dialogue box or the **JPEG Options** dialogue box will be open.

TIFF Options

The settings of this dialogue box also apply to other types of **TIFF**-formats, including multiple-page **TIFF** and **TIFF** with geodata.

If part of a multiple TIFF-file includes images of different types, such as colour and bitonal, then you can adjust separately for each of these types.

There is a separate tab for each of the colour modes. For example, to adjust the saving options for a 256-color image in **TIFF**, it is necessary to use the **256 Indexed** tab.



Options:

Compression: Drop-down list to select the raster degree of compression.

Depending on the selected colour mode, the following parameters in the list are available:

| | |
|-----------------------------------|---|
| No compression | Raster is saved without compression. In this case, the file will be of a large size. At the same time the file can be read by any program that supports working with TIFF |
| CCITT modified Huffman RLE | The compression type, which combines Huffman and RLE algorithms. It is used to compress the bitonal rasters. |
| CCITT Group 3 fax encoding | The compression type, which uses the Huffman algorithm with a fixed table for compressing bitonal rasters. The following options are available for this compression type: 2-D Encoding – The most effective data compression. Fill To Byte Boundary - Controls the method of defining the line start. When this option is on, a new line always starts with the bits number of a multiple byte. |
| CCITT Group 4 fax encoding | The compression type that is optimal for the bitonal rasters. It is supported by most raster editors and ensures the best compression of bitonal data. |
| Macintosh RLE (Packbits) | The compression type that is optimal for colour images. It is supported by most raster editors. At the Striped or Tiled internal organisation, the use of this compression type in some cases can give an increase of file size compared to the saved |

file without compression.

ZIP (Deflate compression)

The compression type that uses an algorithm similar to that used in the ZIP archiver.

Can be used with any raster types.

Versions of the AutoCAD 2005 and earlier don't support reading TIFF-files with ZIP-compression.

Versions of the Spotlight 6.0, RasterID 3.0 and earlier also cannot read files of this compression type. In some cases, attempting to open files of this compression type can lead to fatal errors.

Use the slider to control the degree of compression:



Set the maximum degree of compression to reduce the file size. But this leads to an increase in the time taken for reading/saving the raster image.

Lempel-Ziv & Welch

The compression type that uses a universal compression algorithm without data loss.

This algorithm has a high work speed when compressing and when decompressing. The inconvenience is the low degree of compression in comparison with a scheme of two stage encoding.

This compression type is used also in GIF and PDF formats.

JPEG DCT compression

The compression type that uses a JPEG algorithm.

The same algorithm is used in the similarly-named JPEG format.

It is designed to compress colour rasters.

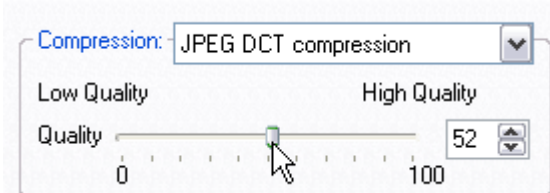
It allows the highest degree of compression to be achieved.

Also, as described above for the ZIP-compression, JPEG-compression is not supported by all raster editors.

JPEG-compression, unlike any other compressions which are used in TIFF-format, is performed with some data loss. This compression type is very useful to create a reduced size file, for example, for Internet transmission. But it is not recommended to use it to store important information.

Indeed, whenever you save a TIFF-file to JPEG-compression, recompression is performed, in which the quality of raster deteriorates.

Use the slider to control the degree of compression:



Increasing the degree of compression reduces the file size, but this leads to a reduction of its quality. At the maximum degree of compression you can get raster that may be deformed beyond recognition.

Organization: A drop-down list to select the type of internal organization of the TIFF-file.

The following options are available:

Row – All data within the file are written in one block. This organisation type is also called *blocked*. TIFF-files of this organisation have the highest compatibility with other programs designed for viewing and editing of rasters.

Striped – All data within the file are written in the form of individual portions of lines. In this way, in some cases, the downloading and viewing of images are accelerated. But in some cases, a combination of the row and striped organisation and one of the compression types, for example, Macintosh RLE (Packbits), leads to an increase of the file size instead of the expected reduction.

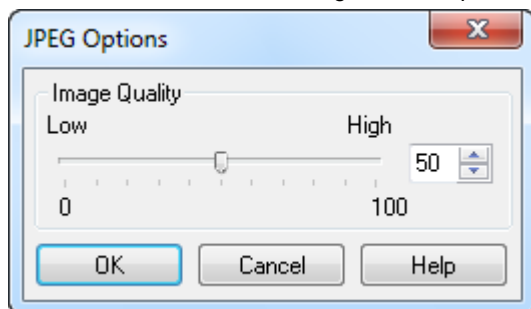
Tiled – All data within the file are written in the form of fragment, so-called *tiles*. This organisation type is not supported by all raster editors. The use of the tile organisation can speed up viewing of rasters.

Byte Order: A drop-down list to select the byte order in the word, which is different for PC and Macintosh. Two options are available:
Intel (little-endian) - Byte order in which the junior (least significant) byte is written first.
Motorola (big-endian) - Byte order in which the senior (most significant) byte is written first.

Save Thumbnail Switching on/off the mode to save a file with a reduced copy of the image (thumbnail). Thumbnail is used to quickly view the file content in the file open dialogue box. It should be remembered that when you switch on the Save Thumbnail mode, the TIFF-file will be saved as multipage and it cannot be read by all raster editors.

JPG and JPEG Options

You can change the degree of compression for JPG and JPEG file formats. Use the slider to control the degree of compression:



Increasing the degree of compression reduces the file size, but this leads to a reduction in its quality. At the maximum degree of compression you can get a raster that may be deformed beyond recognition.

Show boundary

Shows a set of raster images, blocks or external references in the enclosed area. (see «[Set show boundary for a viewport](#)» section («*THE COMPONENT LAYOUT AND PLOT DRAWING*» - «*Viewports*» - «*Edit Layout Viewports*»)).

The show boundary defines the visibility of objects in the current document. The original objects are not changed. Setting of the clip boundary is achieved using the clipping contour.

Clipped blocks or external references, raster images and viewports can be edited as well as not clipped ones.

Setting of the show boundary for the block or external reference



Menu: **Modify – Clip > XRef**



Command line: **XCLIP**

The command sets the clipping contour of the show boundary to display the section of the inserted block or external reference.

The command doesn't change the objects of the block or external reference (the definition of the block or external reference remains unchanged). Creation of the clipping contour affects only the display block or an external reference in the current document.

If a block or external reference has been inserted more than once, it is possible to specify a different clipping contour for each entry, but each entry can have only one contour.

Clipping of the block or external reference is carried out by a polygonal contour: rectangle, polygon or closed polyline.

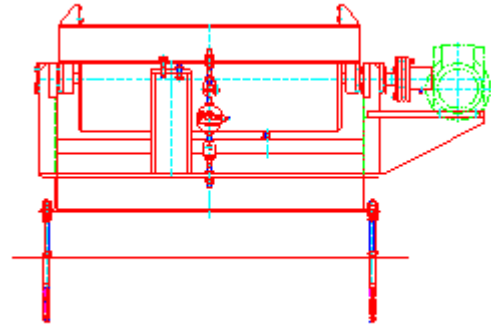
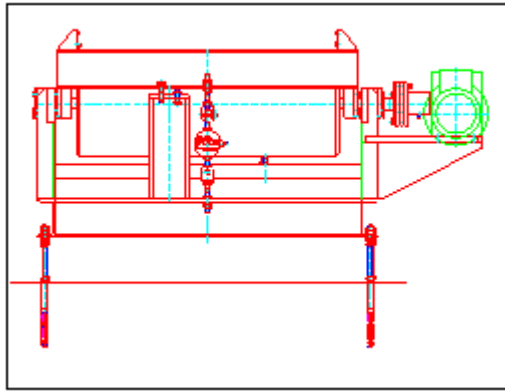
The [XCLIPFRAME](#) system variable is designed to set the visibility of the boundary of the clipping contour. If the system variable is set to a value of 1, the boundary of the clipping contour will be displayed on the screen. You can select it and print it out. If the system variable is set to a value of 0, the visibility of the boundary will be turned off (set by default).

Display of the boundary of the clipping contour is turned on

(value of the system variable [XCLIPFRAME](#) =1)

Display of the boundary of the clipping contour is turned off

(value of the system variable [XCLIPFRAME](#) =0)



You can turn off the clipping of the block or external reference to display the full entry and then turn it on again to display only the clipped area.

A clipped section of the block or external reference can be copied, moved and rotated in the same way as an entry of the block or external reference that is not clipped. The clipping contour is copied, moved or rotated with the entry. The clipping options also extend to attached references: when the main reference is clipped, all attached references will be clipped too.

The clipping contour can be redefined. Set the new clipping contour to remove the old contour.

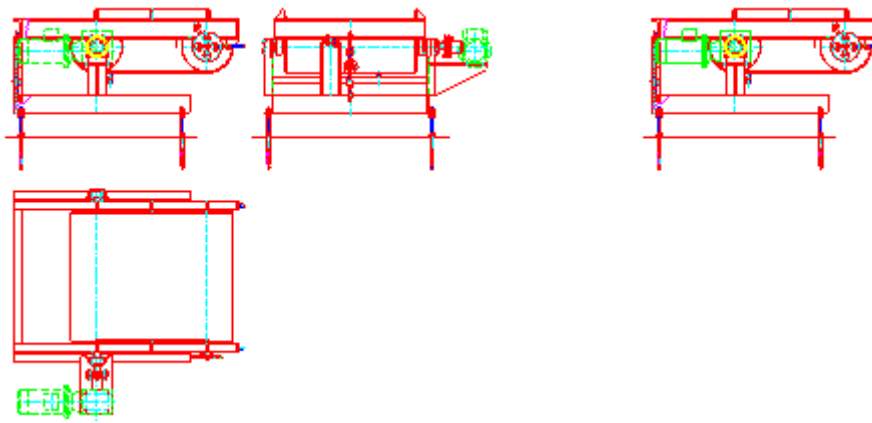
After removing the clipping contour, the block or external reference is displayed on the screen in full.

Options:

| | |
|---------------------------------|--|
| <u>On</u> | Mode displays a clipped section of the block or external reference. |
| <u>Off</u> | Mode displays the full entry of the block or external reference. |
| <u>Clip depth</u> | Set near and far clipping planes for the external reference or block. Objects beyond the limit of the contour set by planes of the space will not be displayed on the screen. |
| <u>Distance</u> | Creating a clipping plane, passing at a prescribed distance parallel to the clipping contour. |
| <u>Remove</u> | Delete near and far clipping planes. |
| <u>Delete</u> | Delete the clipping contour for the selected entry of the block or external reference. This option deletes the clipping contour and clipping plane. |
| <u>Generate Polyline</u> | To automatically create a polyline whose vertices match the vertices of the clipping contour. This option is used to change the current clipping contour: the created polyline is edited by the PEDIT command (the Modify menu – Object > Polyline). Then the edited polyline is used to change the existing clipping contour so that it is based on the new contour. |
| <u>New</u> | Create a new clipping contour. |
| <u>Select polyline</u> | Set the limits contour by the selected closed polyline. The polyline should be created previously and consist of straight-line segments. |
| <u>Polygonal</u> | Set the polygonal contour of the show boundary by sequenced specifying of the polygonal vertices. |
| <u>Undo</u> | Sequenced cancelling of the specified points of the polygonal vertices. Specified first point cannot be cancelled. |
| <u>Rectangular</u> | Set the rectangular contour of the show boundary by sequenced specifying of the opposite rectangular vertices. |

External reference to setting of the show boundary

External reference after the setting of the show boundary



The command prompts when you specify a rectangular boundary:

Select block or X-references:

Select the block or reference and press **ENTER**.

Enter clipping options

[ON/OFF/Clip_depth/Delete/generate_Polyline/New]

<New>: N

[Select polyline / Polygonal / Rectrangular]

<Rectangular>:

Choose Rectangular.

Specify first corner:

Specify the first corner.

Specify opposite corner:

Specify the opposite corner.

The command prompts when you setting the polygonal boundary:

Select block or X-references:

Select the block or reference and press **ENTER**.

Enter clipping options

[ON/OFF/Clip_depth/Delete/generate_Polyline/New]

<New>: N

[Select polyline / Polygonal / Rectrangular]

<Rectangular>:

Choose Polygonal.

Specify first point:

Specify the first point.

Specify next point or [Undo]:

Specify the second point.

...

...

Specify next point or [Undo]:

Specify the end point and press **ENTER**.

The command prompts when you specify a boundary by polyline:

Select block or X-references:

Select the block or reference and press **ENTER**.

Enter clipping options

[ON/OFF/Clip_depth/Delete/generate_Polyline/New]

<New>: N

[Select polyline / Polygonal / Rectrangular]

<Rectangular>:

Choose Select polyline.

Select_polyline:

Select polyline.

To turn on/off clipping of the block or external reference:

- In the **Modify** menu click **Clip** and then **On** or **Off** commands.
- In reply to the prompt in the command line **Select block or X-reference:** Select the entry and press **ENTER**.

To change the clipping contour of the block or external reference:

- In the **Modify** menu click **Clip** and then the **New** command.
- In reply to the prompt in the command line **Select block or X-reference:** Select the entry and press **ENTER**.
- In reply to the prompt in the command line **Delete old boundary(s)? [Yes/No] <Yes>:** Select **Yes** or press **ENTER**.
- In reply to the prompt in the command line [Select polyline / Polygonal / Rectangular] <Rectangular>: Select the required option and set the new clipping contour.

Note: It is possible to create a new clipping contour if the old contour is deleted.

To delete the clipping contour:

- In the **Modify** menu click **Clip** and then **Delete** command.
- In reply to the prompt in the command line **Select block or X-reference:** Select entry and press **ENTER**.

Setting of the show boundary for a raster image



Menu: **Modify – Clip > New**



Command line: **NEWCLIP**

The command allows you to insert a clip of the raster image into the drawing, to set the display on the screen and print only the required part of the raster image.

Setting of the show boundary of the raster image affects its display in the current document, but does not change the source image.

Clipping of the raster image is done using a polygonal outline (rectangular, polygonal or closed polyline), whose vertices are inside the image boundary.

It is possible to specify a different clipping contour for each entry of the same raster image, but each entry can have only one contour.

The clipping contour can be turned off to display the original image and then back on to show the clipping image.

The clipping contours can be redefined. When you specify a new clipping contour, the old contour should be deleted.

After deleting the clipping contour, the raster image is displayed on the screen in its original boundaries.

Options:

Select polyline

Set the limits contour using the selected closed polyline.

The polyline should be created previously and consist of straight-line segments.

Polygonal

Set the polygonal contour of the show boundary by sequenced specifying of the polygonal vertices.

Undo

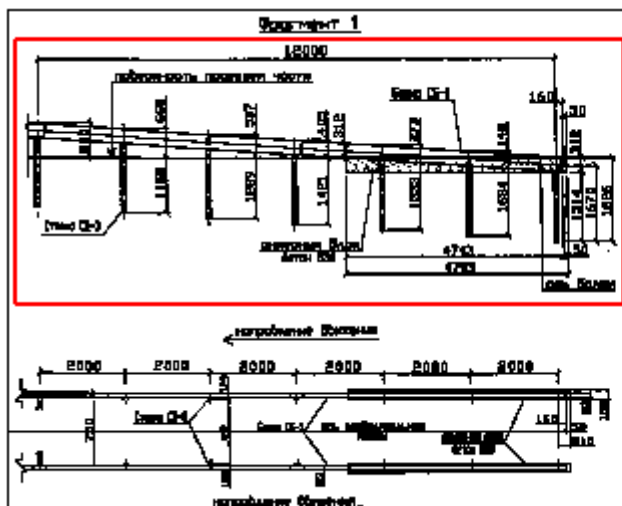
Sequenced cancelling of the specified points of the polygonal vertices.

Specified first point cannot be cancelled.

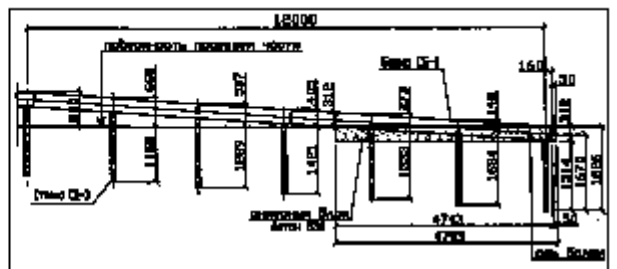
Rectangular

Set the rectangular contour of the show boundary by sequenced specifying of the opposite rectangular vertices.

Setting the rectangular contour of the show boundary of the raster image



Displaying of the raster image after setting the show boundary



The command prompts when setting the rectangular boundary:

Select block or X-references:

[Select polyline / Polygonal / Rectangular]
<Rectangular>:

Specify first corner:

Specify opposite corner:

Select raster and press **ENTER**.

Press **ENTER**.

Specify the first corner.

Specify the opposite corner.

The command prompts when setting the polygonal boundary:

Select block or X-references:

[Select polyline / Polygonal / Rectangular]
<Rectangular>:

Specify first point:

Specify next point or [Undo]:

...

Specify next point or [Undo]:

Select raster and press **ENTER**.

Choose Polygonal.

Specify the first point.

Specify the second point.

...

Specify the end point and press **ENTER**.

The command prompts when specifying a boundary by a polyline:

Select block or X-references:

[Select polyline / Polygonal / Rectangular]
<Rectangular>:

Select_polyline:

Select the raster and press **ENTER**.

Choose Select polyline.

Select the polyline.

To turn on/off the clipping contour:

- In the **Modify** menu, click **Clip** and then the **On** or **Off** commands.
- In reply to the prompt in the command line *Select block or X-reference:* Select the raster image and press **ENTER**.

To change the clipping contour:

- In the **Modify** menu, click **Clip** and then the **New** command.
- In reply to the prompt in the command line *Select block or X-reference:* Select the raster image and press **ENTER**.
- In reply to the prompt in the command line *Delete old boundary(s)? [Yes/No] <Yes>:* Select **Yes** or press **ENTER**.
- In reply to the prompt in the command line [Select polyline / Polygonal / Rectangular]
<Rectangular>: Select the required option and set the new clipping contour.

Note: It is possible to create a new clipping contour if the old contour is deleted.


To delete the clipping contour:

- In the **Modify** menu, click **Clip** and then the **Delete** command.
- In reply to the prompt in the command line *Select block or X-reference:* Select the raster image and press **ENTER**.

Execution of Drawings commands

The Filling and Hatch commands



Menu: **Draw** –  **Hatch**



Toolbar: **Draw** – 



Command line: **BHATCH, H, HATCH**

Hatching in nanoCAD means filling of the selected area with the predefined *pattern*.

The *Hatch pattern* is a predefined pattern used to represent different materials such as steel, concrete, glass and so on. A solid fill can also be used as a hatch pattern.

The hatch can be *associative* or *annotative*.

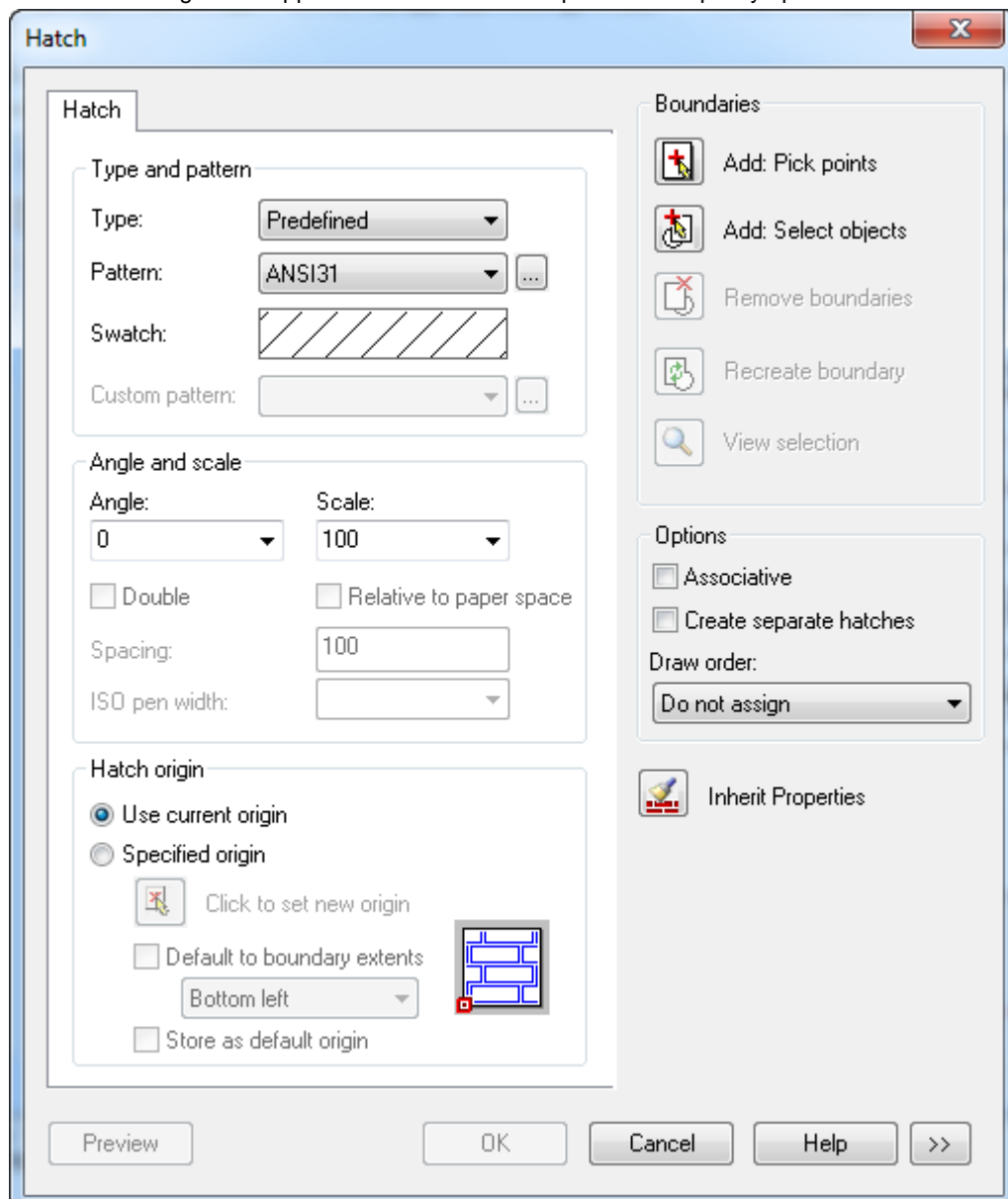
An *associative* hatch is updated when you change the boundary.

An *annotative* hatch does not depend upon the boundary.

You can use several methods to specify the boundaries of a hatch:

- specify a point in an area that is enclosed by objects,
- or select objects that enclose an area.

The **Hatch** dialogue box appears to select the hatch pattern and specify options:



Options:

Type and pattern

- Type:** Drop-down list to select the type of hatch pattern:
The following types are available:
Predefined
User defined
Custom
- Pattern:** Drop-down list to select the available predefined patterns.
The **Pattern** option is available only if you set the Type to **Predefined**.



This button opens the **Hatch pattern** dialogue box.

- Swatch:** Displays a preview of the selected pattern.
Click the swatch to display the **Hatch pattern** dialogue box.

- Custom pattern:** Drop-down list to display the available custom patterns.



This button opens the **Hatch pattern** dialogue box.

Angle and scale

- Angle:** Drop-down list to specify an angle for the hatch pattern relative to the **X** axis of the current UCS.
Values can be input from the keyboard.
- Scale:** Drop-down list to expand or contract a predefined or custom pattern. Values can be input from the keyboard.
This option is only available if you set the Type to **Predefined** or **Custom**.
- Double** Turns on/off the mode to draw a second hatch positioned at 90 degrees to the original hatch.
This option is only available if you set the Type to **User Defined**.

Hatch origin Setting the start point to create the hatch.
Some hatches, such as brick patterns, need to be aligned with a point on the hatch boundary.
By default, all hatch origins correspond to the current UCS origin.

- Use current origin** Turns on the mode of origin setting stored in the HPORIGINMODE system variable.
The origin is set to 0,0 by default.

- Specified origin** Specifies a new hatch origin.



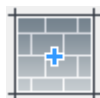
Click to set new origin Specifies the new hatch origin point on the screen using the cursor.

- Default to boundary extents** Turns on/off the mode for calculation of a new origin based on the rectangular extents of the boundary for the hatch.

From the drop-down list the following options are available:

Bottom left
Bottom right
Top right
Top left
Center

The icon displays the current position of the origin point:



- Store as default origin** Turns on/off the mode for saving the value of the new hatch origin in the HPORIGIN system variable.

Boundaries



Add: points **Pick**

Determines a boundary from the existing objects that form an enclosed area around the specified point. The dialogue box closes temporarily and you are prompted to

pick a point.



Add: Select objects

Determines a boundary from selected objects that form an enclosed area. The dialogue box closes temporarily and you are prompted to select objects.



Remove boundaries

Removes from the boundary definition any of the objects that were added previously. This option is unavailable if you have not specified points or not selected objects.



Recreate boundary

Creates a polyline or region around the selected hatch and optionally associates the hatch object with it.

This option is available when you edit the hatch.



View selection

Temporarily closes the Hatch dialogue box and displays the currently defined boundaries with the current hatch settings.

This option is unavailable if you have not specified points or not selected objects.

Options

Associative

Turns the associative hatch mode on/off .

Create separate hatches

Turns on/off the mode for changing a single hatch object that has several separate boundaries into individual hatch objects.

Draw order:

Drop-down list to assign the draw order to a hatch or fill.

From the drop-down list the following options are available:

Do not assign

Send to back

Bring to front

Send behind boundary

Bring in front of boundary



Inherit Properties

Temporarily closes the Hatch dialogue box to specify boundaries using the hatch properties of a selected hatch object.



Preview

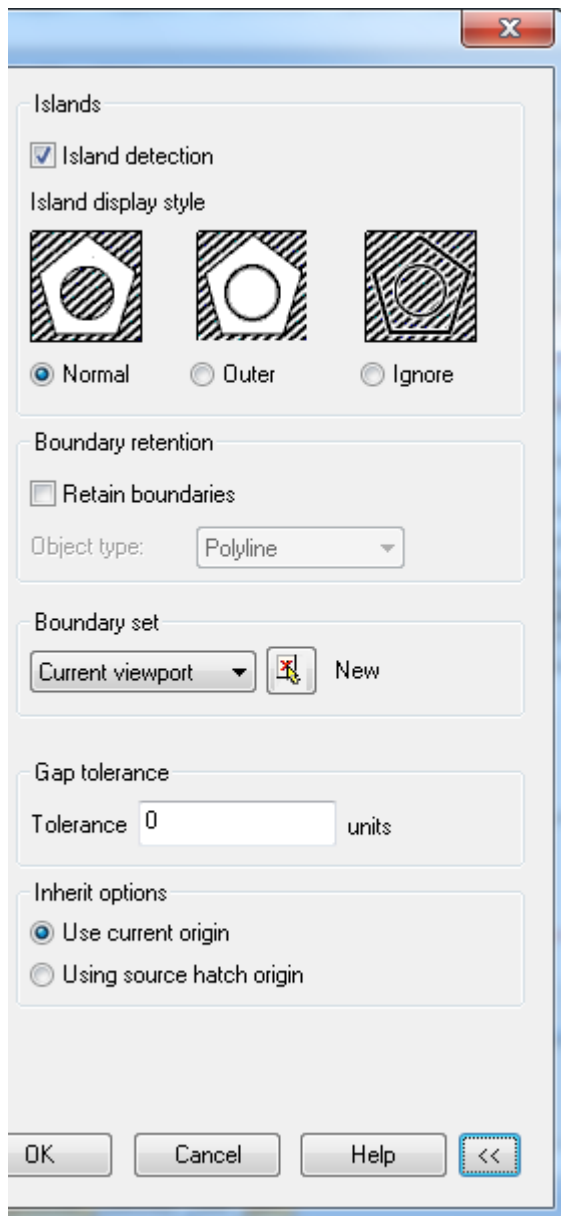
Temporarily closes the Hatch dialogue box and displays the currently defined boundaries with the current hatch settings.

Press **ESC** to return to the dialogue box.



Expands the **Hatch** dialogue box to display more options.

More Options section of the **Hatch** dialogue box:



More Options section of the **Hatch** dialogue box:

Islands

Island detection

Turns on/off the mode to detect internal closed boundaries (islands).

Island display style

Select the island display style



Normal

Hatches inward from the outer boundary.

If the hatch encounters an internal island, it turns off hatching until it encounters another island within the island.



Outer

Hatches inward from the outer boundary. Hatch turns hatching off if it encounters an internal island. This option hatches only the outermost level of the structure and leaves the internal structure blank.



Ignores all internal objects and hatches through them.

Ignore

Boundary retention

Retain boundaries Creates boundary objects from the temporary hatch boundaries and adds them to the drawing.

Object type: Controls the type of the new boundary object.
The following types are available:

Region
Polyline

Boundary set

Defines the set of objects analysed when defining a boundary from a specified point. The selected boundary set has no effect when you use Select Objects to define a boundary.

From the drop-down list the following object sets are available:

Current viewport - Defines the boundary set from everything within the current viewport.

Existing set - Defines the boundary set from the objects that you selected with **New**.



New

Temporarily closes the Hatch dialogue box to select the objects that define the boundary set.

Gap tolerance

Tolerance Sets the maximum size of gaps that can be ignored when objects are used as a hatch boundary.

Enter a value, in drawing units, from 0 to 5000 to set the maximum size of gaps that can be ignored when the objects serve as a hatch boundary.

Any gaps equal to or smaller than the value you specify are ignored and the boundary is treated as closed.





Inherit options

When you use Inherit Properties to create a hatch, these settings control the location of the hatch origin.

Use current origin Uses the current hatch origin setting.

Using source hatch origin Uses the hatch origin of the source hatch.




To create hatch:

- In the **Hatch** dialogue box select the required hatch pattern from the drop-down Pattern list or in the **Hatch pattern** dialogue box opened after you click  icon. The graphic structure of the selected pattern will be displayed in the Swatch field.
- Set the required hatch options in the dialogue box.
- Click **Add: Pick points**  icon and specify points inside the areas that need to be hatched. Or click **Add: Select objects**  icon and select objects that form an enclosed area. The selected boundaries for the hatch are highlighted by a blue solid line when you pick a point inside the area and a white dotted line when you select objects. If the results of highlighting are not satisfactory for any reason, press **ESC** or click **Cancel** to cancel the context menu for the selected boundaries and return to the Hatch dialogue box to reset the selection. The **Preview** and **OK** buttons in the opened dialogue box will be blocked.
- If the highlighted contour is satisfactory, press **ENTER** or click the **Enter** command in the context menu of the **Hatch** dialogue box.
- To preview the hatch result, click . To finish the command without previewing, click **OK**.

When you preview the hatch result:

- If the preview of the hatch is satisfactory, click **Accept** in the command line or click **Enter** or **Accept** in the context menu to finish the command. Pressing **ENTER** also finishes the command.
- If the preview is not satisfactory, select **Reject** in the command line or click **Cancel** or **Reject** in the context menu to return to the dialogue box and change the hatch options,. Pressing **ESC** also returns to the dialogue box.

To create the hatch using the inherited properties of a selected hatch:

- Click the **Inherit properties**  icon. The **Hatch** dialogue box closes temporarily, to select the prototype hatch.
- Select the hatch object whose properties you want the hatch to inherit. The **Preview** and **OK** buttons in the opened dialogue box will be blocked.
- After selection, you can right-click in the drawing area and use the options on the context menu to switch between the **Pick Internal Point**  and **Select Objects**  options to create boundaries
- The order of the rest of the actions corresponds to the order of actions performed when you create a hatch.

Modify Hatched Areas



Menu: **Modify – Object >**  **Hatch...**



Toolbar: **Modify Object –** 



Command line: **HATCHEDIT**



As with any other object, a hatch can be deleted, copied, moved, rotated, etc.

If a hatch is selected, you can change its scale and angle.

You can edit the hatch properties in the **Hatch** dialogue box that is used to create the hatch.

The **Edit Hatches** command allows you to recreate removed hatch boundaries. When you use the **Edit Hatch** command, the **Recreate boundary** option will be available in the **Hatch** dialogue box.

To recreate a boundary:

- Start the **Hatch**  command from the **Modify – Object**
- In the **Hatch** dialogue box, click the **Recreate boundary**  icon.
- In the command line or context menu, select the **Region** or **Polyline** option to specify the object type to recreate the boundary.

Select **Yes** or **No** in the command line `Reassociate hatch with new boundary? [Yes/No] <N>:.`

- In the **Hatch** dialogue box click **OK**.

Shape



Menu: **Draw –**  **Shape...**



Command line: **SHAPE**

Shapes represent the objects described in a special format and are saved in text files with SHP extensions.

Shapes can be part of the description of complex line types (for more information, see the [Line Types](#) section («*Objects Properties* »)).

SHX-fonts are also described and stored in the SHP-files. Each symbol of this font is a special type of figure.

Shapes are objects that you use like blocks. User-defined shapes are helpful when you need to insert a simple part many times and when speed is important. Blocks are more versatile and easier to use and apply than shapes. However, shapes are more efficient to store and draw.

The nanoCAD delivery includes two files (GOST 2.303-68.shx and ltypeshp.shx), containing forms descriptions. After installation, data files are located in a folder:

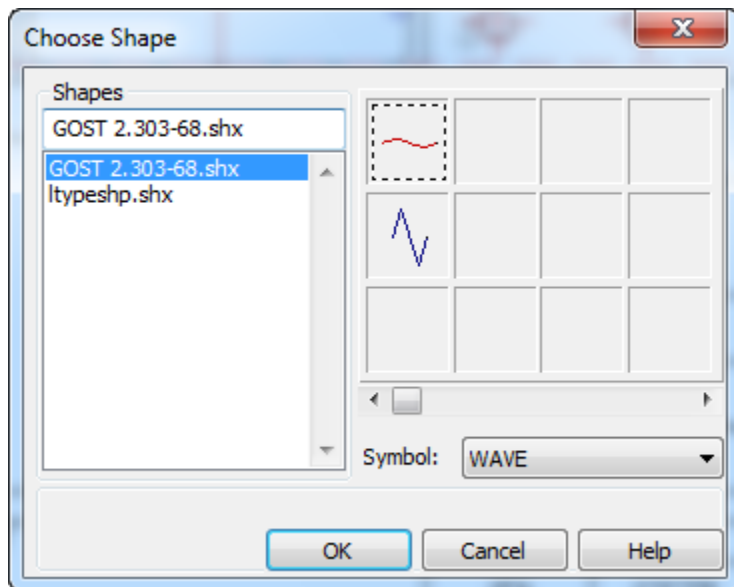
`c:\Documents and Settings\All Users\Application Data\Nanosoft\nanoCAD\SHX.`

The GOST 2.303-68.shx file contains the shapes used to describe complex lines types in the GOST 2.303-68.lin file. nanoCAD supports shapes created for AutoCAD.

You can create the definition files for user-defined shapes with a text editor or word processor that is then saved in the ASCII format. To use the compiled file in nanoCAD, just place it in the folder:

`c:\Documents and Settings\All Users\Application Data\Nanosoft\nanoCAD\SHX.`

The **Shapes** command allows you to insert shapes from (*.SHX) files into the document. Selection of the file containing the shapes is carried out in the **Choose Form** dialogue box.



To insert a shape:

- In the **Shapes** section, choose the file. The slides with graphic images of the available shapes in the file are displayed in the box located in the top right part of the dialog.
- Select the shape to insert and left click on the shape slide or select the shape name from the drop-down **Shape** list.
- Click **OK**.

After closing the dialogue box, perform the appropriate actions on prompts from the command line:

```
Specify insertion point: Specify the point.
Specify shape rotation <0>: Type the angle of rotation.
Specify shape scale <100.0000>: Type the scale factor.
```

Note: You can set the parameter values of the shape using the cursor on the screen, in which case there are dynamic changes in the appearance of the inserted shape, depending on the cursor movement.

FillShapes



Menu: **Draw** –  **FillShapes...**

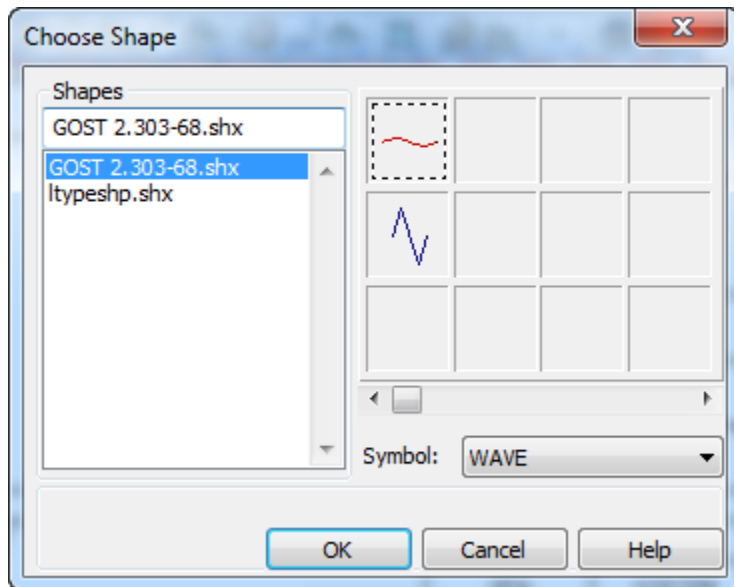


Command line: **FILLSHAPES**

FillShapes can be used as an alternative to the **Hatch** command; for example, to represent solid woodland or a water-logged area in topographical drawings. Unlike Hatch, the **FillShapes** command allows you to set the distance between rows and columns of shapes and also the angle of rotation of the shape in the filling (do not confuse with angle of rotation of the filling).

FillShape is not associative; that is, when you modify the boundary of the filled area (for example, using the grips or the **Stretch** command), the fill is not automatically updated.

After running the command, the **Choose Shape** dialogue box opens to choose the shape which is required to fill the enclosed area:



Note: When you choose a shape to fill the area, the same dialogue box as when choosing a shape to insert into the document is used. To choose a shape as a fill, use the same order as when you choose a shape to insert into the document (for more details, see «[Shape](#)» («*Design Drawings commands*»)).

Options:

Yes Create a block from all the shapes that make up the fill. The block can later be broken into separate shapes by the **Explode** command.


No Create the fill from shapes as separate elements.


Choose the shape for the fill and close the **Choose Shape** dialogue box. Perform the appropriate actions at the prompts from the command line:

| | |
|--|--|
| Click point inside contour: | <i>Specify a point.</i> |
| Specify shape rotation <0>: | <i>Type the value for the rotation angle of the shape.</i> |
| Specify shape scale <100.0000>: | <i>Type the scale factor.</i> |
| Specify shape horizontal offset <16.0000>: | <i>Type the distance between the columns of shapes.</i> |
| Specify shape vertical offset <16.0000>: | <i>Type the distance between the rows of shapes.</i> |
| Specify shape grid angle <0>: | <i>Type the shape fill angle.</i> |
| Combine shapes to block? <Yes> or [Yes / No /]: | <i>Select the necessary options and press ENTER to end the command.</i> |

Note: You can set the parameter values of the fill using the cursor on the screen, in which case there are dynamic changes in the appearance of the fill, depending on the cursor movement.

Boundary

Menu: **Draw** –  **Boundary...**

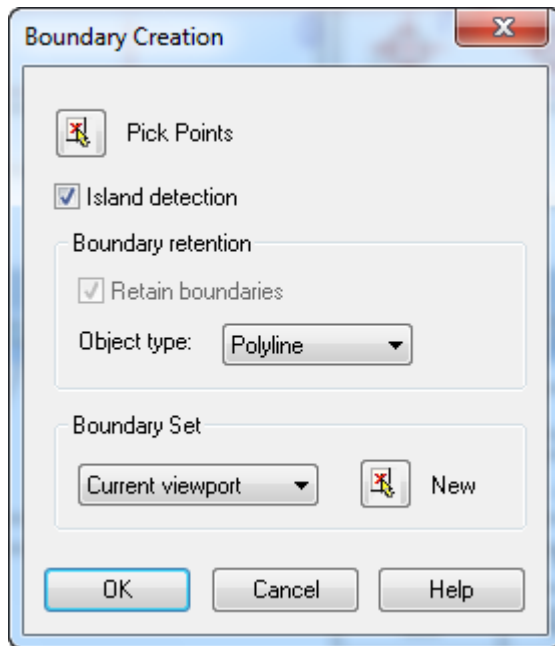
Toolbar: **Draw** – 

Command line: **BO, BOUNDARY**


This command creates a region bounded by a closed polyline (boundary). A boundary can be created from lines, polylines, circles, arcs, ellipses, elliptic arcs, and splines. A boundary can be created from a single closed object or from several intersecting or adjoining end points of the objects bounded by the closed region.

You can create a square or create a hatch for areas bounded by the boundary.

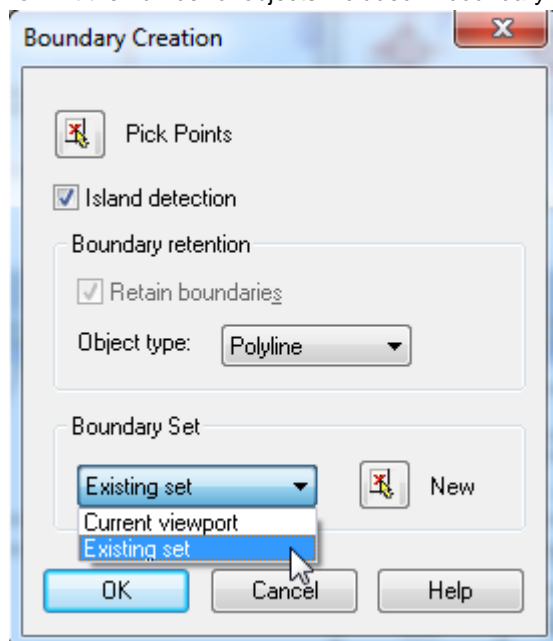
After starting the command, the **Boundary Creation** dialogue box opens:




To create a region or boundary:

- From the **Object type** list select the **Polyline** or **Region**.
- To search for internal closed boundaries (islands), select the **Islands detection** checkbox.
- Click the **Pick Points**  icon.
- Specify a point on the drawing for each internal closed boundary from which you want to create a region or a polyline.
- Click OK.


To limit the number of objects included in boundary determination, you can create a new set of boundaries:



- In the **Boundary Set** section, click the **New**  icon.
- Select the objects on the drawing that define the boundary.

When you select the **Current viewport** option from the list, the current boundaries set is cancelled, and a new boundaries set is created of all objects within the boundaries of the current viewport.

Solid

Menu: **Draw** –  **Solid**

Command line: **SOLID**

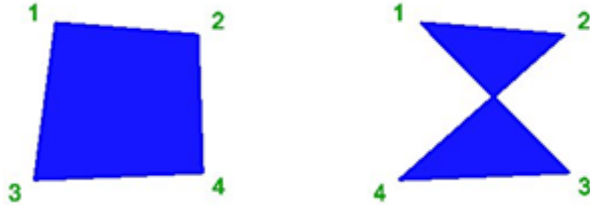
This command creates filled polygons.

The first two points define one edge of the polygon.

Pressing **Cancel** or **Enter** at the fourth point prompt creates a filled triangle. Specifying a fifth point creates a quadrilateral area.

The last two points form the first edge of the next filled area. The prompts *Specify third point: Specify fourth point or [Exit]:* are repeated. Specifying successive third and fourth points creates further connected triangles and four-sided polygons in a single solid object.

The order of specifying the vertices when you create a quadrilateral area affects the shape:



Options:


Exit Ends the command.

The following prompts are displayed:

| | |
|---------------------------------|---|
| Specify first point: | Specify point 1. |
| Specify second point: | Specify point 2. |
| Specify third point: | Specify point 3. |
| Specify fourth point or [Exit]: | Specify point 4 or press ENTER to create a triangle or cancel the command. |
| Specify third point: | Specify the next point. |
| Specify fourth point or [Exit]: | Specify the next point or select Exit to end the command. |

Wipeout



Menu: **Draw** –  **Wipeout**



Toolbar: **Draw** – 



Command line: **WIPEOUT**

This command creates a polygonal area that masks underlying objects with the current background colour.

The wipeout area is bounded by a frame that you can turn on for editing and turn off for plotting. You can convert a closed polyline consisting of line segments into a masking object.

You can create the masking objects in the paper space to hide the objects in the model space.

Options:

Undo Undo the last specified point.
This option allows you to undo all the specified points except the start point.

Close Closes the boundary and ends the command.

Polyline Create a mask boundary from an existing closed polyline.

Frames Select the visibility of the boundaries.

ON Display the boundaries.

OFF Hide the boundaries.

The following prompts are displayed:

| | |
|---|--|
| Specify start point or [Frames / Polyline]: | Specify the point. |
| Specify next point <Start tangent>: | Specify the next point. |
| Specify next point <Start tangent> or [Undo]: | Specify the next point. |
| Specify next point <Start tangent> or [Undo / Close]: | Specify all subsequent points defining the boundary inside which is necessary to hide the objects. |

Specify next point <Start tangent> or [Undo/Close]:

Press **ENTER** or select the Close option to end the command.

When you create the boundary from closed polyline, the following prompts are displayed:

Specify start point or [Frames/Polyline]:

Select the Polyline option.

Select closed polyline:

Select the polyline.

When you create the boundary in the frames mode, the following prompts are displayed:

Specify start point or [Frames/Polyline]:

Select the Frames option.

Enter mode <ON> or [ON/OFF]:

Select the required option.

Note: Selecting the boundary visibility mode affects all mask objects on the drawing.

Revision Cloud



Menu: **Draw –**  **Revision Cloud**



Toolbar: **Draw –** 



Command line: **REVCLOUD**

Revision clouds are polylines that consist of sequential arcs. The maximum arc length cannot be set to more than three times the minimum arc length.

Revision Clouds are used to drawing explanatory labels and markings on the drawings.

When you start the command, the current parameters of the revision cloud are displayed in the command line:

Minimum arc length: 15000.000000 Maximum arc length: 15000.000000 Style: Normal

Options:

Arc length

Specifies the minimum and maximum length of the arcs in a revision cloud.

Object

Specifies a closed object to be converted to a revision cloud or to reverse the direction of the arcs in the revision cloud.

Style

Specifies the style of the revision cloud.

Normal

Normal style.

Calligraphy

Calligraphy style.

Normal Style



Calligraphy



Revision cloud



The following prompts are displayed:

Specify first point [Arc length /Object/Style]<Object>:

Specify the first point.

Guide crosshairs along cloud path...:

Move the cursor to the start point of required contour. When the cursor draws near to the start point, automatic closing of the cloud contour will happen and the command will be finished.

When you convert a closed object to a revision cloud and change the direction of the arcs in the revision cloud to the opposite, the following prompts are displayed:

Specify first point [Arc length /Object/Style]<Object>:

Select the Object option.

Select object:

Select the object.

Reverse direction [Yes/No]<NO>:

Select the required option.

Work with Text

The text you add to your drawings conveys a variety of information. It may be a complex specification, title block information, a label, or even part of the drawing.

In nanoCAD you can create and edit single-line text and multiline text.

For short entries that do not require multiple fonts or lines, create *single-line text*. *Single-line text* is most convenient for titles and labels.

For long, complex entries, such as technical requirements or technical specifications, create *multiline text*.

To input text, the vector fonts with an shx extension are used. These fonts are installed when you install nanoCAD. In addition, it is possible to use the TrueType fonts which are installed in the operating system and have a TTF extension.

The nanoCAD tools for text creation allow you to select the typeface, set and edit the text height, weight and alignment modes.

Text



Menu: **Draw – Text >**  **Single LineText**



Toolbar: **Draw –** 



Command line: **TEXT**

The command allows you to create one or more lines of text. Each text line is an independent object.

To create multiple lines of text, after each line is input of press **ENTER** to move the cursor to the next line. You can also start a new line, by specifying it with the cursor on the screen.

To input «degree», «plus/minus», «diameter», you can use the special symbols: %%d, %%p and %%c.

Press **ESC** to cancel the command and remove the typed text.

To finish text input, press the key combination **CTRL+ENTER**.

By default, when you input text, the text style that is set as current in the **Text Style** dialogue box is used.

If necessary, you can change the text style in the command line immediately after the start of the command.

Options:

Style Type the name of the text style in the command line.

? The display of all available text styles in the command line.

The following prompts are displayed:

Input origin of text or [**Style**]:

Type the text style or ?:

Input origin of text or [**Style**]:

Specify text height <2.5000>:

Specify rotation of text <0>:

Type the text. Finish input with
<Ctrl>+<Enter> or cancel it with
<Esc>...:

Select the **Style** option.

Type the name of the text style or ? symbol to show the names of all available text styles in the command line.

Input the origin of the text on the drawing.

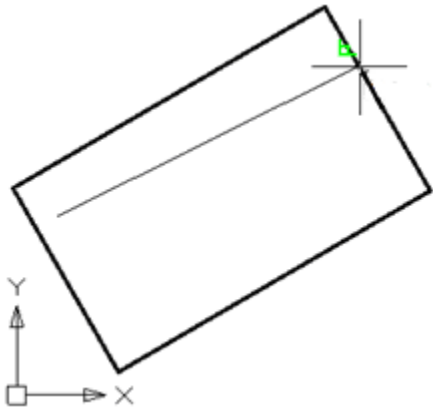
Type in the command line or set by cursor on the screen the text height.

Type in the command line or set by cursor on the screen the rotation of the text.

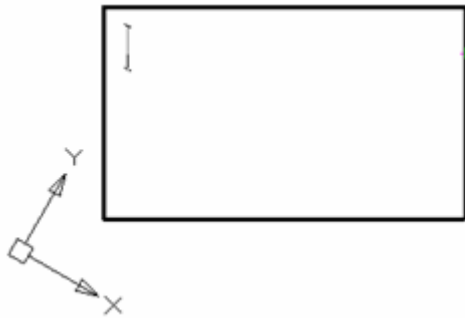
Type the text. Press **CTRL+ENTER** to finish input and end the command.

To create rotated text:

- Input the origin of the text.
- Specify the text height.
- Specify the rotation of the text:



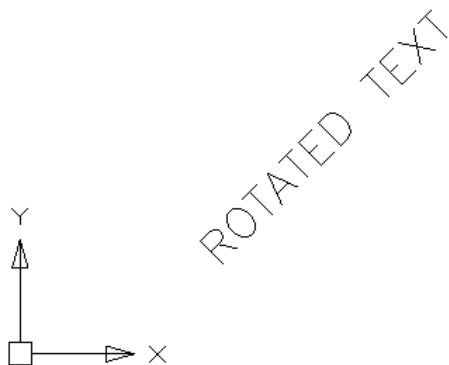
- After specifying the rotation, the text line remains horizontal for convenience during input. The objects on the drawing will be rotated by the specified angle, but in the opposite direction:



- Type the text:



- Finish input with **Ctrl+Enter**. The image on the screen returns to its normal view:



Multiline Text



Menu: **Draw – Text > T Multiline Text...**



Toolbar: **Draw – T**



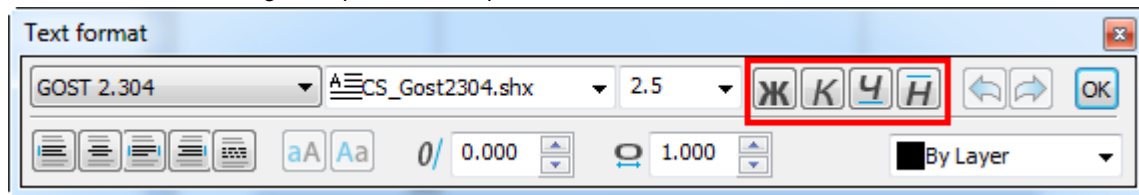
Command line: **MT, MTEXT, T**

Multiline text consists of any number of text lines or paragraphs that fit within a width you specify. It can extend vertically to an indefinite length. Multiline text is a single object.

There are more editing options for multiline text than there are for single-line text. For example, you can apply underlining, fonts, colour, and text height changes to individual characters, words or phrases within a paragraph

Draw a rectangular frame by double clicking a point to place multiline text there. Specify the top left corner first and then the bottom right corner.

In the **Text format** dialog that opens, set the parameters of the multiline text:



Options:

Drop-down list to select the text style.

Drop-down list to select the font file that defines the style of the characters.

Input field for the characters' height.

Text formatting modes



Bold

Turns bold formatting on and off for new or selected text. This option is available only for characters using TrueType fonts.



Italic

Turns italic formatting on and off for new or selected text. This option is available only for characters using TrueType fonts.



Underlined

Turns underlining on or off for new or selected text.



Overlined

Turns overlining on or off for new or selected text.

Text position modes



Align left

Sets the alignment of the text to the left.



Align center

Sets the alignment of the text to the centre.



Align right

Sets the alignment of the text to the right.



Justified

Sets the text mode to justified.



Distributed

Sets the text mode to distributed.

Change the case of selected text



Uppercase

Changes the selected text to uppercase (replace the lowercase characters with uppercase).



Lowercase

Changes the selected text to lowercase (replace the uppercase characters with lowercase).

Additional options

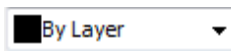
Determines the forward or backward slant of the text.

A positive angle slants text to the right. A negative angle slants text to the left

Widens or narrows the selected characters.

The 1.0 setting represents the normal width of the letter in this font.

Set to more than 1.0 to increase the width, and set to less than 1.0 to decrease the width.



Drop-down list to select the colour for the text.



Undo

Undoes actions in the text editor.



Redo

Redoes actions in the text editor.



CTRL+ENTER

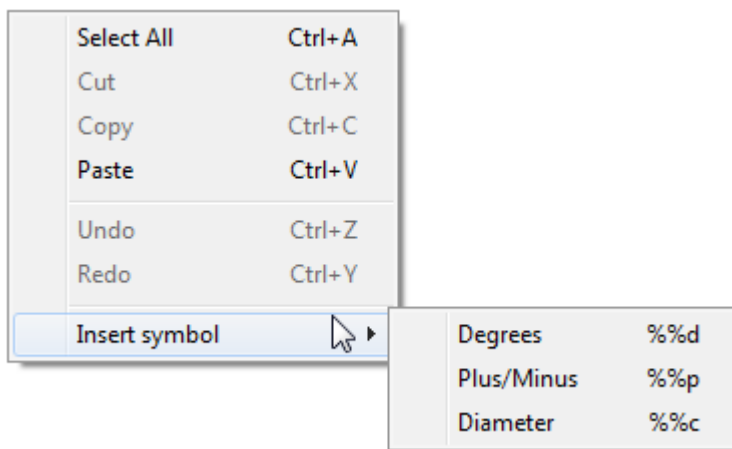
Click button to close the window.

To create multiline text:

- Set the text area by specifying the top left corner first and then the bottom right corner.
- Set the required options in the **Text format** dialogue box.
- Click the cursor inside the text area on the drawing.
- Type the text from the keyboard.

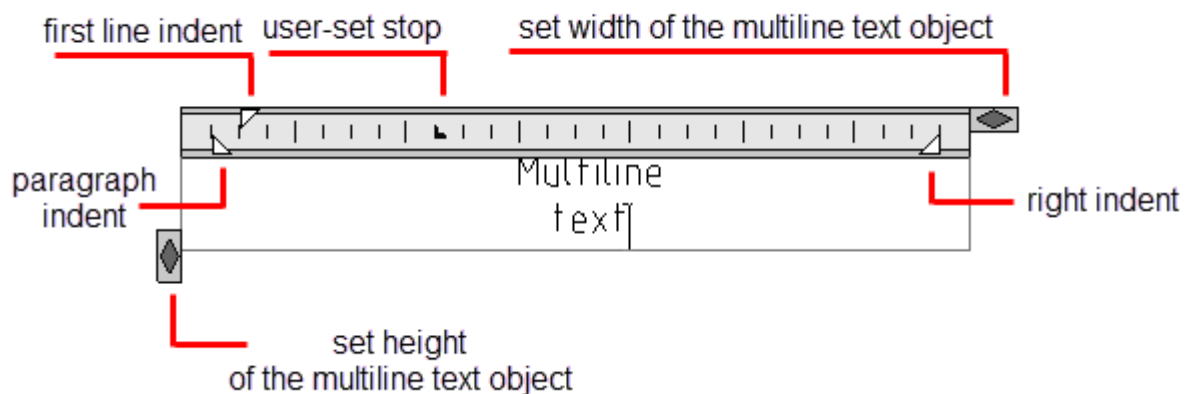
To move to a new line, press **ENTER**.

To insert «degree», «plus/minus», «diameter» symbols, use the **Insert symbol** command from the context menu:



Note: Text can be overlooked if the font size in the **Text format** dialogue box is set too small compared to the scale of the drawing.

- Set the position of the selected text using the text input window elements:



- To complete typing the multiline text, press the key combination **CTRL+ENTER** or click **OK** in the **Text format** dialogue box. You can also click outside the text input area on the drawing to complete the typing of the multiline text.

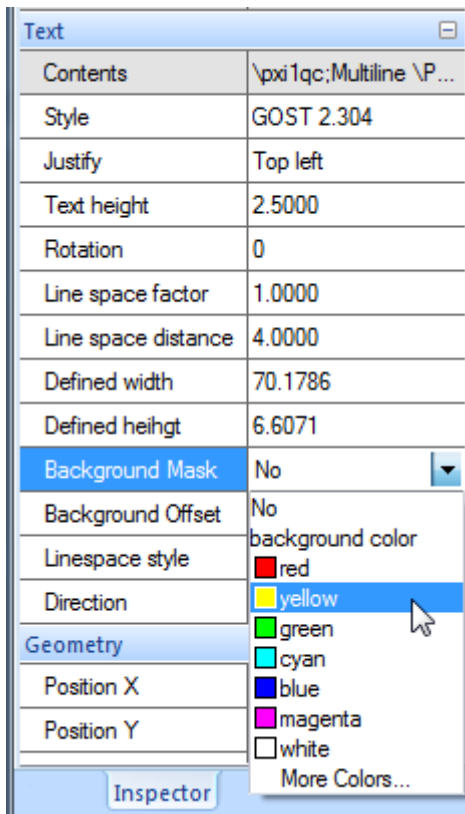
Note: Press **ESC** to cancel the text input and finish the command.

Background Mask

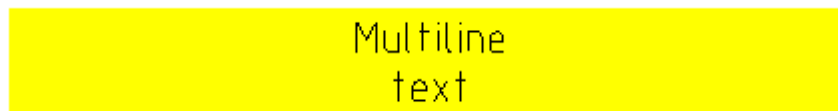
You can use the background mask for multiline text. The background mask allows you to place text on the non-transparent background.

To create a background mask:

- Select the multiline text.
- Select the mask colour from the drop-down **Background Mask** list (the **Text** section) of the **Inspector** window:



The mask is created for the entire text area. When you resize the text area using grips, the size of mask also changes.



Select **No** to cancel the background mask. Select **Background color** to assign the background colour to the mask. The **Background Offset** option of the **Inspector** window sets the fields for the text mask.

The factor value is based on the text height:

- A factor of 1.0 exactly fits the multiline text object.
- A factor of 1.5 (set by default) extends the background by 0.5 times the text height

It is possible to set the background for several text objects at one time.

Editing Text


Text objects created in nanoCAD, are selected with grips and they can be edited like other objects: rotated, removed, copied etc.

The properties of text objects can be changed in the **Inspector** window.

Note: It is only possible to change single-line text properties in the **Inspector** window.

Justification of the text objects



Menu: **Modify – Object > Text >**  **Justifytext**



Toolbar: **Modify Object –** 



Command line: **JUSTIFYTEXT**

Start the command and select the text object on the screen. Select the justification method in the command line or from the context menu.

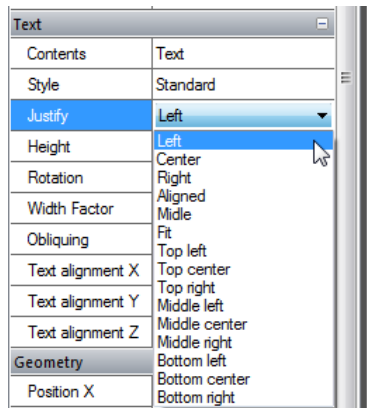
For *single line text*, the **JUSTIFYTEXT** command changes the type of justification, without moving the text (the number and location of the grips will be changed).

For *multiline text*, the **JUSTIFYTEXT** command changes the type of text justification relative to the text boundaries, without moving the text (the number and location of the grips for the text area and the location of the boundaries of the text area relative to the text will be changed).

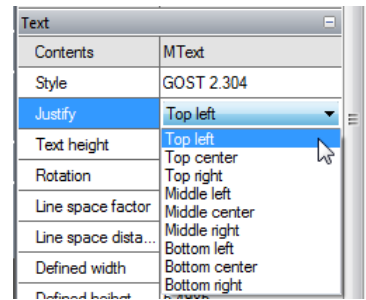
To change the type of justification with a simultaneous change of the text object (for single-line text) and text area (for multiline of text) use the **Inspector** window.

To do this you should first select the text object and then specify the required type of justification in the **Inspector**

window:



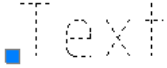





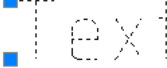
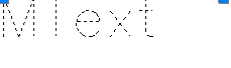


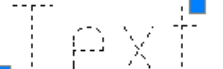





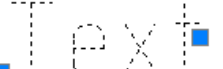

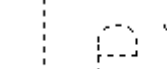
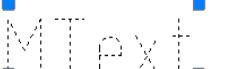
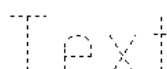



Text



MText

Start the command and select the text object on the drawing. Specify the type of justification in the command line or the context menu.

Options:

| | | | |
|---------------|---|---|--|
| Left |  | - | Left-justifies the text at the baseline. |
| Align |  | - | Inscribes text between two points. The width of each character is calculated automatically so that the text exactly fits into the specified area. The character height is not changed. |
| Fit |  | - | Inscribes text between two points. The height and width of each character is calculated automatically so that the text exactly fits into the specified area. |
| Center |  | - | Aligns text from the horizontal centre of the baseline, which you specify with a point. |
| Middle |  | - | Aligns text at the horizontal centre of the baseline and the vertical centre of the height you specify. |
| Right |  | - | Right-justifies the text at the baseline. |
| TL |  |  | Left-justifies text at a point specified for the top of the text. |
| TC |  |  | Centres text at a point specified for the top of the text. |
| TR |  |  | Right-justifies text at a point specified for the top of the text. |
| ML |  |  | Left-justifies text at a point specified for the middle of the text. |
| MC |  |  | Centres the text both horizontally and vertically at the middle of the text. |
| MR |  |  | Right-justifies text at a point specified for the middle of the text. |
| BL |  |  | Left-justifies text at a point specified for the baseline. |
| BC |  |  | Centres text at a point specified for the baseline. |
| BR |  |  | Right-justifies text at a point specified for the baseline. |

Editing the content of the text objects

Editing single-line text



Menu: **Modify – Object > Text >**  **Edit...**

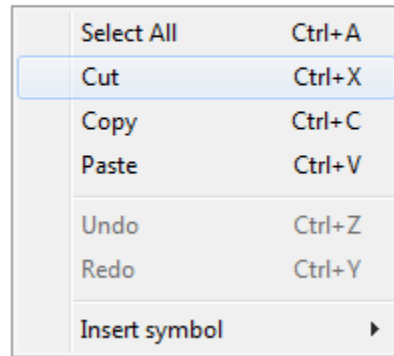


Toolbar: **Modify Object** –



Command line: **DDEDIT, ED**

Editing the content of single-line text is carried out on the screen. The content of the selected single-line text is selected and highlighted automatically to completely edit the text. If necessary, you can insert or delete single characters or fragments within the line. For more convenient editing, use the context menu:

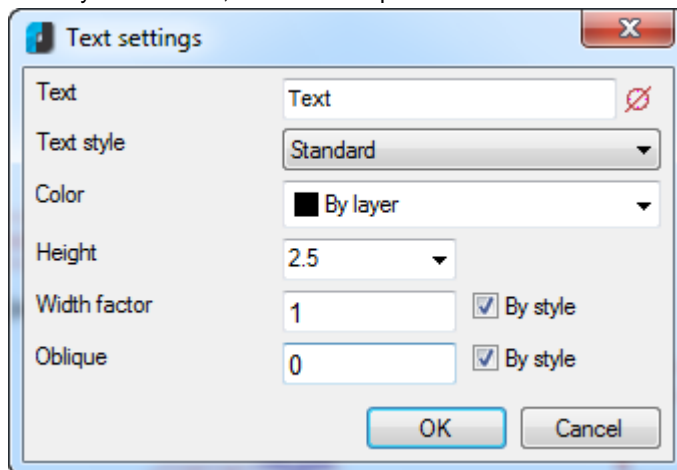


Press **CTRL+ENTER** or click on the drawing out of the single-line text to confirm changes and finish the editing.

To finish the editing without saving changes, press **ESC**.

Note: Press **ENTER** to split the single-line text into two lines, each of which is transformed into a separate text object after the command end.

Text lines can be edited by double clicking. In this case, the **Text settings** dialogue box opens where you can edit not only text content, but also the options:



Options:

Text

Field for display and editing of the text line content.



Insert special symbol

This icon opens an additional menu to insert symbols.

Text Style

Drop-down list to select the text style.

Color

Drop-down list to select the colour of the text line.

Height

Drop-down list to select the height of the characters. Keyboard input is also possible.

Width factor

Sets the character spacing.
Entering a value less than *1.0* condenses the text.
Entering a value greater than *1.0* expands it.

Oblique

Sets the oblique angle of the text.

☒ By style

The fields display/specify the compatibility of the width factor and oblique of the edited text symbols by the value set in the text style.

For text in which the width factor and/or oblique of the symbols correspond to the specified text style, the check boxes are selected in these fields:

| | | |
|--------------|---|--|
| Width factor | 1 | <input checked="" type="checkbox"/> By style |
| Oblique | 0 | <input checked="" type="checkbox"/> By style |


If the width factor and/or oblique of the symbols does not correspond to the specified text style, the check boxes are not selected:

| | | |
|--------------|------|-----------------------------------|
| Width factor | 0.75 | <input type="checkbox"/> By style |
| Oblique | 15 | <input type="checkbox"/> By style |




The current values of the editable text are displayed in the input fields for the width factor and oblique.

To set the width factor and oblique for the editable text to the values which correspond to those set in the text style, it is necessary to select the check boxes. Values are changed automatically:

| | | |
|--------------|---|--|
| Width factor | 1 | <input checked="" type="checkbox"/> By style |
| Oblique | 0 | <input checked="" type="checkbox"/> By style |

The **Insert special symbol**  icon opens an additional menu that allows you to select and insert the special symbols into the current text line:



The buttons , ,  of this menu, in turn, also open:

Menus of some mathematical and other symbols



Uppercase Greek letters menu




Lowercase Greek letters menu



Note: Special characters are displayed correctly only in texts based on vector fonts. When you insert characters based on a TrueType font into the text, question marks instead of the special characters appear on the screen.

To insert symbols:

- Place the cursor in the required position in the text line.
- Click the **Insert special symbol**  icon.
- Select the required symbol.
- When you click on the symbol, the additional menu will be closed and the symbol will be inserted automatically at the current position of the cursor in text line.

Editing multiline text



Menu: **Modify – Object > Text >**  **Edit...**



Toolbar: **Modify Object –** 



Command line: **DDEDIT, ED**

Start the command and select the multiline text. The **Text format** dialogue box opens. You can also activate the editing mode, by double clicking on the multiline text.

To edit the content and options of multiline text:

- Select the text fragment in the text box and replace it with new text or type additional text. When text is selected, the following options are available in the context menu: **Select All, Cut, Copy, Paste, Undo, Redo, Insert symbol.**
- Change the options for the selected text by using the tools of the **Text format** dialogue box.
- To end multiline text editing, press **CTRL+ENTER** or click **OK** in the **Text format** dialogue box. You can also by click out of the text input area on the drawing to complete text the typing.

Note: Press **ESC** to cancel the text input and finish the command.

Creating a text style



Menu: **Format – T Text Style...**



Toolbar: **Styles – T**



Command line: **ST, STYLE**

The text style is a convenient tool that creates text objects using different fonts, font size, obliquing angle, orientation, and other text characteristics.

You can create and use several text styles in one drawing. Each text object in the drawing is created using the current text style. If you want to create text using a different text style, you can make another text style current.

Except for the default *Standard* text style, you must create any text style that you want to use.

You can modify an existing text style in the **Text Style** dialogue box by changing the settings.

A text style name can be up to 255 characters long. It can contain letters, numbers, and the special characters dollar sign (\$), hyphen (-), and underscore (_).

By default, the text styles are automatically named Style1, Style2, Style3, etc.

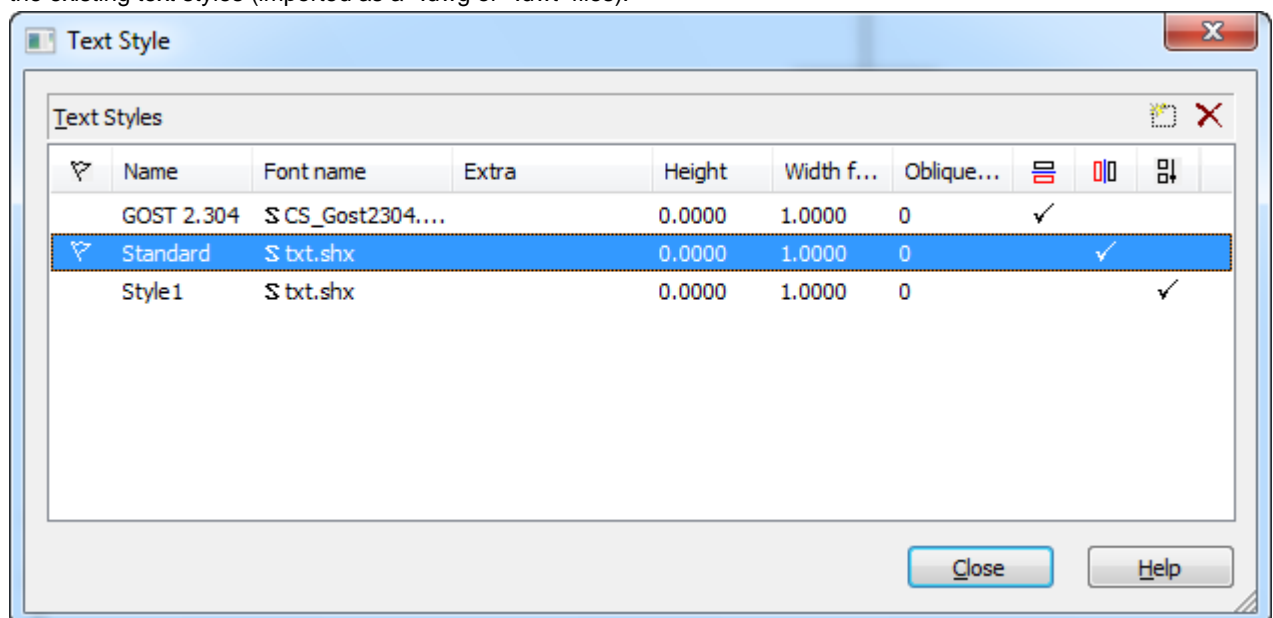
Created text styles are saved with the drawing file.

You can remove unused text styles from your drawing by deleting them from the **Text Style** dialogue box or with the **Purge** command (the **File** menu – **Drawing Utilities**).

Note 1: The *Standard* text style cannot be removed.

Note 2: Changing the **Upside Down** and **Backwards** options has no effect on multiline text objects

The **Text Style** dialogue box allows you to set the current text style, create new text styles, change the parameters of the existing text styles (imported as a *.dwg or *.dwt files):



Options:



Add new style

Creates a new text style.






Delete style

Deletes unused text styles.






Set current

Sets the selected style as current.

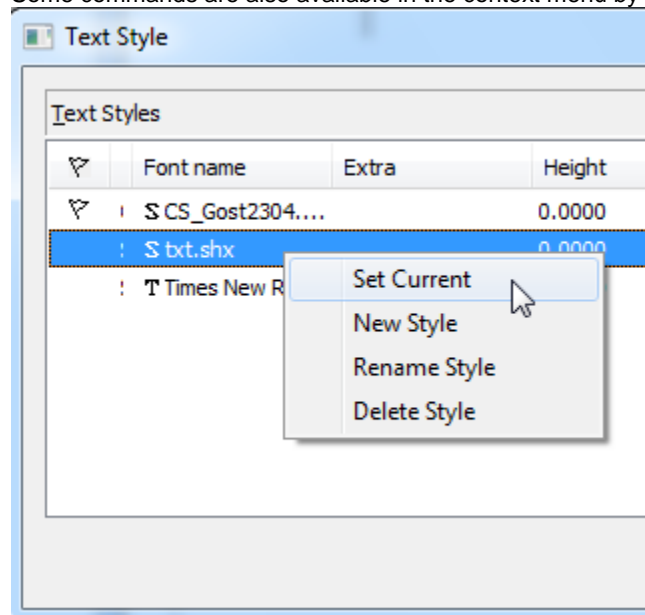
| | |
|--|--|
| Name | Name of text style. |
| Font name | Drop-down list to select the font file that set the character's style. |
| Extra | Drop-down list to select the character's style. The list is empty for shx-fonts. For TTF-fonts the following types are available: Italic Normal Bold Bold Italic |
| Height | Input field for the height of the characters: |
| Width factor | Sets the character spacing. Entering a value less than <i>1.0</i> condenses the text. Entering a value greater than <i>1.0</i> expands it. |
| Oblique angle | Sets the oblique angle of the text. |
|  Upside down | Turn on/off the mode to display the characters upside down. |
|  Backwards | Turn on/off the mode to display the characters backwards |
|  Vertical | Turn on/off the mode to display the characters aligned vertically. Vertical is only available if the selected font supports dual orientation. |

Note: The vector fonts (*.shx) are marked with **S** in the **Font name** list, TrueType-fonts with **T**.

To create a text style:


- Select the text style, which the new text style should be based on (the **Add new style**  icon is available).
- Click the **Add new style**  icon. The new style with **Style1** name will be created.
- To rename the created style, double click on the text style name, type the new name and press **ENTER**.
- From the drop-down list select the font file.
- Specify the required font options (height, extra, oblique angle, etc.).
- To set the created text style as the current one, double click in the left field next to the text style. The flag  in this box indicates that the style is set as the current one.
- Click **Close** to close the dialogue box.

Some commands are also available in the context menu by right clicking in the field of the text styles list:



To delete a text style:

- Select the text style you want to delete (the **Delete style**  icon is available).

- Click the **Delete style**  icon or select **Delete Style** from the context menu (the selected style must not be the current one).
- Click **Close** to close the dialogue box.

Find and Replace Text

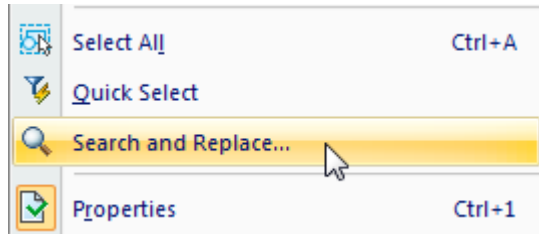


Menu: **Edit – Search and Replace...**

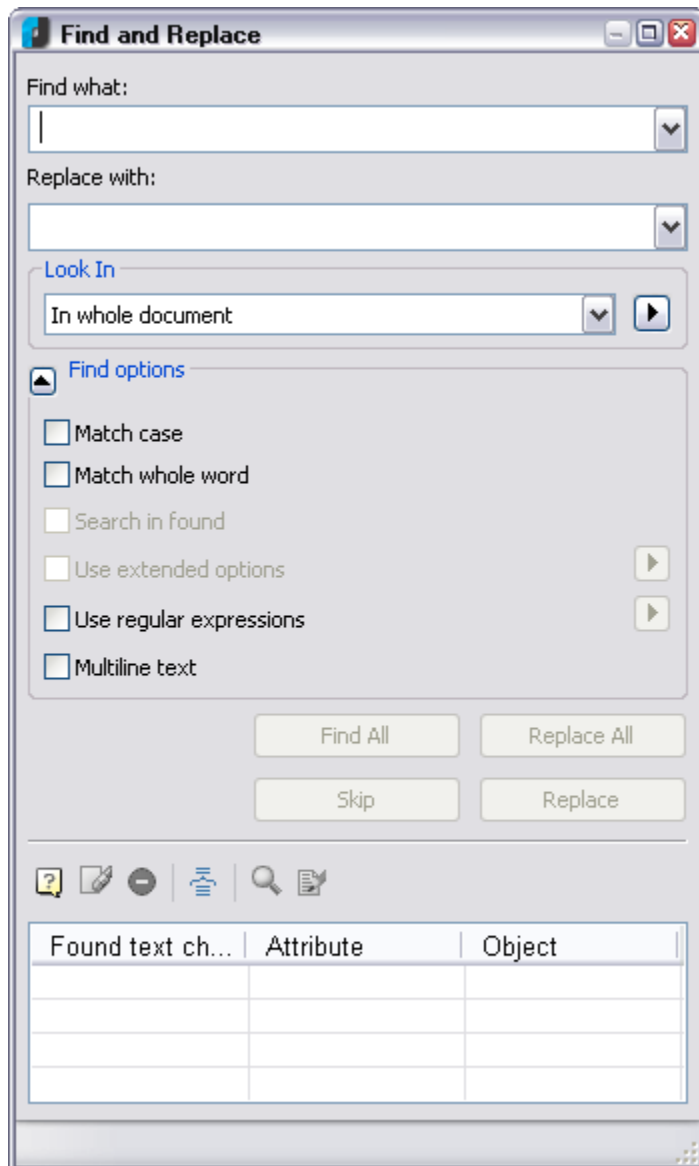
Command line: **FIND**

The **Search and Replace** command is used to find text in the document:

You can also start the command from the context menu:



Start the command to open the **Find and Replace** dialogue box:



Options:

Find what: In the **Find what** field, type the text to be searched for.

Replace with: In the **Replace with** field, type the text that is required to replace the found lines

Look In Set the **Look in** parameter.

The following options are available:

In whole document

In current space

In current selection



Click the icon to open the **Quick selection** dialogue box.

Find options



This icon maximises/minimises the options list.

Match case

If this is checked, the search is made to match the case of the letters.

Match whole word

If this is checked, the search is made for whole words; that is letter combinations separated by stops or spaces.

Search in found

This option is accessible after the line search has been made. It restricts the search range.

Use extended options

This option is accessible when searching in the objects base. Click the icon and in the window that appears, choose the expanded search spaces

Use regular expressions

Switch to the search mode with additional choices of conditions.

Click the icon and in the window that appears, choose the required expression.

| Regular expressions | |
|---------------------|--|
| \ | Marks the next character as either a special character or a liter... |
| ^ | Matches the beginning of input. |
| \$ | Matches the end of input. |
| * | Matches the preceding character zero or more times. For exam... |
| + | Matches the preceding character one or more times. For example, "zo+" matches "zoo" but not "z". |
| ? | Matches the preceding character zero or one times. For exam... |
| . | Matches any single character except a newline character. |
| (Pattern) | Matches pattern and remembers the match. The matched subst... |
| (?:Pattern) | Matches pattern but does not capture the match, that is, it is a ... |
| (?=Pattern) | Positive lookahead matches the search string at any point whe... |
| (?!Pattern) | Negative lookahead matches the search string at any point whe... |
| x y | Matches either x or y. For example, "z[food]" matches "z" or "fo... |
| {n} | n is a nonnegative integer. Matches exactly n times. For exam... |
| {n,} | n is a nonnegative integer. Matches at least n times. For exam... |
| {n,m} | m and n are nonnegative integers. Matches at least n and at m... |
| [xyz] | A character set. Matches any one of the enclosed characters. ... |
| [^xyz] | A negative character set. Matches any character not enclosed. ... |
| [a-z] | A range of characters. Matches any character in the specified r... |
| [^m-z] | A negative range characters. Matches any character not in the ... |
| \b | Matches a word boundary, that is, the position between a word... |
| \B | Matches a nonword boundary. "ea*r\b" matches the "ear" in "n... |
| \d | Matches a digit character. Equivalent to [0-9]. |
| \D | Matches a nondigit character. Equivalent to [^0-9]. |
| \f | Matches a form-feed character. |
| \n | Matches a newline character. |
| \r | Matches a carriage return character. |
| \s | Matches any white space including space, tab, form-feed, etc. ... |
| \S | Matches any nonwhite space character. Equivalent to "[^\f\n\r]... |
| \t | Matches a tab character. |
| \v | Matches a vertical tab character. |
| \w | Matches any word character including underscore. Equivalent t... |
| \W | Matches any nonword character. Equivalent to "[^A-Za-z0-9_]". |
| \num | Matches num, where num is a positive integer. A reference bac... |

For guidance on the elements of the list, there is a help with more detailed explanations.

The expression is selected with a left-click. It is possible to use some regular expressions.

Multiline text

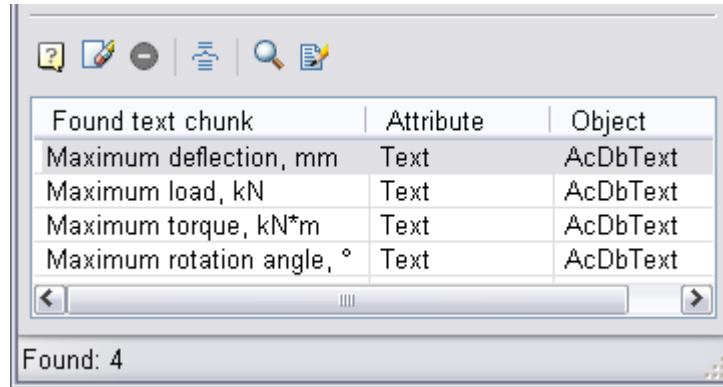
Switch on/off the multiline text search mode.

When the mode is on, the **Find what** and the **Replace with** fields take the form:

| | |
|---------------|--|
| Find what: | |
| <div></div> | |
| Replace with: | |
| <div></div> | |

Find All

Starts the search procedure.
The search result will be shown in the table.



| Found text chunk | Attribute | Object |
|---------------------------|-----------|----------|
| Maximum deflection, mm | Text | AcDbText |
| Maximum load, kN | Text | AcDbText |
| Maximum torque, kN*m | Text | AcDbText |
| Maximum rotation angle, ° | Text | AcDbText |

Found: 4

Replace All

Replaces all found fragments with the new value.

Replace

Starts the process of consecutive replacement of the found fragments.

Skip

Passes the next found fragment in the list without replacing it with the new value.



Clear All

Click this icon to clear the list of found text fragments.



Stop

Click this icon to stop the find and replace process.



Group results by objects

Click this icon to group results by objects.



Show object

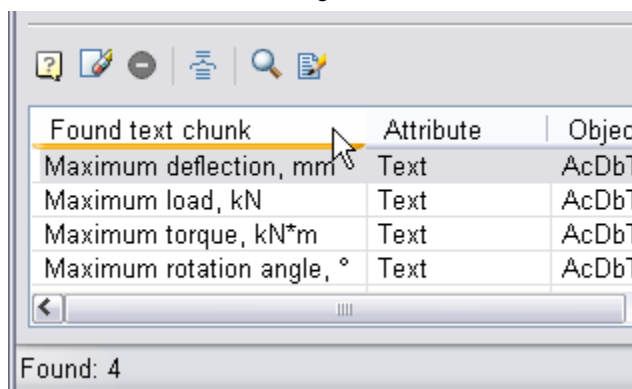
Click this icon to show the selected text fragment in the table on the drawing or in the database of objects.



Edit object

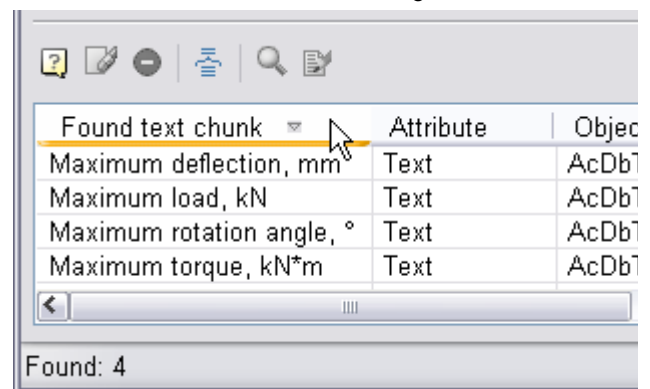
Click this icon to open the **Text settings** dialogue box to edit the found text fragment on the drawing.

Note: Left-click on the headings of the table columns of the search results to sort the found text fragments:



| Found text chunk | Attribute | Object |
|---------------------------|-----------|----------|
| Maximum deflection, mm | Text | AcDbText |
| Maximum load, kN | Text | AcDbText |
| Maximum torque, kN*m | Text | AcDbText |
| Maximum rotation angle, ° | Text | AcDbText |

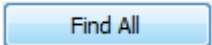
Found: 4

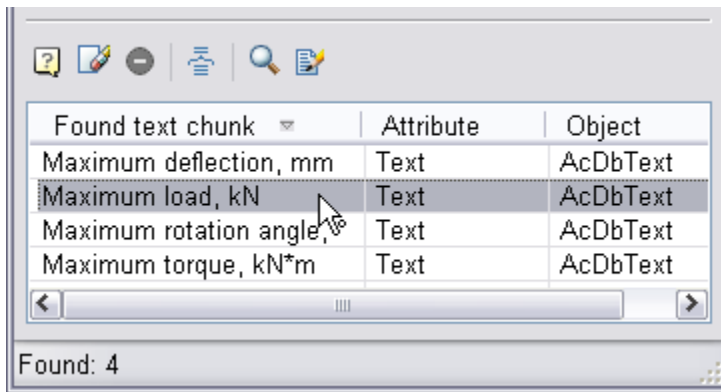



| Found text chunk | Attribute | Object |
|---------------------------|-----------|----------|
| Maximum deflection, mm | Text | AcDbText |
| Maximum load, kN | Text | AcDbText |
| Maximum rotation angle, ° | Text | AcDbText |
| Maximum torque, kN*m | Text | AcDbText |

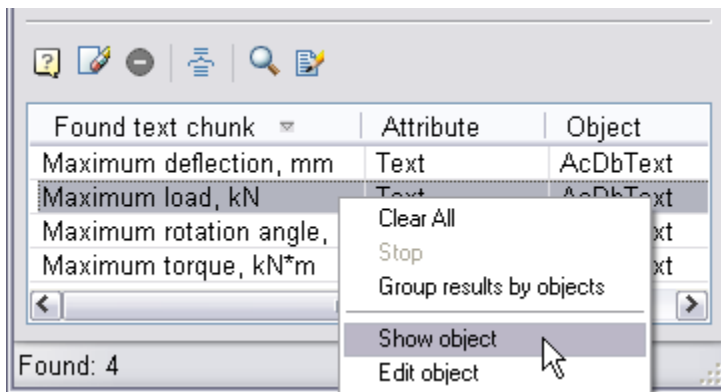
Found: 4

To find and edit the text fragment on the drawing:

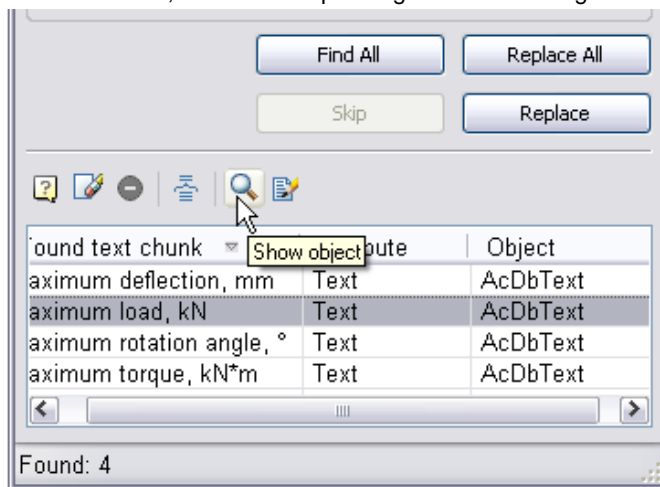
- Click the **Search and Replace** command in the context menu.
- Type the text, for example, *maximum*, into the **Find what** field.
- Click the button .
- Select the required line in the table of search results.



- Double click on the required line, or click the **Show object**  icon, or select the **Show object** command from context menu:



- As a result, there is auto-panning of the found fragment that contains the selected text on the drawing:



Parameter

Maximum deflection, mm


Maximum rotation angle, °

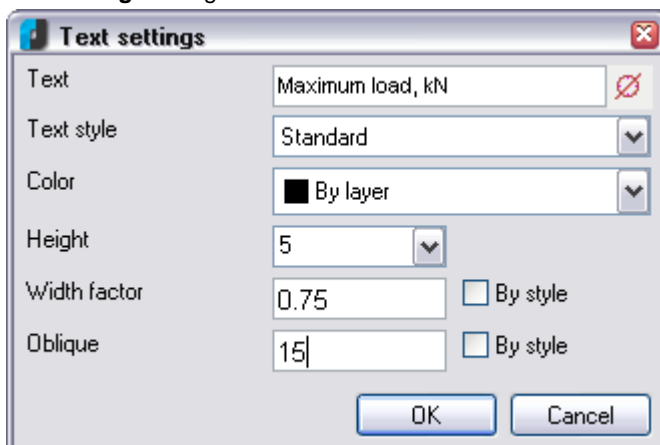
*Maximum torque, kN*m*

Maximum load, kN

Shaft weight, kg

Length, mm

- Click the **Edit object**  icon or select the **Edit object** command from the context menu to open the **Text settings** dialogue box:



- Type the required changes and click **OK**.

Dimensioning

Dimensions display the geometrical attributes of the objects on the drawing, as well as the distances and angles between them.

The dimensions are part and parcel of any drawing.

In general, dimensions can consist of the following items:

- *The dimension line* indicates the direction and extent of a dimension. For angular dimensions, the dimension line is an arc.
- *The extension line* is drawn from the measured object to the dimension line.
- *Arrows* are displayed at the ends of the dimension line. You can use different types of arrows, including tick marks and points.
- *The dimension text* displays the numerical value of the measured object. The text can also include prefixes and suffixes, for example, symbols of the radius, diameter, degree, etc., as well as tolerances.
- *The leader* is the line joining together the dimension text and the dimension line to which it belongs. Leaders can be created automatically (when the corresponding options are set), when the text size does not fit between the extension lines or when you manually drag the dimension text (with grips) to another place.

The four basic types of dimensioning are:

- [Linear dimensions](#) display the distance between the specified points. This type includes the following dimensions:
 - [horizontal](#),
 - [vertical](#),
 - [aligned](#),
 - [ordinate](#),
 - [group dimension](#),
 - [base dimension](#) and
 - [dimensions chain](#).
- [Radial dimensions](#) indicate the radii and diameters of arcs and circles. These include:
 - [diameter](#),
 - [radius](#),
 - [big radius](#).
- [Angular dimensions](#) are used to indicate the angles between two segments or three points.
- [Arc dimension](#) displays the length of an arc or an arc segment of a polyline.

Dimensions can be associative, non-associative or exploded. Associative dimensions adjust to changes in the geometric objects that they measure.

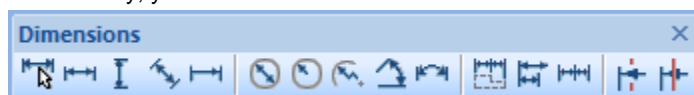
- *Associative dimensions*. Automatically adjust their locations, orientations and measurement values when the geometric objects associated with them are modified.
- *Non-associative dimensions*. Selected and modified with the geometry they measure. Non-associative dimensions do not change when the geometric objects they measure are modified.
- *Exploded dimensions*. Contain a collection of separate objects: lines, arrows, arcs and text, rather than a single dimension object. Exploded dimensions do not change when the geometric objects they measure are modified.

It is not recommended to disable the associative dimensioning mode which is used by default or to explode the associative dimensions without a strong reason.

In nanoCAD, the dimensioning commands are available from the **Dimensions** menu and from the **Utilities** toolbar:



If necessary, you can use the **Dimensions** toolbar:



Some features of nanoCAD's dimensioning

Set the Scale for Dimensions

You can specify the size of dimensions in your drawing. Set the scale value using the **Scale** icon in the status line.

Dimension scale affects the size of the dimension geometry relative to the objects in the drawing. At dimensioning, all

size elements (height of the dimension text, size of the arrows etc.) are automatically scaled corresponding to the current *dimension scale*.

The dimension scale is useful to dimension fragments drawn in the model space at the 1:1 scale. Their scale will change at arrangement on the worksheet.

For example, two views are drawn in the model space at the 1:1 scale. The first view will be placed on the layout at 1:1, the second view (based on its actual size) at 1:10. For dimensioning in the model space, you must specify the dimension scale as 1:1 for the first view and 1:10 for the second view. All elements of dimensioning of the first view will have values determined by the dimension style (for example, the height of the dimension text – 2.5 mm, the length of the arrows – 2.5 mm, etc.). The value of the second view dimensions will be automatically increased by 10 times (the height of the dimension text in the model space will be 25 mm, the length of the arrow – 25 mm), so that the dimensions are displayed correctly (the height of the dimension text – 2.5 mm, the length of the arrows – 2.5 mm, etc.) when this view is inserted on the layout.

When you change the *dimension scale*, the dimensions are not recalculated automatically.

To change any size of dimension scale, it is necessary to select it and select the required scale in the **Measurement scale** menu.

To set drawn dimensions to the current dimension scale, it is necessary to select the **Set to selection** command in **Measurement scale** menu and select the required dimensions on the drawing.

For more information on using scale, see «[Symbol scale and measurement scale](#)» («*TUNING NANOCAD*»).

Dimensioning with a single command



Menu: **Dimensions** –  **Auto**



Toolbar: **Utilities** – 



Command line: **D, DIMLINEAR, DLI**

1. You can set all dimensions in nanoCAD using a single command – **Auto** (the **Dimensions** menu) or **Dimensions** (on the **Utilities** and the **Dimensions** toolbars).

It is recommended to turn on the snap mode to maximise the facilities for dimensioning with a single command; set the required type of snap as a permanent snap. Switch on the **To turn the object snap “Nearest” on automatically during an insert of objects** option in the **nanoCAD – Options** dialogue box on the **Main** tab of the **Edit** section (the Tools menu – [Settings parameters](#)).

2. Start the **Auto** command and select any dimensions from the context menu by right clicking:

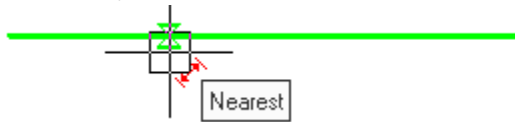


or by selecting the appropriate option in the command line:








Insert dimension [Auto /aLigned /Horizontal /Vertical /Radial /Diametral /BigRadius /arC/U-angUlar/Ordinate /CHain/Base/Properties /Grouped] :

3. You can also set dimensions on the drawing directly after starting the **Auto** command.

When you move the cursor over any graphic entity (segment, segment of polyline, arc or circle) it will be highlighted automatically:

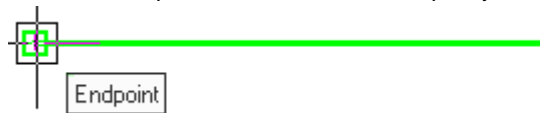


When the graphic entities are highlighted, nanoCAD displays the secondary symbols near the cursor that serve as prompts for the user. The secondary symbols indicate what dimension will be drawn if you left click on the graphic entity:







-  Aligned dimension.
-  Horizontal dimension.
-  Vertical dimension.
-  Diameter dimension.
-  Radius dimension.
-  Angle dimension.
-  Base dimension.

This method is used for dimensioning relating to a graphic primitive.

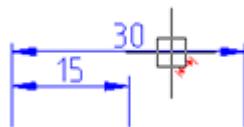
4. To quickly draw the dimensions that relate to a graphic primitive, you can turn off the object snap mode.
5. When you move cursor along the highlighted primitive, the corresponding snap markers are displayed at its characteristic points. You can use it to specify the initial points of the extension lines:



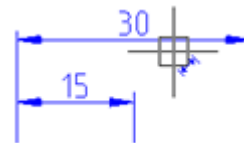
This method is used for dimensioning the elements of a drawing consisting of several graphic primitives.

6. nanoCAD allows you to place the dimensions at given distances from each other, not only in base dimensions, but also at linear dimensioning. The distance by which the new dimension should be spaced from the existing one, is specified by a *base-line spacing* option in the **Modify dimension style** dialogue box on the [Lines](#) tab. To do this, it is necessary to specify the position of the dimension line and slowly move the cursor from the existing dimension line. When you draw near a specified distance, the new dimension line will be “attracted” to the required position. The colour of the secondary character displayed near the cursor changes from red (,  or ) to blue (,  or ):

Dimension line is not at the specified distance



Dimension line is at the specified distance



7. At dimensioning, you can set the extension line oblique by holding the **CTRL** key and moving the cursor in the desired direction.
8. To change the position of the dimension text, hold the **SHIFT** key and move the cursor to the first or second extension line (by default, the dimension text is located in the centre of the dimension line).
9. During dimensioning, you can use the [Edit dimension](#) dialogue box to specify the required properties and options for the executable dimension. To do this, it is necessary to select the **Properties** option in the command line or context menu. The dimensioning command is not interrupted.
10. To finish dimensioning, press **ESC** or select **Cancel** from the context menu.
11. You can dimension chamfers and fillets during their creation. Turn on the *Measure chamfer* or *Measure fillet* mode in the [Chamfer](#) or [Fillet](#) dialogue box.

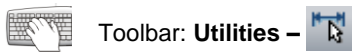
Linear dimensions

Horizontal, vertical and aligned dimensioning

In the following examples, horizontal, vertical and aligned dimensions are created by using the **Auto** command.



Menu: **Dimensions** –  **Auto**



Toolbar: **Utilities** –



Command line: **D, DIMLINEAR, DLI**

To dimension you can use also the **Horizontal**, **Vertical** and **Aligned dimension** commands.



Menu: **Dimensions** – **Horizontal**



Toolbar: **Utilities** –



Command line: **DIMHOR**



Menu: **Dimensions** – **Vertical**



Toolbar: **Utilities** –



Command line: **DIMVER**



Menu: **Dimensions** – **Aligned dimension**



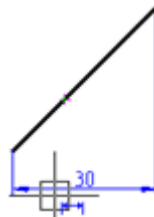
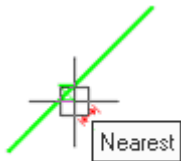
Toolbar: **Utilities** –



Command line: **DAL, DIMALIGNED**

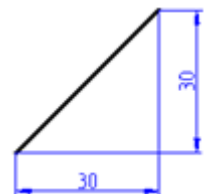
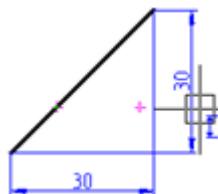
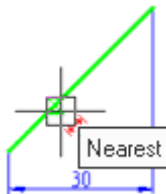
To specify the horizontal dimension of the line:

- Start the **Auto** command.
- Place the cursor over the line to show its dynamic highlighting and display the character . Left click to confirm the dimensioning:
- Move the cursor down to change the character to :
- Left click to set the position of the dimension line:



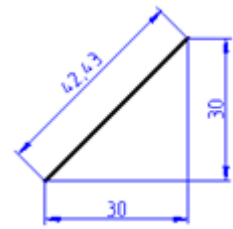
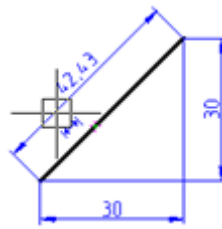
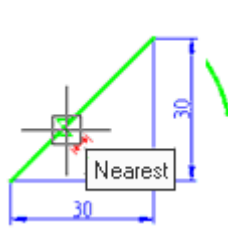
To specify the vertical dimension of the line:

- Place the cursor over the line to show its dynamic highlighting and display the character . Left click to confirm the dimensioning:
- Move the cursor to the right to change the character to :
- Left click to set the position of the dimension line:



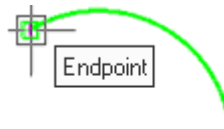
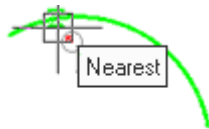
To specify the aligned dimension of the line:

- Place the cursor over the line to show its dynamic highlighting and display the character . Left click to confirm the dimensioning:
- Move the cursor to the left to change the character to :
- Left click to set the position of the dimension line:

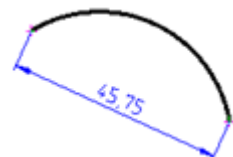
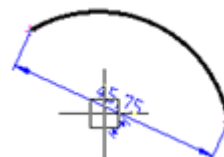
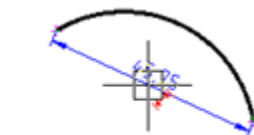


For dimensioning the arc length using characteristic points:

- Start the **Auto** command.
- Place the cursor over the arc to show its dynamic highlighting:
- Move the cursor near to the endpoint of the arc. When the snap marker appears, left click to choose the endpoint of the first extension line of the dimension:
- Move the cursor to the other endpoint of the arc and left click to choose the endpoint of the second extension line of the dimension:

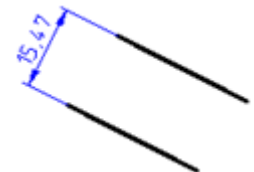
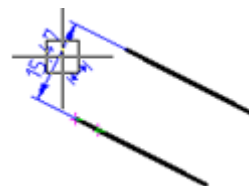
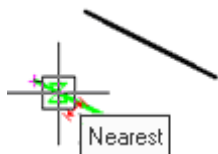


- Move the cursor to the middle of the arc to display the character π :
- Move the cursor down and to the left to change the character to $\frac{\pi}{2}$:
- Left click to set the position of the dimension line:



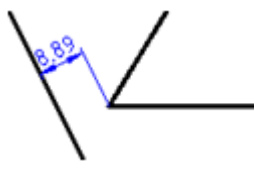
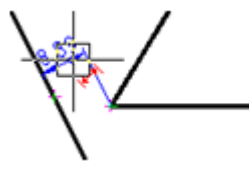
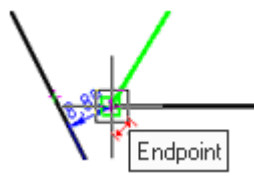
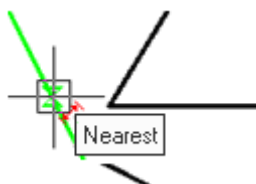
For dimensioning between two parallel line segments:

- Start the **Auto** command.
- Select the lower line segment by highlighting it and left click:
- Select the top line segment by highlighting it and left click when the horizontal character appears:
- Move the cursor up and to the left to change the character to $\frac{\pi}{2}$:
- Left click to set the position of the dimension line:



To draw the dimension from a point to a line segment:

- Start the **Auto** command.
- Select the lower line segment by highlighting it and left click::
- Select the endpoint of the second line segment using the snap:
- Move the cursor up and to the left:
- Left click to set the position of the dimension line:



Ordinate dimensioning



Menu: **Dimensions** –  **Ordinate**



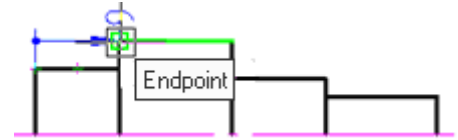
Toolbar: **Utilities** –



Command line: **DIMORDINATE**

To specify the chain of linear ordinate dimensions:

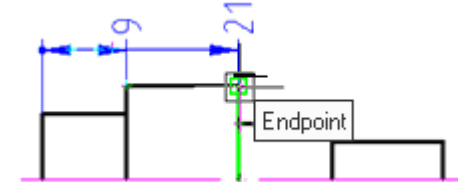
- Start the **Auto** command.
- Select the **Ordinate** option in the command line or context menu.
- Specify the first point of the first dimension:
- Specify the second point of first dimension:



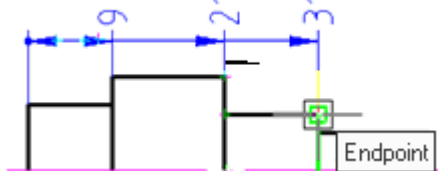
- Specify the dimension line location:



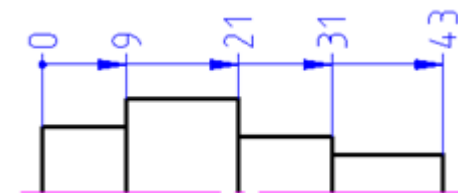
- Select the **Base** option in the command line or context menu and specify the endpoint of the second ordinate dimension:



- Specify all the endpoints of the other ordinate dimensions:

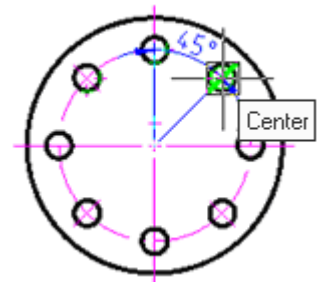
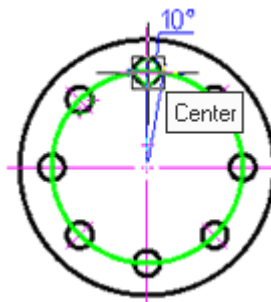
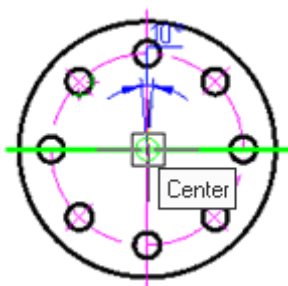


- Press **ENTER** to finish the command:



To specify the chain of angular ordinate dimensions:

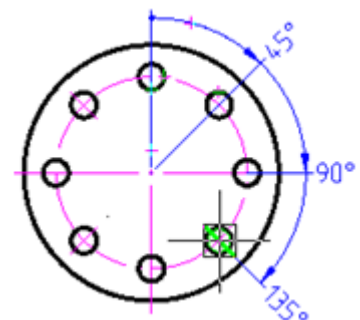
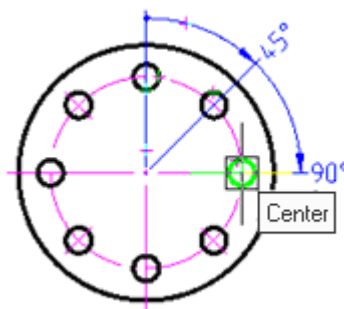
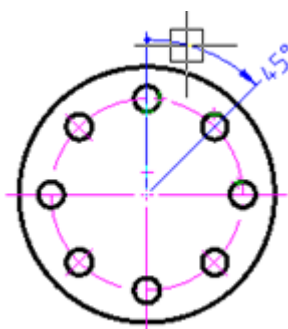
- Start the **Auto** command.
- Select the **angular** option in the command line or context menu.
- Specify the first point of the angular dimension (vertex of angle):
- Specify the second point of the angular dimension:
- Specify the third point of the angular dimension:



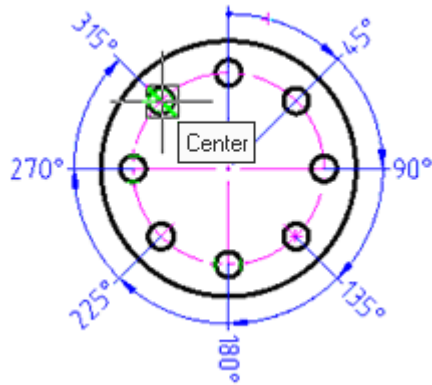
- Select the **Ordinate** option in the context menu and specify the dimension line location:

- Select the **CHain** option in the command line or context menu and specify the endpoint of the second dimension:

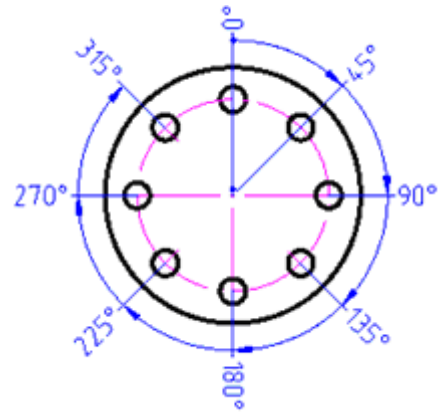
- Specify the endpoint of the third dimension:



- Specify the endpoint of the last dimension:



- Press **ENTER** to finish the command:



Group dimensioning



Menu: **Dimensions** – **Group dimension**



Toolbar: **Utilities** –

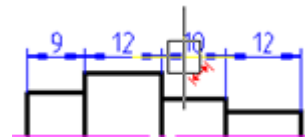


Command line: **QDIM**

The **Group dimension** command is particularly useful for creating a series of vertical or horizontal dimensions.

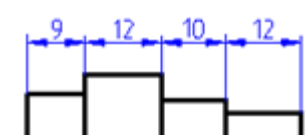
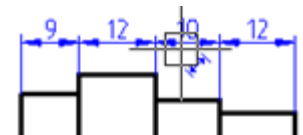
To specify the group dimensions:

- Start the **Auto** command.
- Select the **Grouped** option in the command line or context menu.
- Select the objects you want to dimension:
 - Press **ENTER** to end the selection of objects:

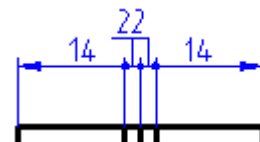
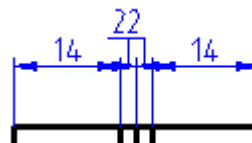
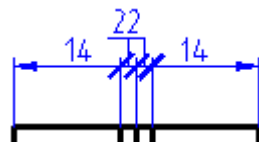


Depending on the direction of the cursor movement (vertical or horizontal), the chain of vertical or horizontal dimensions will be dynamically displayed.

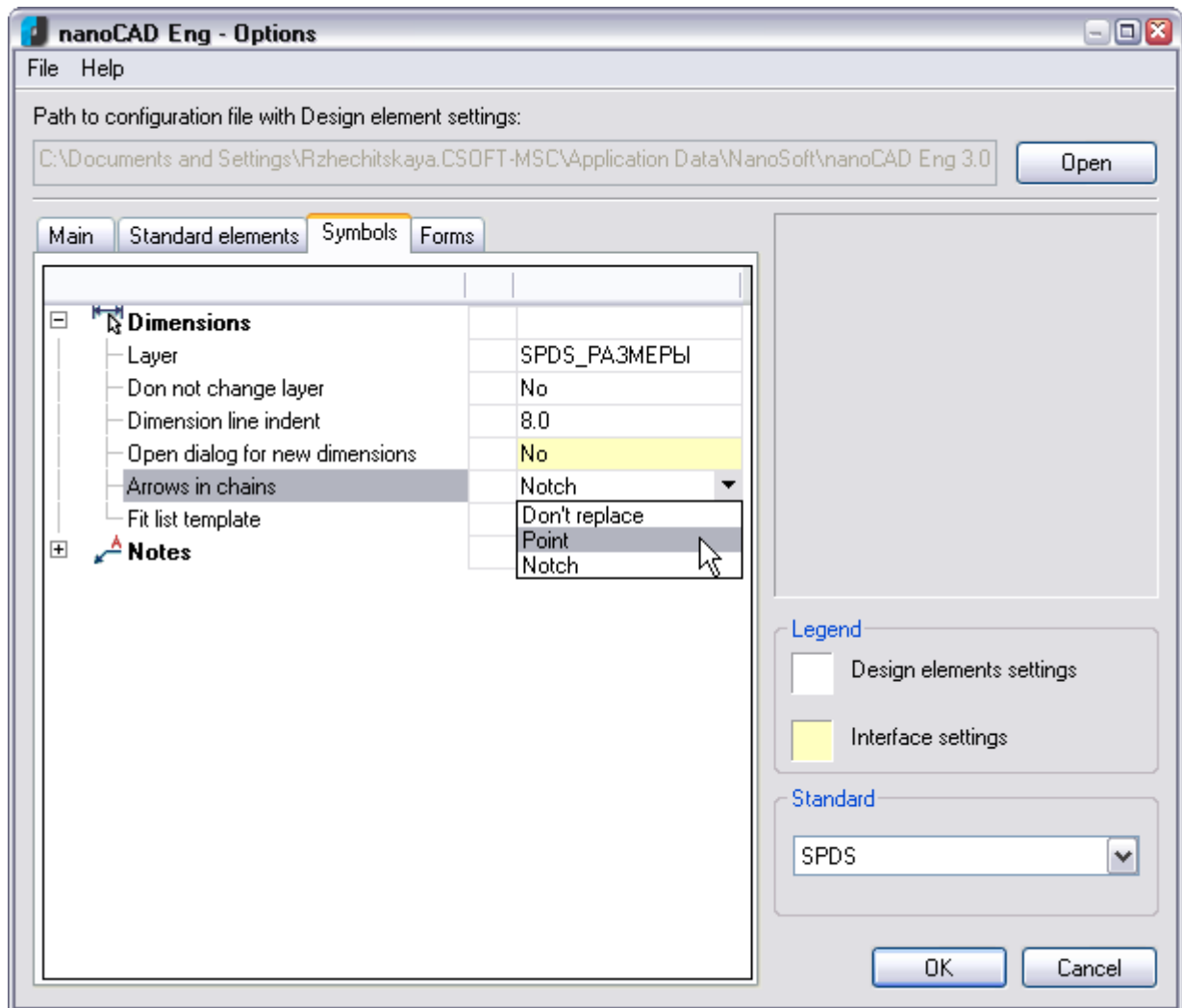
- Move the cursor to change the character from red to blue to blue :
- Left click to specify the location of the dimension lines:



When dimensioning the objects that have a small length, the crossed arrows are automatically replaced by notches or points:



The setting for replacement of intersecting arrows is performed in the **nanoCAD – Options** dialogue box (The **Tools** menu - [Settings Parameters](#)):



Base dimensioning



Menu: **Dimensions** –  **Base dimension**



Toolbar: **Utilities** – 



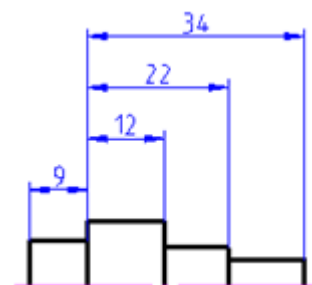
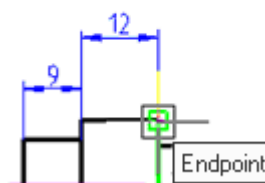
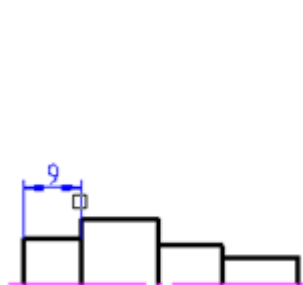
Command line: **DBA, DIMBASELINE**

Base dimensions are a sequence of dimensions measured from a reference point.

A *base dimension* creates a linear, angular or ordinate dimension from the baseline of the previous or selected dimension.

To draw the base dimensions:

- Start the **Auto** command.
- Select the **Base** option in the command line or context menu.
- Select the extension line of the previous dimension as a base:
- Specify the endpoint of the first dimension:
- Specify the endpoints for all the other dimensions and press **ENTER** to finish the command:



The default spacing between the baseline dimensions can be set from the **Modify dimension Style**, [Lines](#) tab, **Baseline Spacing**.

Chain dimensioning



Menu: **Dimensions** –  **Dimensions chain**



Toolbar: **Utilities** – 



Command line: **DCO, DIMCONTINUE**

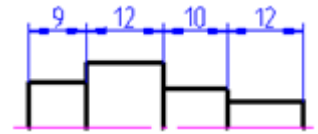
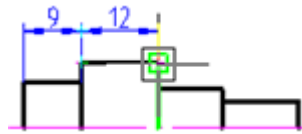
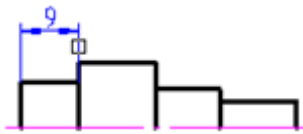
The *dimensions chain* creates a dimension that starts from an extension line of a previously created dimension.

The *dimensions chain* can be a linear, angular or ordinate.

To draw the *dimensions chain*, the object must have least one linear, angular or ordinate dimension.

To draw the dimensions chain:

- Start the **Auto** command.
- Select the **CHain** option in the command line or context menu.
- Select the previously created dimension as the base:
- Specify the endpoint of the first dimension:
- Specify the endpoints for all the other dimensions and press **ENTER** to finish the command:



Radial dimensions

Diametrical dimensioning



Menu: **Dimensions** –  **Diameter dimension**




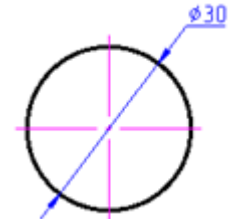
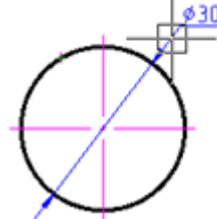
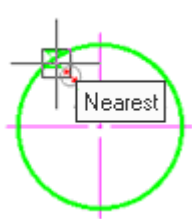
Toolbar: **Utilities** – 




Command line: **DIMDIAMETER**

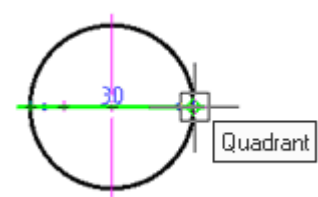
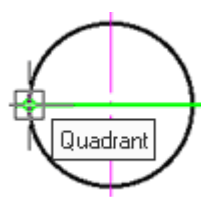
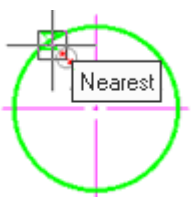
For dimensioning the diameter of a circle:

- Start the **Auto** command.
- Place the cursor over the circle to show its dynamic highlighting and display the character . Left click to confirm the dimensioning:
- Choose the location of the dimension:
- Left click to fix the chosen location of the dimension:

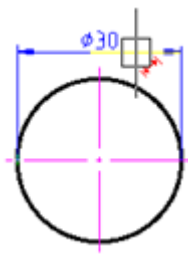


For dimensioning the diameter of a circle using characteristic points:

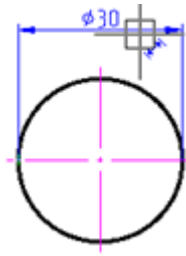
- Turn on the **Quadrant** snap.
- Start the **Auto** command.
- Place the cursor over the circle to show its dynamic highlighting and display the character . Left click to confirm the dimensioning:
- Move the cursor to the first characteristic point of the circle and left click to specify the endpoint of the first extension line of the dimension:
- Move the cursor to the second characteristic point of the circle and left click to specify the endpoint of the second extension line of the dimension:



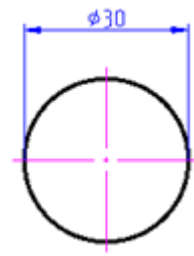
- Move the cursor up to display the character



- Move the cursor up to change the character from red to blue:



- Left click to set the location of the dimension line:



Radial dimensioning



Menu: **Dimensions** –  **Radius dimension**



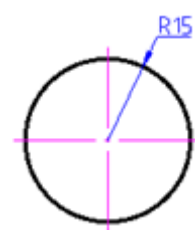
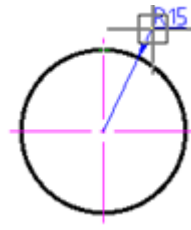
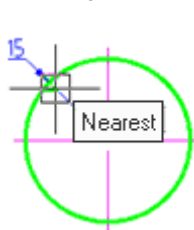
Toolbar: **Utilities** – 




Command line: **DIMRADIUS, DRA**

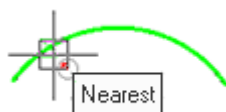
To draw the radius of a circle:

- Start the **Auto** command.
- Select the **Radial** option in the command line or context menu.
- Place the cursor over the circle to show its dynamic highlighting. Left click to confirm the dimensioning:
- Choose the location of the dimension:
- Left click to fix the chosen location of the dimension:

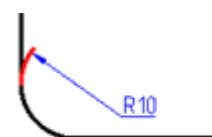
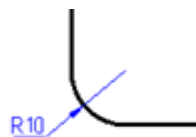
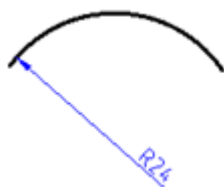


To draw the arc radius:

- Start the **Auto** command.
- Place the cursor over the arc to show its dynamic highlighting and display the character . Left click to confirm the dimensioning:
- Choose the location of the dimension:
- Left click to fix the chosen location of the dimension:



Options for dimensioning the arc radius:



Big radius dimensioning



Menu: **Dimensions** –  **Big radius**



Toolbar: **Utilities** – 

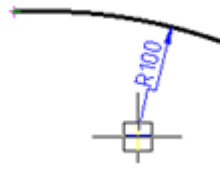
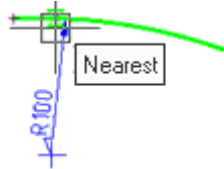


Command line: **DIMJOGGED, DJO**

The *big radius* command creates jogged dimensions for circles and arcs.

To draw the big radius:

- Start the **Auto** command.
- Select the **BigRadius** option in the command line or context menu.
- Place the cursor over the arc to show its dynamic highlighting. Left click to confirm the dimensioning:
- Choose the location of the dimension:
- Left click to fix the chosen location of the dimension:



Angular dimensions



Menu: **Dimensions** –  **Angle dimension**





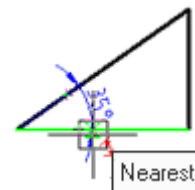
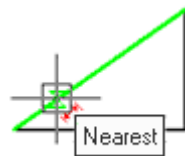
Toolbar: **Utilities** – 



Command line: **DAN, DIMANGULAR**

To draw the angle between two segments:

- Start the **Auto** command.
- Place the cursor over one of the segments to show its dynamic highlighting and display the character . Left click to confirm the dimensioning:
- Place the cursor over the second segment to show its dynamic highlighting and display the character . Left click to confirm the dimensioning:

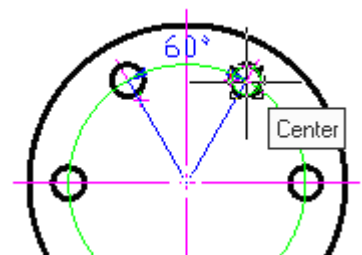
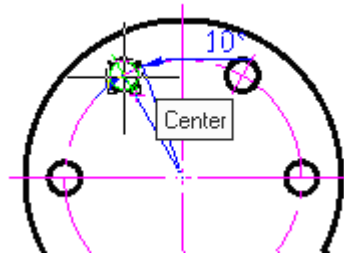
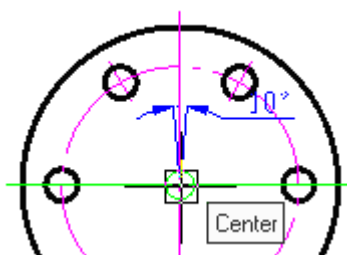


- Choose the location of the dimension:
- Left click to set the location of the dimension line:

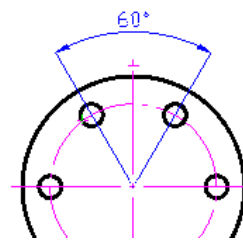
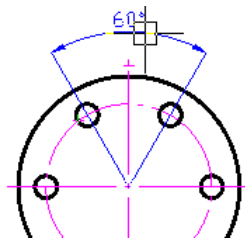


For dimensioning the angular dimension using characteristic points:

- Start the **Auto** command.
- Select the **anGular** option in the command line or context menu.
- Specify the first point of the angular dimension (vertex of angle):
- Specify the second point of the angular dimension:
- Specify the third point of the angular dimension:



- Specify the location of the dimension line:
- Press **ENTER** to finish the command:



Arc length



Menu: **Dimensions** –  **Arc**



Toolbar: **Utilities** – 



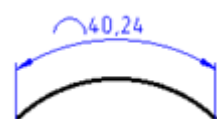
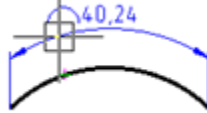
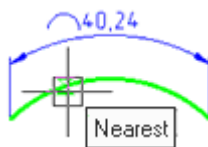
Command line: **DAR, DIMARC**

This command creates an arc length dimension. Arc length dimensions measure the distance along an arc or polyline arc segment.

To differentiate between the arc dimensions and the linear or angular dimensions, the arc symbol is displayed above the dimension text by default.

For dimensioning the arc length dimension:

- Start the **Auto** command.
- Select the **arC** option in the command line or context menu.
- Place the cursor over the arc to show its dynamic highlighting. Left click to confirm the dimensioning:
- Choose the location of the dimension:
- Left click to fix the chosen location of the dimension:



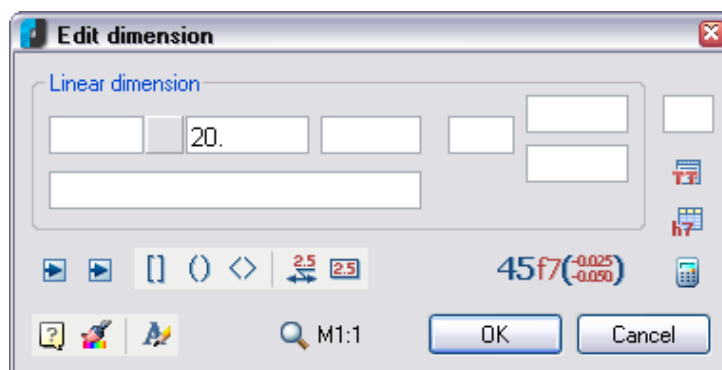
Dimensions editing

You can edit the existing dimensions using the [Inspector](#) window or the **Edit dimension** dialogue box.

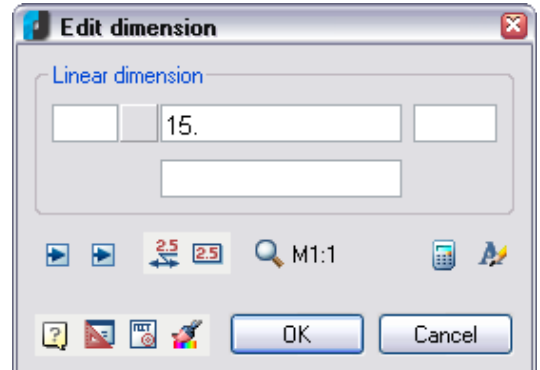
The content of the **Inspector** window depends on the type of dimension selected.

The form of the **Edit dimension** dialogue box is different for different dimension styles. For the *Standard* and *SPDS* dimension styles, the **Edit dimension** dialogue box will have a more simplified form than for the *ESKD* dimension style.

ESKD dimension style



Standard and *SPDS* dimension styles

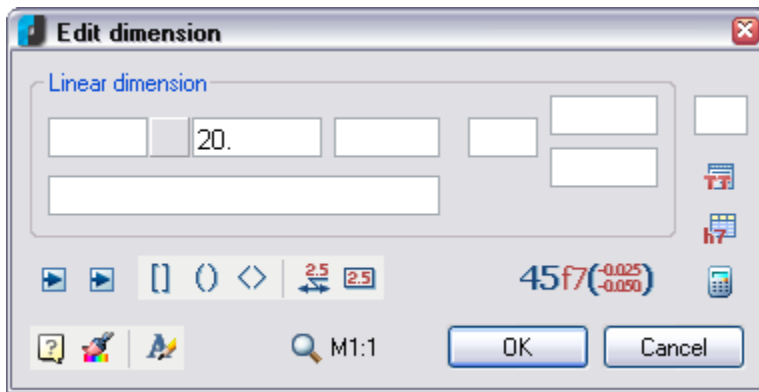


The options and input fields, coupled with the fit settings and buttons for the **Technical conditions**, **Fit table**, **Square brackets**, **Round brackets**, **Pointed brackets** are not included in the **Edit dimensions** dialogue box for the *Standard* and *SPDS* dimension styles.

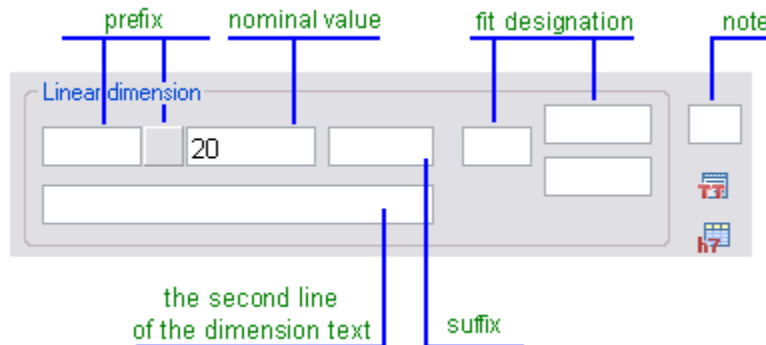
Below is a description of the **Edit dimension** dialogue box for the *ESKD* dimension style:

You can open the **Edit dimension** dialogue box in different ways:

- Double click on the dimension.
- Place the cursor over the dimension and press the right mouse button.
- Select the dimension, press the right mouse button and select the **Edit** command from the context menu.



The structure of the input fields for the dimension text:




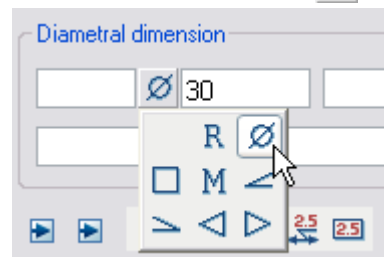
Options:

Dimension type In this section the dimension type (for example, *Linear dimension*, *Diametral dimension*, *Angular dimension* etc.) and values of the dimension text are displayed.

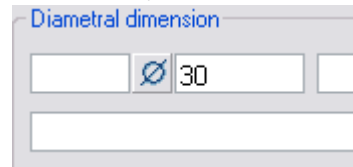
Prefix

The prefix consists of the text input field and the *Symbol* button.

If the dimension does not have a special symbol that is set as the default prefix, the button is displayed without an image: . Click the icon to open the panel to select a symbol:

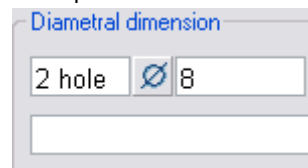


If a special symbol from the panel is set by default, it appears on the button:



A prefix specified in the **Edit dimension** dialogue box has precedence over a prefix set by default.

Example of dimension text with a prefix consisting of the text and special symbol:



Nominal value Field to display and edit the nominal value of the dimension text.

Suffix This field displays the suffix of the dimension text set by default, such as the chamfer angle

designation:

Linear dimension

In the same field, you can set the value of a custom symmetrical fit of the dimension:

Linear dimension

☒

Fit designation

The fields display the specified fit values of the dimension.

Depending on the way the fits are written (the **Fit view** button), the values in the fields can be displayed in different ways:

Diametral dimension

☒

Diametral dimension

☒

Diametral dimension

☒

Note

The input field for a note for the dimension is used to create the hyperlink to the item that contains the technical conditions that define the general requirements for several dimensions. For example, in this field you can type the star symbol (*) to denote the reference dimension (having the corresponding item in the technical conditions).

The second line of the dimension text

An example displaying the dimension text consisting of two lines:

Linear dimension

Buttons



These buttons are used to change the arrow type.
Click to open the panel and select the required arrow type:



These buttons are used to turn on/off the modes for placing text in the *Square*, *Round* or *Pointed* brackets.

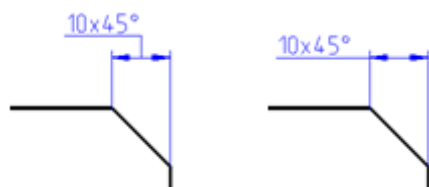


This button is used to turn on/off the mode for placing the text on the leader.

Example:

Mode is on

Mode is off

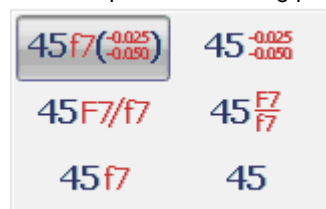


This button is used to turn on/off the mode for placing the text in a rectangle.



Use this button to choose the method of writing the fit.

Click to open the following panel:



Opens the **Technical conditions** dialogue box.



Opens the **Fits** dialogue box.



Opens the [Calculator](#).



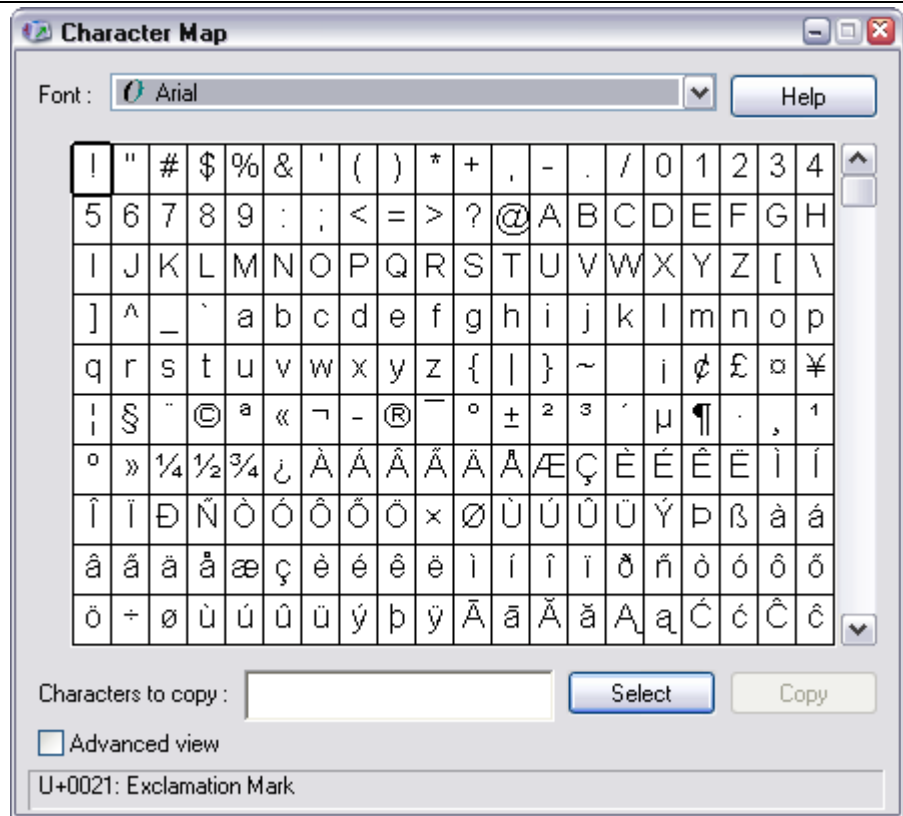
This button opens the **Text settings** dialogue box to change the style, height and colour of the dimension text.



The **Match Properties** button temporarily closes the **Edit dimension** dialogue box to select the dimension whose properties should be copied to the editable dimension.

Please note that context menus, which include the following commands, are available in the input fields of the dimension text :

| | |
|---|--|
| <div>History ▶</div> <div>Recent ▶</div> | <p>These commands are used to collect inputted information and insert it into the input fields if necessary.</p> |
| <div>Superscript Ctrl+Up</div> <div>Subscript Ctrl+Down</div> <div>Insert division</div> <div>Pick from drawing</div> <div>Create hyperlink...</div> <div>Symbols ▶</div> | <p>The functions of the first three commands of this section are as their names suggest.</p> <p>The Pick from drawing command temporarily closes the Edit dimension dialogue box and opens the Value picker dialogue box to get different objects properties from the drawing. You can then insert them into the Edit dimension dialogue box.</p> <p>The Symbols command allows you insert various symbols into the input fields, including characters from the Windows table.</p> |



| | |
|------------|--------|
| Undo | Ctrl+Z |
| Cut | Ctrl+X |
| Copy | Ctrl+C |
| Paste | Ctrl+V |
| Delete | Del |
| Select all | Ctrl+A |

The commands from this section allow you to perform operations using the clipboard.

☒ 0
☐ 0.0
☐ 0.00
☐ 0.000

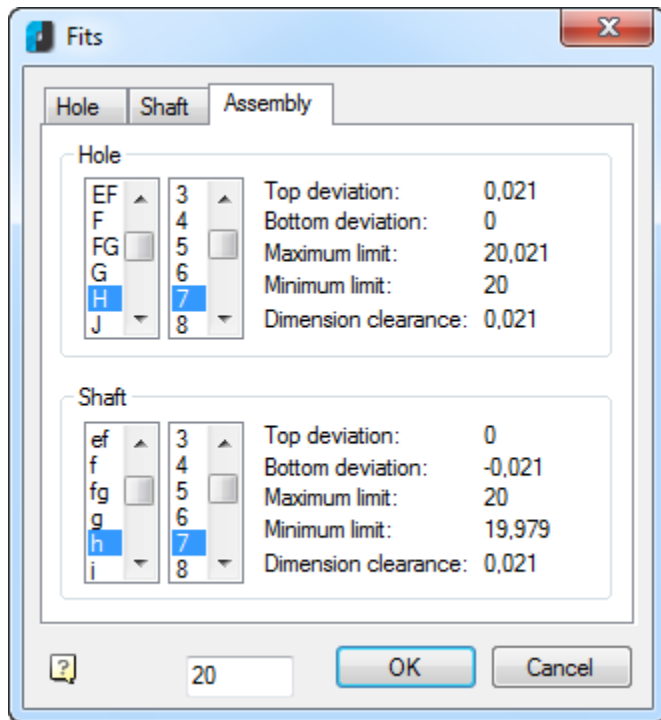
Sets the precision of the nominal dimension.

To set the dimension fit:

- Click the button
- In the panel that appears, select the method of writing the tolerances:



- Click the button
- In the opened **Fits** dialogue box, select the required values:



Break and Restore dimensions

If necessary, you can use breaks to improve the readability of the graphical information and exclude errors in the dimensions on the drawing. The **Break dimension** command, unlike the [Explode](#) command from the **Modify** menu, does not break the completeness of the dimension and does not result in a loss of associativity.

Essentially, the **Break dimension** command does not remove the dimension part, but places a mask on it.

To break the dimension (or rather, to place the mask), it is necessary to select two points on the dimension or extension line that define the location and length of the mask.

When you edit the dimension or an object that intersects dimension, the dimension mask (break) will not be updated automatically. So after moving the dimension or editing an object that intersects dimension, you should first restore the dimension and then add a dimension mask (break) again.

Dimension Break

Menu: **Dimensions** – **Break dimension**

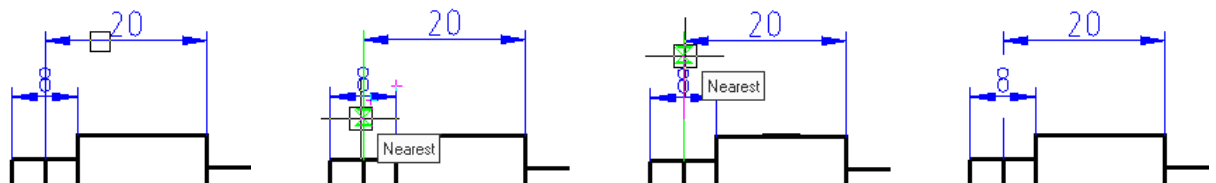
Toolbar: **Utilities** –

Command line: **DIMBREAK**

Creates a break in the dimension. When you create a break, turn on the **Nearest** snap (or switch on the **To turn the object snap “Nearest” on automatically during an insert of objects** option in the **nanoCAD – Options** dialogue box on the **Main** tab of the **Edit** section (the Tools menu – [Settings parameters](#)).

Options:

Unbreak Removes the breaks in the selected dimension.



The following prompts are displayed:

Select dimension to break:

Select the dimension.

Select first point [**Unbreak**]:

Select the first point.

Select second point [**Unbreak**]:

Select the second point.

When you select a dimension, the **Break line** command will be available from context menu.

Dimension Restore



Menu: **Dimensions** –  **Restore dimension**



Toolbar: **Utilities** – 



Command line: **DIMUNBREAK**

Removes the dimension breaks created by the [Break dimension](#) command.

The following prompts are displayed:

Select dimensions to unbreak:

*Select the dimensions and press **ENTER**.*

When you select dimension, the **Unbreak lines** command will be available from context menu.

Explode dimensions

In some cases, it is necessary to explode a dimension into separate parts – lines, arrows, arcs and dimension text. To perform this operation, use the [Explode](#) command from the **Modify** menu.

It is strongly recommended not to explode the dimensions unless there is a specific need.

Dimension styles



Menu: **Format** –  **Dimensions styles...**



Menu: **Dimensions** –  **Dimensions styles...**



Toolbar: **Styles** – 

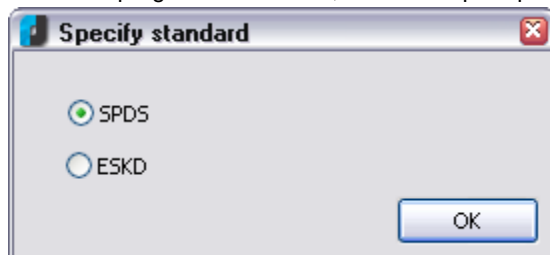


Command line: **DIMSTYLE**

A dimension style is a named collection of dimension settings that control the appearance of dimensions, such as arrowhead style, text location and lateral tolerances.

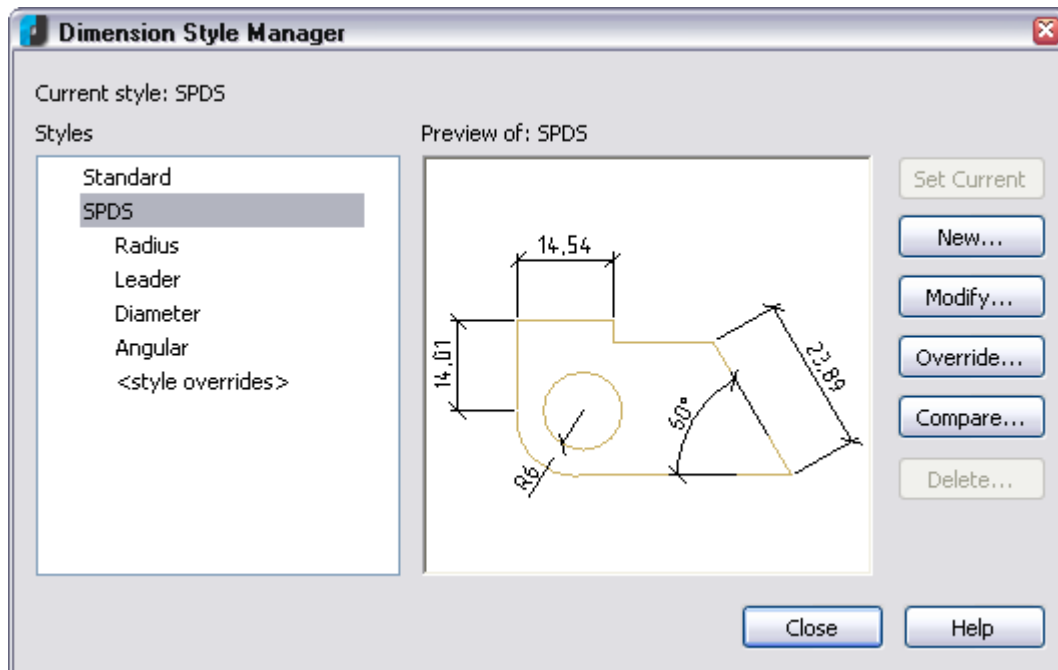
nanoCAD is supplied with two dimension styles: ESKD and SPDS, corresponding to the requirements of the Unified Design Documentation System and Construction Design and Estimate Documentation.

When the program is installed, the user is prompted to choose which style will be used by default:



If necessary, you can change the default style in the **Main** tab of the **nanoCAD – Options** dialogue box (the **Tools** menu – the **Settings Parameters** command).

Controlling of the dimension styles is carried out in the **Dimension Style Manager** dialogue box:



Options:

Current style: Displays the name of the current dimension style .

Styles: Displays a list of the dimension styles in the drawing

Preview of: Shows a graphic representation of the style selected in the **Styles** list.

Set Current

Sets the style selected under Styles as the current one.

New...

Displays the **Creating style** dialogue box where you can define a new dimension style.

Modify...

Changes the options for the style selected in the list.

Override...

Changes the options for the current dimension style.

Compare...

Compares the properties of two dimension styles.


Delete...

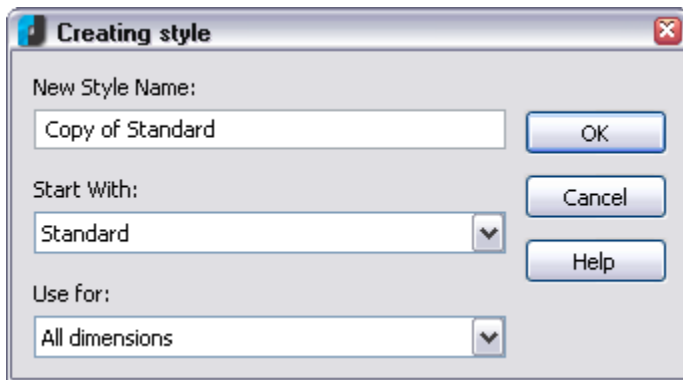
Deletes the selected dimension style.

To set the current dimension style:

- Select the required style from the **Styles** list.
- Click the button .

To create a new dimension style:

- In the **Styles** list, select the style from which you want to create a new one.
- Click the button .
- In the **Creating style** dialogue box, type a name for the new style:



Options:

New style name:

Specifies the new dimension style name.

Start with:

Sets the style to use as a basis for the new one.

Use for:


Creates a *dimension substyle* that applies only to specific dimension types.

The following *substyles* are available:



- All dimensions**
- Linear dimensions**
- Angular dimensions**
- Diametral dimensions**
- Radial dimensions**
- Ordinate dimensions**
- Leader dimensions**

- Click **OK**.
- In the **Modify dimension style** dialogue box that appears, set the required options for the new dimension style.


To modify a dimension style:

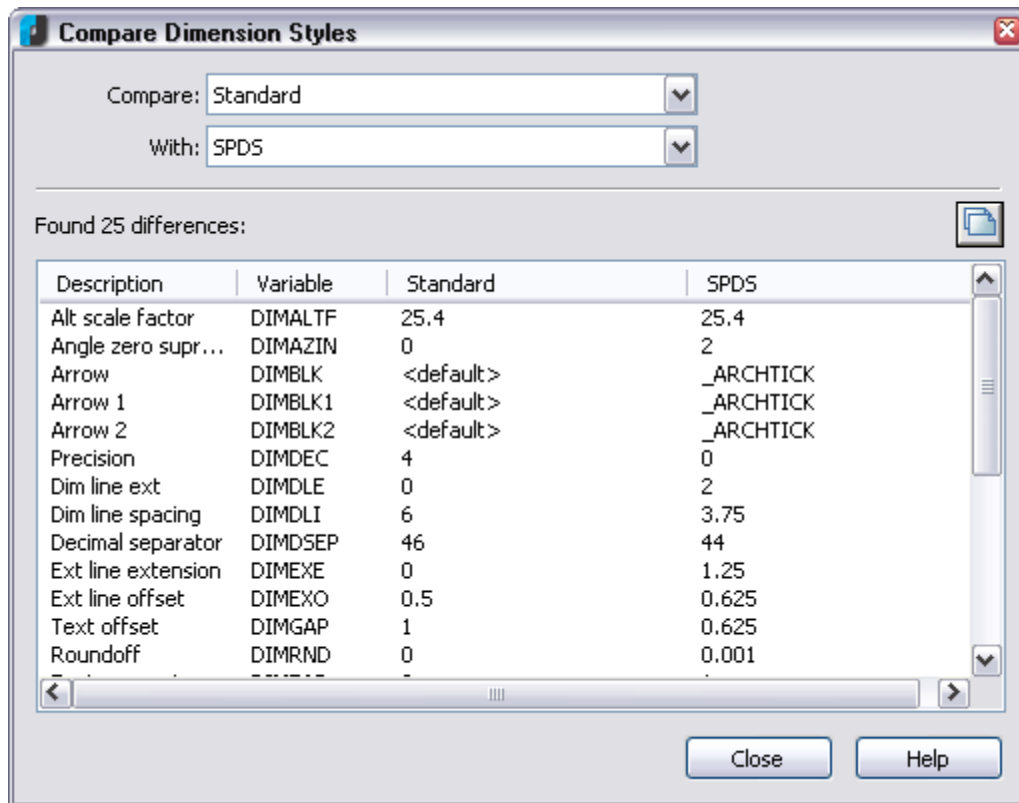
- In the **Styles** list, select the required style.
- Click the button .
- In the **Modify dimension style** dialogue box that appears, set the new options for the dimension style to be modified.

To change the options for the current dimension style:

- In the **Styles** list, select the current dimension style (if you select any other style, the **Override** button  will not be available).
- Click the button .
- In the **Modify dimension style** dialogue box that appears, override the options for the current dimension style.

To compare two dimension styles:

- In the **Styles** list, select the first dimension style which you want to compare.
- Click the button .
- In the **Compare Dimension Styles** dialogue box, from the **With** list select the second dimension style for comparing:



Options:

Compare: Drop-down list to select the first dimension style for comparing.

With: Drop-down list to select the second dimension style for comparing.

Found differences:

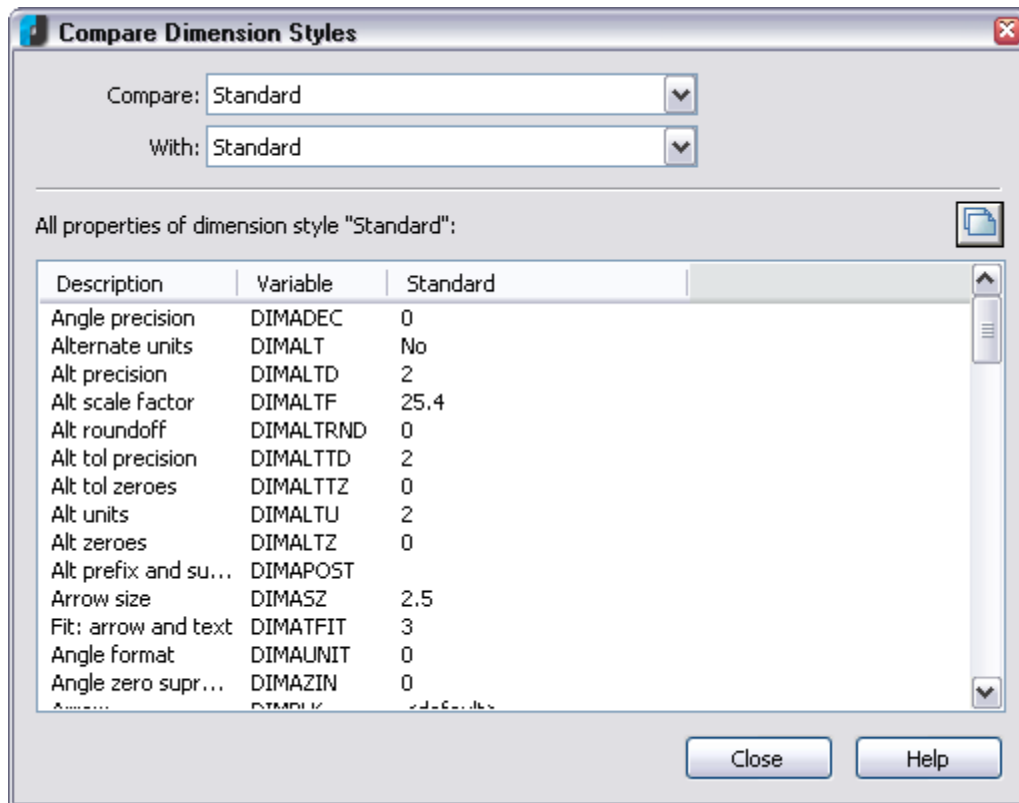
Description List of the properties of the compared dimension styles.

Variable List of the dimension variables that define the compared properties.




Use this button to copy the comparison results to the clipboard.

The **Compare Dimension Styles** dialogue box can be used to view a list of all the properties of any dimension style. To do this, select the same dimension style in the **Compare** and **With** lists:



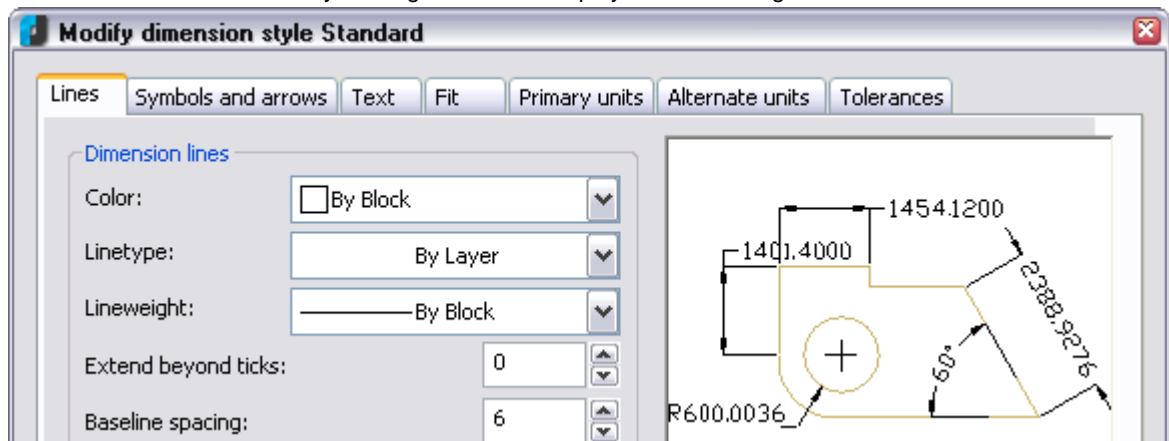
To delete a dimension style:

- In the **Styles** list, select the style that you want to delete.
- Click the button .

Modify a dimension style

The properties of new dimensions are set and options for existing dimension styles are modified in the **Modify dimension style** dialogue box.

The name of the dimension style being modified is displayed in the dialogue box title:



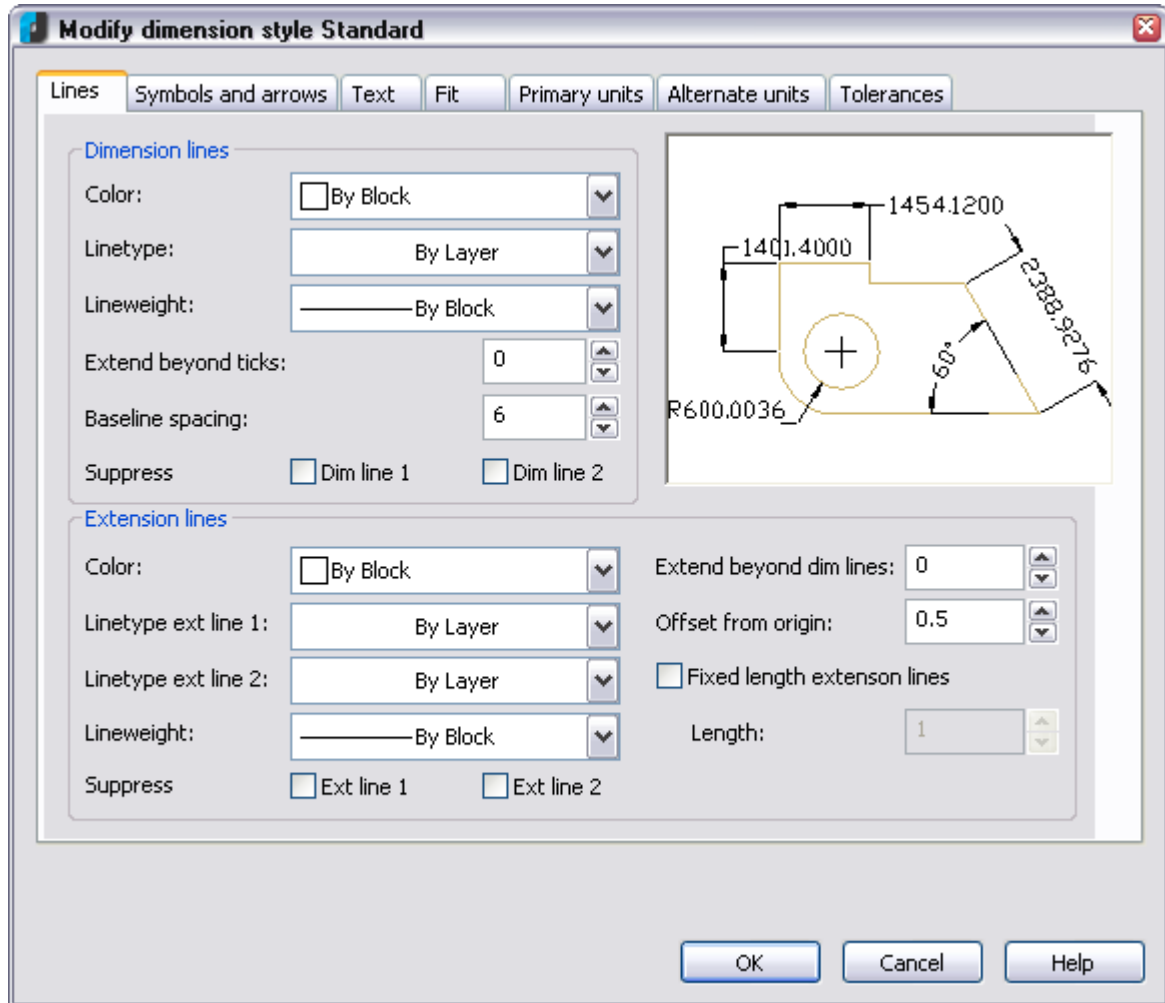
The **Modify dimension style** dialogue box contains the following tabs:

- [Lines](#)
- [Symbols and arrows](#)
- [Text](#)
- [Fit](#)
- [Primary units](#)
- [Alternate units](#)
- [Tolerance](#)

The window in the upper right corner of each tab displays a graphical preview of the properties of the dimension style being modified.

The Lines tab

Sets the properties of dimension lines and extension lines.



Options:

Dimension lines

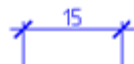
Color: Displays and sets the colour for the dimension line.

Linetype: Sets the linetype of the dimension line.

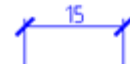
Lineweight: Sets the lineweight of the dimension line.

Extend beyond ticks: Specifies the distance to extend the dimension line past the extension line when you use ticks and no marks for arrowheads.

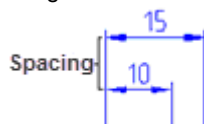
Extend beyond ticks: 2



Extend beyond ticks: 0

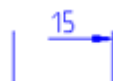


Baseline spacing: Sets the spacing between the dimension lines of a baseline dimension.



Suppress: Suppresses the display of dimension lines.

Dim line 1, Dim line 2 Dim Line 1 suppresses the first dimension line



Dim Line 2 suppresses the second dimension line



Extension lines

Color: Sets the colour for the extension lines.

Linetype ext line 1: Sets the linetype of the first extension line.

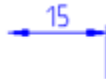
Linetype ext line 2: Sets the linetype of the second extension line.

Lineweight: Sets the lineweight of the extension line.

Suppress: Suppresses the display of extension lines.

Ext line 1, Ext line 2 Ext Line 1 suppresses the first extension line

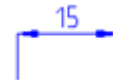
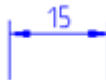
Ext Line 2 suppresses the second extension line



Extend beyond dim lines: Specifies the distance to extend the extension lines above the dimension line.

Extend beyond dimension line: 1.25

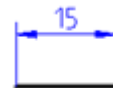
Extend beyond dimension line: 0



Offset from origin: Sets the distance to offset the extension lines from the points on the drawing (object) that define the dimension.

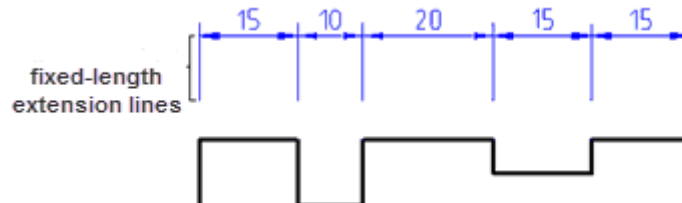
Offset from object: 0.625

Offset from object: 0



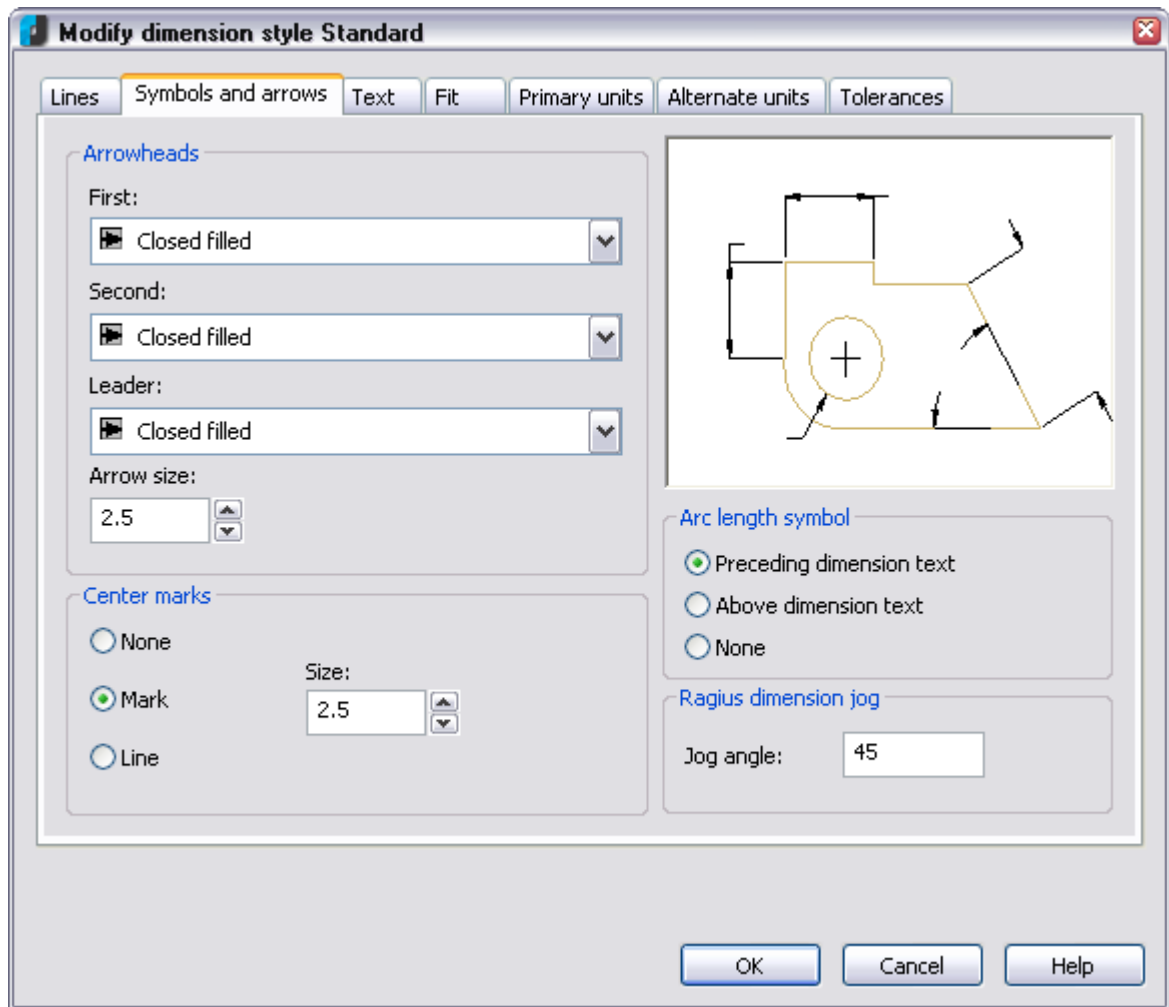
Fixed Length Extension Lines: Enables fixed length extension lines.

Length: Sets the total length of the extension lines.



The Symbols and arrows tab

Sets the format and placement for arrowheads, centre marks, arc length symbols and jogged radius dimensions.



Options:

Arrowheads

- First:** Sets the arrowhead for the first dimension line.
When you change the first arrowhead type, the second arrowhead automatically changes to match it.
- Second:** Sets the arrowhead for the second dimension line.
When you change the second arrowhead type, the first arrowhead does not automatically change to match it.
- Leader:** Sets the arrowhead for the leader line.
- Arrow size:** Displays and sets the size of arrowheads.

Center marks

- None** Creates no centre mark or centreline.
- Mark** Creates a centre mark.
- Line** Creates a centreline.
- Size:** Displays and sets the size of the centre mark or centreline.

Arc length symbol

- Preceding Dimension Text** Places arc length symbols before the dimension text.
- Above Dimension Text** Places arc length symbols above the dimension text.

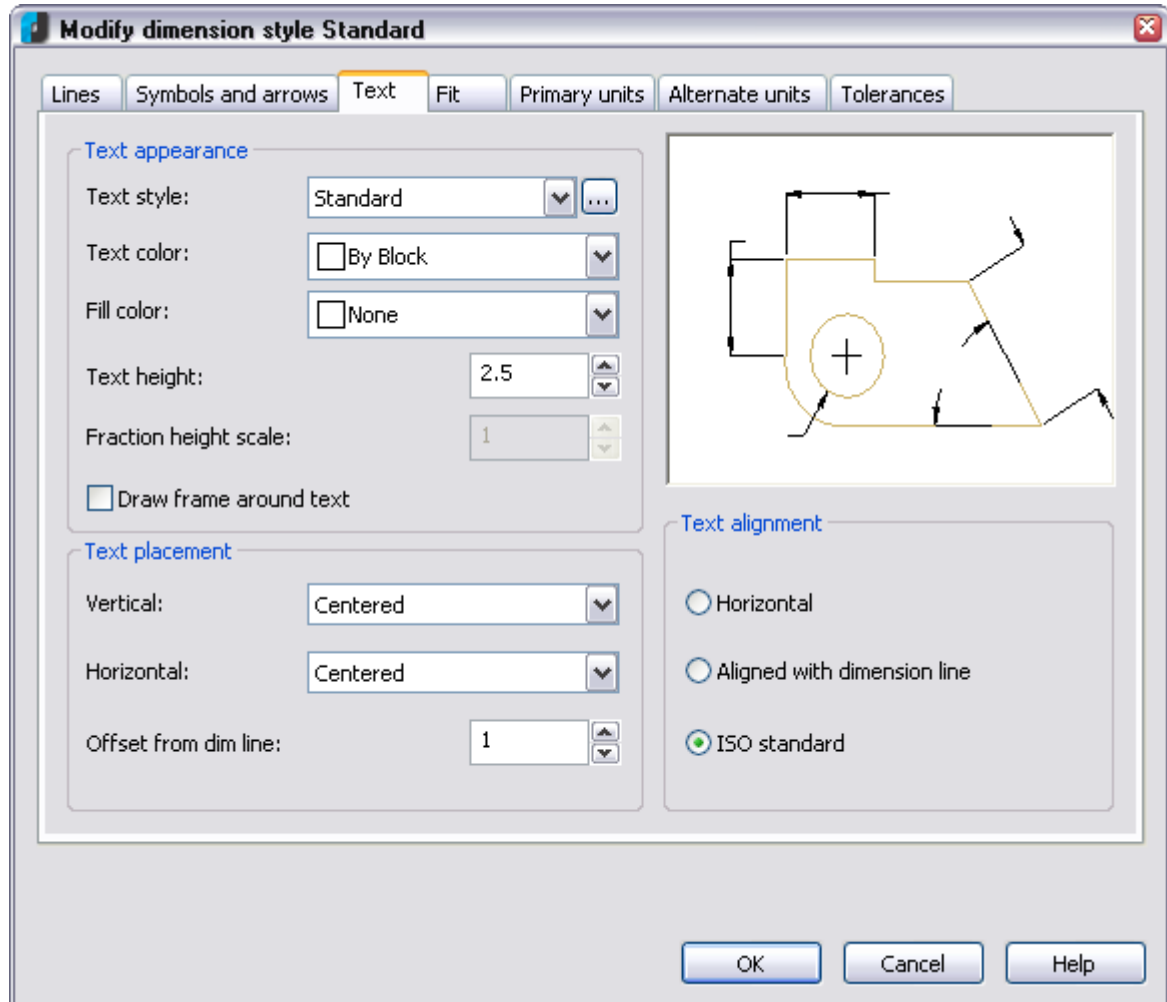
None Suppresses the display of arc length symbols.

[Radius dimension jog](#)

Jog angle: Determines the angle of the transverse segment of the dimension line in a jogged radius dimension.

The Text tab

Sets the format, placement, and alignment of dimension text:



Options:

[Text appearance](#)

Text style: Lists the available text styles.



Displays the [Text styles](#) dialogue box where you can create or modify text styles.

Text color: Sets the colour for the dimension text.

Fill color: Sets the colour for the text background in dimensions.

Text height: Sets the height of the current dimension text style.
If a fixed text height is set in the Text Style (that is, the text style height is greater than 0), that height overrides the text height set here.

Fraction height scale: Sets the scale of fractions relative to the dimension text.
This option is available only when **Fractional** is selected as the **Unit Format** on the [Primary Units](#) tab. The value entered here is multiplied by the text height to determine the height of dimension fractions relative to dimension text.

Draw frame When selected, draws a frame around the dimension text.

around text

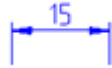
Text placement

Vertical:

Controls the vertical placement of the dimension text in relation to the dimension line:



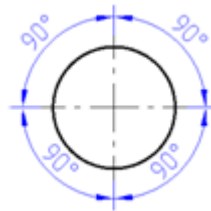
Centered – Centres the dimension text between the two parts of the dimension line.



Above - Places the dimension text above the dimension line. The distance from the dimension line to the baseline of the lowest line of text is the current text gap.



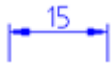
Outside - Places the dimension text on the side of the dimension line farthest away from the first defining point.



JIS - Places the dimension text to conform to a Japanese Industrial Standards (JIS) representation.

Horizontal:

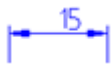
Controls the horizontal placement of the dimension text along the dimension line in relation to the extension lines:



Centered – Centres the dimension text along the dimension line between the extension lines.



At Ext Line 1 - Left-justifies the text with the first extension line along the dimension line. The distance between the extension line and the text is twice the arrowhead size plus the text gap value.



At Ext Line 2 - Right-justifies the text with the second extension line along the dimension line. The distance between the extension line and the text is twice the arrowhead size plus the text gap value.



Over Ext Line 1 - Positions the text over or along the first extension line.



Over Ext Line 2 - Positions the text over or along the second extension line.

Offset from dim line:

Sets the current text gap, which is the distance around the dimension text when the dimension line is broken to accommodate the dimension text.

This value is also used as the minimum length required for dimension line segments.

Text is positioned inside the extension lines only if the resulting segments are at least as long as the text gap. Text above or below the dimension line is placed inside only if the arrowheads, dimension text and a margin leave enough room for the text gap.

Offset from dim line: **0.625**



Offset from dim line: **0**



Text alignment

Horizontal

Places text in a horizontal position.



Aligned with dimension line

Aligns text with the dimension line.

ISO standard

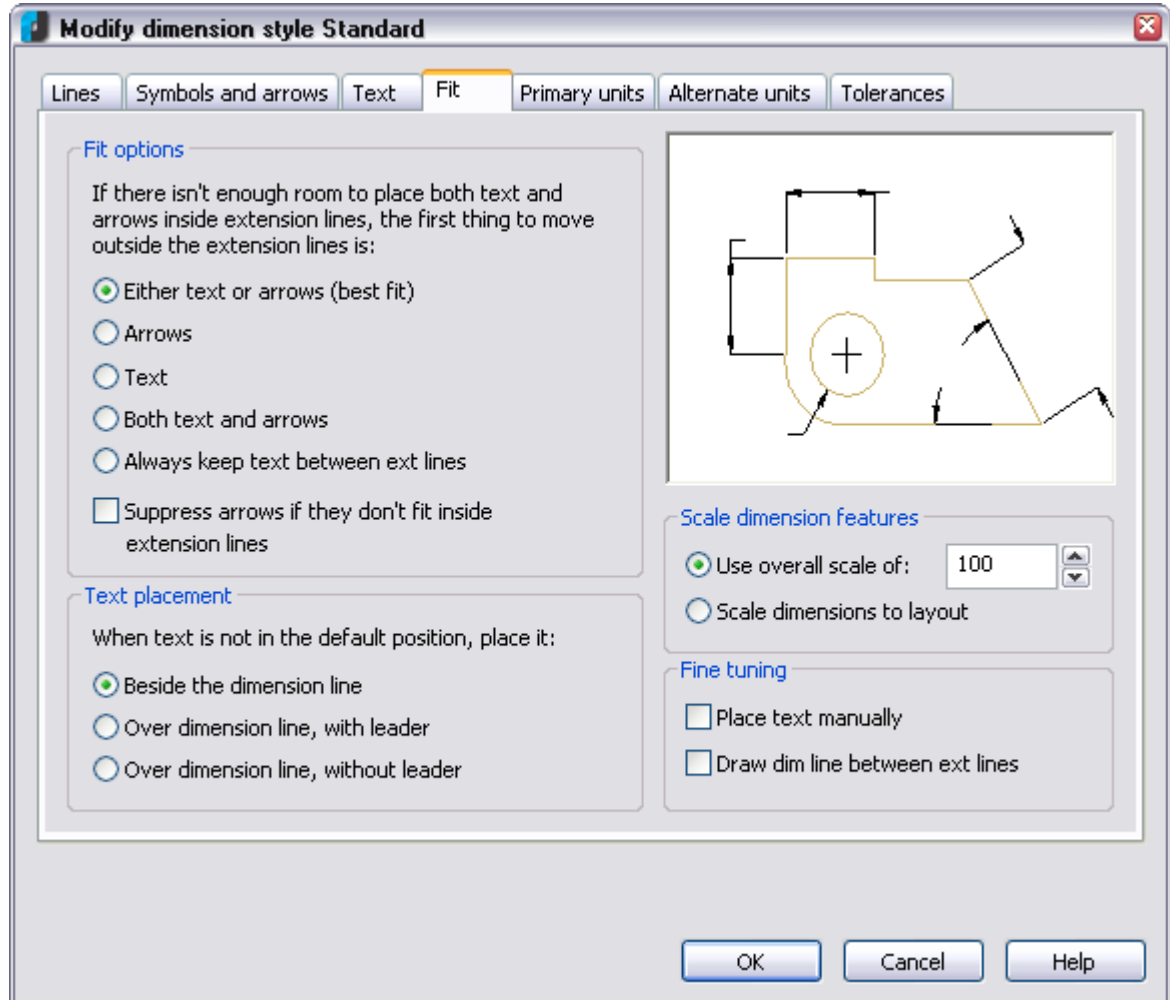
Aligns text with the dimension line when text is inside the extension lines, but aligns it

horizontally when text is outside the extension lines.



The Fit tab

Controls the placement of the dimension text, arrowheads, leader lines and the dimension line:



Options:

Fit options

Controls the placement of text and arrowheads based on the space available between the extension lines.

Either text or arrows (best fit)

Moves either the text or the arrowheads outside the extension lines based on the best fit.

Arrows

Moves the arrowheads outside the extension lines first, then the text.

Text

Moves the text outside the extension lines first, then the arrowheads.



Both text and arrows

When not enough space is available for text and arrowheads, both are moved outside the extension lines.



Always keep text

Always places the text between the extension lines.

between ext lines

Suppress arrows if they don't fit inside extension lines

Suppresses arrowheads if not enough space is available inside the extension lines.

Text placement

Sets the placement of the dimension text when it is moved from the default position; that is, the position defined by the dimension style.

Beside the dimension line

If selected, moves the dimension line whenever the dimension text is moved.



Over dimension line, with leader

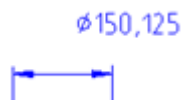
If selected, dimension lines are not moved when text is moved. If text is moved away from the dimension line, a leader line is created connecting the text to the dimension line.

The leader line is omitted when text is too close to the dimension line.



Over dimension line, without leader

If selected, dimension lines are not moved when text is moved.



Scale dimension features

Use overall scale of:

Sets a scale for all dimension style settings that specify size, distance or spacing, including text and arrowhead sizes.

Scale dimensions to layout

Determines a scale factor based on the scaling between the current model space viewport and the paper space.

Fine tuning

Place text manually

Places the text at the position you specify at the **Dimension Line Location** prompt.

Ignores any horizontal justification settings.

Draw dim line between ext lines

Draws dimension lines between the measured points, even when the arrowheads are placed outside the measured points.

The Primary units tab

Sets the format and precision of the primary dimension units and sets prefixes and suffixes for the dimension text:

Modify dimension style Standard

Lines Symbols and arrows Text Fit Primary units Alternate units Tolerances

Linear dimensions

Unit format: Decimal

Precision: 0.0000

Fraction format: Horizontal

Decimal separator: '.' (Period)

Round off: 0

Prefix:

Suffix:

Measurement scale

Scale factor: 1

☐ Apply to layout dimension only

Zero suppression

☐ Leading ☒ 0 feet

☐ Trailing ☒ 0 inches

Angular dimensions

Units format: Decimal Degrees

Precision: 0

Zero suppression


☐ Leading

☐ Trailing

OK Cancel Help

Options:

Linear dimensions

- Unit format:** Sets the current units format for all dimension types except Angular.
- Precision:** Displays and sets the number of decimal places in the dimension text.
- Fraction format:** Sets the format for fractions.
Options are available if you have set the **Fractional** or **Architectural** values in the **Unit format** option.
- Decimal separator:** Sets the separator for decimal formats.
Options are available if you have set the **Decimal** value in the **Unit format** option.
- Round off:** Sets rounding rules for dimension measurements for all dimension types except Angular.
For example:
1. If you enter a value of **0.25**, all distances are rounded to the nearest **0.25** unit.
2. If you enter a value of **1.0**, all dimension distances are rounded to the nearest integer.
The number of digits displayed after the decimal point depends on the **Precision** setting.
- Prefix:** Includes a prefix in the dimension text.
You can enter text or use control codes to display special symbols. For example, entering the control code **%%c** displays the diameter symbol.
When you enter a prefix, it overrides any default prefixes such as those used in diameter and radius dimensioning.
- 
- Suffix:** Includes a suffix in the dimension text.
You can enter text or use control codes to display special symbols.

When you enter a suffix, it overrides any default suffixes.



Measurement scale

Scale factor: Defines the linear scale options.
Scale factor set by default – 1.
For example, if you enter 2, the dimension for a 1-millimetre line is displayed as two millimetres.
The value does not apply to angular dimensions and is not applied to rounding values or to plus or minus tolerance values.

Apply to layout dimension only

Applies the measurement scale factor only to dimensions created in layout viewports.
Except when using non-associative dimensions, this setting should remain unchecked.

Zero suppression

Leading Suppresses leading zeros in all decimal dimensions.
For example, **0.3000** becomes **.3000**.

Trailing Suppresses trailing zeros in all decimal dimensions.
For example, **30.0000** becomes **30**.

0 feet Suppresses the feet portion of a feet-and-inches dimension when the distance is less than one foot.
For example, **0'-6 1/2"** becomes **6 1/2"**.
Options are available if you have set the **Engineering** or **Architectural** values in the **Unit format** option.

0 inches Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet.
For example, **1'-0"** becomes **1'**.
Options are available if you have set the **Engineering** or **Architectural** values in the **Unit format** option.

Angular dimensions

Unit format: Sets the angular units format.

Precision: Sets the number of decimal places for angular dimensions.

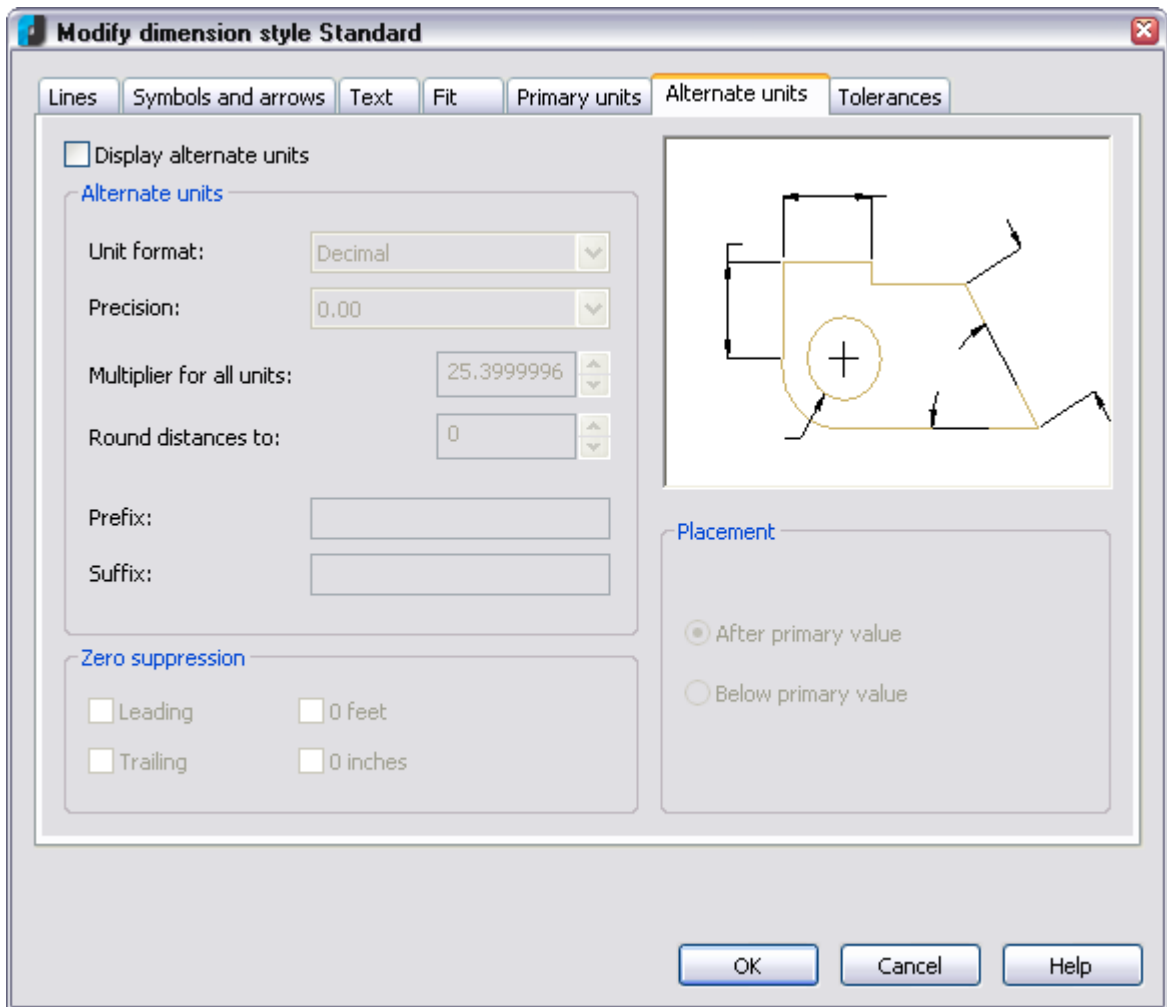
Zero suppression

Leading Suppresses leading zeros in angular decimal dimensions.
For example, **0.3000** becomes **.3000**.

Trailing Suppresses trailing zeros in angular decimal dimensions.
For example, **30.0000** becomes **30**.

The Alternate units tab

Specifies the display of alternative units in dimension measurements and sets their format and precision:



Options:

Display alternate Adds alternative measurement units to the dimension text.

Alternate units

Unit format: Sets the unit format for alternative units.

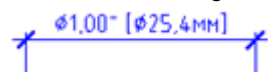
Precision: Sets the number of decimal places for alternative units.

Multiplier for all units: Specifies the multiplier used as the conversion factor between primary and alternative units. For example, to convert inches to millimetres, enter **25.4**. The value has no effect on angular dimensions and it is not applied to the rounding value or the plus or minus tolerance values.

Round distance to: Sets rounding rules for alternative units for all dimension types except Angular. For example:

1. If you enter a value of **0.25**, all alternate measurements are rounded to the nearest **0.25** unit.
2. If you enter a value of **1.0**, all dimension measurements are rounded to the nearest integer. The number of digits displayed after the decimal point depends on the **Precision** setting.

Prefix: Includes a prefix in the alternative dimension text. You can enter text or use control codes to display special symbols. For example, entering the control code **%%c** displays the diameter symbol. When you enter a prefix, it overrides any default prefixes such as those used in diameter and radius dimensioning.



Suffix: Includes a suffix in the alternative dimension text. You can enter text or use control codes to display special symbols.

When you enter a suffix, it overrides any default suffixes.



Zero suppression

| | |
|-----------------|---|
| Leading | Suppresses leading zeros in all decimal dimensions. For example, 0.3000 becomes .3000 . |
| Trailing | Suppresses trailing zeros in all decimal dimensions. For example, 30.0000 becomes 30 . |
| 0 feet | Suppresses the feet portion of a feet-and-inches dimension when the distance is less than 1 foot. For example, 0'-6 1/2" becomes 6 1/2" . Options are available if you have set the Engineering or Architectural values in the Unit format option. |
| 0 inches | Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet. For example, 1'-0" becomes 1' . Options are available if you have set the Engineering or Architectural values in the Unit format option. |

Placement

| | |
|----------------------------|---|
| After primary value | Places the alternative units after the primary units in the dimension text. |
| Below primary value | Places the alternative units below the primary units in the dimension text. |

The Tolerances tab

Controls the display and format of the dimension text tolerances:

Modify dimension style Standard

Lines Symbols and arrows Text Fit Primary units Alternate units **Tolerances**

Tolerance format

Method:

Precision:

Upper value:

Lower value:

Scaling for height:

Vertical position:

Zero suppression

☐ Leading ☐ 0 feet
☐ Trailing ☐ 0 inches

Alternate units tolerance

Precision:

Zero suppression

☐ Leading ☐ 0 feet
☐ Trailing ☐ 0 inches

OK Cancel Help

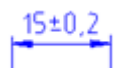
Options:

Tolerance format

Method: Sets the method for calculating the tolerance:



None - Does not add a tolerance.



Symmetrical – Adds a plus/minus expression of tolerance in which a single value of variation is applied to the dimension measurement.

A plus-or-minus sign appears after the dimension.

Enter the tolerance value in **Upper Value**.

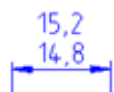


Deviation – Adds a plus/minus tolerance expression.

A plus sign (+) precedes the tolerance value entered in **Upper Value**, and a minus sign (-) precedes the tolerance value entered in **Lower Value**.

Note 1: If you input a minus sign (-) before an upper maximum deviation value, the value will be displayed with a minus sign (-) on the drawing.

Note 2: If you input a minus sign (-) before the a lower maximum deviation value, the value will be displayed with a plus sign (+) on the drawing.



Limits– Creates a limit dimension.

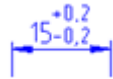
A maximum and a minimum value are displayed, one over the other.

The maximum value is the dimension value plus the value entered in **Upper Value**. The minimum value is the dimension value minus the value entered in **Lower Value**.

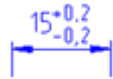


Basic – Creates a basic dimension, which displays a box around the full extent of the dimension.

| | |
|----------------------------|---|
| Precision: | Sets the number of decimal places. |
| Upper value: | Sets the maximum or upper tolerance value. When you select Symmetrical in Method, this value is used for the tolerance. |
| Lower value: | Sets the minimum or lower tolerance value. |
| Scaling for height: | Sets the current height for the tolerance text. The ratio of the tolerance height to the main dimension text height is calculated. |
| Vertical position: | Controls the text justification for symmetrical and deviation tolerances: |



Bottom Aligns the tolerance text with the bottom of the main dimension text.



Middle - Aligns the tolerance text with the middle of the main dimension text.



Top Aligns the tolerance text with the top of the main dimension text.

Zero suppression

| | |
|-----------------|---|
| Leading | Suppresses leading zeros in all decimal dimensions. For example, 0.3000 becomes .3000 . |
| Trailing | Suppresses trailing zeros in all decimal dimensions. For example, 30.0000 becomes 30 . |
| 0 feet | Suppresses the feet portion of a feet-and-inches dimension when the distance is less than 1 foot. For example, 0'-6 1/2" becomes 6 1/2" . Options are available if you have set the Engineering or Architectural values in the Unit format option. |
| 0 inches | Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet. For example, 1'-0" becomes 1' . Options are available if you have set the Engineering or Architectural values in the Unit format option. |

Alternate units tolerance

| | |
|-------------------|---|
| Precision: | Displays and sets the number of decimal places. |
|-------------------|---|

Zero suppression

| | |
|-----------------|---|
| Leading | Suppresses leading zeros in all decimal dimensions. For example, 0.3000 becomes .3000 . |
| Trailing | Suppresses trailing zeros in all decimal dimensions. For example, 30.0000 becomes 30 . |
| 0 feet | Suppresses the feet portion of a feet-and-inches dimension when the distance is less than 1 foot. For example, 0'-6 1/2" becomes 6 1/2" . Options are available if you have set the Engineering or Architectural values in the Unit format option. |
| 0 inches | Suppresses the inches portion of a feet-and-inches dimension when the distance is an integral number of feet. For example, 1'-0" becomes 1' . Options are available if you have set the Engineering or Architectural values in the Unit format option. |

Notes

Leader note



Menu: **Draw – Notes** >  **Leader notes ...**

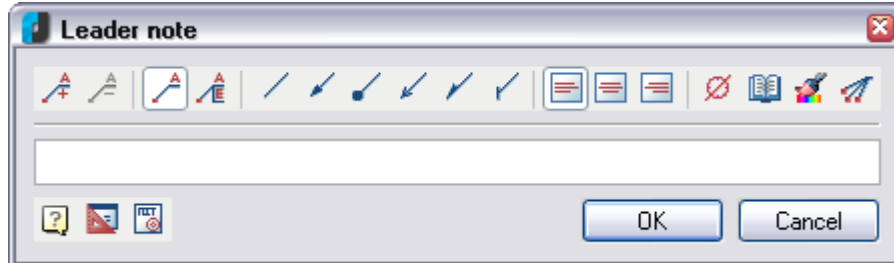


Toolbar: **Utilities** – 



Command line: **NOTE**

This command opens the **Leader note** dialogue box to set the leader note options:



Options:

Use the icons to add/remove text input fields and to add a border:



Add string.



Remove string.



Simple note.



Multiline note.

Use the icons to select the style of the extension line:



None.



Arrow.



Point.



Open arrow.



Half-arrow.



Oblique.

Use the icons to select the text alignment method:



By left edge.



By centre.



By right edge.

Other icons and options:



The **Insert special symbol** icon opens the panel with the table of special symbols, to select and insert them at the current cursor position in the text input field.



The **Notepad** icon opens the [Notepad](#) dialogue box.



The **Match properties** icon temporarily closes the dialogue box to specify the inserted leader whose properties should be copied and applied to the newly-created leader.



The **Add extension line** icon is used to insert additional extension lines. The icon is enabled when you edit a leader inserted into the drawing.

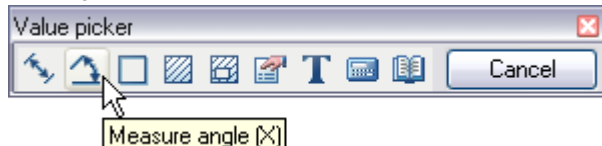
Right-click in the text field and choose the required menu item:



The **History** item contains a list of recently input text lines.

The **Recent** item is a list specified by the user.

The **Pick from drawing** item opens the [Value picker](#) toolbar which allows you to copy values from the objects on the drawing:



To create a universal note:

- Type the required text into the text fields.
- Select the required leader options.
- Click **OK**.
- Specify a point on the object to which the leader arrow will be directed.
- The following arrow type switching options are available in the command line and context menu:
 - None**— creates the extension line without an arrow,
 - Arrow**— creates the extension line with an arrow,
 - Point**— creates the extension line with a point.
- Select an option and specify the leader position on the drawing.

Position note



Menu: **Draw – Notes >**  **Position notes...**

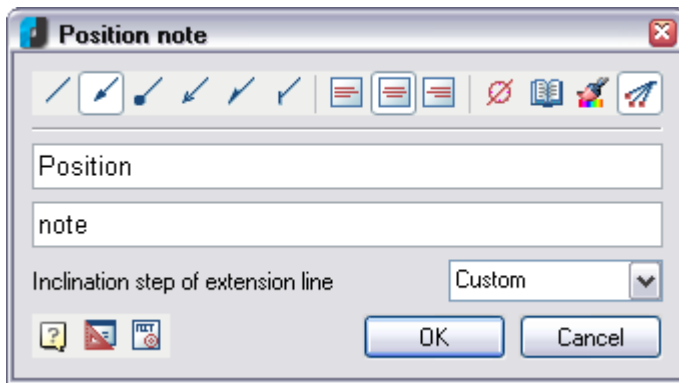


Toolbar: **Utilities –** 









Command line: **MLD, MLEADER**

This command opens the **Position note** dialogue box to set the note options:




Option:





Use the icons to select the style of the extension line:

-  None.
-  Arrow.
-  Point.
-  Open arrow.
-  Half-arrow.
-  Oblique.

Use the icons to select the text alignment method:

-  By left edge.
-  By centre.
-  By right edge.

Other icons and options:

-  The **Insert special symbol** icon opens the panel with the table of special symbols, to select and insert them at the current cursor position in the text input field.
-  The **Notepad** icon opens the [Notepad](#) dialogue box.
-  The **Match properties** icon temporarily closes the dialogue box to specify the inserted leader whose properties should be copied and applied to the newly-created leader.
-  The **Add extension line** icon is used to insert additional extension lines. The icon is enabled when you edit a leader inserted into the drawing.

Inclination step of extension line

Drop-down list to select inclination.

In the list the following inclinations are available:

- Custom** - the extension line is placed arbitrarily (by default);
- 15** - the extension line is placed in step multiples of 15°;
- 30** - the extension line is placed in step multiples of 30°;
- 45** - the extension line is placed in step multiples of 45°;
- 90** - the extension line is placed in step multiples of 90°.

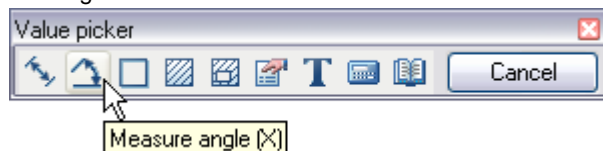
Right-click in the text field and choose the required menu item:

| | |
|---------------------|-----------|
| History | ▶ |
| Recent | ▶ |
| Template | ▶ |
| <hr/> | |
| Superscript | Ctrl+Up |
| Subscript | Ctrl+Down |
| Insert division | |
| Pick from drawing | |
| Insert object... | |
| Create hyperlink... | |
| Symbols | ▶ |
| <hr/> | |
| Undo | Ctrl+Z |
| Cut | Ctrl+X |
| Copy | Ctrl+C |
| Paste | Ctrl+V |
| Delete | Del |
| <hr/> | |
| Select all | Ctrl+A |

The **History** item contains a list of recently input text lines.

The **Recent** item is a list specified by the user.

The **Pick from drawing** item opens the [Value picker](#) toolbar which allows you to copy values from the objects on the drawing:



To create a position note:

- Type the required text into the text fields.
- Select the required note options.
- Click **OK**.
- Specify a point on the object to which the leader arrow will be directed.
- Specify the shelf position on the drawing.

Comb leader note



Menu: **Draw – Leaders >**  **Comb leader notes...**

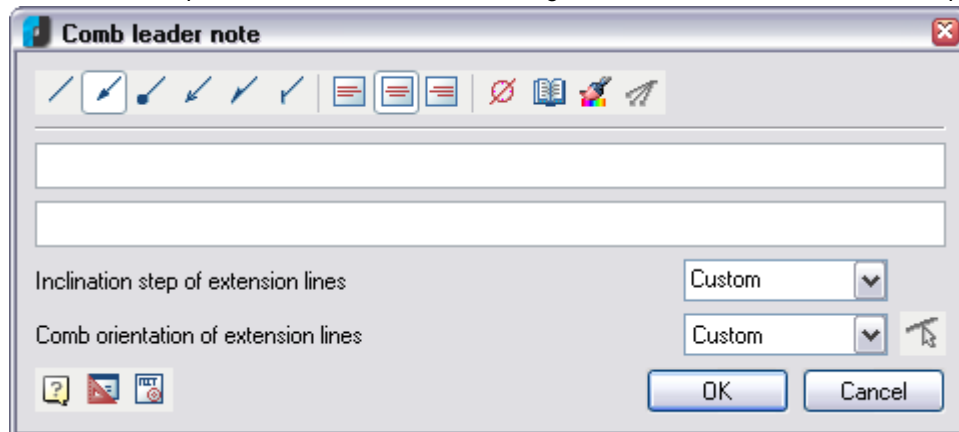


Toolbar: **Utilities –** 



Command line: **NOTEC**

This command opens the **Comb leader note** dialogue box to set the comb leader note options:



Options:

Use the icons to select the style of the extension line:



None.



Arrow.



Point.



Open arrow.



Half-arrow.



Oblique.

Use the icons to select the text alignment method:



By left edge.



By centre.



By right edge.

Other icons and options:



The **Insert special symbol** icon opens the panel with the table of special symbols, to select and insert them at the current cursor position in the text input field.



The **Notepad** icon opens the [Notepad](#) dialogue box.



The **Match properties** icon temporarily closes the dialogue box to specify the inserted leader whose properties should be copied and applied to the newly-created leader.



The **Add extension line** icon is used to insert additional extension lines. The icon is available when you edit a leader inserted into the drawing.



The **Select line** icon is used to switch the comb orientation parallel to the specified line on the drawing. The icon is available when you edit the comb leader note inserted into the drawing.

Inclination step of extension lines

Drop-down list to select inclination.

In the list the following inclinations are available:

Custom - the extension line is placed arbitrarily (by default);

15 - the extension line is placed in step multiples of 15°;

30 - the extension line is placed in step multiples of 30°;

45 - the extension line is placed in step multiples of 45°;

90 - the extension line is placed in step multiples of 90°.

Comb orientation of extension lines

Drop-down list to select the comb orientation of the extension line.

The following options are available in the list:

Custom

Horizontal

Vertical

Right-click in the text field and choose the required menu item:



By left edge.



By centre.



By right edge.

Use the icon to select the secant type:



Single-stroked line.



Double-stroked line.

Other icons:



The **Insert special symbol** icon opens the panel with the table of special symbols, to select and insert them at the current cursor position in the text input field.



The **Notepad** icon opens the [Notepad](#) dialogue box.

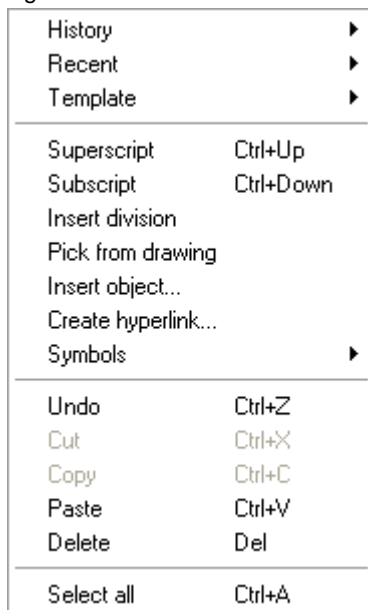


The **Match properties** icon temporarily closes the dialogue box to specify the inserted leader whose properties should be copied and applied to the newly-created leader.



The **Select line** icon is used to override the first and second lines of breaking construction. The icon is available when you edit the node secant note inserted into the drawing.

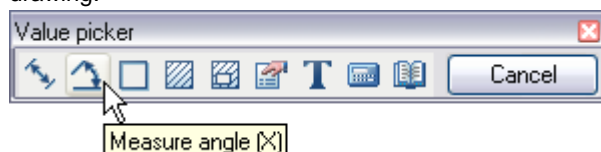
Right-click in the text field and choose the required menu item:



The **History** item contains a list of recently input text lines.

The **Recent** item is a list specified by the user.

The **Pick from drawing** item opens the [Value picker](#) toolbar which allows you to copy values from the objects on the drawing:



To create a node secant note:

- Type the required text into the text fields.
- Select the required note options.
- Click **OK**.
- Specify the first line of breaking construction, perpendicular to which the secant line will be located.

- Specify the second line of breaking construction.
- Specify the shelf position on the drawing.

Note for multilayered constructions



Menu: **Draw – Notes** >  **Notes for multilayered constructions...**

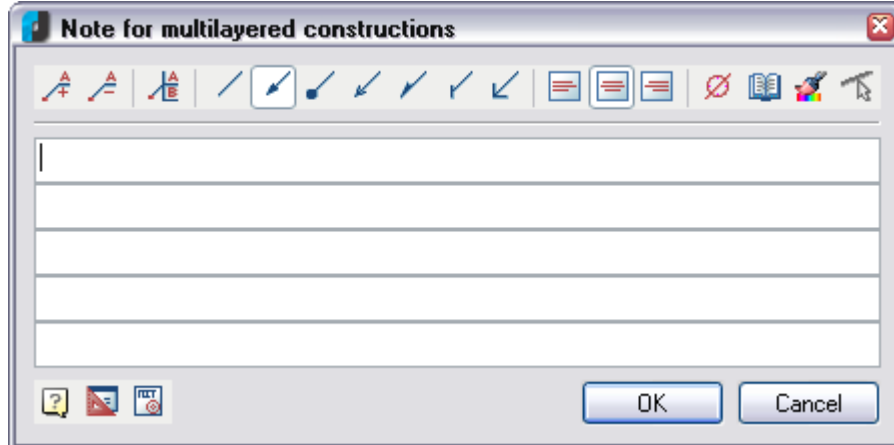


Toolbar: **Utilities** – 



Command line: **QLEADER**

This command opens the **Note for multilayer constructions** dialogue box to set the note options:



Options:

Use the icons to add/remove text input fields and to add borders:



Add string.



Remove string.



Line on first string.

Use the icons to select the style of the extension line:



None.



Arrow.



Point.



Open arrow.



Half-arrow.



Oblique.



Right angle.

Use the icons to select the text alignment method:



By left edge.



By centre.



By right edge.

Other icons:



The **Insert special symbol** icon opens the panel with the table of special symbols, to select and insert them at the current cursor position in the text input field.



The **Notepad** icon opens the [Notepad](#) dialogue box.

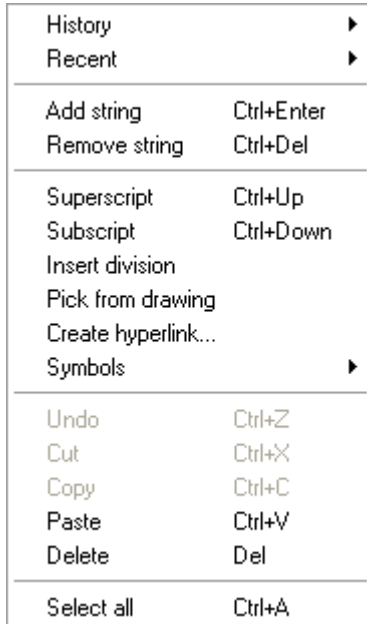


The **Match properties** icon temporarily closes the dialogue box to specify the inserted leader whose properties should be copied and applied to the newly-created leader.



The **Select line** icon is used to override the insertion point of the extension line. The icon is available when you edit the note for multilayer constructions inserted into the drawing.

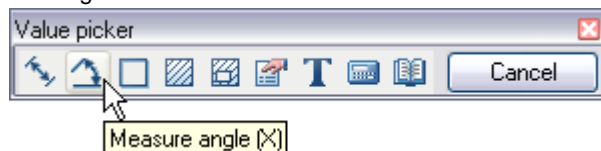
Right-click in the text field and choose the required menu item:



The **History** item contains a list of recently input text lines.

The **Recent** item is a list specified by the user.

The **Pick from drawing** item opens the [Value picker](#) toolbar which allows you to copy values from the objects on the drawing:



To create a note for multilayer constructions:

- Type the required text into the text fields.
- Select the required note options.
- Click **OK**.
- Specify a point on the object to which the leader arrow will be directed.
- Specify the shelf position on the drawing.

Node note



Menu: **Draw – Notes >** **Node notes...**

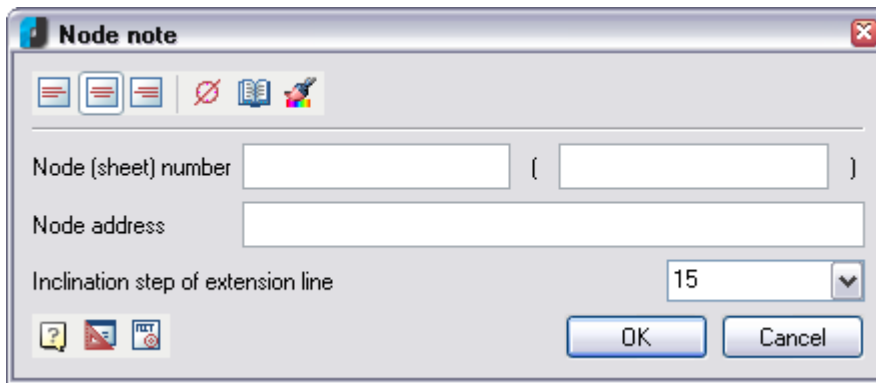


Toolbar: **Utilities –**



Command line: **NLD, NLEADER**

This command opens the **Node note** dialogue box to set the note options:



The 'Node note' dialog box has a title bar with a close button. Below the title bar is a toolbar with icons for text alignment (left, center, right), a red 'X' icon, a notepad icon, and a match properties icon. The main area contains three input fields: 'Node (sheet) number' with a text box and a parentheses button, 'Node address' with a text box, and 'Inclination step of extension line' with a dropdown menu showing '15'. At the bottom are 'OK' and 'Cancel' buttons.

Options:

Use the icons to select the text alignment method:



By left edge.



By centre.



By right edge.

Other icons and options:



The **Insert special symbol** icon opens the panel with the table of special symbols, to select and insert them at the current cursor position in the text input field.



The **Notepad** icon opens the [Notepad](#) dialogue box.



The **Match properties** icon temporarily closes the dialogue box to specify the inserted leader whose properties should be copied and applied to the newly-created leader.

Inclination step of extension lines

Drop-down list to select the inclination.

In the list the following inclinations are available:

Custom - the extension line is placed arbitrarily (by default);

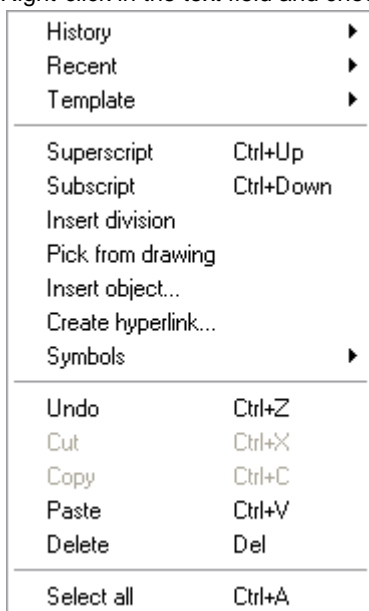
15 - the extension line is placed in step multiples of 15°;

30 - the extension line is placed in step multiples of 30°;

45 - the extension line is placed in step multiples of 45°;

90 - the extension line is placed in step multiples of 90°.

Right-click in the text field and choose the required menu item:

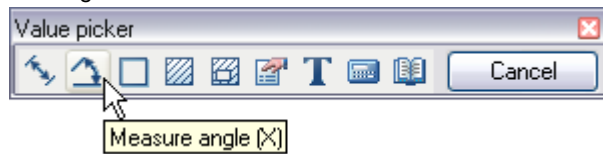


A right-click context menu is shown with the following items: History, Recent, Template, Superscript (Ctrl+Up), Subscript (Ctrl+Down), Insert division, Pick from drawing, Insert object..., Create hyperlink..., Symbols, Undo (Ctrl+Z), Cut (Ctrl+X), Copy (Ctrl+C), Paste (Ctrl+V), Delete (Del), and Select all (Ctrl+A).

The **History** item contains a list of recently input text lines.

The **Recent** item is a list specified by the user.

The **Pick from drawing** item opens the [Value picker](#) toolbar which allows you to copy values from the objects on the drawing:



To create a node note:

- Type the required text into the text fields.
- Select the required note options.
- Click **OK**.
- Specify the oval/circle centre.
- Specify the oval/circle size.
- Specify the shelf position and/or the leader pitch angle.

Mark of linear designs



Menu: **Draw – Notes >**  **Marks of linear designs...**

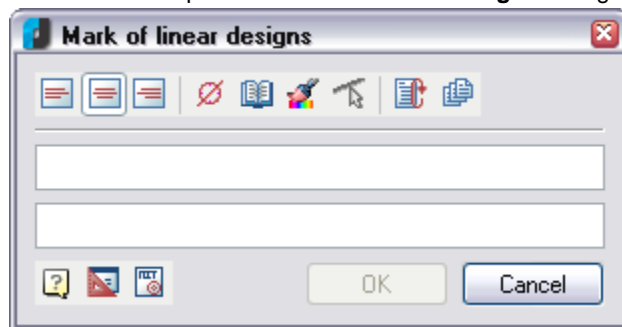


Toolbar: **Utilities –** 



Command line: **LINM**

This command opens the **Mark of linear designs** dialogue box to set the note options:



Options:

Use the icons to select the text alignment method:



By left edge.



By centre.



By right edge.

Other icons:



The **Insert special symbol** icon opens the panel with the table of special symbols, to select and insert them at the current cursor position in the text input field.



The **Notepad** icon opens the [Notepad](#) dialogue box.



The **Match properties** icon temporarily closes the dialogue box to specify the inserted leader whose properties should be copied and applied to the newly-created leader.



The **Select line** icon is used to override the insertion place of the extension line. The icon is available when you edit the marks of linear constructions inserted into the drawing.

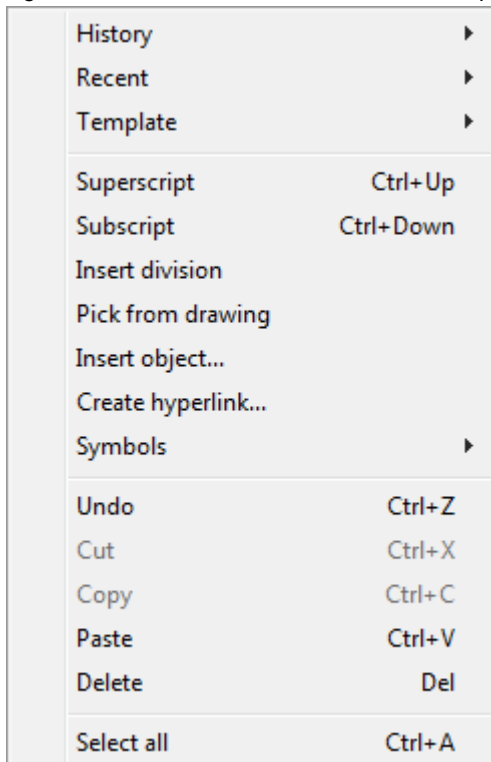


The **Auto repeat** icon allows you to mark several linear constructions without a repeated command call. For each new leader, the **Linear constructions marking** dialogue box will open to set new options for the leader, for example, new text.



The **Multiple insert** icon allows you to mark several linear constructions without a repeated command call. All leaders are drawn with same options and with the same text.

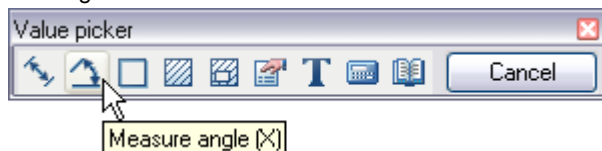
Right-click in the text field and choose the required menu item:



The **History** item contains a list of recently input text lines.

The **Recent** item is a list specified by the user.

The **Pick from drawing** item opens the [Value picker](#) toolbar which allows you to copy values from the objects on the drawing:



To create a note:

- Type the required text into the text fields.
- Select the required note options.
- Click **OK**.
- Select line of construction, on which the mark will be located.
- Specify the text position.

Chain note



Menu: **Draw – Notes >** **Chain notes...**

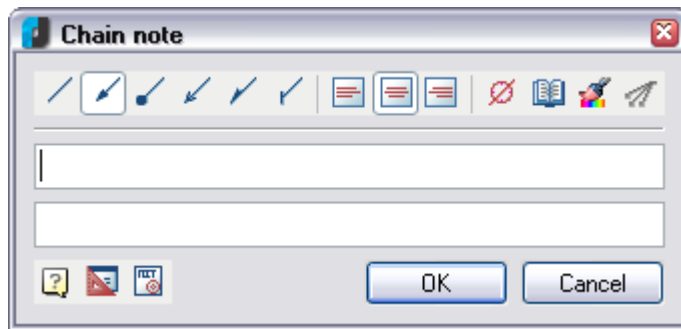


Toolbar: **Utilities –**









Command line: **NOTEH**

The command opens the **Chain note** dialogue box to set the note options:



Options:





Use the icons to select the style of the extension line:

-  None.
-  Arrow.
-  Point.
-  Open arrow.
-  Half-arrow.
-  Oblique.

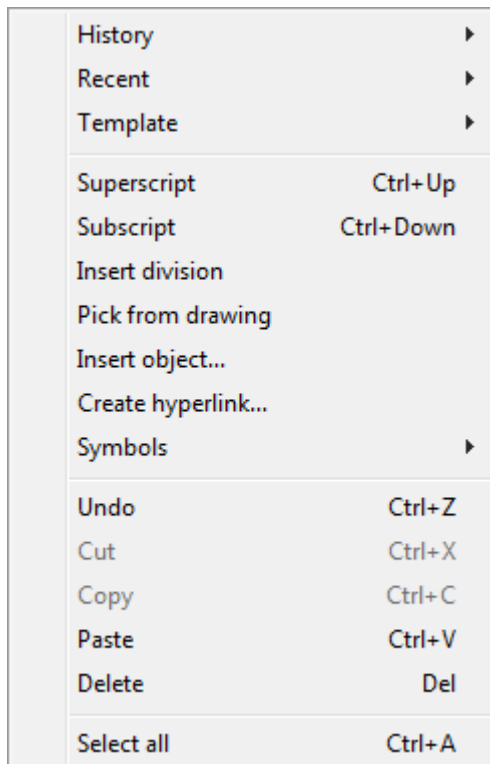
Use the icons to select the text alignment method:

-  By left edge.
-  By centre.
-  By right edge.

Other icons:

-  The **Insert special symbol** icon opens the panel with the table of special symbols, to select and insert them at the current cursor position in the text input field.
-  The **Notepad** icon opens the [Notepad](#) dialogue box.
-  The **Match properties** icon temporarily closes the dialogue box to specify the inserted leader whose properties should be copied and applied to the newly-created leader.
-  The **Add extension line** icon is used to insert additional extension lines. The icon is available when you edit a chain note inserted into the drawing.

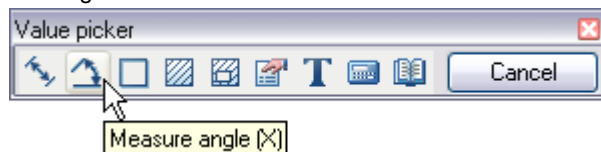
Right-click in the text field and choose the required menu item:



The **History** item contains a list of recently input text lines.

The **Recent** item is a list specified by the user.

The **Pick from drawing** item opens the [Value picker](#) toolbar which allows you to copy values from the objects on the drawing:



To create a chain note:

- Type the required text into the text fields.
- Select the required note options.
- Click **OK**.
- Specify the first leader node.
- Specify the next leader nodes.
- Specify the last node and press **ENTER**.
- Specify the shelf position.

If the first leader node is placed on the line, the extension line will be perpendicular to this line.

Editing the leaders

You can edit the leaders by double clicking. When editing the leaders, the same dialogue box as when creating these leaders will be opened, but some additional icons that are blocked when creating leaders will be available (for example, the **Add extension line**, **Select line** and others).

It is very easy to edit leaders with grips (for more information, see "[Advanced grips](#)" («*EXECUTION OF DRAWINGS COMMAND*» - «*Notes*»)).

You can also detach, append and edit extension lines with the appropriate commands from the **Draw** menu – the **Notes** item or from the **Utilities** toolbar.

Detach leader



Menu: **Draw – Notes >**  **Detach leader**



Toolbar: **Utilities –** 




Command line: **PLD**

To detach a leader, it is necessary to select it in the drawing. The leader will be detached immediately after its selection.

Append leader



Menu: **Draw – Notes >**  **Append leader**



Toolbar: **Utilities –** 



Command line: **PL**

To append leader:

- Specify a point on the object.
- If necessary, specify the number of intermediate points forming a break of leader.
- Specify a point on the required leader inserted in the drawing, to finish the command.

Edit leader



Menu: **Draw – Notes >**  **Edit leader**



Toolbar: **Utilities –** 



Command line: **PLR**

To edit leader:

- Select a leader in the drawing.
- Specify a point on the required leader to fix a new position for the leader.

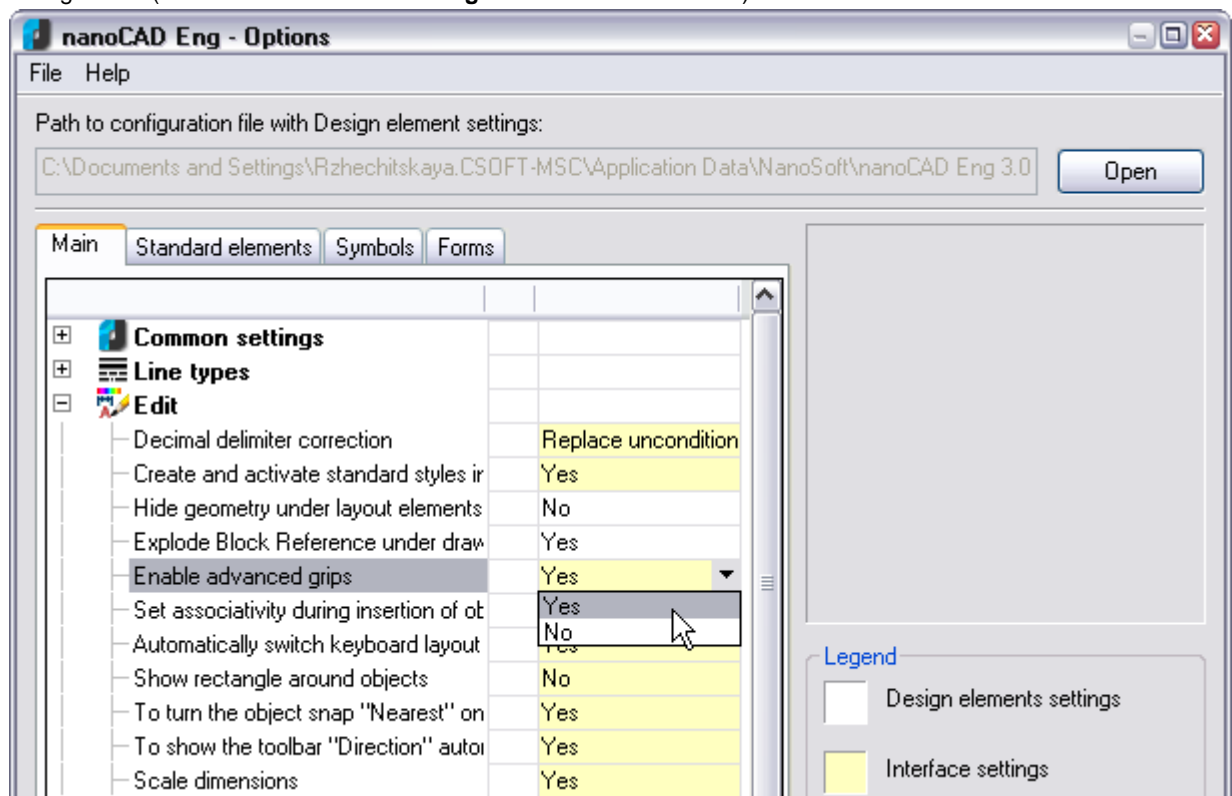
Advanced grips for design elements

Advanced grips are assigned for editing objects on the screen without using the Edit window.

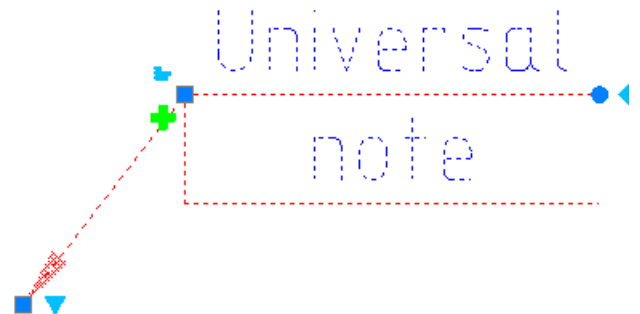
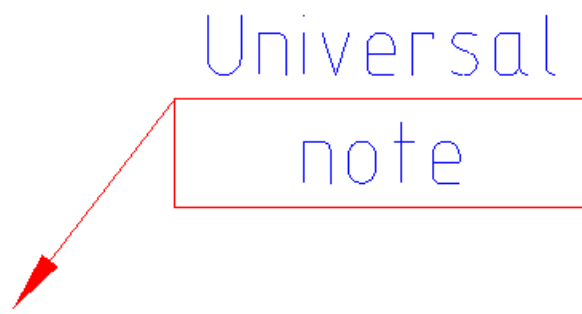
The advanced grips are different in shape (round, triangular, rhombic, etc.) and, in some cases, in colour from the ordinary grips.

When the advanced grips operating mode is on, selection of design elements is made first and then they are edited with the mouse.

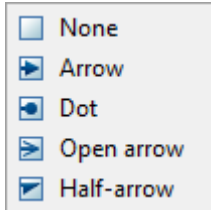
You can turn the advanced grips operating mode on/off in the **Edit** item in the **Main** tab of the **nanoCAD – Options** dialogue box (the **Tools** menu – the **Settings Parameters** command):



Universal note

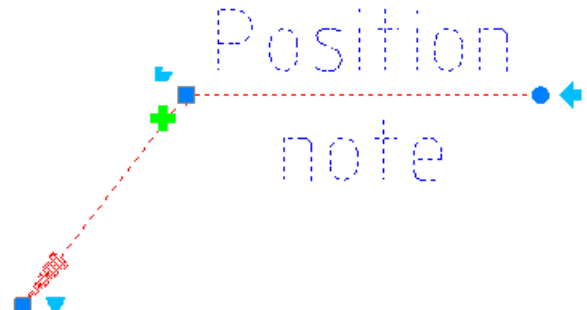
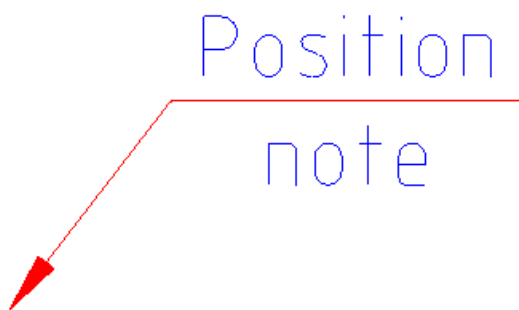


- ▼ Use this grip to select the arrow type.
Click on the grip to open the menu for arrow type selection:

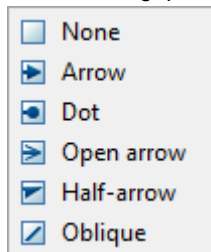


- ✚ Use this grip to add a leader.
- ≡ Use this grip to align text.
There are 3 alignment types: by left edge, by centre and by right edge.
- Use this grip to rotate a shelf.
- ◀ Use this grip to mirror a shelf.

Position note

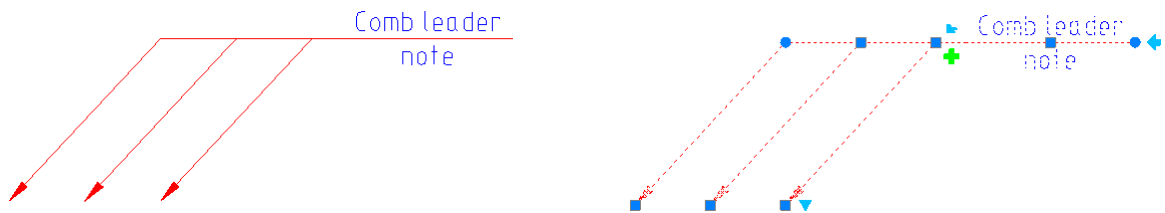


- ▼ Use this grip to select the arrow type.
Click on the grip to open the menu for arrow type selection:



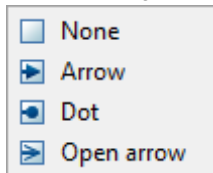
- ✚ Use this grip to add a leader.
- ≡ Use this grip to align text.
There are 3 alignment types: by left edge, by centre and by right edge.
- Use this grip to rotate a shelf.
- ◀ Use this grip to mirror a shelf.

Comb leader note



Use this grip to select the arrow type.

Click on the grip to open the menu for arrow type selection:



Use this grip to add a leader.




Use this grip to align text.

There are 3 alignment types: by left edge, by centre and by right edge.



Use this grip to rotate a comb line.



Use this grip to rotate a shelf (located next to the  grip).



Use this grip to mirror a shelf.

Node secant note

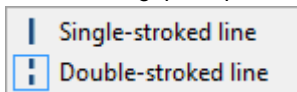


Node secant
note



Use this grip to select the stroke type.

Click on the grip to open the menu for stroke type selection:



Use this grip to align text.

There are 3 alignment types: by left edge, by centre and by right edge.



Use this grip to rotate a shelf.

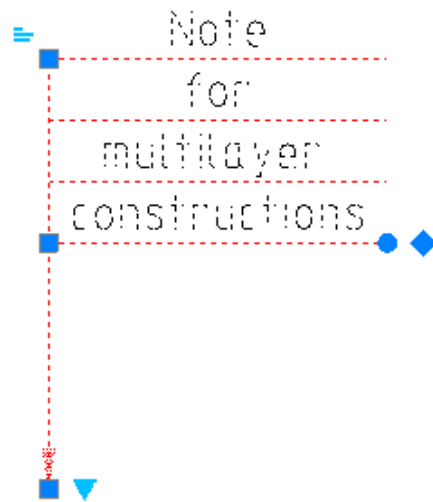
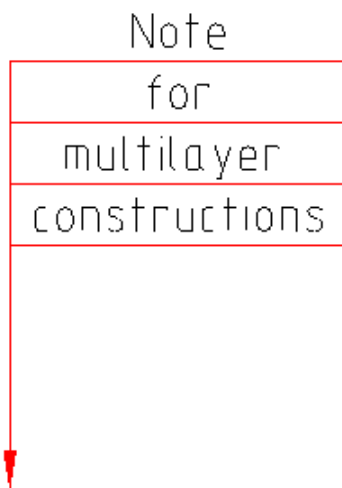


Use this grip to mirror a shelf.



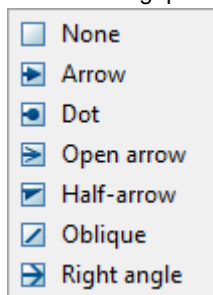
Use this grip to move a shelf.

Note for multilayer constructions



Use this grip to select the arrow type.

Click on the grip to open the menu for arrow type selection:



Use this grip to align text.

There are 3 alignment types: by left edge, by centre and by right edge.

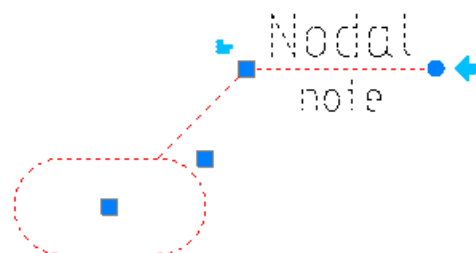


Use this grip to rotate shelves.



Use this grip to change the position of shelves.

Nodal note



Use this grip to align text.

There are 3 alignment types: by left edge, by centre and by right edge.

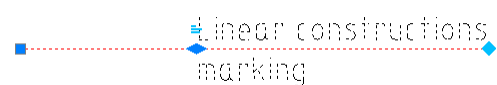
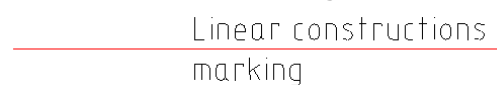


Use this grip to rotate a shelf.



Use this grip to mirror a shelf.

Linear constructions marking



Use this grip to align text.

There are 3 alignment types: by left edge, by centre and by right edge.

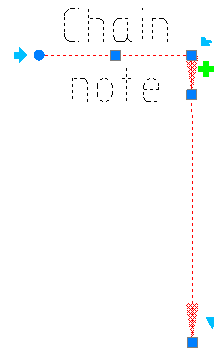
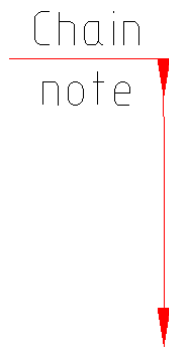


Use this grip to change the shelf position.



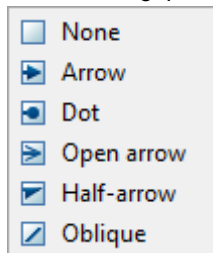
Use this grip to change the text position.

Chain note



Use this grip to select the arrow type.

Click on the grip to open the menu for arrow type selection:



Use this grip to add a leader.



Use this grip to align text.

There are 3 alignment types: by left edge, by centre and by right edge.



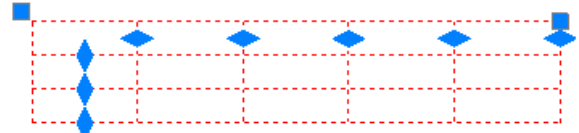
Use this grip to rotate a shelf.



Use this grip to mirror a shelf.

Table

| | | | | |
|--|--|--|--|--|
| | | | | |
| | | | | |
| | | | | |



Use these grips to edit the linear dimensions of columns.



Use these grips to edit the linear dimensions of rows.

Tables

- ☐ [Editing Tables on the Drawing](#)
- ☐ [The Interface](#)
- ☐ [Basic Tools](#)
- ☐ [Editing the Size of Rows and Columns](#)
- ☐ [Import, Export from MS Excel](#)
- ☐ [Page Division](#)
- ☐ [Cell properties](#)
- ☐ [Snap to cell](#)
- ☐ [Expression builder interface](#)
- ☐ [Function in expression builder](#)
- ☐ [Report creation](#)



Menu: **Draw – Tables >**  **Tables...**



Toolbar: **Draw –** 



Command line: **TABLE, TB**

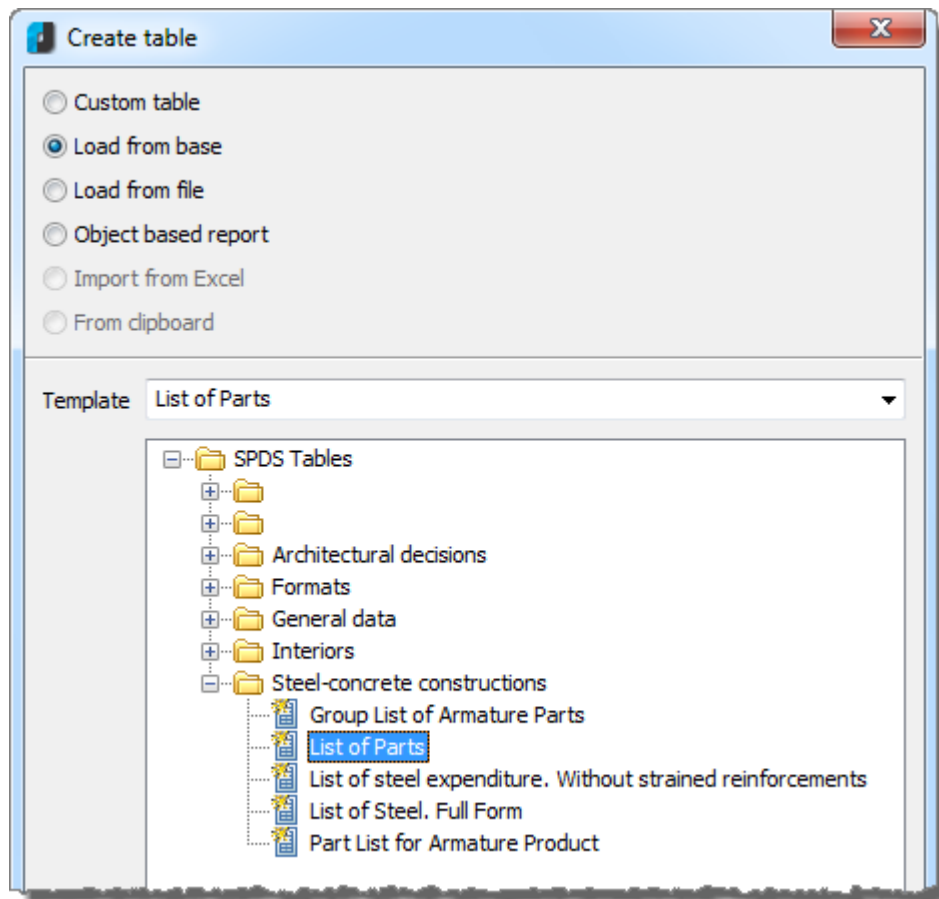
The command opens the *Create table* dialogue box:

The 'Create table' dialog box is shown with the 'Custom table' radio button selected. The 'Title' field is empty. The 'Columns' field shows '25' and '5'. The 'Rows' section has three sub-fields: 'Header rows' with '12' and '0', 'Data rows' with '8' and '3', and 'Footer rows' with '12' and '0'. A grid of 25 columns and 12 rows is visible, with the first 5 columns and the first 3 rows highlighted in blue.

- ☐ To create a non-standard table, click the **Custom table** item. Use the numeric entry fields to set the parameters for the custom table. The number of rows or columns and cell sizes can be altered later when the table is first edited.

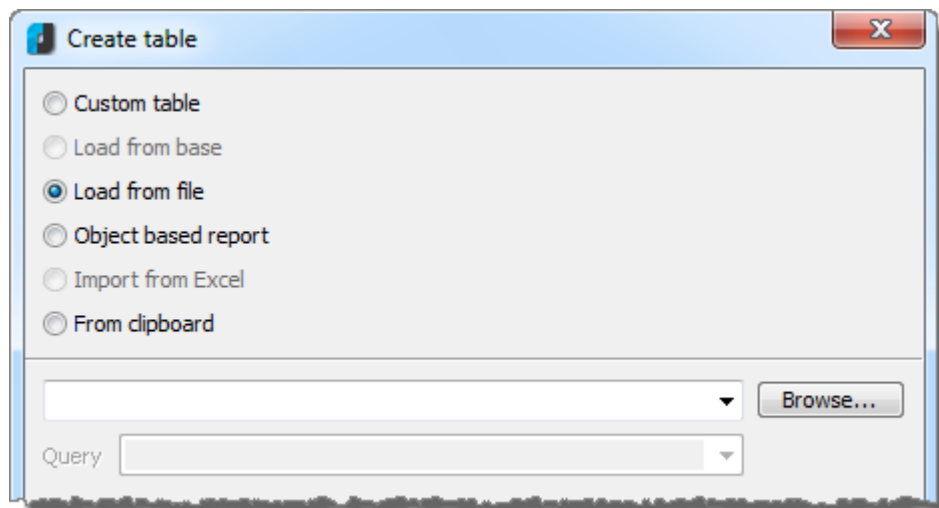
The 'Create table' dialog box is shown with the 'Load from base' radio button selected. The 'Title' field is empty. The 'Columns' field shows '25' and '5'. The 'Rows' section has three sub-fields: 'Header rows' with '12' and '0', 'Data rows' with '8' and '3', and 'Footer rows' with '12' and '0'. A grid of 25 columns and 12 rows is visible, with the first 5 columns and the first 3 rows highlighted in blue.

- ☐ To insert a standard table, click the **Load from base** item. A standard table can be inserted from the nanoCAD library.



Choose the desired table type in the dialogue box. All basic standard tables are present in the nanoCAD library.

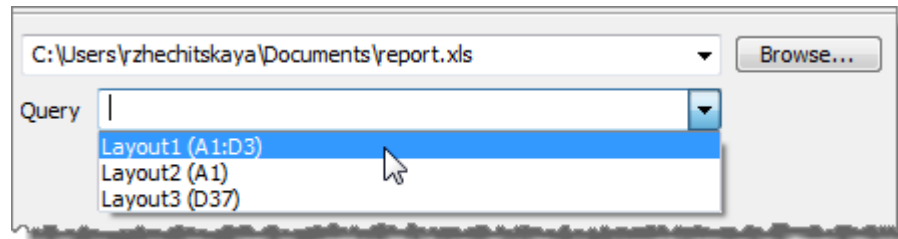
- ☐ To insert a table from a file, click the **Load from file** item.



Click the **Browse** button and select the table file
Supported formats:

- ☐ tbl - tables format;
- ☐ dat - data file or text file;
- ☐ mdb - Microsoft Access database;
- ☐ xls - Microsoft Office Excel table;
- ☐xlsx - Microsoft Office Excel 2007 table;
- ☐ csv - table, cells are divided by commas;
- ☐ txt - standard text file;
- ☐ xml - XML document.


When loading the table from xlsx or xls files, it is necessary to select the list in the Excel document.



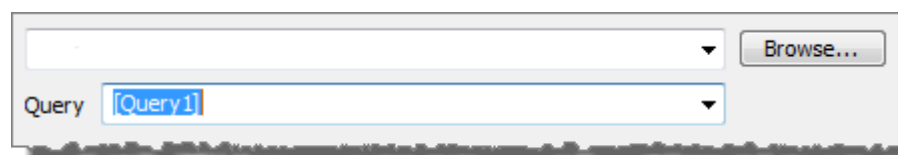
It is important! The list choice can be carried out after loading the table from a file.

| General | |
|-----------------|-----------------------------------|
| Color | <input type="checkbox"/> white |
| Layer | 0 |
| Linetype | —— By Layer |
| Linetype Scale | 1.0000 |
| Line Weight | —— 0.00 |
| Transparency | 0 |
| Hyperlink | |
| nanoCAD Eng | |
| Title | |
| Scale | 1:1 |
| Row count | 3 |
| Col count | 4 |
| Text style | GOST 2.304 |
| Text height | 2.5000 |
| Text color | <input type="checkbox"/> By Layer |
| Text weight | —— By Layer |
| Width factor | 0.0000 |
| Text indent | 0.6000 |
| From up to down | Yes |
| Source file | C:\Users\rzhechitskaya\Do... |
| Source query | Layout 1 (A1:D3) |

In the table properties, the **Source file** line displays the path to the initial table file.

In the **Source query** line, enter the required list from the document. Then in the **Table edit** dialogue box click the **Update table from external source**  icon.

When loading the table from an mdb file, the list of base queries is displayed in the drop-down list.

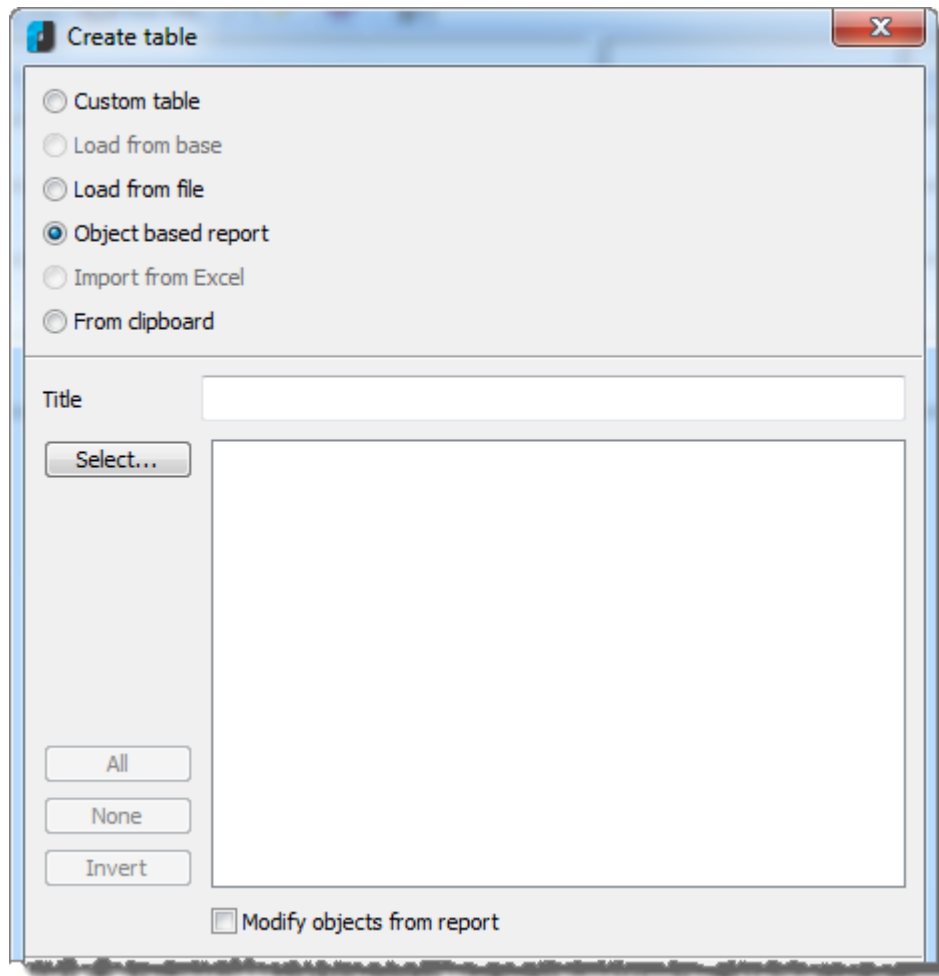


To import tables from files, it is possible to drag a file from the browser to the [Table edit](#) dialogue box.

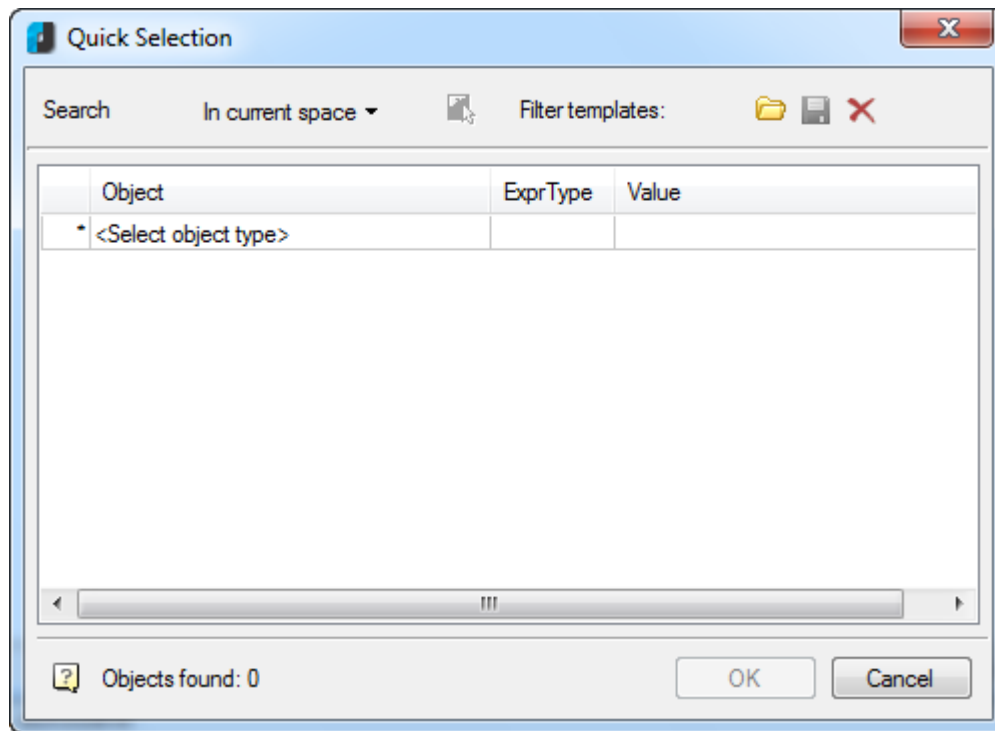
When dragging a file from the browser into the table field, the imported table is added to the existing table.

When dragging a file from the browser into the dialogue box field, the imported table replaces the existing table

- To generate an object base report, click the **Object based report** item.



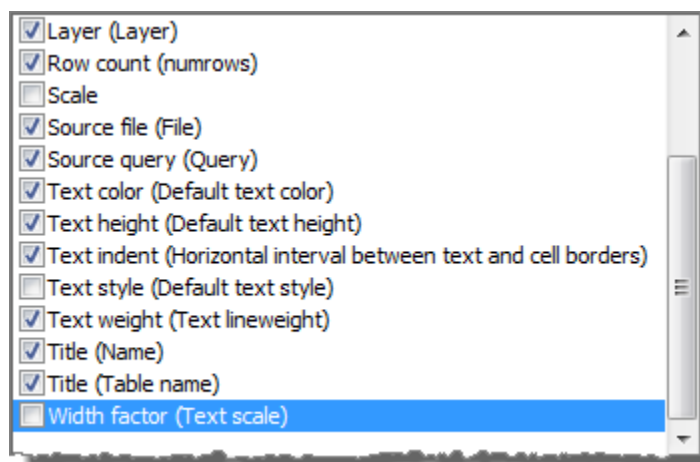
To choose the required objects, click the **Select** button.



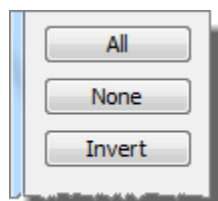
Setting of

search conditions is carried out in the [Quick Selection](#) dialogue box.

Which attributes of the chosen objects the report is based on are also set in this dialogue box.



The attributes to be included in the report are switched by tags in the list or switches.



- ☐ **All** - all attributes are selected
- ☐ **None** - the choice is removed from all attributes
- ☐ **Invert** - selection of attributes is inverted

The report with a template of a view is created in the table:

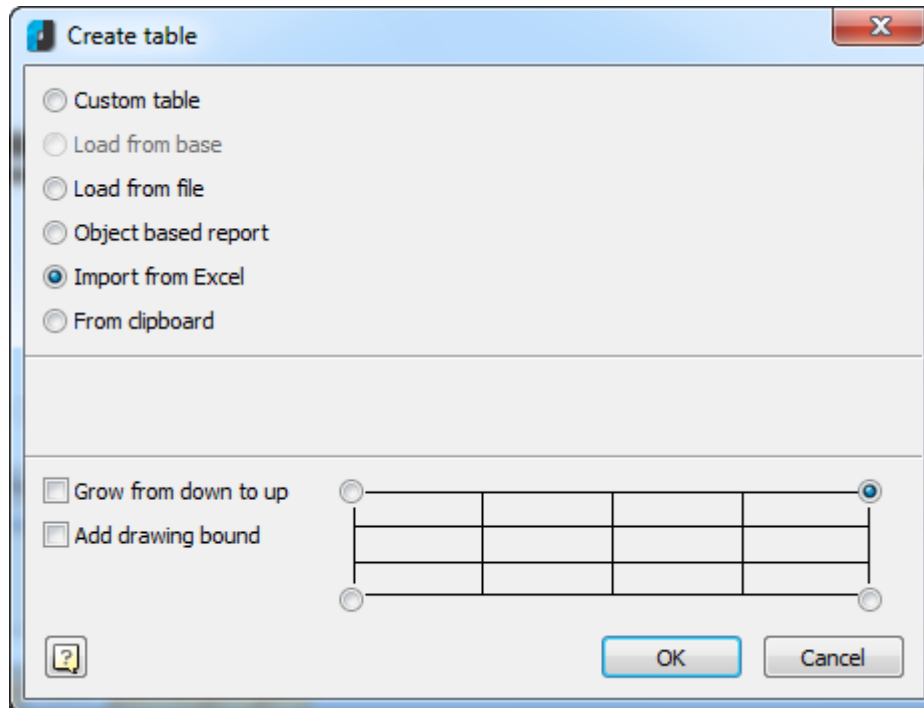
```
=Iff(Exist(Object."AttributeName");Object."AttributeName";"")
```

This expression checks the existence of the given attribute line with the *AttributeName* name and uses its value in the cell. Otherwise, it leaves a cell empty.

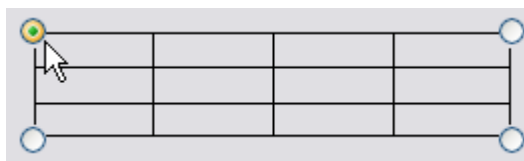
| | A | B | C | D | E |
|---|----------------------------------|----------------------------------|------------------------------------|------------------------------------|----------------------------------|
| | Report template | | | | |
| | Section properties | | | | |
| 1 | =Object.numcols; «» otherwise | =Object.«Decimal «» otherwise | =Object.Direction; «» otherwise | =Object.Name; if E «» otherwise | =Object.numrows; «» otherwise |
| | Report | | | | |
| 2 | 7 | 17 | true | 45 | C:\D... |

The number of columns in the table is defined by the number of chosen attributes.

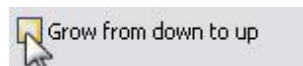
- ☐ To import a table from Excel, select the **Import from Excel** item. The document should be opened to make this item active.




- ☐ Choose the location of the base point.



- ☐ Choose the row numbering method.



- ☐ If the *Grow Down to Up* box is checked, the rows will be numbered in reverse order. Check the ☐ Add drawing bound box to start the [Format](#) dialogue box when you insert a table.
- ☐ Click the OK button and pick the insertion point in the drawing.

Note: If the objects group was previously selected on the drawing and the **Tables**  command is activated, then you will be offered the option to create an object-based report.

Editing Tables on the Drawing

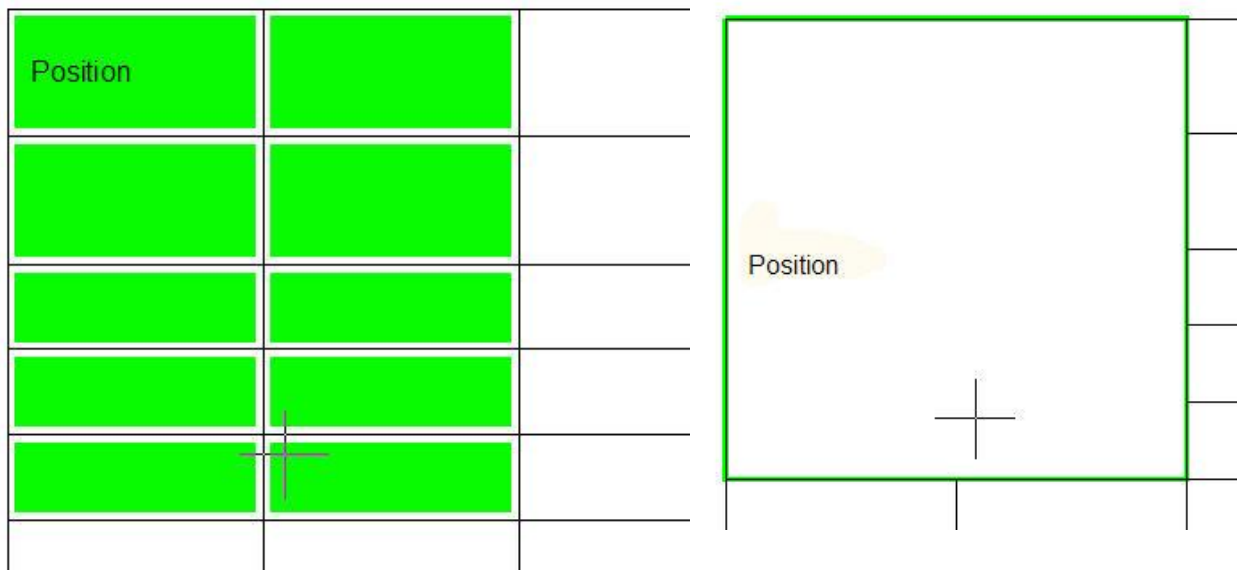
The on-screen table editor is opened by left-clicking the table frame while holding down the CTRL key. Content can be entered into cell by placing the cursor above the cell and left-clicking. The active cell is then highlighted in green. The text entered in a cell is automatically condensed to fit the cell width.

The **Table Edit** toolbar is shown when a table is being edited on-screen.



Group cells. Use this button to merge adjacent cells.

After clicking this icon, the cell located under the cursor is made active and shown in green. Hold down the left mouse button and select the cells you wish to merge. Left-click again to confirm selection - the selected cells will be merged together.



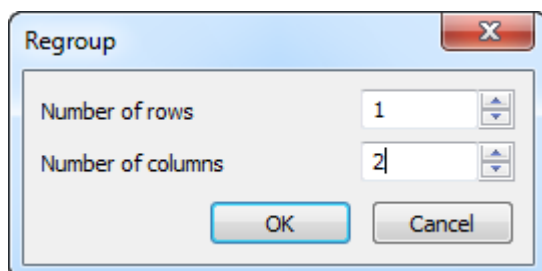
Ungroup cells. Use this button to restore previously merged cells into single cells.

After clicking the icon, move the cursor to a cell which was created by merging multiple cells. It is made active and highlighted in green. Left-click inside the cell. The cell breaks up into its original smaller cells



Split cells. This button divides a cell into smaller cells.

After clicking the icon, place the cursor over a cell and left-click inside the cell. The application prompts for two points within the cell - pick these two points by left-clicking each. The **Regroup** dialogue box will appear.

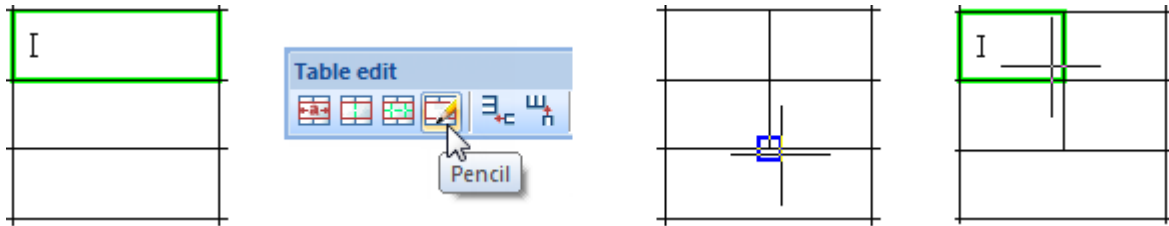



Enter the desired division values. Multiple cells within a single row or column can be split simultaneously.





Pencil. Use this button to split a cell into multiple cells by drawing additional cell borders.


Move the cursor over a cell and left-click inside the cell. Click the icon. Draw the division line by picking points on the existing cell borders (use nanoCAD object snap). The cell or cells will be split into new cells of arbitrary size. Each new cell is independent.





 **Add row** adds a row to the bottom of the table.


 **Add column** adds a column to the end of the table.


 **Insert row** adds a row to the table at the cursor position.

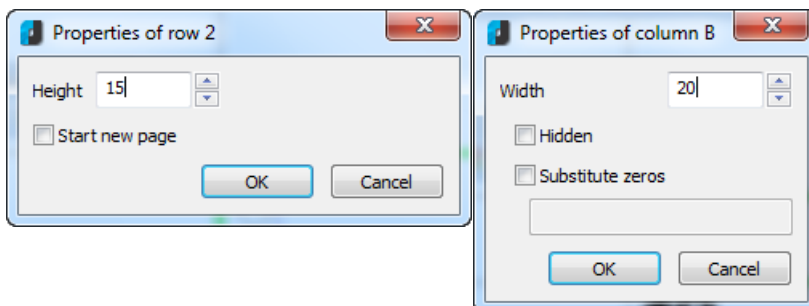
 **Insert column** adds a column to the table at the cursor position.


 **Delete row** deletes the row containing the active cell.


 **Delete column** deletes the column containing the active cell.

 **Row properties** use to alter the height of the row containing the active cell. A dialogue box appears in which the row height, in millimetres, can be specified.

 **Column properties** use to alter the width of the column containing the active cell. A dialogue box appears in which the column width, in millimetres, can be specified.



 **Cell properties** opens the [Cell Properties](#) dialogue box. Use this to set the properties of the selected cell.

 **Cells properties** use this to set the properties of multiple cells. Select the desired cells and set the parameters in the corresponding dialogue box.

 **Sum.**

 **Selective sum.**

Use these buttons to automatically add up the values from the selected cells and display the sum in a blank cell at the end of the selection. When using these functions, do not enter any sum expressions in the **Cell Properties** dialogue box.

 [Notepad](#) opens the **Default – Notepad** dialogue box.



[Calculator](#) opens the **Calculator** dialogue box.



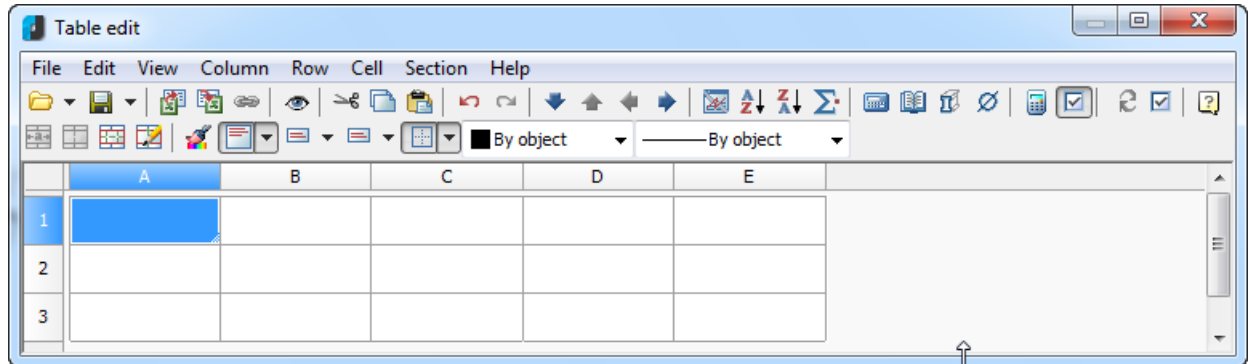
This button allows a special symbol to be inserted from the menu.



Exit exits the on-screen table editor.

The Interface

The **Table edit** dialogue box:



The dialogue box contains:

- ☐ standard pull-down menus

File Edit View Column Row Cell Section Help ;

- ☐ [tool palettes](#) buttons;
- ☐ rulers with sliders allowing column width or row height to be quickly adjusted;
- ☐ the cell grid with name headers.

| | A | B | C | D | E |
|---|---|---|---|---|---|
| 1 | | | | | |
| 2 | | | | | |
| 3 | | | | | |

Each table section has a header:

- ☐ First page header;
- ☐ Header;
- ☐ Last page header;
- ☐ Report header;
- ☐ Report template;
- ☐ Report;
- ☐ Report sum;
- ☐ First page footer;
- ☐ Footer;
- ☐ Last page footer.

Click the left mouse button on a row or column name to select it.

| | A | B | C | D | E |
|---|---|---|---|---|---|
| 1 | | | | | |
| 2 | | | | | |
| 3 | | | | | |

| | A | B | C | D | E |
|---|---|---|---|---|---|
| 1 | | | | | |
| 2 | | | | | |
| 3 | | | | | |

To select the table, click the rectangle at the intersection of the lines and columns names.

| | A | B | C | D | E |
|---|---|---|---|---|---|
| 1 | | | | | |
| 2 | | | | | |
| 3 | | | | | |

Names (addresses) of table cells are fully equivalent to those used in MS Excel: columns are marked alphabetically (A, B, C, D, ... , Z; AA, BB etc.) while rows are marked with ordinal numbers.

Filling of adjacent cells with data.

To accelerate the data input in the table, it is possible to use the function of automatic data filling. The table editor can automatically continue a line of numbers, number combinations and text with a set pattern. By selecting several cells and dragging the filling marker, it is possible to quickly fill in the data lines with the different types.

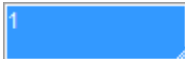


Filling cells with sequences of numbers or combinations of numbers and text with a set pattern.

1. Select the first of the filled cells.



2. Enter the initial value for the values line.



3. Enter a value in the following cell to set the filling pattern.

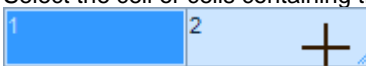


For example, if it is required to set the sequence 1, 2, 3, 4, 5..., enter in the first two cells values of 1 and 2. If the sequence 2, 4, 6, 8... is required, enter 2 and 4. If the sequence 2, 2, 2, 2... is required, the second cell can be left empty.

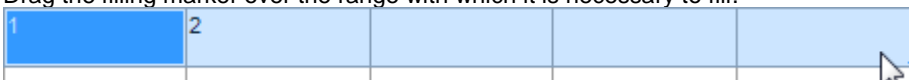
The sequence filling proceeds as shown in the table below.

| Initial value | Line extension |
|----------------------------|-----------------|
| 1, 2, 3... | 4, 5, 6... |
| First period Second period | Third period... |
| Article 1 Article 2 | Article 3... |

4. Select the cell or cells containing the initial values.



5. Drag the filling marker over the range with which it is necessary to fill.



The cells will be filled with the set sequence of numbers.



To fill in an increasing order, drag the marker down or to the right. To fill in a decreasing order, drag the marker up or to the left.

Use the autofilling to continue lists containing the values from collections (main menu/rows/user sorting).

Cell Colours

| | |
|---------------|---|
| =ObjectName | - cell containing an expression;; |
| CSN M 12x120# | - not edited cell, for example, in report ; |
| #ERR | - cell with a mistake in an expression; |
| 60 | - cell containing data of report section ; |

Basic Tools



Open table allows a data set from a previously created *.dat file to be imported, or a standard table template from the nanoCAD library to be imported. Clicking the black triangle shows additional buttons which are used to select the data source.



Insert a table from the library.



Insert a table from an external file *.tbl, *.dat, *.mdb, *.txt, *.csv, *.xml, *.xls, *.xlsx. If an Excel file is chosen, only the first list will be inserted.



Save to file exports the cell data to a special *.dat file format or saves it in the nanoCAD library. Clicking the black triangle shows additional buttons used to select the desired destination.



Save table in nanoCAD library.



Export table to an external file *.dat, *.txt, *.csv, *.xml, *.xls.



Export to Excel exports the table data to MS Excel. Clicking this button creates a new Excel worksheet containing all the table data with identical formatting.



Import from Excel imports calculated data from an open MS Excel worksheet.

The standard windows clipboard tools:



Cut



Copy



Paste



Undo last change.



Redo last change.

Formatted cell navigation tools:



Move row down



Move row up



Move column left



*Move column
right*



Page division. This tool is intended for dividing a table into multiple fragments without losing its integrity. Use this function to split a large table in order to place it on a smaller sheet of paper, while still being able to edit it as a whole

Tools used for sorting cells by contents:



Sort ascend




Sort descend



Create summary function summarises the contents of the chosen cells.

| | A | B | C | D | E |
|---|--------------|---|---|---|---|
| 1 | =summ(A2:E2) | | | | |
| 2 | 1 | 2 | 3 | 4 | 5 |

Select the cell in which it is required to calculate the sum and click the button . Select the cells which are to be summarised and press Enter.



Open calculator opens the [Calculator](#) window.

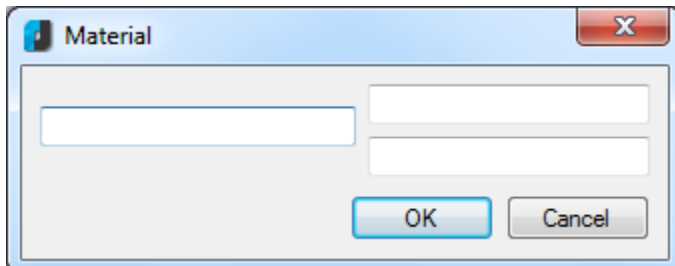


Open Notepad opens the [Notepad](#) window.



Insert material.

Use this tool to insert formatted rows of material into the table. Click the icon to open the [Material](#) dialogue box



The Material dialog box has a title bar with a close button (X). It contains two empty text input fields. At the bottom, there are two buttons: 'OK' and 'Cancel'.



Special Symbols button allows a special symbol to be inserted from the menu.



Recalculate Table is used to manually update the calculated cell data after editing reference cell values or altering expressions.



Automatic calculation operates a mode of automatic recalculation of the values of the table cells. By default, the automatic calculation mode is included



Update reports. Click this button to update the report.



Automatic report update operates a mode of automatic recalculation of the report. By default, the automatic report update mode is included.



Group selection merges multiple cells into a single cell.

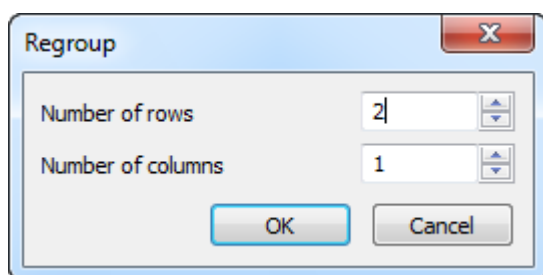


Ungroup selected cells restores the original cells from merged cells.



Change count of rows and columns. Use to change the count of rows and (or) columns in the chosen cells range.

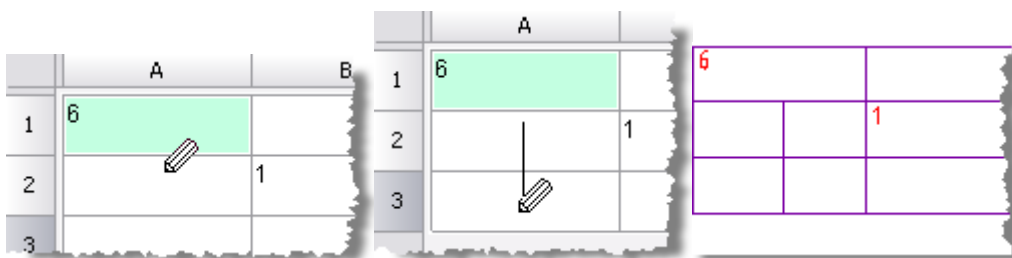
Select one or several cells and click the **Change count of rows and columns** button. In the **Regroup** dialogue box that appears, enter the required values for breakdown.



The **Regroup** dialog box has a title bar with a close button (X). It contains two input fields: "Number of rows" with a value of 2 and "Number of columns" with a value of 1. Below these fields are "OK" and "Cancel" buttons.



Split table cells with pencil tool use to insert new cells by manually drawing new grid lines.



Specify the start and end point of a line which will divide each cell that is crossed by it into two. Right-click on a cell border to delete this border.

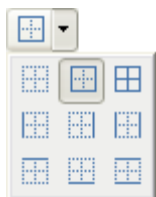


Text alignment use to control the text alignment in the selected cells. Clicking the black triangle brings up additional buttons.





Borders use to display or hide the borders of the selected cells. Clicking the black triangle brings up additional buttons used to toggle border visibility.



No border lines.



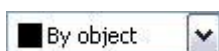
Display all lines around the selected cells.



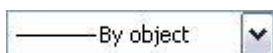
Display all borders - inner and outer.


The remaining tool icons are self-explanatory.

The **Border Colour** pull-down menu:



The **Border Lineweight** pull-down menu:

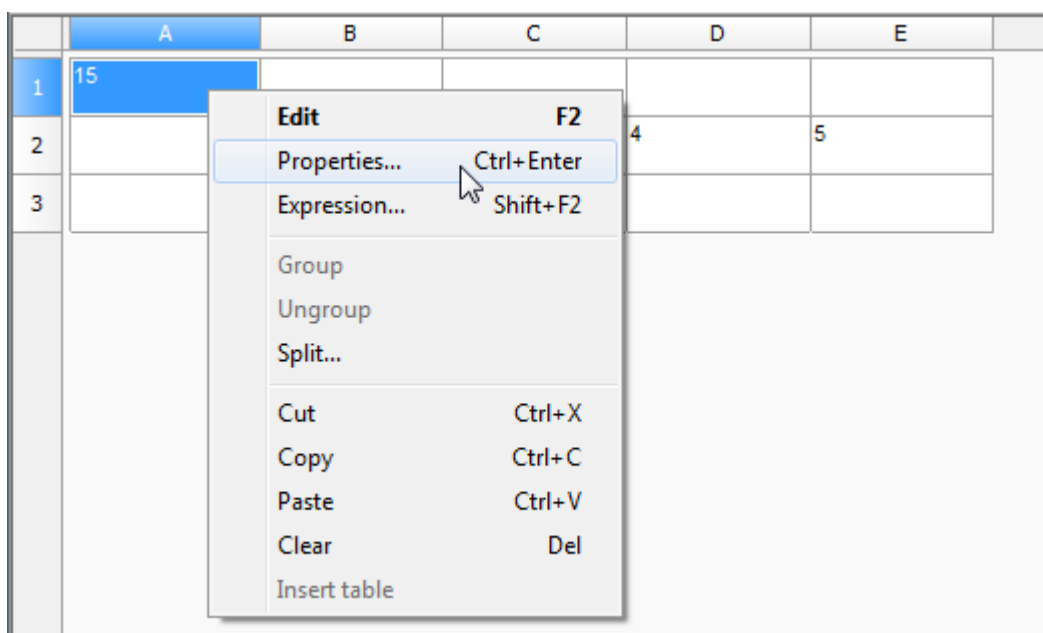


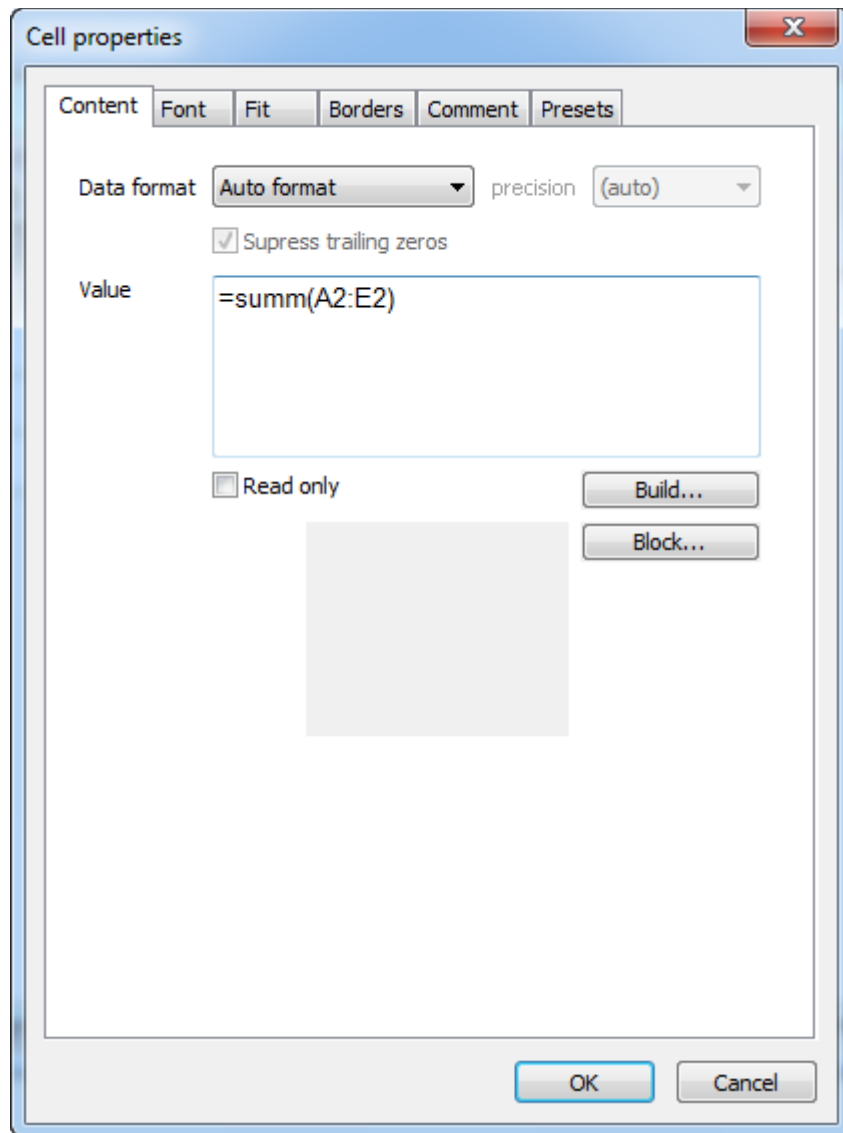
To set the border parameters, it is necessary to select the colour or lineweight again and then to choose the border (use the Borders  command) to which it is necessary to apply the set parameters.

Cell Properties. The parameters of the table cells are set in the [Cell properties](#) dialogue box.

To edit the properties of a single cell using the all-function editor, select the desired cell and open [Cell properties](#) from the context menu. The effect of this command is the same as in the on-screen table editor.

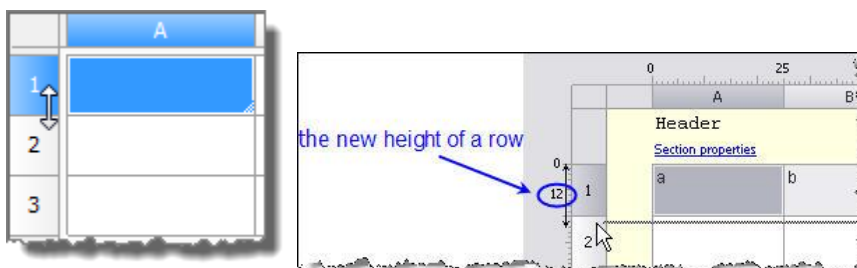
To edit multiple cells in the all-function editor, select the desired cells and open [Cell properties](#) from the context menu or use the **Ctrl+Enter** key combination. The effect of this command is the same as in the on-screen table editor.





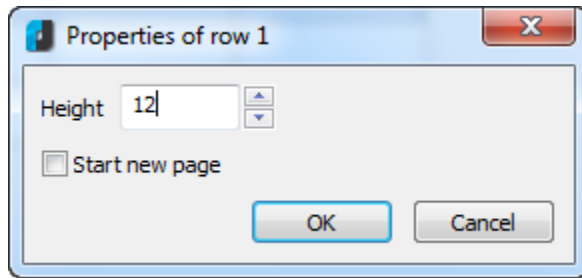
Editing the Size of Rows and Columns

To adjust the height of a row or the width of a column, use the sliders located on the rulers (horizontal and vertical). Move the slider by clicking it and holding down the left mouse button whilst dragging it. While adjusting the height or width, their values are highlighted in colour for easy monitoring.

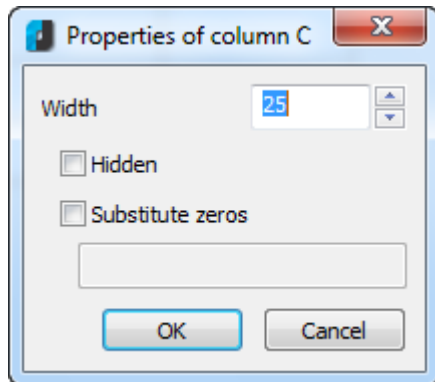


Row height and column width may also be set in the property dialogue box that can be opened by right-clicking the header of a row or column.

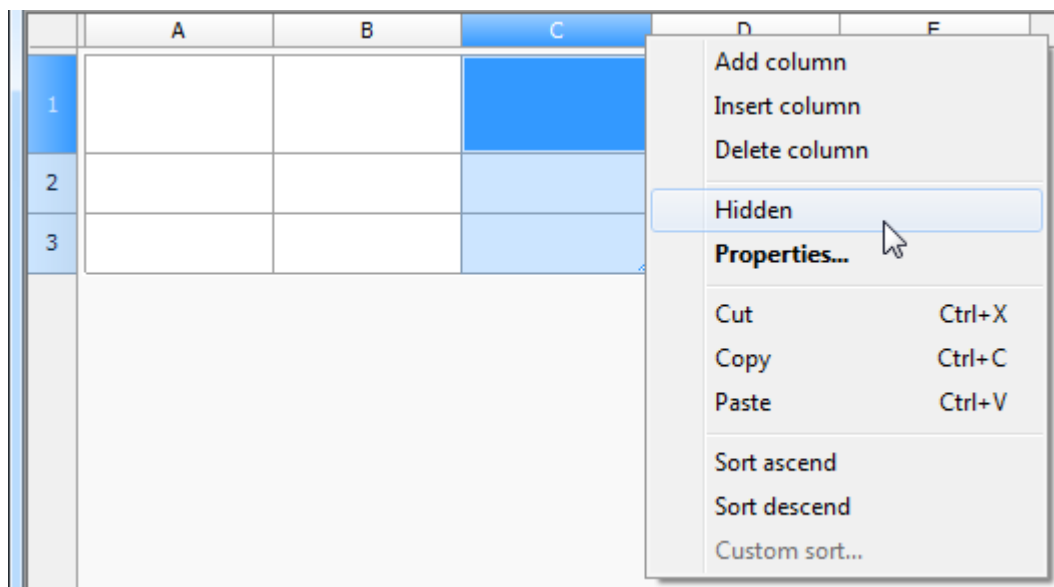
1. In this dialogue box, enter the desired width/height in millimetres.



2. In the **Properties of column** dialogue box, the **Hidden** checkbox is available. Using it, you can hide the display of the selected column on the drawing. It can be displayed in the editor mode.



It is also possible to hide a column using the column context menu.





The **Substitute zeros** command (in numerical cells with a zero value, the cells should have a numerical format) replaces the cells' contents with the set text.


Note! The format of the cell text value is established in the cell properties on the [Content](#) tab.

Close the dialogue box by clicking OK.

Import, Export from MS Excel

The **Import table from Excel**  button imports calculated data from an open MS Excel worksheet.


- ☐ In the open Excel list, select the cell data from which it is required to import to the nanoCAD table.
- ☐ Click the **Import table from Excel**  button in the **Table Edit** dialogue box.
- ☐ The data will be transferred into the table.

The **Export table to Excel**  button exports the table data to MS Excel. Clicking this button creates a new Excel worksheet containing all the table data with identical formatting.

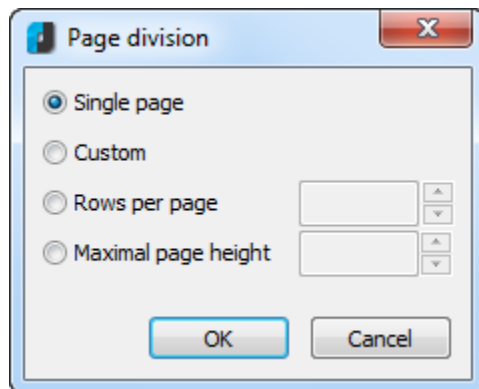
Note! The following data are not exported to Excel from the nanoCAD tables:

- ☐ nanoCAD blocks
- ☐ Formatted text
- ☐ Expression mismatched Excel format

Page Division

 **Page division** divides a table into multiple fragments without losing its integrity. Use this function to split a large table in order to place it on a smaller sheet of paper, while still being able to edit it as a whole.

- ☐ Invoking this command brings up the **Page division** dialogue box.



- ☐ Check the **Single page** box to disable page division.

☒ Single page



- ☐ Choose the division method:

1. **Custom**. The table can be divided into any parts.

☒ Custom

The page break before a row is set by the **Start new page** command from the context menu.

| | A | B | C | D |
|----|--------|---|---|---|
| | Header | | | |
| 1 | Header | | | |
| | Data | | | |
| 3 | a | g | n | |
| 4 | b | h | o | |
| 5 | c | j | p | |
| 6 | d | k | q | |
| 7 | e | l | r | |
| 8 | f | m | s | |
| | Footer | | | |
| 10 | Footer | | | |

| | |
|------------------|---|
| Add row | |
| Insert row | |
| Delete row | |
| Hide row | |
| Show hidden rows | |
| Start new page |  |
| Properties... | |
| Convert to |  |
| Cut | Ctrl+X |
| Copy | Ctrl+C |
| Paste | Ctrl+V |

Close the dialogue box. In the Table edit window the break will be marked by a border, using which it is possible to change a row height.

| | A | B | C | D |
|----|--------|---|---|---|
| | Header | | | |
| 1 | Header | | | |
| | Data | | | |
| 3 | a | g | n | |
| 4 | b | h | o | |
| 5 | c | j | p | |
| 6 | d | k | q | |
| 7 | e | l | r | |
| 8 | f | m | s | |
| | Footer | | | |
| 10 | Footer | | | |

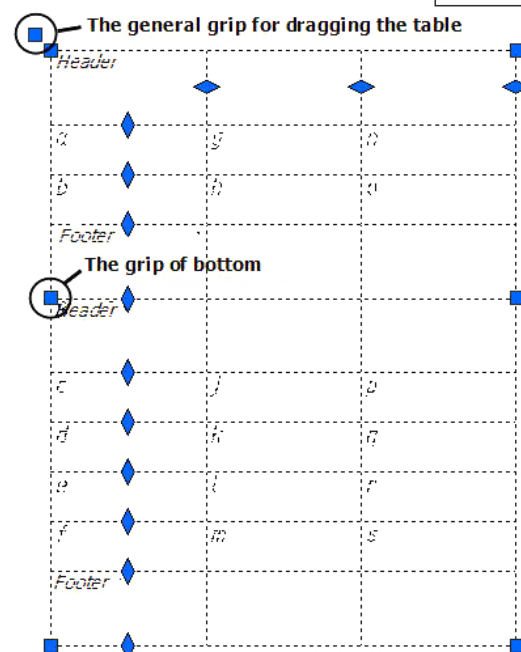
On the drawing, the table with the break will look as follows:

The table without break

| | | |
|--------|---|---|
| Header | | |
| a | g | n |
| b | h | o |
| c | j | p |
| d | k | q |
| e | l | r |
| f | m | s |
| Footer | | |

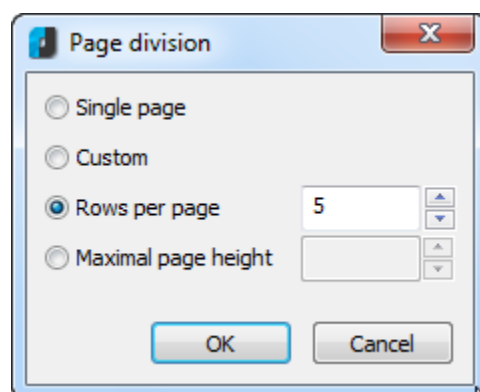
The table with break

| | | |
|--------|---|---|
| Header | | |
| a | g | n |
| b | h | o |
| Footer | | |
| Header | | |
| c | j | p |
| d | k | q |
| e | l | r |
| f | m | s |
| Footer | | |

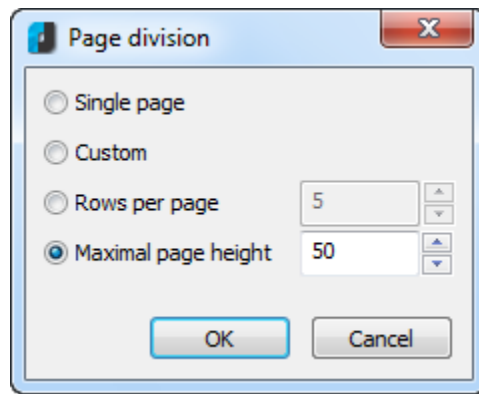


The parts of the divided table can be moved independently of each other within the drawing. Double click on any part to open the table editor.

2. **By row count** enter the desired number of rows in the fragment. Headers do not count.



3. **By maximal page height** enter the total height of the fragment, in millimetres, using the current scale.




Convert table



Menu: **Draw – Tables >**  **Convert table...**



Toolbar: **Tables –** 



Command line: **CONVERTTABLE**

This command is used to convert AutoCAD tables into nanoCAD tables. Further editing of AutoCAD tables can be carried out using nanoCAD tools.

To convert a table, launch the *Convert table*  command and select the table.

The actual sizes of the initial graphics are multiplied by the drawing scale. If the table will have row heights with a zero value, it is necessary to change the drawing scale according to the size of the initial graphics.

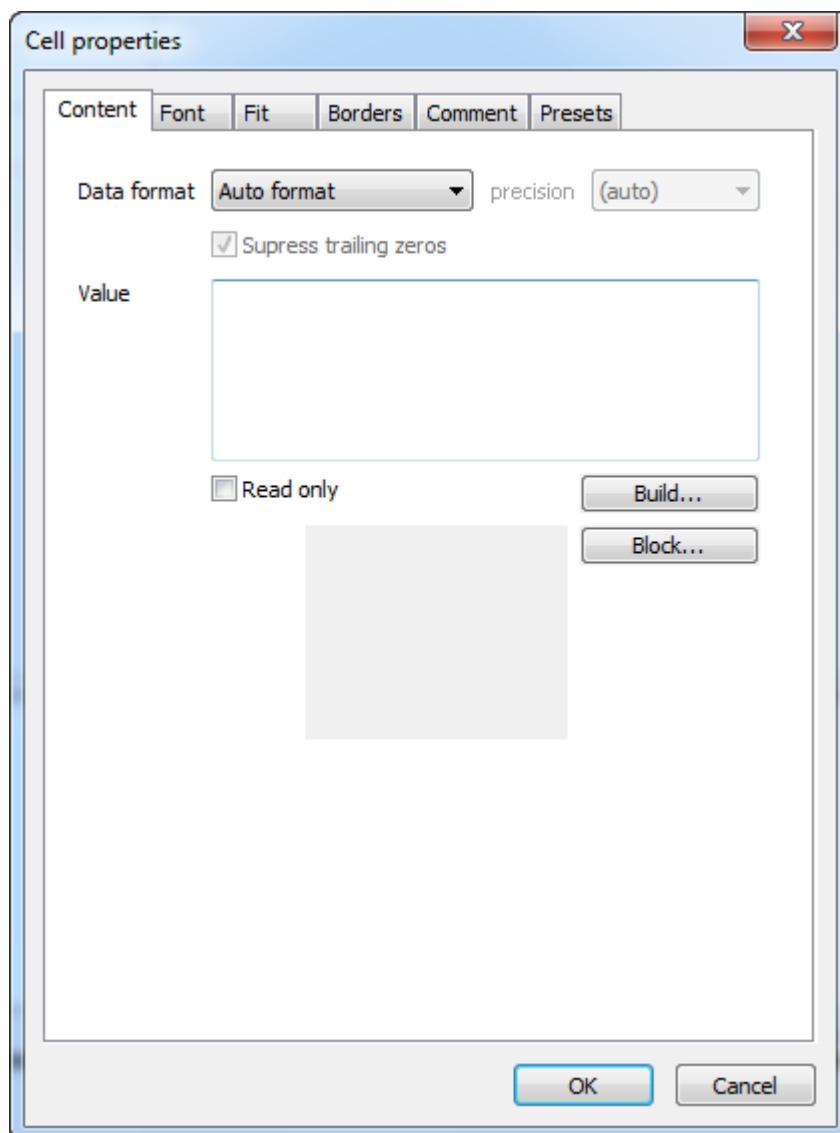
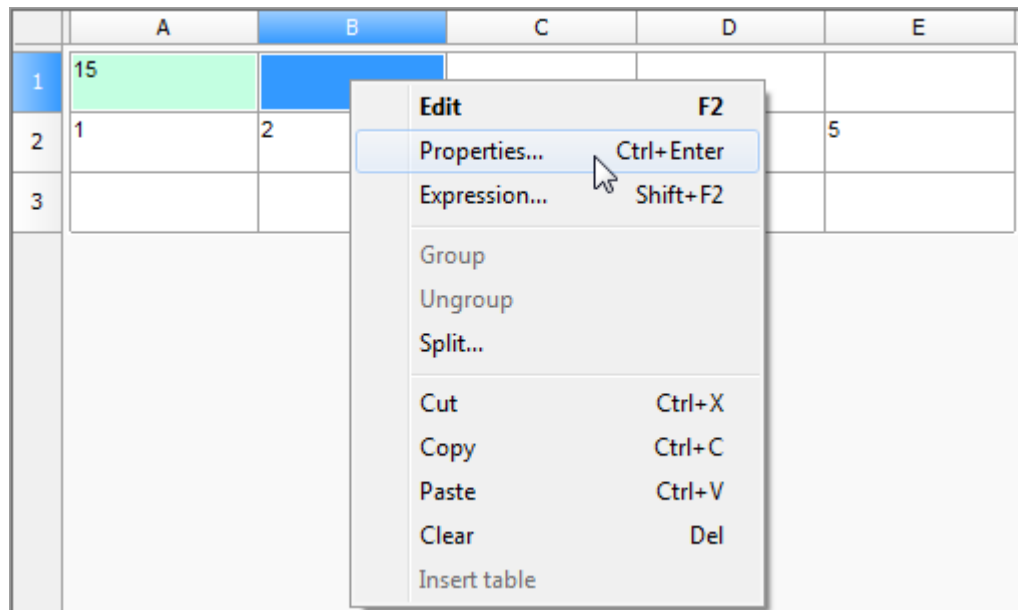
When you convert nanoCAD 3.0 tables to an AutoCAD table, all dynamical links will be destroyed.

Cell properties

You can set the properties of table cells in the **Cell Properties** dialogue box.

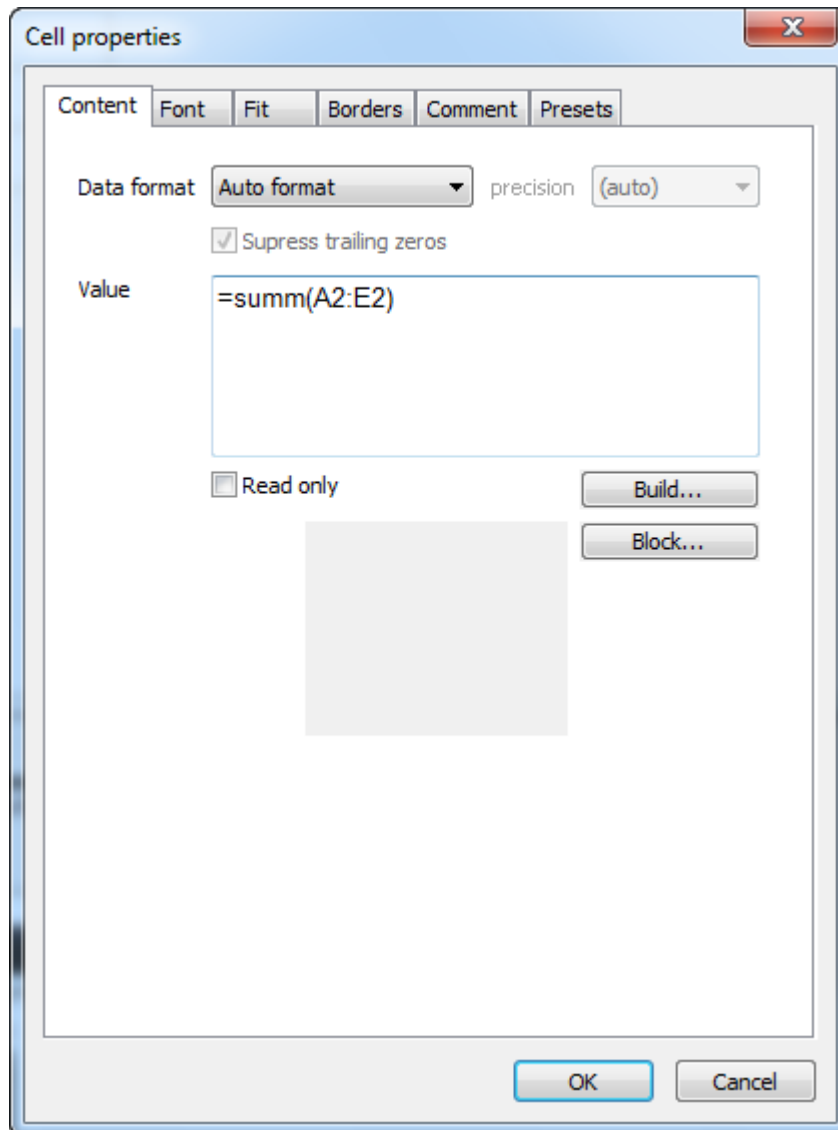
To edit the properties of a cell, select it in the **Table edit** dialogue box and select the **Properties** command from the right-button menu or use the Ctrl+Enter key combination or double click on the cell. The action of this command is similar to **Cell Properties** on the **Table edit** toolbar.

To edit several table cells, select the required cells and select the **Properties** command from the right-button menu. The effect of this command is similar to **Cells Properties** on the **Table edit** toolbar.

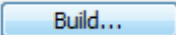


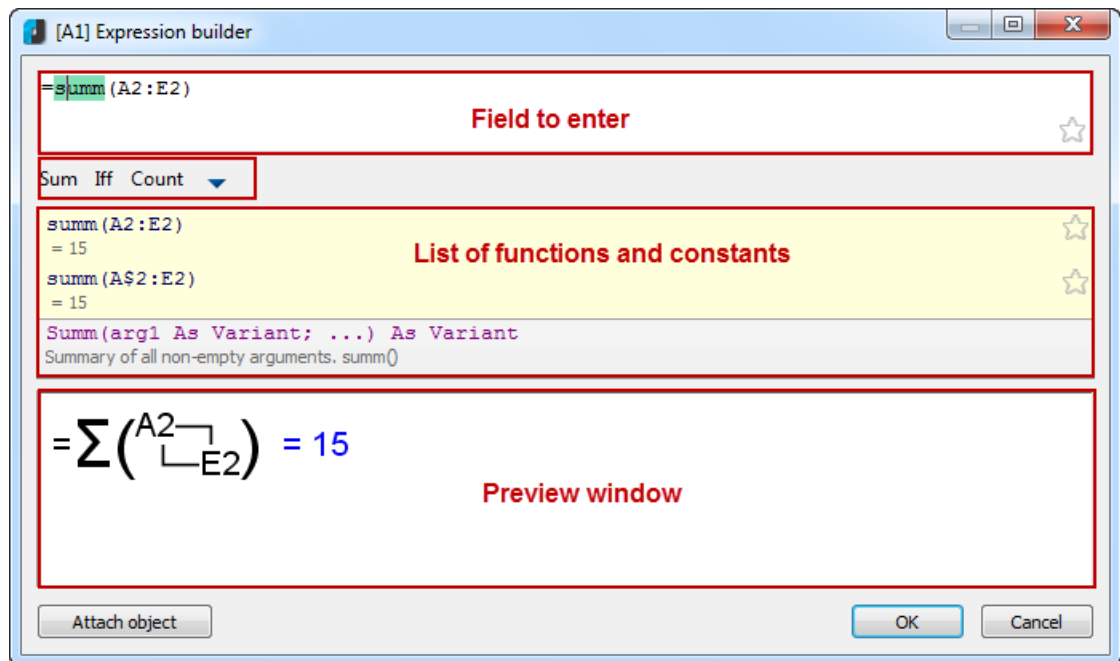
There are six tabs:

- In the **Content** tab you can specify the data format and formula to calculate a value.



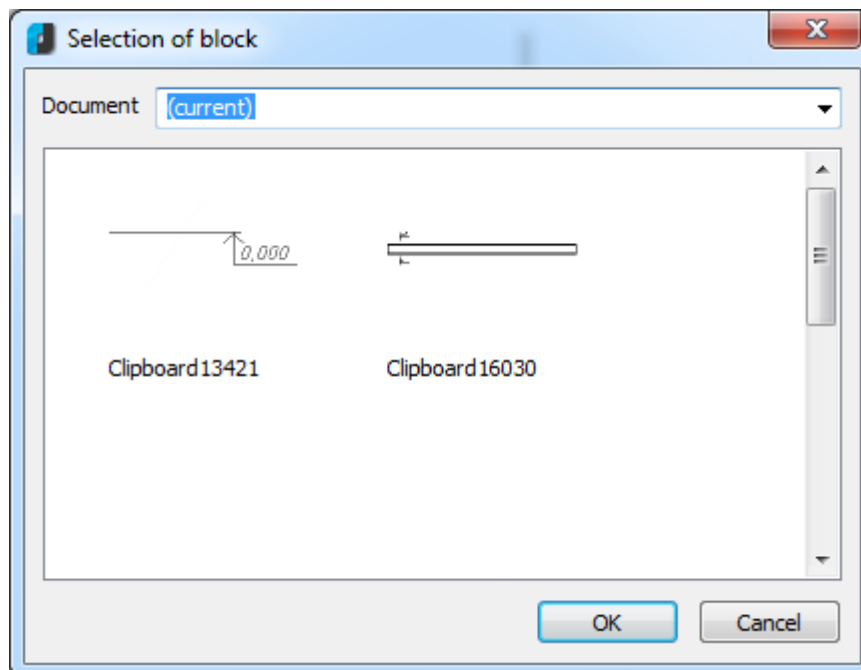
You can specify the data format and enter the value for the cell. Select the **Read only** checkbox to prevent cell editing. The cell is highlighted.

 - - Opens the [Expression builder](#).

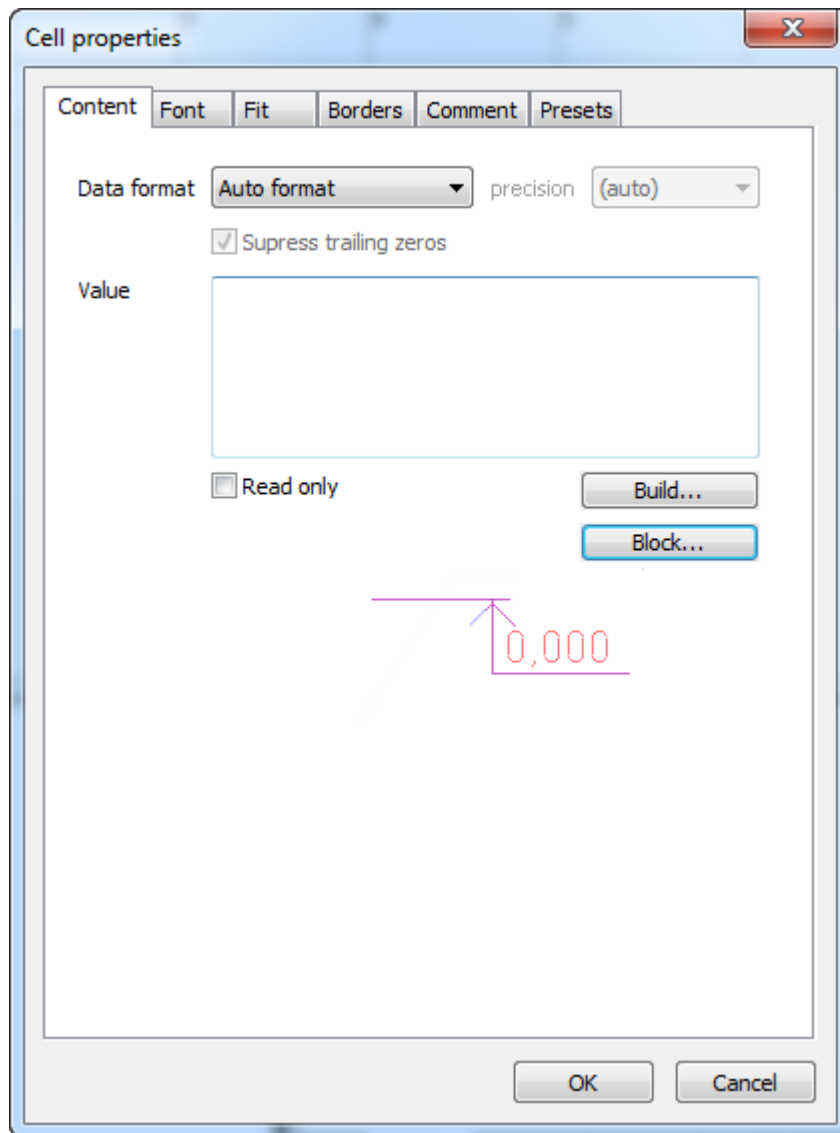


Block... - Insert block into the cell.

Select a block in the current drawing. You can select another file using the *Open* menu.



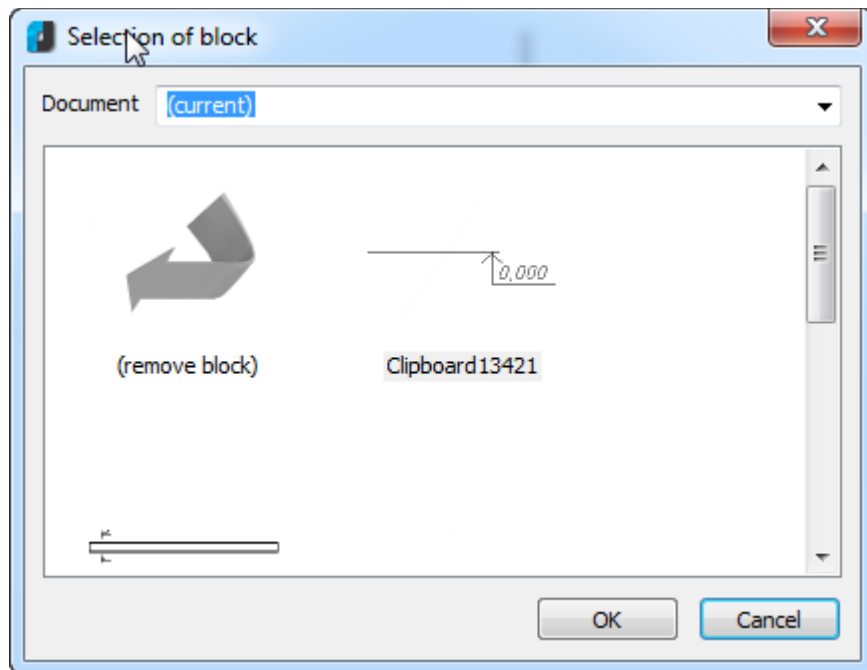
After a block is selected, it is displayed in the cell properties dialogue box and in the table cell.



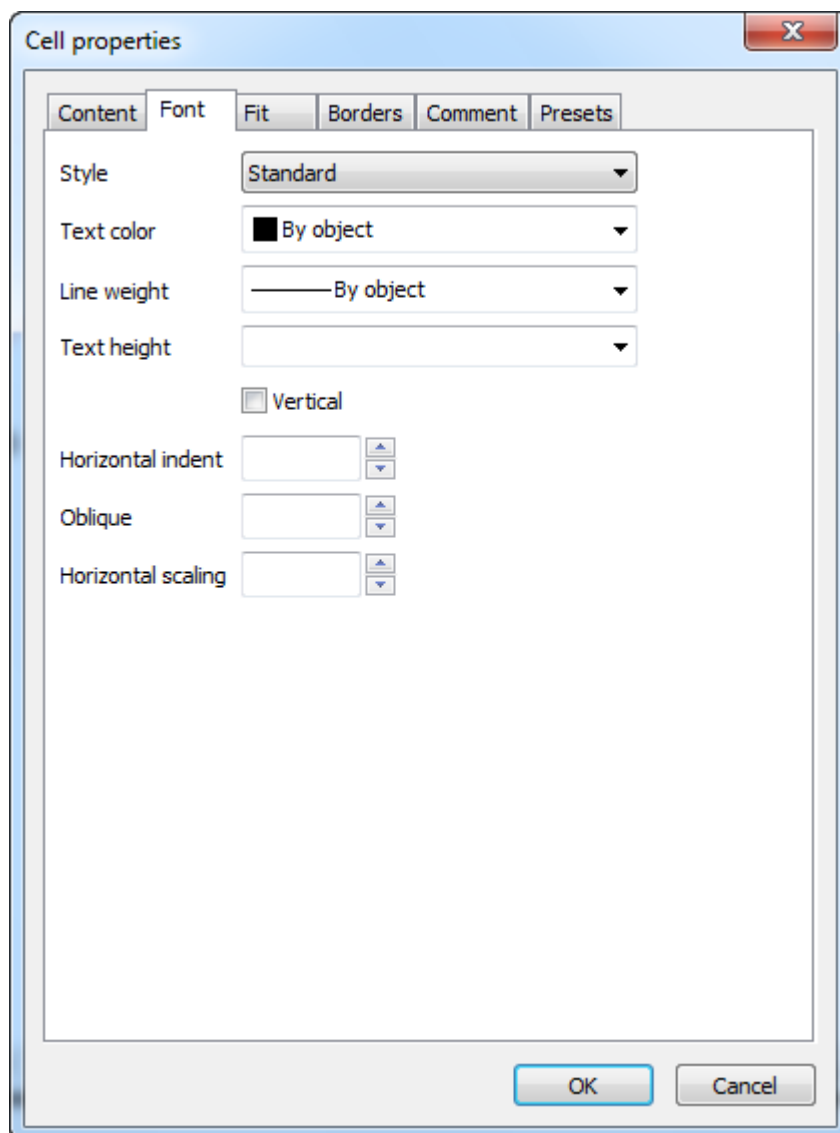
Blocks



To detach a block, select **Remove block** in the **Selection of block** dialogue box.



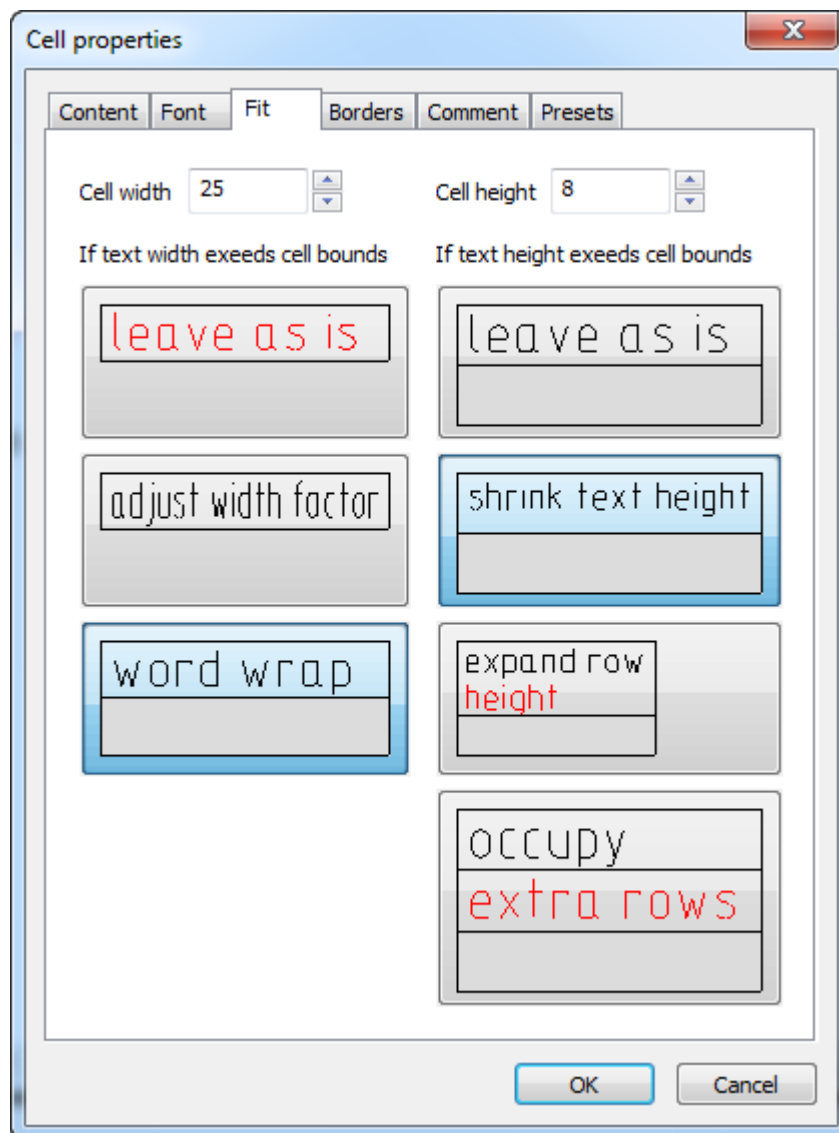
- In the **Font** tab you can specify the text font, symbol colour, line weight, indent, text angle and text scaling.



The **Vertical** checkbox changes the text direction to vertical.

If the **Oblique** and **Horizontal scaling** fields are empty, their values are taken from the text style. If the **Horizontal indent** field is empty, its value is taken from the table settings.

- In the **Fit** tab you can specify the **Cell width** and **height** and fitting parameters.



If text width is more than cell width:

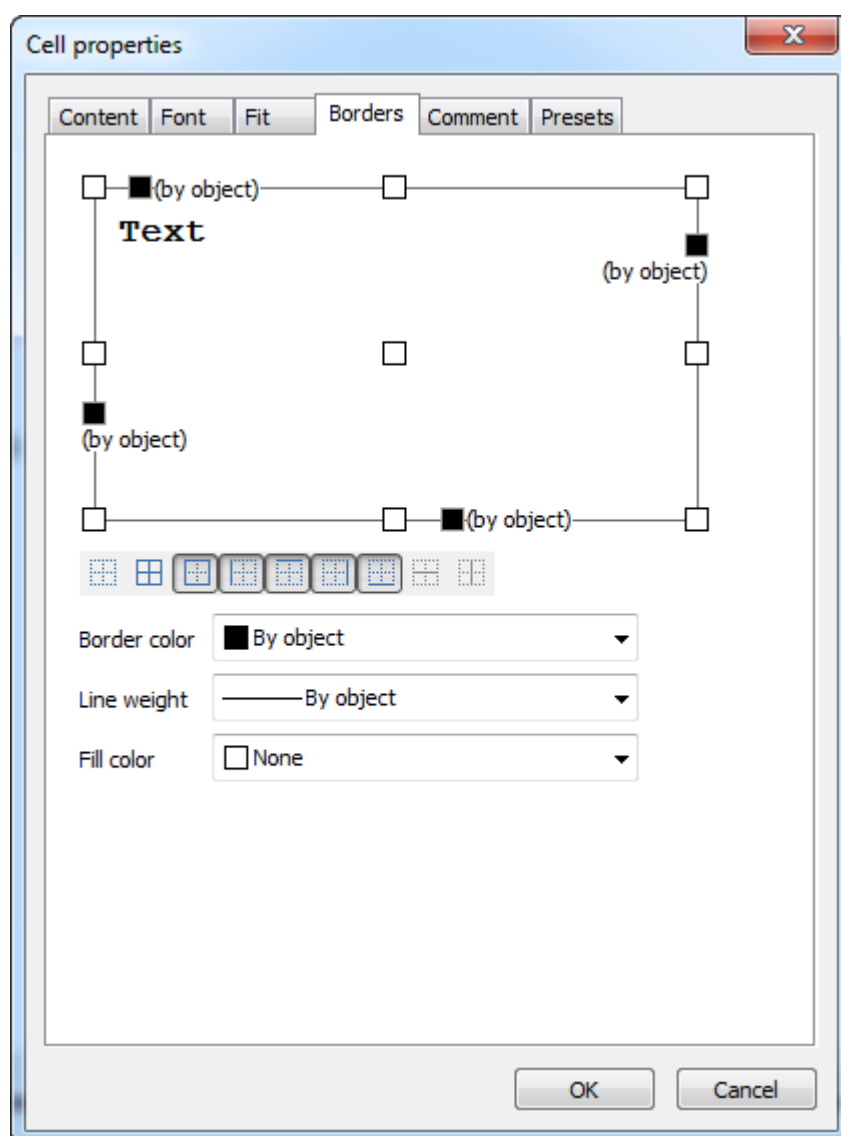
| | |
|--------------------|--|
| leave as is | |
| adjust with factor | |
| word wrap | |

If text height is more than cell height:

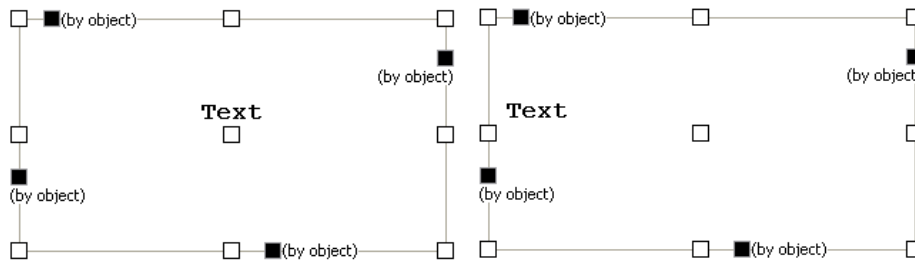
| | | |
|--------------------|--|--|
| leave as is | | |
| shrink text height | | |
| expand | | |
| row | | |
| height | | |
| occupy | | |
| extra | | |
| rows | | |


Occupy extra rows does not change the row's number in the table; the required row is made higher by the required number of times and lined.

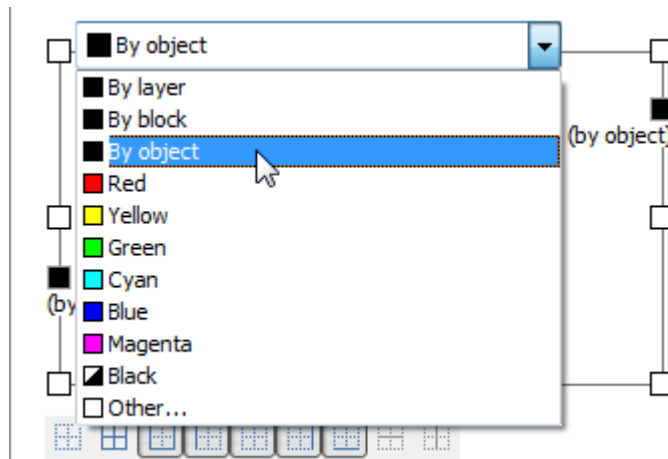
- In the **Borders** tab you can specify the type, weight and colour of the lines of the selected cell. You can control the display of cell borders. To switch border display on/off, select one of the buttons.



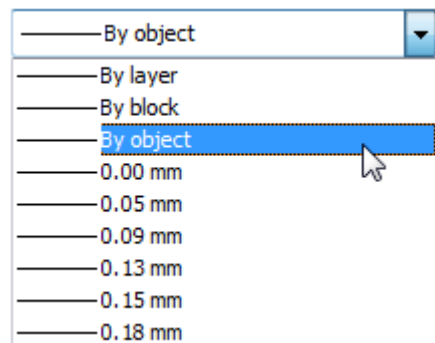
or click near the cell border in the preview area. To align text in the cell, click the symbol □ in the cell.





To specify the colour of the border, click the symbol  and select a colour from the list.



To specify the line weight of the border, select the *Line weight* field and select the weight from the list.



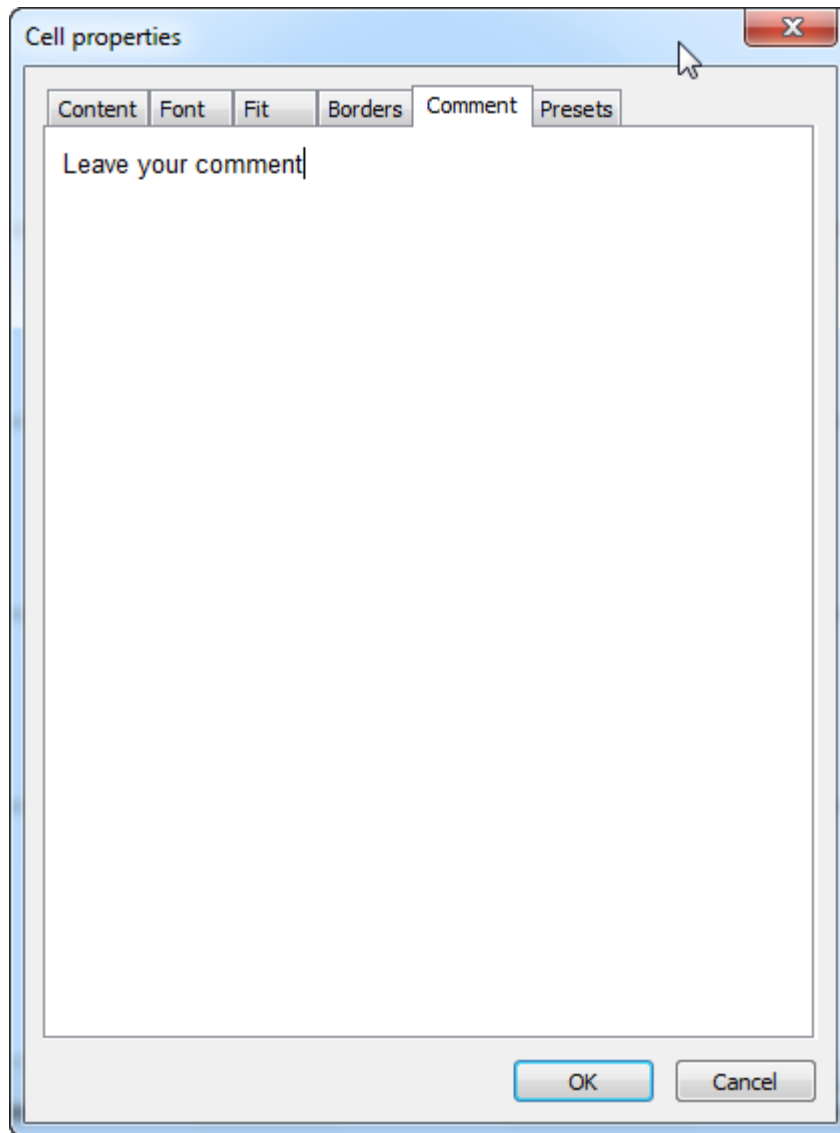
The colour and weight of cells and their fill colour can be specified in the tab.

| | |
|--------------|---|
| Border color |  By object |
| Line weight | —— By object |
| Fill color |  None |

To apply a new colour and weight, click the border or use the buttons.



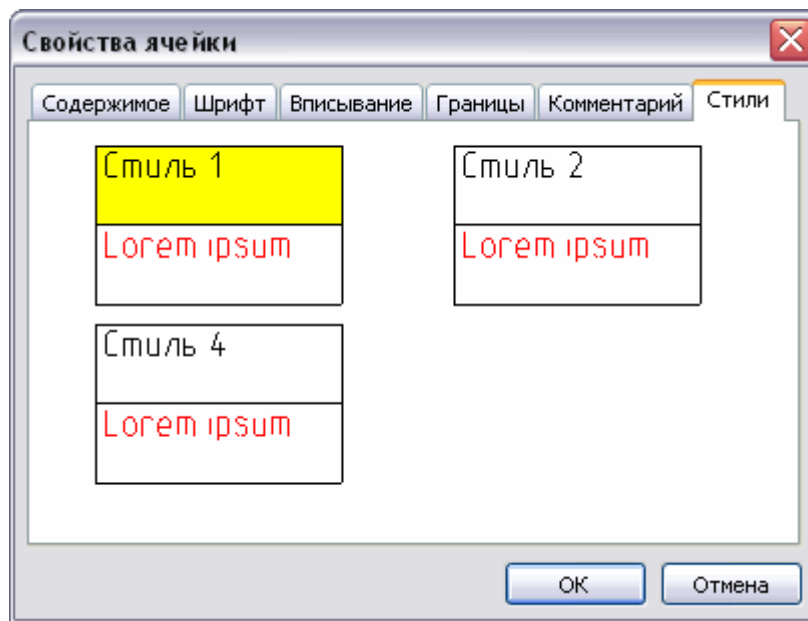
☐ **Comment** tab. Field to enter a comment.



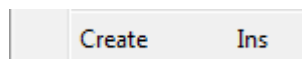
A cell with a comment is displayed with a green label in the editor and when moving the cursor over it the comment is shown.

| | A | B | C | D | E |
|---|----|---|--------------------|---|---|
| 1 | 15 | | | | |
| 2 | 1 | 2 | Leave your comment | | 5 |
| 3 | | | | | |

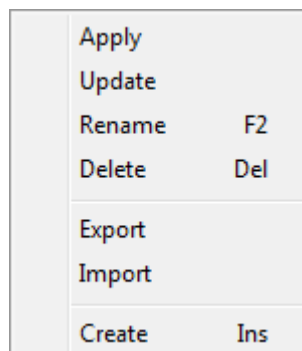
- ☐ In the **Presets** tab you can specify the style for a cell.



To create a style, click on an empty space on the **Presets** tab and from the context menu select **Create**.



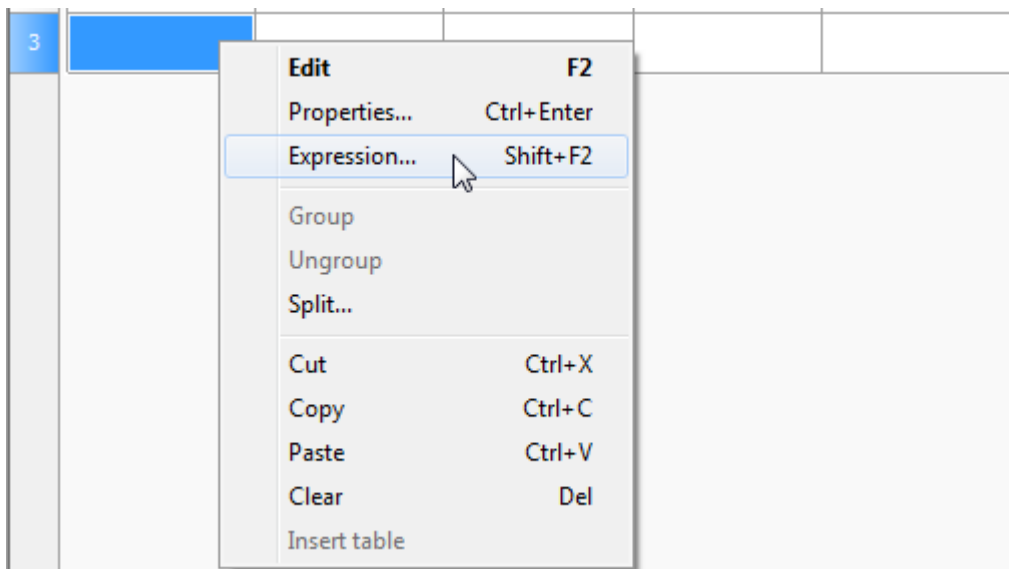
To work with style templates, open the context menu of a style.



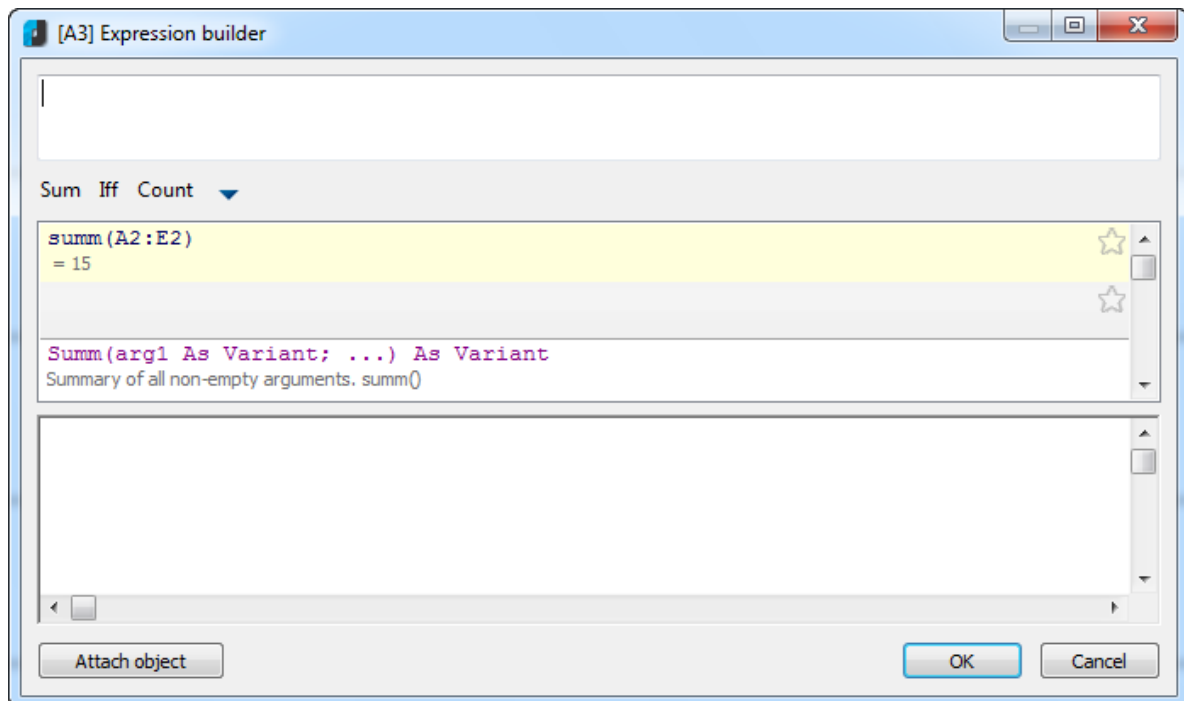
Snap to cell

In every cell formula you can use other object properties. If automatic table recalculation is switched on, the object formula is automatically recalculated when the object is changed. You can attach one or several objects to every cell. Objects have names: Object1, Object2, Object3, ... There is continuous numbering in the table. If an object is not used in any formulas, it is detached from the table during the following recalculations and the object references are renumbered.

To snap object properties to a specified cell, use **Expression** from the right-button menu or press the key combination **Shift+F2**.

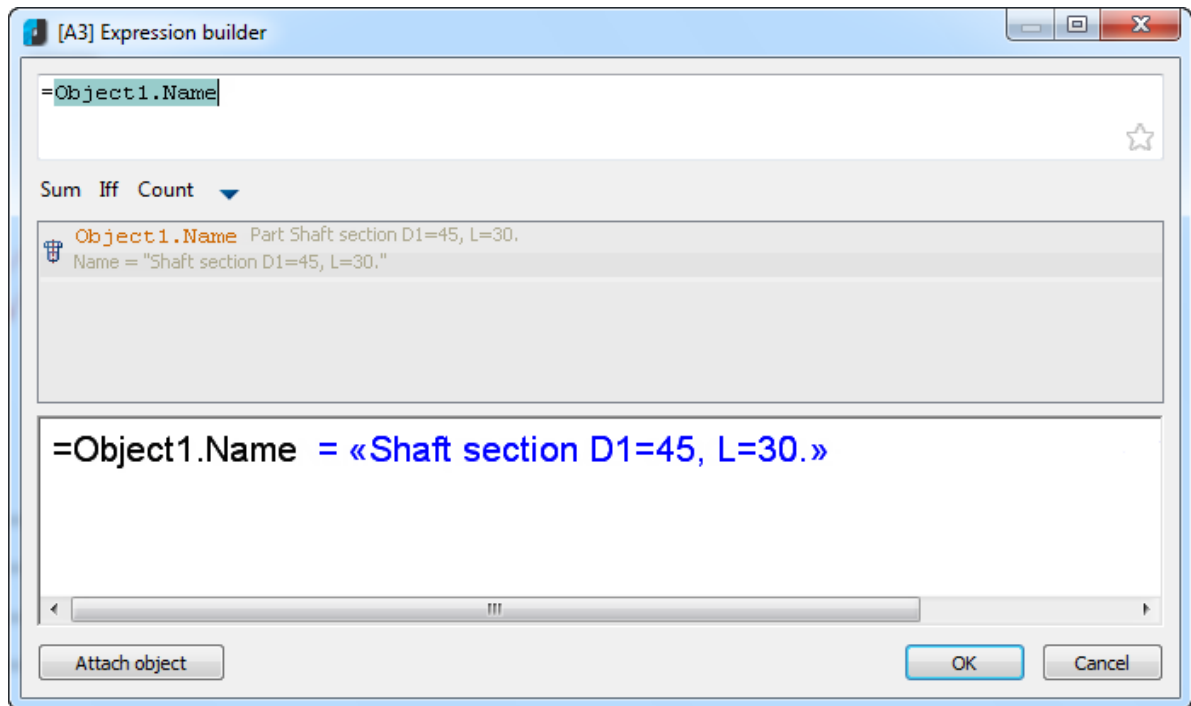


[Expression builder](#) dialogue box opens.



Click the **Attach object** button.

Select the object whose properties you want to snap to the cell. In the properties list you will see the selected object's properties. Double-click it and the property will be added to the cell field. Click **OK**



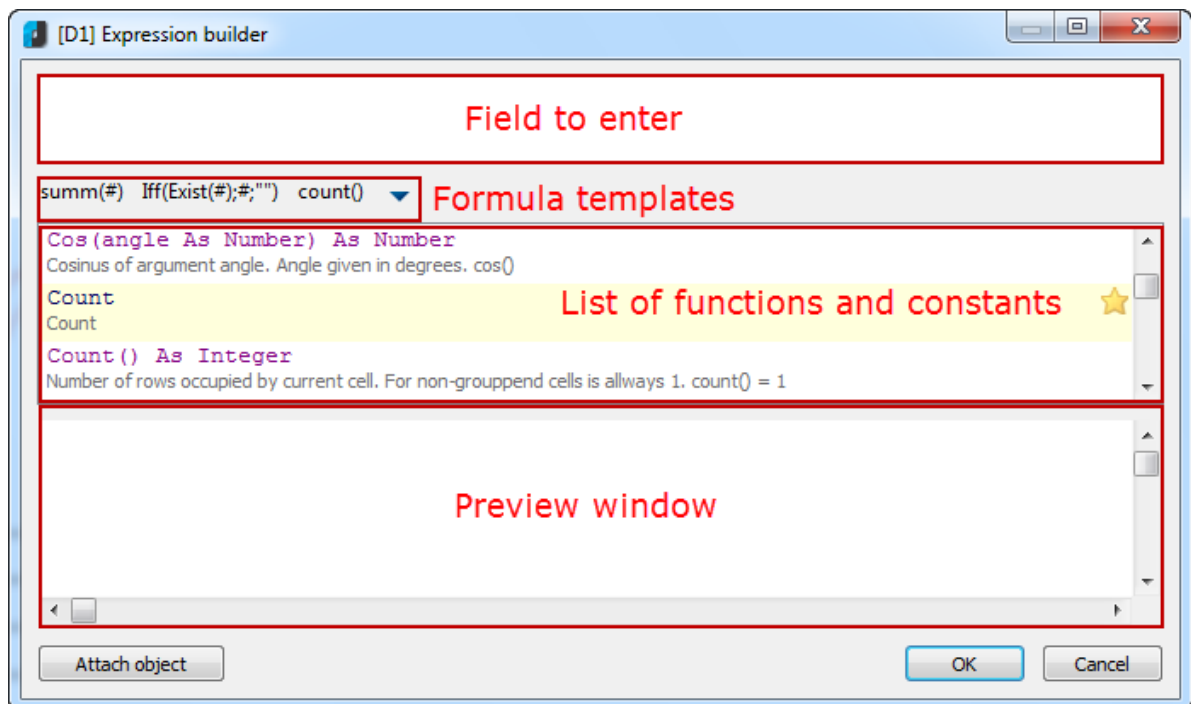
After the object is attached to the cell, the cell colour is changed (it means that there is a formula in the cell) and the calculated result will be displayed:



Expression builder interface

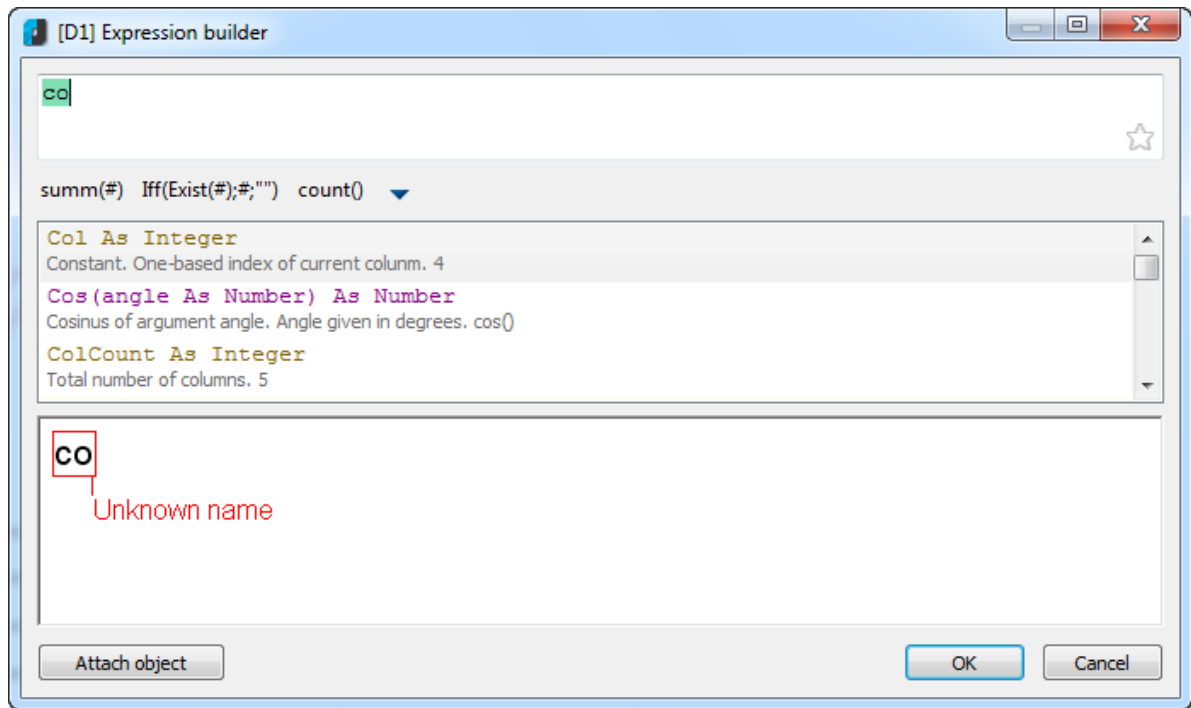
With expression builder you can specify parameters, arithmetic expressions and references to object properties for the selected cell.

You can open expression builder for a table cell and for the **Attach object** dialogue box. Press Shift+F2 to open the dialogue box.

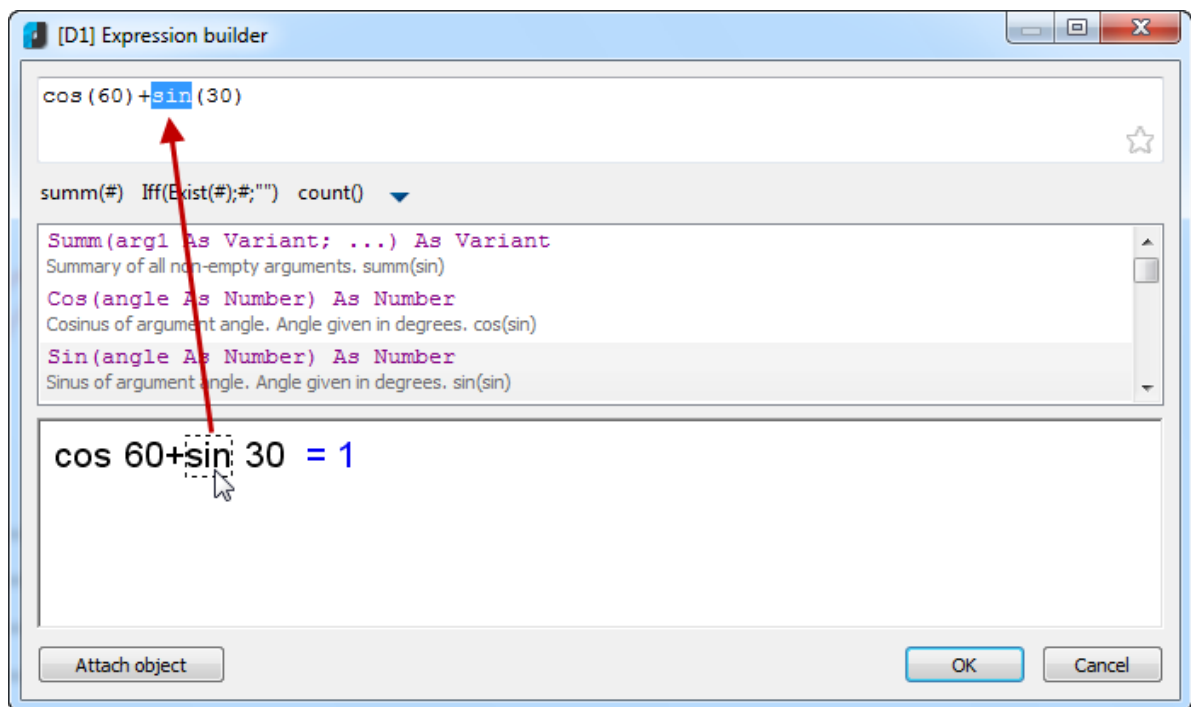


In the top part of the dialogue box there is a field to enter any arithmetic expressions and use any constants and object properties.

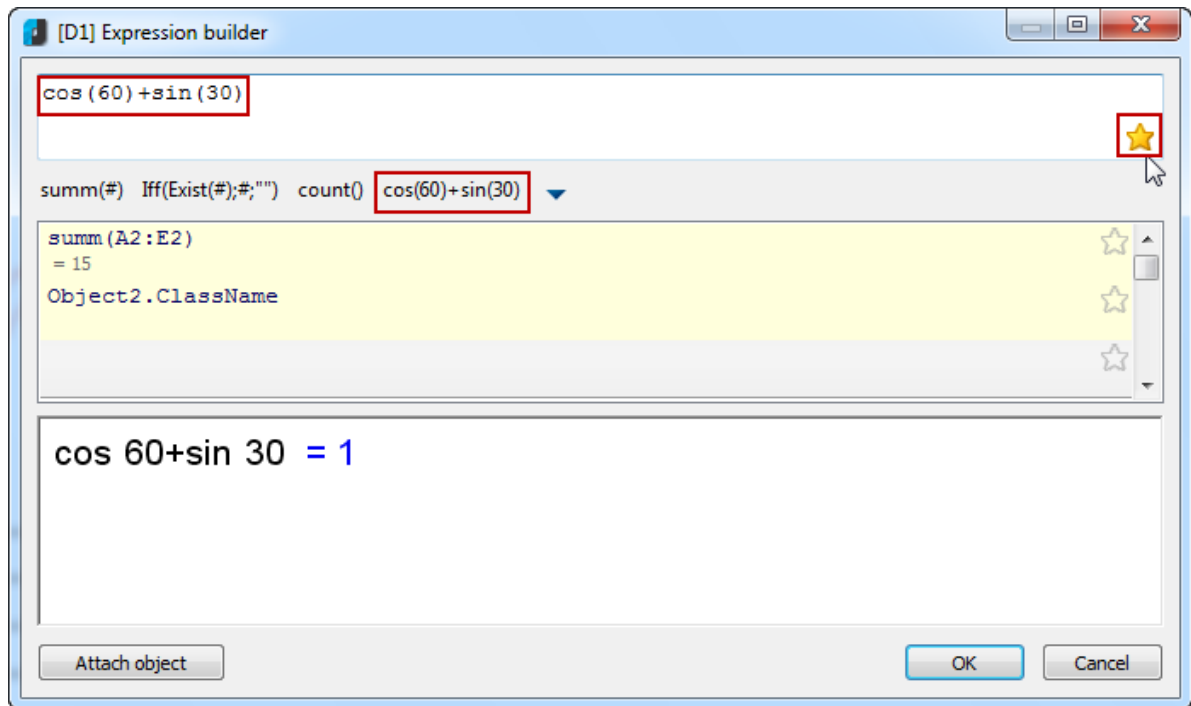
During input, in the list of functions and constants, all items containing input are displayed. If the input is incorrect, a message about the error or a tooltip will be displayed.



Click in the preview and the expression in the input field will be selected.



Press the ★ button and the entered expression will be saved on the template formula bar.



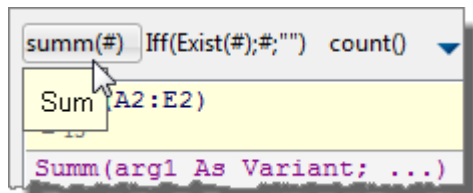
Expressions from the formula template bar are marked with a star in the list of functions and constraints.

Formula templates

These allow expressions to be saved in templates for further use.

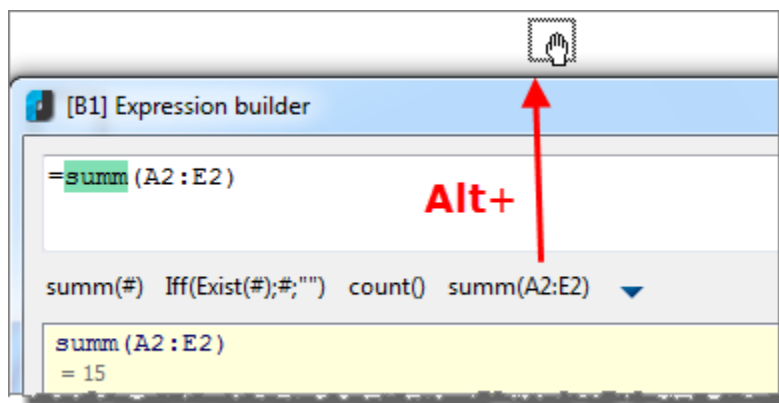
To save an expression, type it in the field and select the  button. The expression's button will be displayed in the templates list.

There is a tooltip when you move the cursor over the item.



To insert a template, select the button of the saved template.

To delete a template, press the *Alt* button and drag the template button outside the **Expression builder** dialogue box.



If there are "\$" symbols in the template, after insertion they are changed to the selected text. For example, we create the *Iff(Exist(\$);\$;"")* template then type the text *Object.Name*, select it and insert the *Iff* template and the symbols are changed to the text - *Iff(Exist(Object.Name;Object.Name;"")*.

Snap to objects

 - [attaches object](#).

Functions in the expression builder

Mathematical operations:

+ Addition
- Subtraction
* Multiplication

/ Division
^ Raising to the power
sqrt() Square-root generation

Logical operations:

== equals
> more
< less
! nor
| or
!= not equals
>= not less
<= not more
|| or
&& and

Functions:

Mathematical:

Cos, Sin, Tg - trigonometric function; argument is specified in degrees.

Acos, Asin, Atg - arc trigonometric functions, the result received in degrees.

Abs - modulus.

Int – rounding of number.

Summ - returns the sum of the variables' values if the cells are grouped.

When entering a function, pay attention to brackets.

For example:

Summ(Row) - returns the sum of the numbers of grouped rows.

Summ(Detal.L) - returns the sum of the parameter L values of the "Detail" object in grouped rows

Data conversion:

Str - data conversion to string type.

Num - data conversion to numeric type.

Frm - numerical value conversion to string type according to the parameters of the table column.

For example:

Frm(0.001230) returns 0.0012 string if zero suppression mode and 0,0000 accuracy are set for the column containing the cell.

Selection and comparing:

FmtText - addition of formatted strings.

FmtSub - creation of lower index.

FmtSuper - creation of upper index.

FmtDigit - number conversion to typesetting form.

FmtRaw - not formatted text.

DmtDiv - creation of fraction.

If/Iff - function of logical decision. Format:

if(Logical_Condition; If_True; If_False,

Where:

Logical_Condition – logical condition with logical operations for comparing;

If_True – returned value, if logical condition is held

If_False - returning value, if logical condition is not held

For example:

if(object == Marker; Marker.Position; "Not determined"). If the object type attached to the column cell has a Marker value, the function returns its Position value. If the object has another type, the *Not determined* value is returned.

Exist - Checks the existence of a constant: =IF(EXIST(Object.Name);Object.Name;0)

Min/Max - returns the minimum/maximum value listed in the brackets

For example:

Min(maximum_value; minimum_value) – returns minimum_value;

Max(maximum_value; minimum_value) - returns maximum_value.

Count - returns the number of objects (for grouped rows of a table).

For example:

Count() - returns number of objects attached to the cells of the grouped rows.

Avg - calculates arithmetical average of arguments. It can use any number of arguments of any ranges. Null arguments are not used.

Off - returns a cell value, specified by a relative index. The index format is:

Off(row; column)

For example:

Off(-1;2) - returns the cell value located one row above and two columns to the right of the current cell

Cell returns the cell value specified by an absolute index.

Merge (Cells,Expression) - merges the Cells in a range. If the Expression is not equal to zero, it returns the Expression value.

For example:

=merge(A5:C5; "Name") – merges cells from A5 to C5 and in the results cell, the “Name” text is displayed.

SetHeight() – specifies the column height. Height value is specified in brackets.

Val() - calculated argument.

For example:

`=val("A"+"1")` - calculates "A1", and uses it as an expression (value in A1 cell). If argument is not a row, it is returned.
`=val(10+2)` – is the same as `=10+2`.

`=val("summ(A"+Str(off(0;-1))+":D"+Str(off(0;-1))+")")` - sum of cells from A to D of row, whose number is in the cell to the left of the current one. In general, you can do without `val`. It is needed when you have to enter a cell address in user form to take the value from it further. Create the `Addr` variable, attach it to the input field in the form and enter in the table: `=val(Addr)`. Function allows recurrence: `=val(val("A"+"1"))` - takes its value from the cell whose address is in cell A1. Number of enclosures is limited to 64.

Geometry(Object) - works as **Attach object**. The height of the attached object is fitted to the cell height. The object is taken from the report and attached to the cell.

SUMM(Section(-1)) - Sum of the current column's cells in the section, following after the current section.

COUNT(Section(A3)) - Number of rows in the section containing cell A3.

Reserved variables:

Pi - pi character.

Row - returns the row number for every cell (rows are numbered from 1, the header row is not numbered).

Col - returns the column number for every cell (column "A" is number 1).

Object - returns the object type, attached to row cells.

Object1, Object2, ... - objects attached to the cell.

Title - table name.

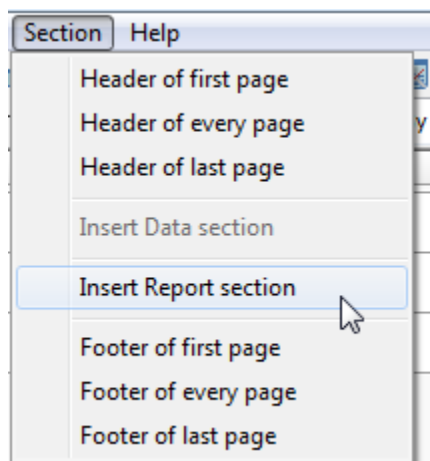
RowCount - general number of rows.

ColCount - general number of columns.

Report creation

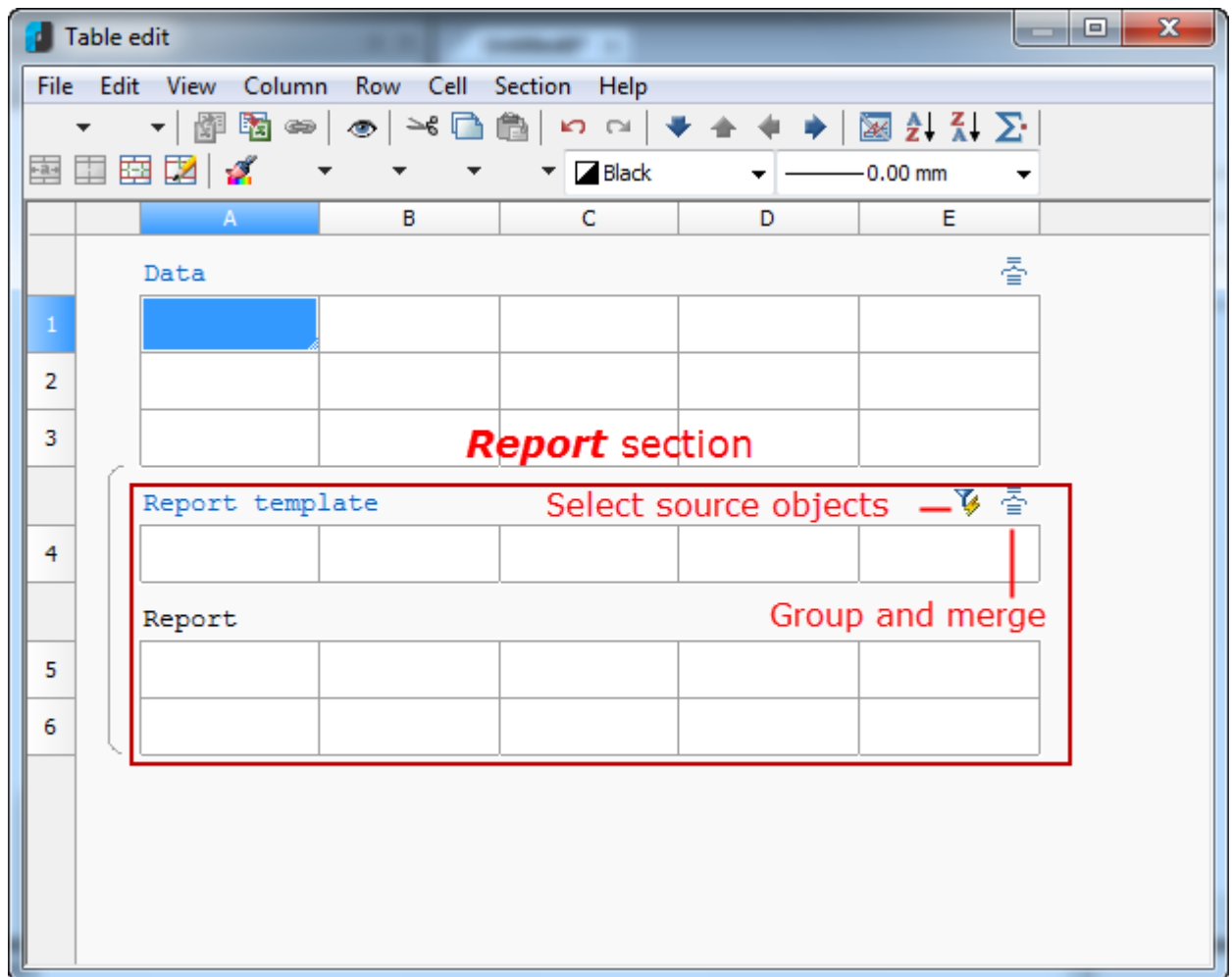
Reports are used to arrange information about objects in the drawing.

To create a report, select **Insert Report section** from the **Section** menu in the table editor.



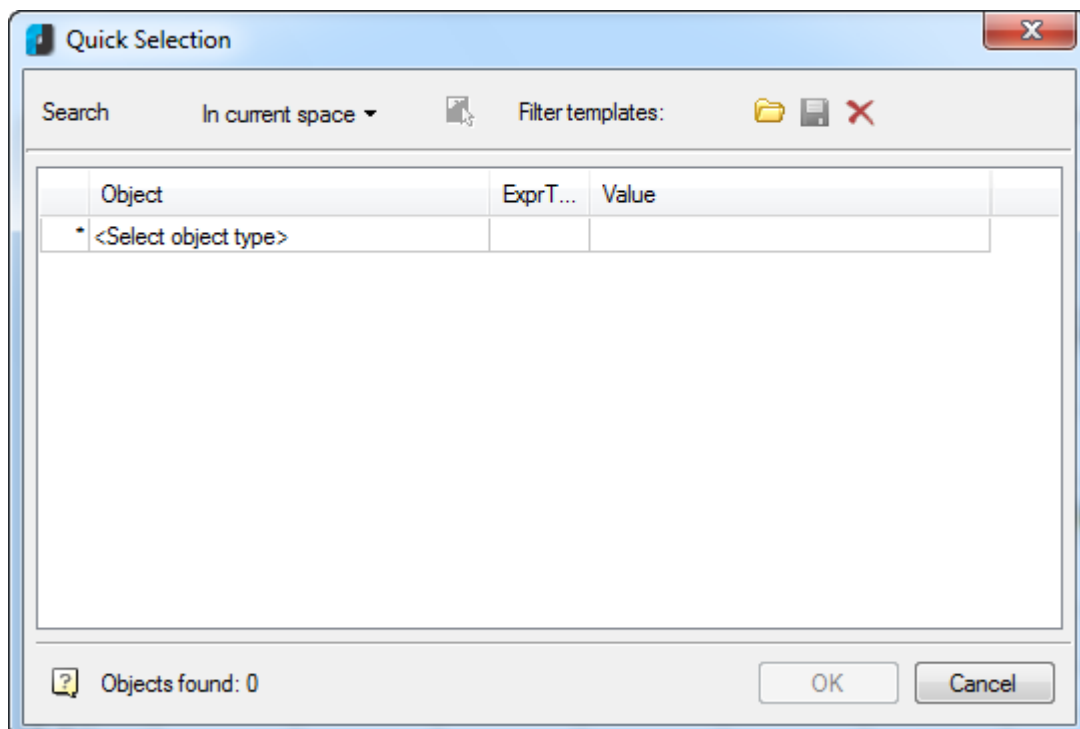
A report consists of two sections: **Report template** and **Report**.


The report template defines how the content is displayed in the report and can contain one or several lines. A copy of the template rows is created for every object. Formulas are calculated according to the properties of the selected objects.



The report template contains the variables of the selected objects; the values of the template variables are displayed in the report.



Using the **Select source objects**  tool you can select inserted objects according to the required conditions.



In the [Grouping and merging](#)  dialogue box you can specify the settings for grouping and merging table cells.

Report template menu:

- **Object filter** opens **Quick selection**;
- Merging and grouping allows grouping and merging cells by properties;
- Freeze report switches off **Automatic update**;
- Report header adds a report header row to the beginning of the report;
- Report footer adds a report summary row to the end of the report;

| | A | B | C | D | E |
|----|---|---|---|---|---|
| | Report template   | | | | |
| 4 | | | | | |
| 5 | | | | | |
| | Report | | | | |
| 6 | | | | | |
| 7 | | | | | |
| 8 | | | | | |
| 9 | | | | | |
| 10 | | | | | |
| 11 | | | | | |
| 12 | | | | | |
| 13 | | | | | |
| | Report summary | | | | |
| 15 | | | | | |

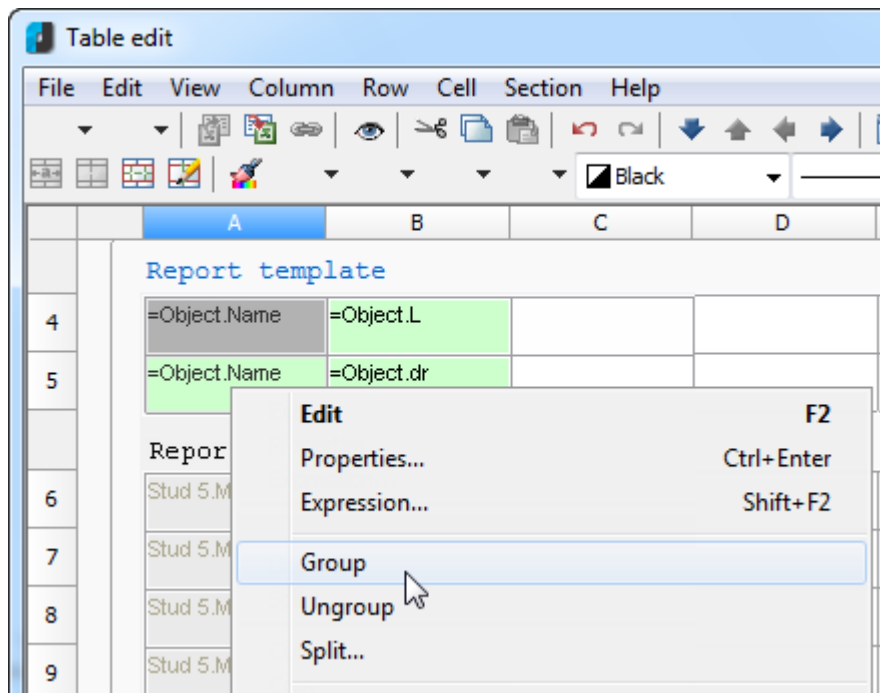
Context menu of the **Report summary** cells:

| |
|---------|
| Sum |
| Average |
| Minimum |
| Maximum |
| Count |

The action applied to the column fragment in one section.

Vertical grouping

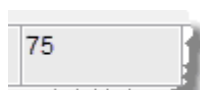
A report template can contain several lines. Cells with identical values can be merged. From the right-button menu select **Insert row** or **Add row**.



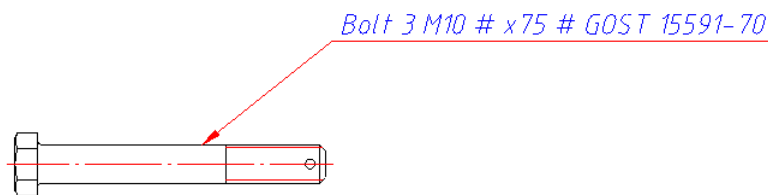
The selected cells will be grouped. The table will be displayed as follows:

| | |
|---|-----|
| Stud 5.M24x500 # GOST 24379. 1-80 | 500 |
| | 24 |
| Stud 5.M42x500 # GOST 24379. 1-80 | 500 |
| | 42 |
| Stud 8. M30x300 # GOST 24379.1- 80 | 300 |
| | 30 |
| Stud 8. M16x300 # GOST 24379.1- 80 | 300 |
| | 16 |

There are cells in the report highlighted with special colours.



If you edit such cells, changes are applied to the object to which they belong.



Create a table and report template.

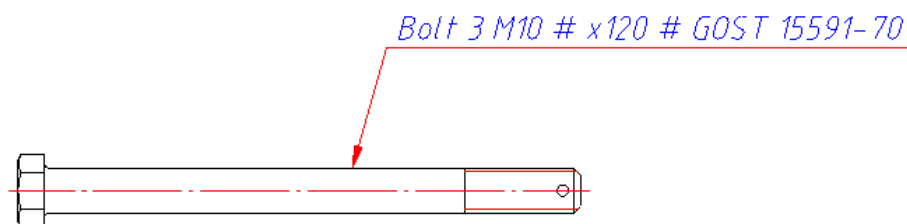
Add the parameters: **=Object.Name** and **=Object.L**.

| A | B | C |
|------------------|-----------|---|
| Report template | | |
| =Object.Name | =Object.L | |
| Report | | |
| Bolt 3 M10 # x75 | 75 | |

Enter a new length, for example 120.

| A | B | C |
|-------------------|-----------|---|
| Report template | | |
| =Object.Name | =Object.L | |
| Report | | |
| Bolt 3 M10 # x120 | 120 | |

The bolt's length is changed.

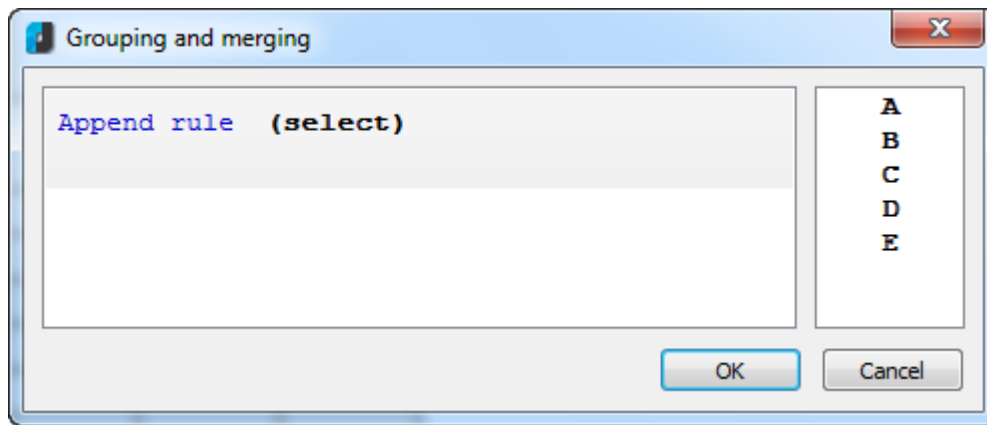


merging cells

Group and merge  button. Report template.

In the *Grouping and merging* dialogue box you can specify settings for grouping and merging table cells. Grouping and merging are only applied to cells in the [report template](#).

Grouping and



Study the example for how to merge cells.

Table edit

File Edit View Column Row Cell Section Help

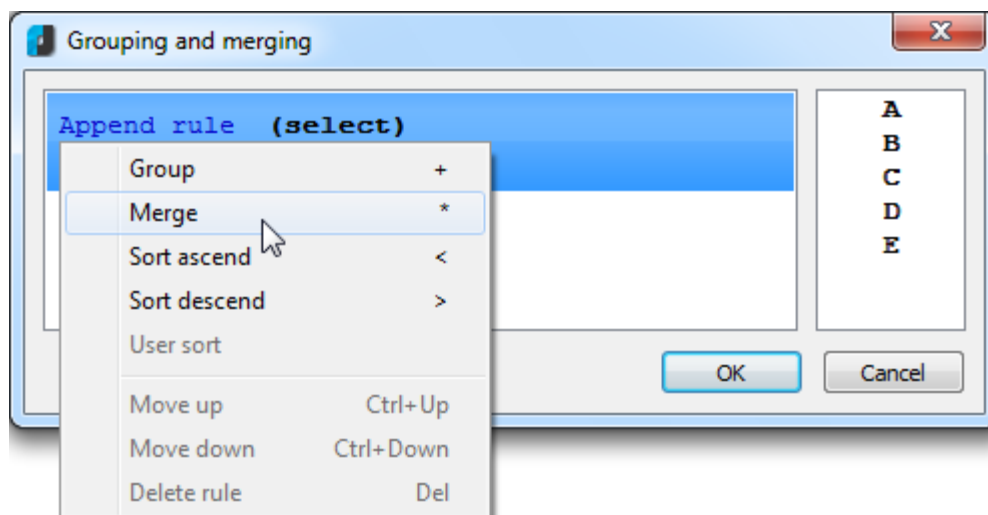
Report template

| | A | B | C | D |
|----|-------------------|----------------|------------|-----------|
| 4 | =Object.ObjectDes | =Object.Name | =Object.dr | =Object.H |
| 5 | Report | | | |
| 6 | GOST 12204-72 | Leg7034-0596GO | 10 | 63 |
| 7 | GOST 12204-72 | Leg7034-0597GO | 10 | 80 |
| 8 | GOST 12204-72 | Leg7034-0599GO | 12 | 63 |
| 9 | GOST 12204-72 | Leg7034-0591GO | 8 | 32 |
| 10 | GOST 12204-72 | Leg7034-0594GO | 10 | 40 |
| 11 | GOST 12204-72 | Leg7034-0598GO | 12 | 50 |
| 12 | GOST 12204-72 | Leg7034-0595GO | 10 | 50 |
| 13 | GOST 12204-72 | Leg7034-0593GO | 8 | 50 |
| 14 | GOST 12204-72 | Leg7034-0592GO | 8 | 40 |

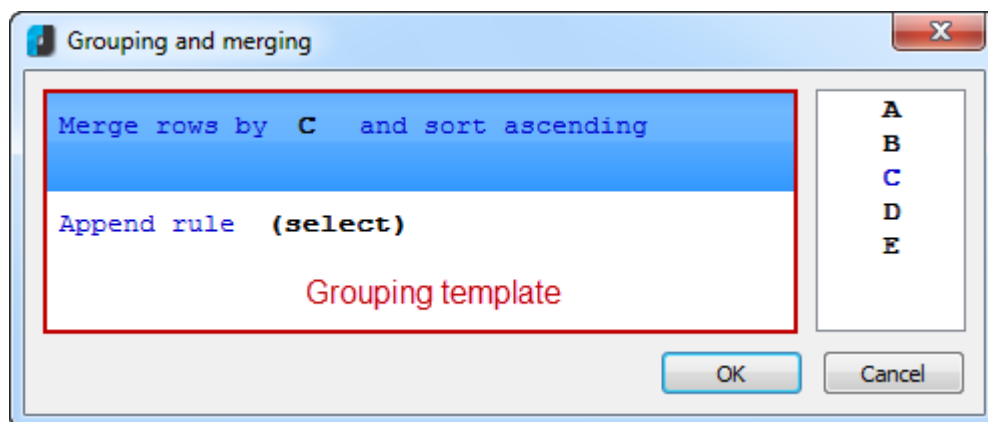
| | | | |
|---------------|---------------------------|----|----|
| GOST 12204-72 | Leg7034-0596GOST 12204-72 | 10 | 63 |
| GOST 12204-72 | Leg7034-0597GOST 12204-72 | 10 | 80 |
| GOST 12204-72 | Leg7034-0599GOST 12204-72 | 12 | 63 |
| GOST 12204-72 | Leg7034-0591GOST 12204-72 | 8 | 32 |
| GOST 12204-72 | Leg7034-0594GOST 12204-72 | 10 | 40 |
| GOST 12204-72 | Leg7034-0598GOST 12204-72 | 12 | 50 |
| GOST 12204-72 | Leg7034-0595GOST 12204-72 | 10 | 50 |
| GOST 12204-72 | Leg7034-0593GOST 12204-72 | 8 | 50 |
| GOST 12204-72 | Leg7034-0592GOST 12204-72 | 8 | 40 |

Merging

1. In the **Grouping and merging** dialogue box select **Append rule** and select **Merge**.



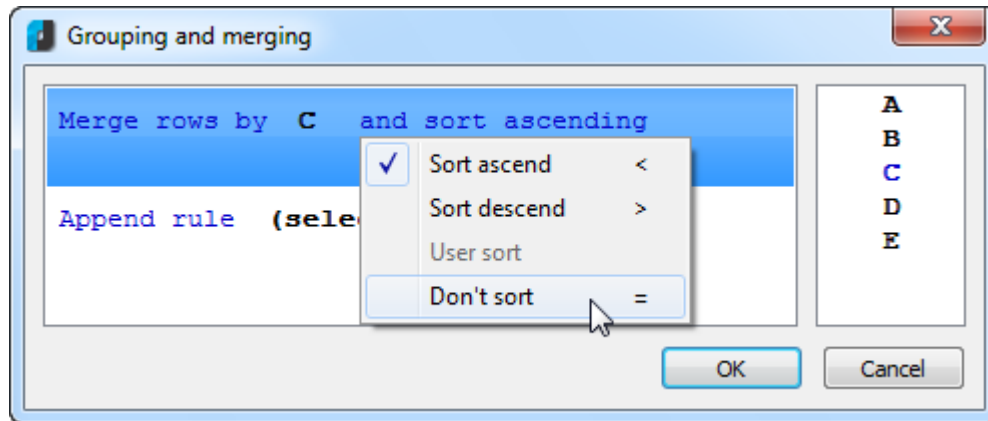
2. Select the column name for the cells you want to merge. Selection is made by double clicking on the column name or by dragging it into the **Select** field.



Note! To select another column, you should drag it into the column list and then select it. You can select several columns then place them in the required order.

The columns are processed according to the rules of the **grouping template**.

3. If you want to sort, select **and don't sort** and select sort type.

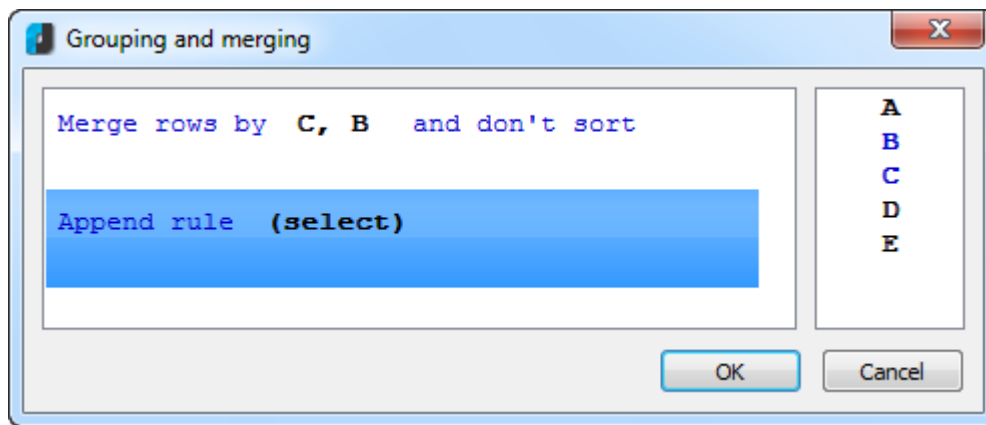


4. Click **OK** to check the result in the table editor. Cells having identical values in the D column will be merged.

Table edit

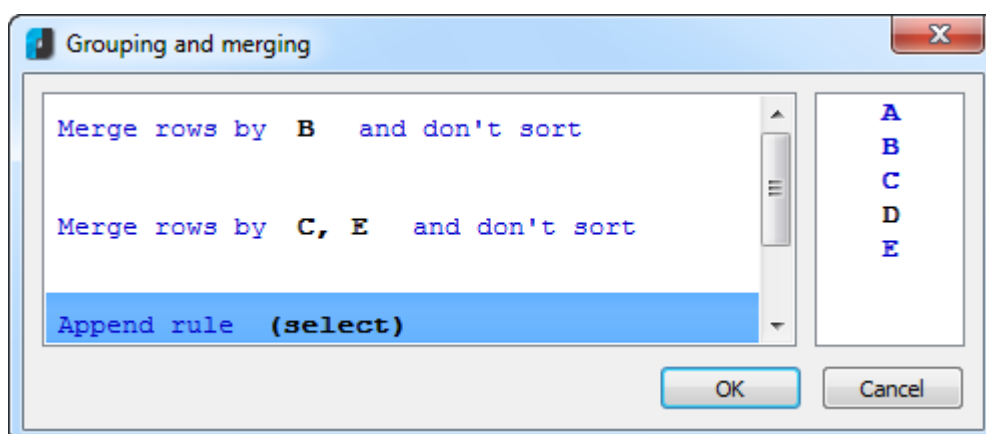
File Edit View Column Row Cell Section Help

| | A | B | C | D |
|--------|-------------------|----------------|------------|-----------|
| 1 | =Object.ObjectDes | =Object.Name | =Object.dr | =Object.H |
| Report | | | | |
| 2 | GOST 12204-72 | Leg7034-0591GO | 8 | 32 |
| 3 | GOST 12204-72 | Leg7034-0593GO | | 50 |
| 4 | GOST 12204-72 | Leg7034-0592GO | | 40 |
| 5 | GOST 12204-72 | Leg7034-0596GO | 10 | 63 |
| 6 | GOST 12204-72 | Leg7034-0597GO | | 80 |
| 7 | GOST 12204-72 | Leg7034-0594GO | | 40 |
| 8 | GOST 12204-72 | Leg7034-0595GO | | 50 |
| 9 | GOST 12204-72 | Leg7034-0599GO | 12 | 63 |
| 10 | GOST 12204-72 | Leg7034-0598GO | | 50 |



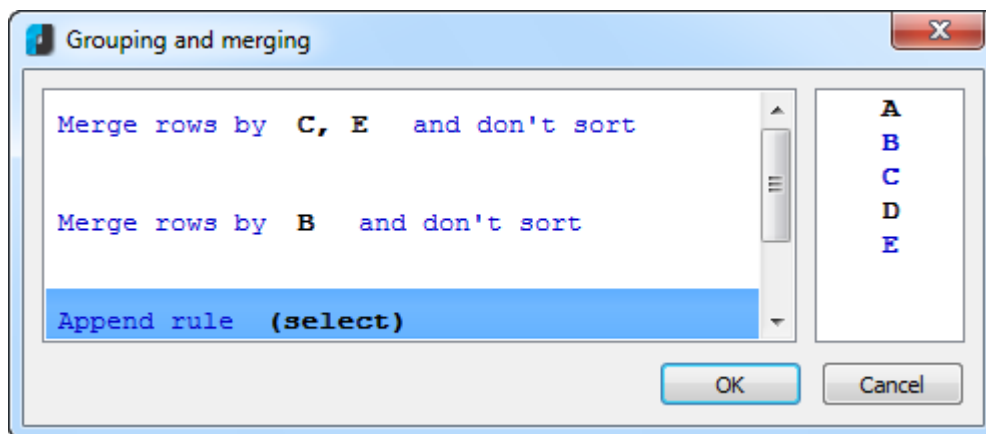
Rows, which have identical values in B and identical values in C columns, are merged. The order of specifying columns is unimportant.

| Name | Description | Thread diameter | Length | Diameter |
|---------------------------|---------------|-----------------|--------|----------|
| Leg7034-0591GOST 12204-72 | GOST 12204-72 | 8 | 32 | 12 |
| Leg7034-0592GOST 12204-72 | | | 40 | 12 |
| Leg7034-0593GOST 12204-72 | | | 50 | 12 |
| Leg7034-0594GOST 12204-72 | GOST 12204-72 | 10 | 40 | 14 |
| Leg7034-0595GOST 12204-72 | | | 50 | 14 |
| Leg7034-0596GOST 12204-72 | | | 63 | 16 |
| Leg7034-0597GOST 12204-72 | | | 80 | 16 |
| Leg7034-0599GOST 12204-72 | GOST 12204-72 | 12 | 63 | 18 |
| Leg7034-0598GOST 12204-72 | | | 50 | 18 |



Rows, which have identical values in B are merged, then cells in C and E are merged. The order of specifying columns is unimportant.

| <i>Name</i> | <i>Description</i> | <i>Thread diameter</i> | <i>Length</i> | <i>Diameter</i> |
|----------------------------------|----------------------|------------------------|---------------|-----------------|
| <i>Leg7034-0591GOST 12204-72</i> | <i>GOST 12204-72</i> | 8 | 32 | 12 |
| <i>Leg7034-0592GOST 12204-72</i> | | | 40 | |
| <i>Leg7034-0593GOST 12204-72</i> | | | 50 | |
| <i>Leg7034-0594GOST 12204-72</i> | <i>GOST 12204-72</i> | 10 | 40 | 14 |
| <i>Leg7034-0595GOST 12204-72</i> | | | 50 | |
| <i>Leg7034-0596GOST 12204-72</i> | <i>GOST 12204-72</i> | 10 | 63 | 16 |
| <i>Leg7034-0597GOST 12204-72</i> | | | 80 | |
| <i>Leg7034-0599GOST 12204-72</i> | <i>GOST 12204-72</i> | 12 | 63 | 18 |
| <i>Leg7034-0598GOST 12204-72</i> | | | 50 | |

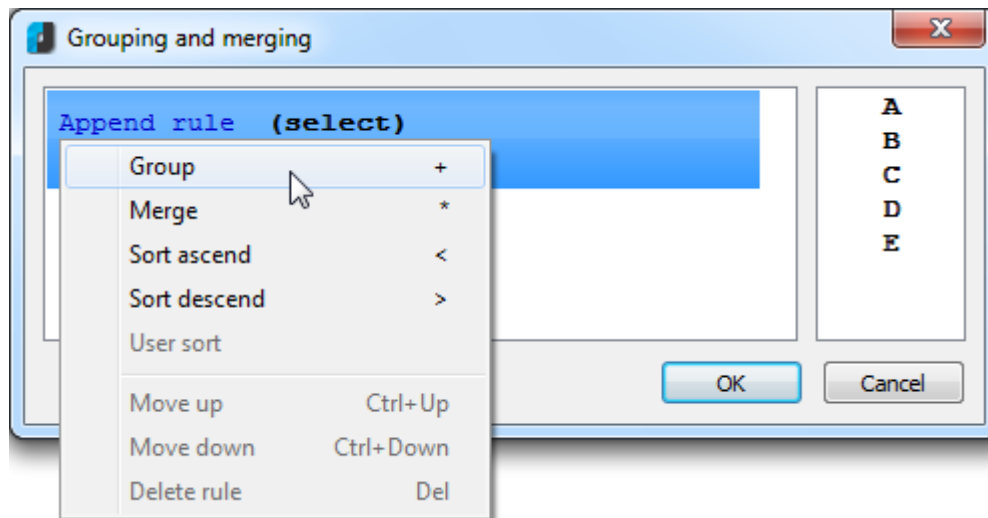


Rows, which have identical values in C, are merged, then cells in E and B are merged. The order of specifying columns is unimportant.

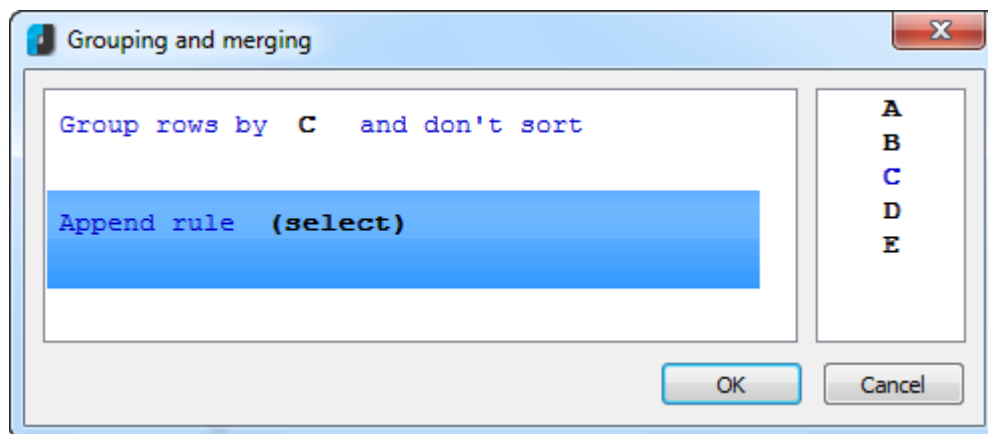
| <i>Name</i> | <i>Description</i> | <i>Thread diameter</i> | <i>Length</i> | <i>Diameter</i> |
|----------------------------------|----------------------|------------------------|---------------|-----------------|
| <i>Leg7034-0591GOST 12204-72</i> | <i>GOST 12204-72</i> | 8 | 32 | 12 |
| <i>Leg7034-0592GOST 12204-72</i> | | | 40 | |
| <i>Leg7034-0593GOST 12204-72</i> | | | 50 | |
| <i>Leg7034-0594GOST 12204-72</i> | <i>GOST 12204-72</i> | 10 | 40 | 14 |
| <i>Leg7034-0595GOST 12204-72</i> | | | 50 | |
| <i>Leg7034-0596GOST 12204-72</i> | <i>GOST 12204-72</i> | 10 | 63 | 16 |
| <i>Leg7034-0597GOST 12204-72</i> | | | 80 | |
| <i>Leg7034-0599GOST 12204-72</i> | <i>GOST 12204-72</i> | 12 | 63 | 18 |
| <i>Leg7034-0598GOST 12204-72</i> | | | 50 | |

Grouping

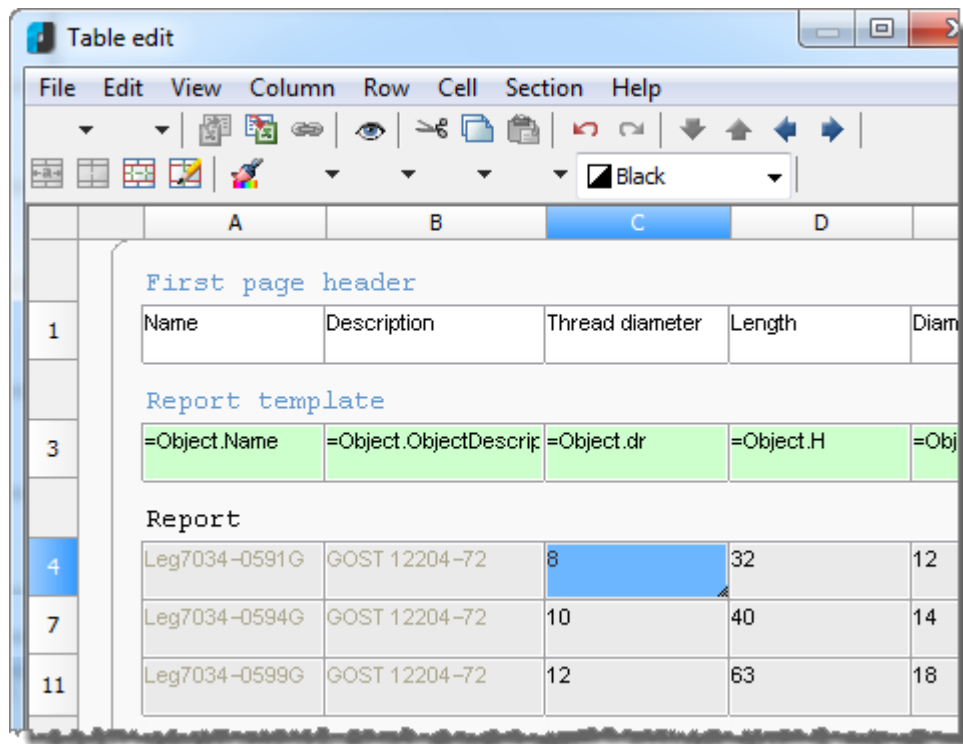
1. In the *Grouping and merging* dialogue box select **Append rule** and select **Group**.



2. Select the column name of the cells you want to group. Selection is made by double clicking on the column name or by dragging it into the **Select** field.



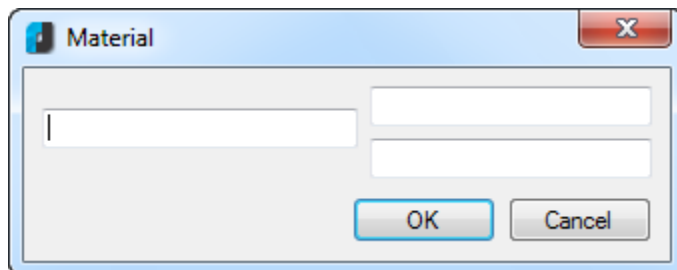
3. If you want to sort, select **and don't sort** and select a sort type.
4. Click **OK** to check the result in the table editor. Cells having identical values in the C column will be grouped.



Insert material


Insert material  button.

Use this tool to insert formatted row of material in the table. Click the icon to open the **Material** dialogue box.



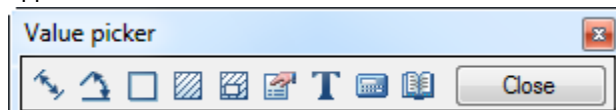
Measure

Menu: **Tools – Inquiry >  Measure...**

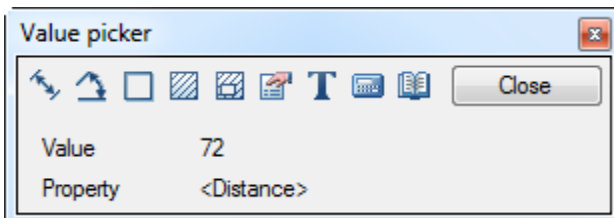
Toolbar: **Main – **

Command line: **MEASURE**

Take from drawing allows the taking of properties from a drawing or from standard details. The *Value picker* toolbar appears



Measured value displayed in the same **Value picker** dialogue box.



The imaging precision of measurements and calculations for the commands of the **Value picker** dialogue box corresponds to the value set for the current dimension style (the **Precision** options from the [Primary units](#) tab of the **Modify dimension style** dialogue box).

Options:



Measure distance

Measures the distance between specified points, radius or diameter. You can switch the method of measurement from the context menu or the command line.



Measure angle

Measures the angle. You can switch the method of measurement from the context menu or the command line.



Measure perimeter

Measures the perimeter of the closed area or the length of the closed polyline or circle.

To measure:

- click inside the enclosed area (the measured contour is highlighted),
- click on the closed polyline or circle.



Measure area

Measures the area of the closed contour.

To measure:

- click inside the enclosed area (the measured area is indicated by hatching),
- click on the closed polyline or circle.



Complex area

Measures some areas of the closed contours.

To measure:

- click inside each enclosed area (the measured areas are indicated by hatching),
- click on all the closed polylines or circles.

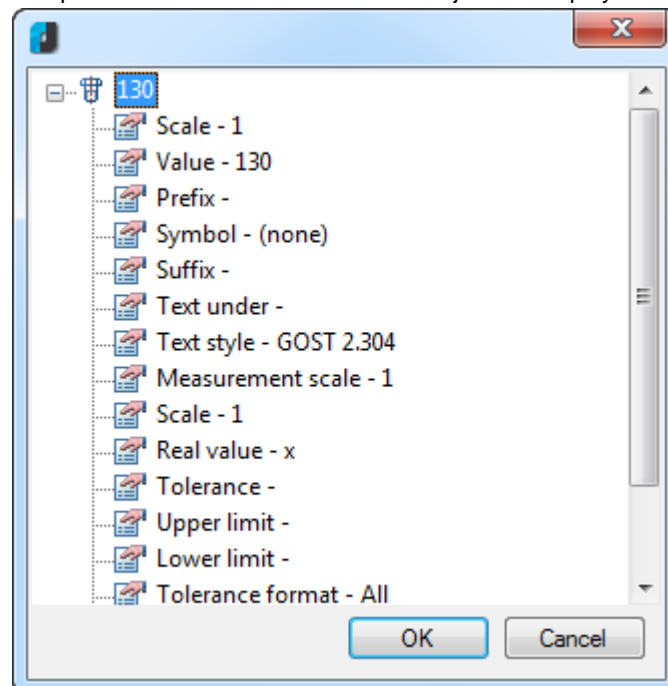
Press **ENTER** to finish selection of the closed areas (of polylines and circles). The calculated value of the complex area is displayed in the dialogue box.



Take from property

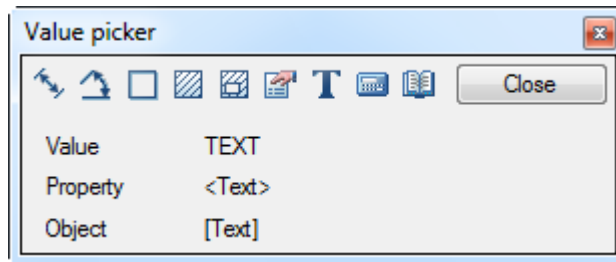
The command takes the values of any parameter from the selected nanoCAD objects on the drawing (dimension, leader, table, etc.).

The parameters values of the selected object are displayed in the list in the new window:



Take from text

Takes the values of the selected single line or multiline text:



To select text, place the cursor over it. The selection is highlighted by a frame.



Calculate

Opens the [Calculator](#).



Take from notes

Opens the [Notepad](#).

Note: If the **Value picker** dialogue box was called up at the runtime of another command (for example, from the **Position note** dialogue box of the [Pick from drawing](#) command from the context menu), the measured value is passed to this dialogue box.

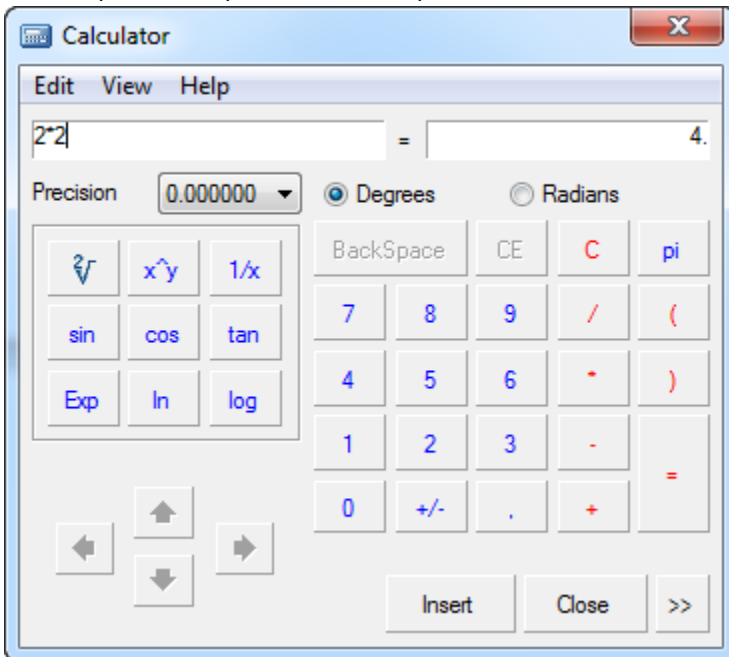
Calculator





Button:  **Calculator**

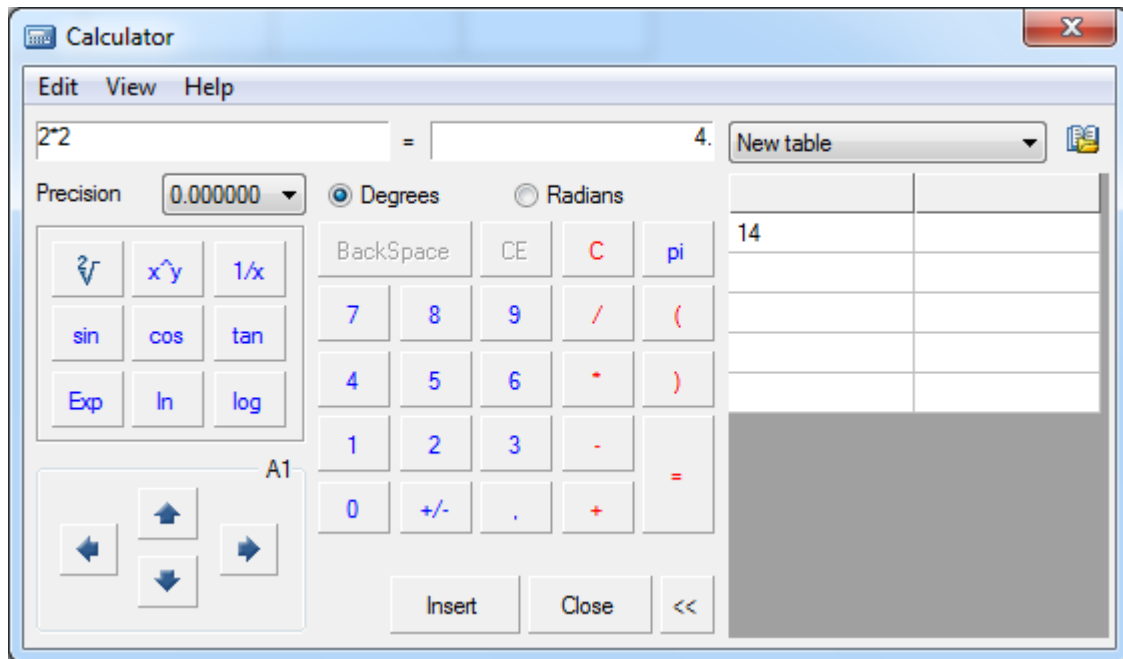
The calculate button is available in the [Value picker](#) dialogue box, the [Edit dimension](#) dialogue box and the [Table edit](#) toolbar.

You can perform simple and more complex mathematical calculations in the table cells using the calculator functions.



It is possible to connect the calculator to the **Notepad**  tool.

To access the [Notepad](#) button, click the button  :



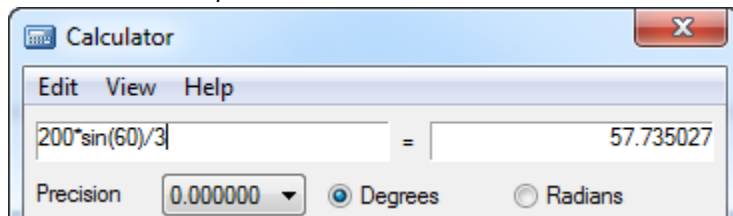
The drop down list displays the tables from the **Notepad**. This tool is useful when you frequently perform standard calculations.

The **Insert** button is used to insert the result of the calculation from the calculator into the edited table cell or input field.

You can use the arrow buttons from the bottom left corner to select table cells. The name of the current cell is also displayed:



You can use the *Expression* command from the *View* menu to calculate formulas:



Use the left field to enter the formula to be calculated. The result will be displayed in the right field.

The **Precision** option sets the rounding level of the calculation result.

Notepad



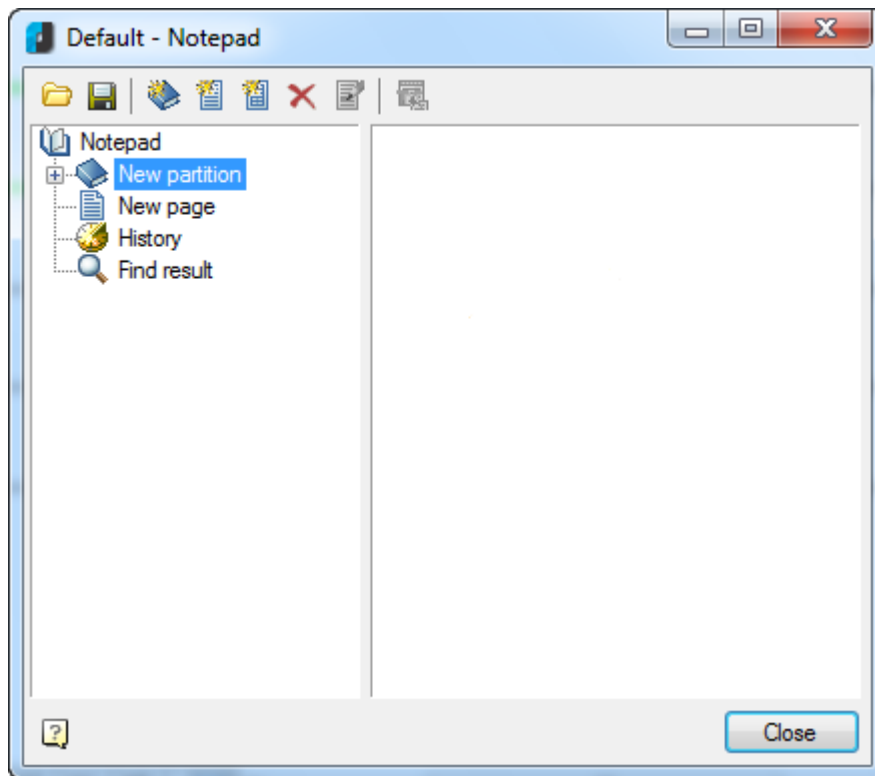
Button:  **Notepad**

The **Notepad** button is available in the [Value picker](#) dialogue box and in the [Table edit](#) toolbar.

Notepad is used to enter temporarily used text (acronyms, symbols etc.). It is a universal tool for text input.

Notepad allows the saving of a set of expressions and their positional structure in a file; entering and placing of text expressions and small tables; searching and editing of previously created notes.

There is a content tree in the left part of Notepad. The user can select partitions, pages and tables there. There are control buttons in the top part:



Buttons:



Load from file

Opens the Notepad file saved on the disk.



Save

Saves changes in the Notepad file.

These buttons control the structure of the Notepad:



Add partition



Add page



Add table



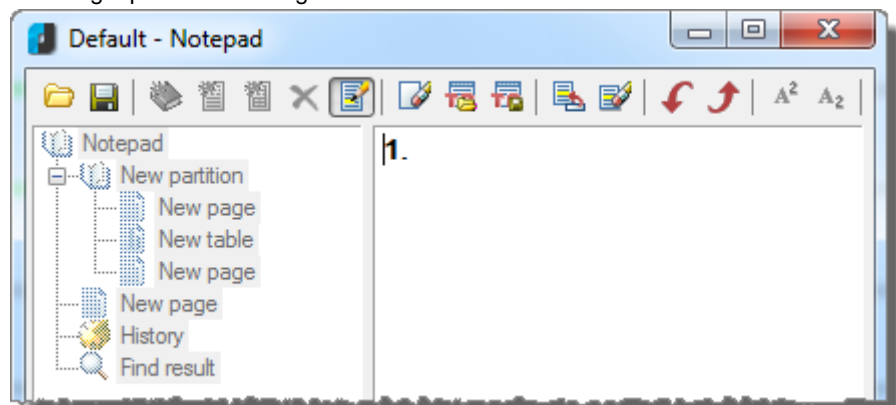
Delete tree item



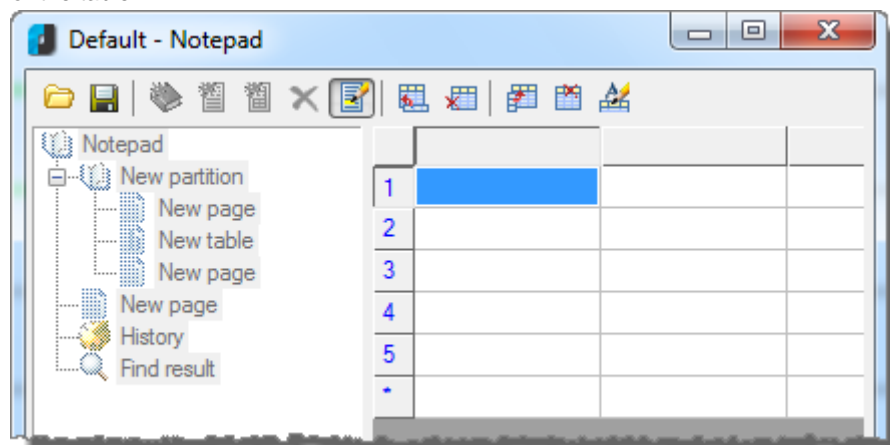
Edit page

Switches on page and table editing mode. This button is available if a page or table is selected in the tree.

When editing a page, the tree becomes unavailable and the edit toolbar buttons show in the right part of the dialogue box:



or the table:



Page editing tools:



Clear

Clears the current page.



Load from file

Allows importing of text from an MS Word RTF-file.



Save to file

Allows writing of the page content into an RTF-file for using in MS Word.



Add technical condition



Delete technical condition



Move down



Move up



Superscript

Turn on/off the input mode for uppercase index.



Subscript

Turn on/off the input mode for bottom index.



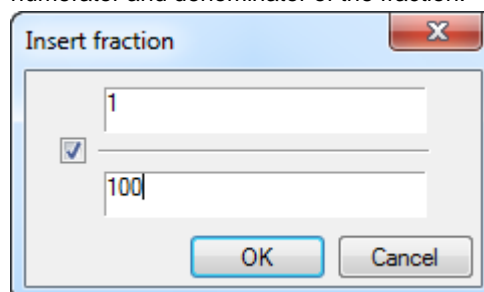
Insert special symbol

Opens the panel to select and insert [special symbols](#).



Insert fraction

Opens the **Insert fraction** dialogue box to set the numerator and denominator of the fraction:



The switch ☒ controls the display of the fraction line.



Insert tolerance

Opens the **Fit** dialogue box.



Insert material

Opens the **Material** dialogue box.



Calculator

Opens the [Calculator](#) dialogue box.

Table editing tools:



Insert row



Delete row



Insert column

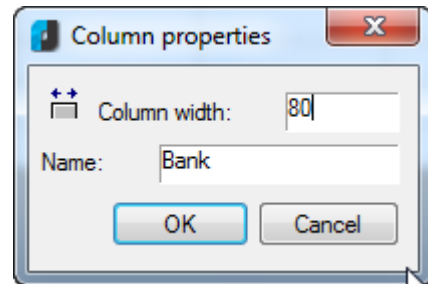


Delete column



Rename column


Opens the **Column properties** dialogue box, in which you can set the width and name of the selected column:



Select/Transfer selected text

Transfers the selected text into the edited table cell.

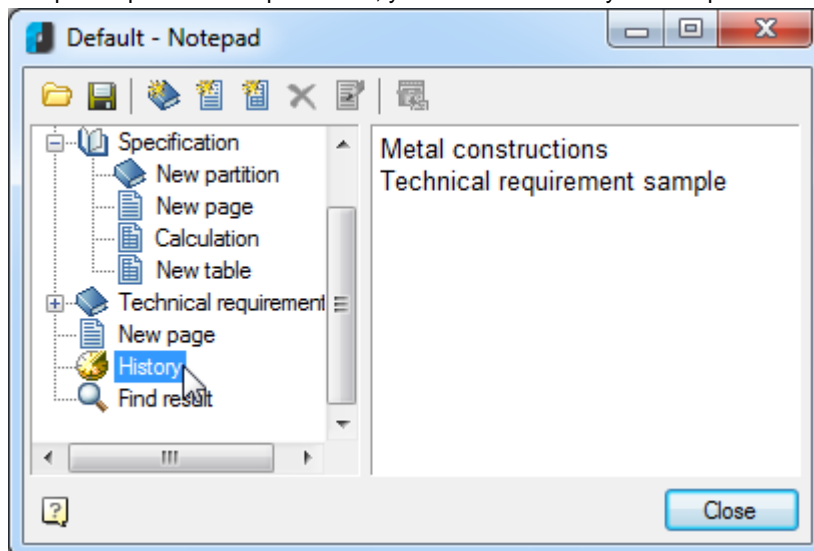
If page or table is selected in the tree, its content is displayed in the right part of the dialogue box.

Select the text and select the  button.

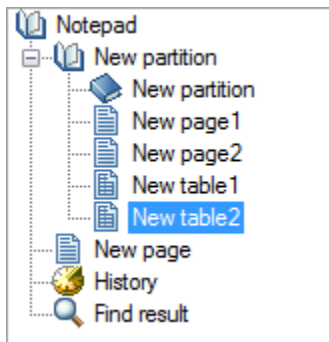
Note: When you use the **Notepad** for the first time, it is recommended to save the new notepad in a separate file using the **Save** button.

The dialogue box has a special search tool to search the text line. Right click on any partition or page of the partition and select the **Find** command from the context menu. The found links are placed in the *Find result* section.

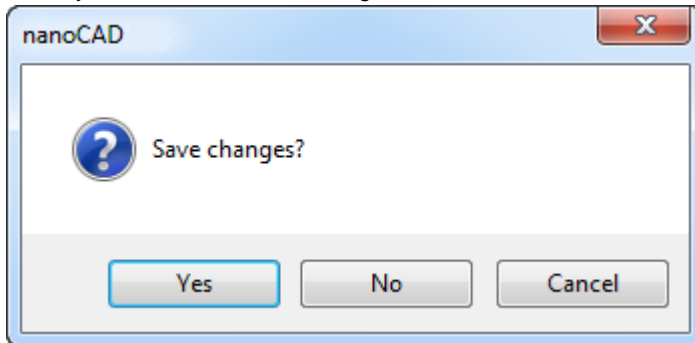
To speed up the search procedure, you can view recently used expressions in the *History* section:



You can create an unlimited number of sections and subsections, pages and tables in the **Notepad**:



You can move the partitions, pages and tables in the structural tree by using drag and drop. To rename the partitions, pages and tables, use the **Rename** command from the context menu. When you want to close the dialogue box, it offers to save the changes in the Notepad:



Features of the design elements

In place edit

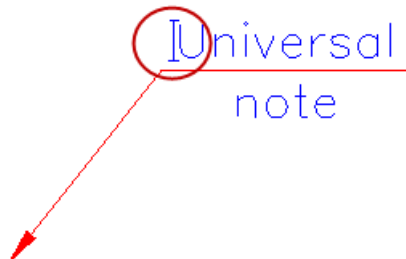


Button:  **In place edit**



Command line: **IPEDIT**

This command allows you to edit all nanoCAD's objects, containing text, directly in the drawing. To quick-start the command, press and hold down the **CTRL** key and left click on the object. After running the command, the cursor appears in the text line of the edited object:



Parameters redefinition



Button:  **Parameters redefinition**

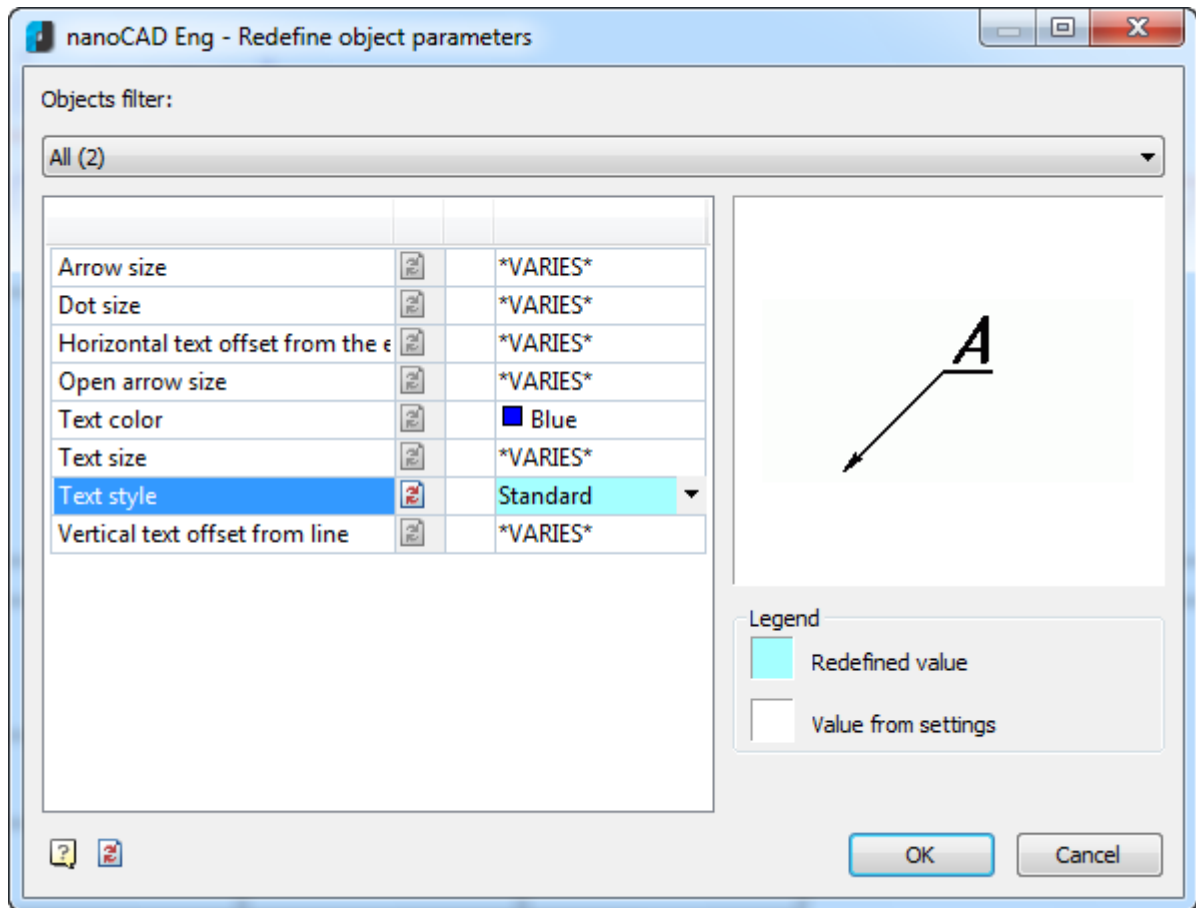


Command line: **PR**

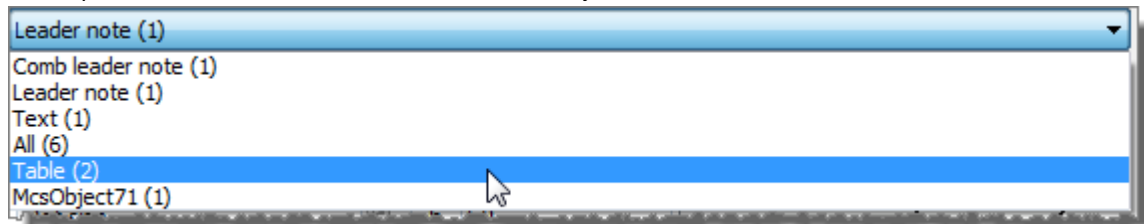
This command is used to define the variations in the settings of the nanoCAD design objects in the current drawing. The redefinition is controlled by the parameters set in the settings of the design elements (the **Tools** menu – the [Settings Parameters](#) command).

The command allows you to check the redefinition status of one, several or all objects in the drawing.

The **Parameters redefinition** command opens the **nanoCAD - Redefine object parameters** dialogue box:

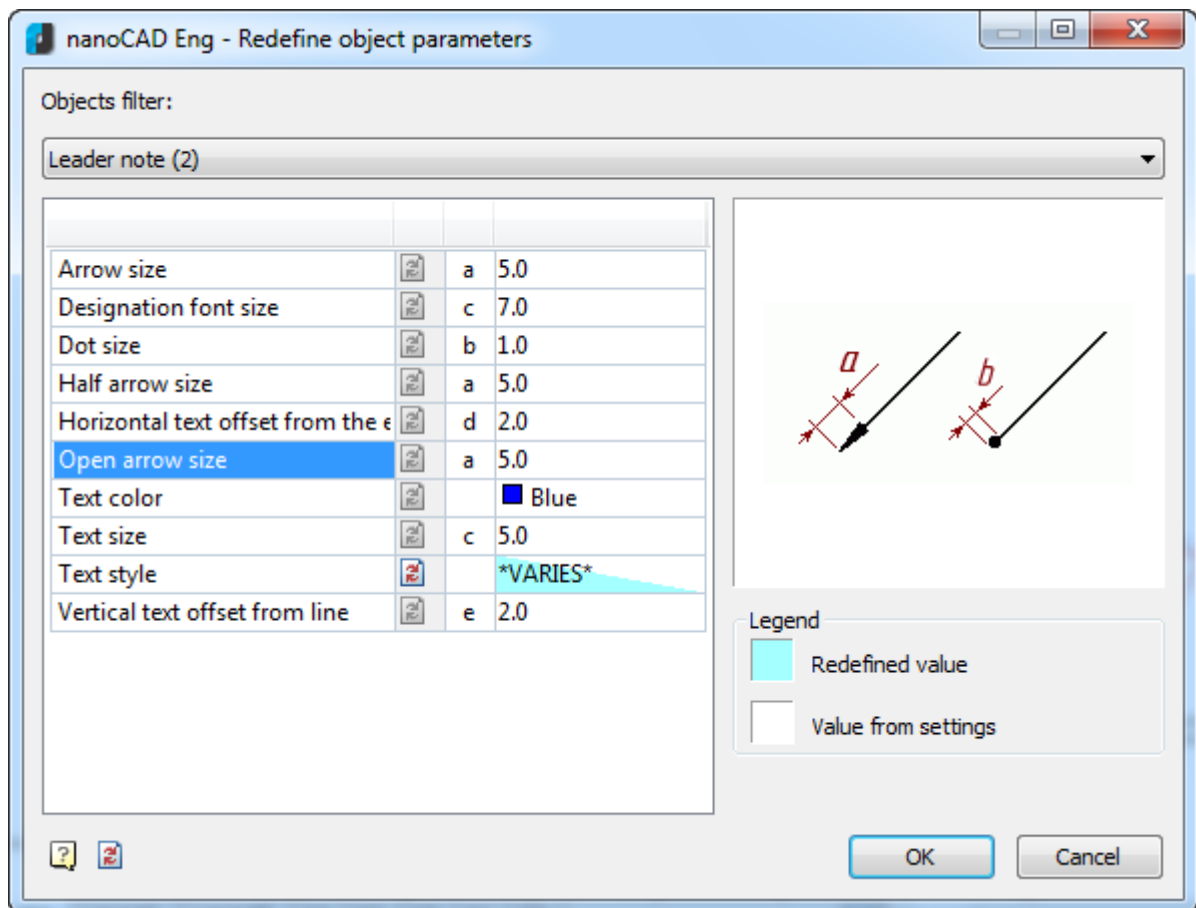


The objects are grouped by type in the dialogue box. The objects' parameters are grouped by name. To set parameters or remove redefinitions, use the *Object filter* list:



The total number of found objects in the drawing is displayed in the brackets for each object type.

When an object type is selected, its properties will be displayed in the parameters table and its graphical view will be displayed in the right box of the dialogue box.

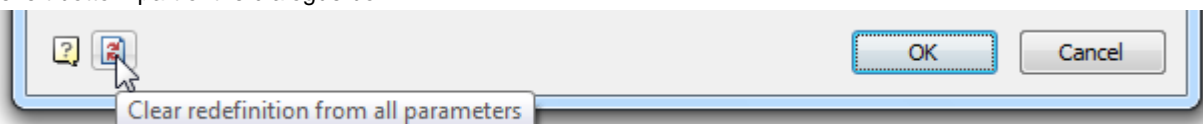


The redefined parameters are highlighted in blue. If, for the selected objects group, only some parameters have redefinitions, then the redefinition highlighting will be diagonal.

| | | | |
|------------|--|---|----------|
| Text size | | c | 7.0 |
| Text style | | | *VARIES* |

To remove the redefinition, it is necessary to click the icon

To remove the redefinition from all objects, it is necessary to click the **Clear redefinition from all parameters** icon in the left bottom part of the dialogue box:



It is possible to enter the new value for the selected parameter from the keyboard or select from the drop-down list:


| | | | |
|--------------------------------------|--|---|----------|
| Arrow size | | a | 5.0 |
| Designation font size | | c | 7.0 |
| Dot size | | b | 1.8 |
| Half arrow size | | a | 2.5 |
| Horizontal text offset from the line | | d | 3.5 |
| Open arrow size | | a | 7 |
| Text color | | | 10 |
| Text size | | c | 14 |
| Text style | | | 20 |
| Vertical text offset from line | | e | *VARIES* |

If the new value aligns with a value specified by default in the **nanoCAD – Options** dialogue box (the **Tools** menu – the [Settings Parameters](#) command), the blue highlighting will not be displayed.


To change the parameters of one or several design elements, it is necessary to:

- Select the objects on the drawing.
- Select the **Parameters redefinition** command from the context menu.

- In the opened dialogue box set new values for the settings. The changes apply only to the selected objects.

The **Parameters redefinition**  command is useful to check documents for compliance of design elements with corporate company standards.

To check the redefinition status of all objects on the drawing:

- Start the **Parameters redefinition** command.
- Press **Enter** to select all nanoCAD design elements. They will be highlighted on the drawing.
- Click the **Clear redefinition from all parameters**  icon to clear redefinition from all parameters. As a result, you obtain a document designed according to the settings set by your company (consisting in part of nanoCAD elements).

Regenerate



Button:  **Regenerate**



Command line: **REGENOBJ**

This command is used to:

- regenerate the nanoCAD objects,
- regenerate overlapping of the nanoCAD objects,
- regenerate the tables,
- update automatically calculated attribute values.

Create 3D objects

Surfaces

nanoCAD allow you to create a polygon or polyface mesh by specifying vertices. The mesh density controls the number of facets in legacy polygonal and polyface meshes. Density is defined in terms of a matrix of M and N vertices, like a grid consisting of columns and rows. M and N specify the column and row position, respectively, of any given vertex.

Mesh objects do not have the mass and volume properties of 3D solids.

In the **Draw** menu – **Meshes** item, the commands for the 3D meshes creation in the form of elementary surfaces are presented – box, wedge, cone, sphere, torus, pyramid, dish, dome.

You can control whether the mesh is displayed as a wireframe, hidden, or conceptual image by changing the visual style.

Box



Menu: **Draw – Meshes > Box**



Command line: **3D**

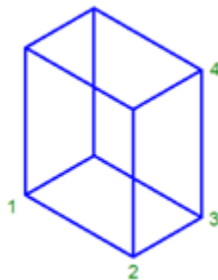
The command creates a 3D box.

Options:

Cube Switch to create a cubic box with edges of equal length.

Reference Specify the rotation angle of the box, measured from the reference angle.

Points Specify the rotation angle, measured from the reference angle to the segment, defined by two points.



Command prompts:

Specify corner point of box:

Specify point 1.

Specify length of box:

Specify point 2.

Specify width of box or [Cube]:

Specify point 3 or select the Cube option.

Specify height of box:

Specify point 4.

Specify rotation angle of box about the Z axis or [Reference]:

Specify the rotation angle or select the Reference option.

The rotation angle is set relative to the first corner specified (point 1) and is measured from the X-axis on the XY plane.

When you select the Reference option, the following prompts are displayed:

Укажите опорный угол
<0>:

Specify a point or enter the angle value.

Укажите новый угол или
[Точки]:

Specify a point or enter the angle value or select the Points option.

Note: The **Reference** and **Points** options are useful to define the box orientation relative to an existing object.

Wedge

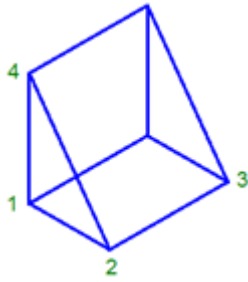


Menu: **Draw – Meshes> Wedge**



Command line: **3D**

The command creates a right-angled wedge-shaped polygon.



Command prompts:

Specify corner point of wedge:

Specify point 1.

Specify length of wedge:

Specify point 2.

Specify width of wedge:

Specify point 3.

Specify height of wedge:

Specify point 4.

Specify rotation angle of wedge about the Z axis or **[Reference]**:

Specify a point or enter an angle value. The rotation angle is set relative to the first corner specified (point 1) and is measured from the X-axis on the XY plane.

Cone



Menu: **Draw – Meshes> Cone**

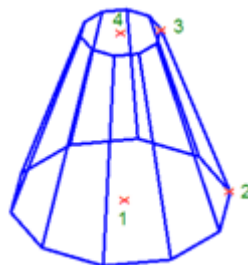


Command line: **3D**

The command creates a cone-shaped polygon mesh.

Option:

Diameter Switch to specify the diameter.



Command prompts:

Specify center point for base of cone:

Specify the centre point for the base of the cone (point 1).

Specify radius for base of cone or **Diameter**:

*Specify the radius for the base of the cone (point 2) or select the **Diameter** option.*

Specify radius for top of cone or **Diameter**<0>:

*Specify the radius for the top of the cone (point 3) or select the **Diameter** option. A value of 0 (set by default) produces a cone. A value greater than 0 produces a truncated cone.*

Specify height of cone:

Specify the height of the cone (point 4).

Enter number of segments for surface of cone <16>:

Enter the number of segments.

Sphere



Menu: **Draw – Meshes> Sphere**

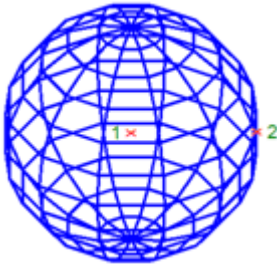


Command line: **3D**

The command creates a spherical polygon mesh.

Option:

Diameter Switch to specify the diameter.



Command prompts:

Specify center point of sphere:

Specify radius of sphere or [Diameter]:

Enter number of longitudinal segments for surface of sphere <16>:

Enter number of latitudinal segments for surface of sphere <16>:

Specify the centre point of the sphere (point 1).

Specify the radius of the sphere (point 2) or select the Diameter option.

Enter the number of longitudinal segments for the surface of the sphere.

Enter the number of latitudinal segments for the surface of the sphere.

Torus



Menu: **Draw – Meshes> Torus**

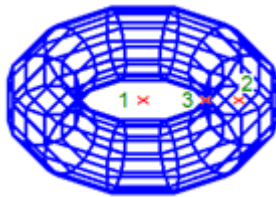


Command line: **3D**

The command creates a toroidal polygon mesh.

Option:

Diameter Switch to specify the diameter.



Command prompts:

Specify center point of torus:

Specify radius of torus or [Diameter]:

Specify radius of tube or [Diameter]:

Enter number of segments around torus circumference <16>:

Enter number of segments around tube circumference <16>:

Specify the centre point of the torus (point 1).

Specify the radius of the torus (point 2) or select the Diameter option.

Specify the radius of the tube (point 3) or select the Diameter option.

Enter the number of segments around the torus circumference.

Enter the number of segments around the tube circumference.

Pyramid



Menu: **Draw – Meshes> Pyramid**



Command line: **3D**

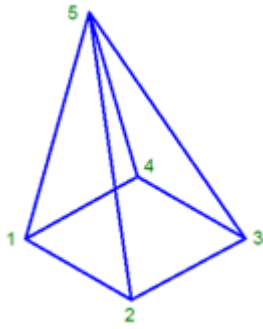
The command creates a pyramid or a tetrahedron.

Options:

Tetrahedron Switch to create the tetrahedron.

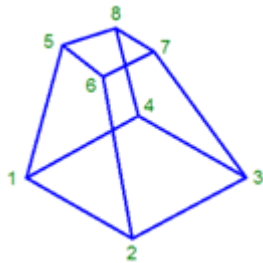
Ridge Switch to create a pyramid with a top with a ridge form.

Top Switch to create a truncated pyramid.



When you create a pyramid, the following prompts are displayed:

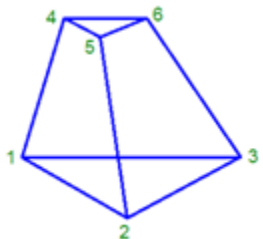
| | |
|--|------------------|
| Specify first corner point for base of pyramid: | Specify point 1. |
| Specify second corner point for base of pyramid: | Specify point 2. |
| Specify third corner point for base of pyramid: | Specify point 3. |
| Specify fourth corner point for base of pyramid or [Tetrahedron]: | Specify point 4. |
| Specify apex point of pyramid or [Ridge/Top]: | Specify point 5. |



When you create a truncated pyramid, the following prompts are displayed:

| | |
|--|-------------------------------|
| Specify first corner point for base of pyramid: | Specify point 1. |
| Specify second corner point for base of pyramid: | Specify point 2. |
| Specify third corner point for base of pyramid: | Specify point 3. |
| Specify fourth corner point for base of pyramid or [Tetrahedron]: | Specify point 4. |
| Specify apex point of pyramid or [Ridge/Top]: | Select the Top option. |
| Specify first corner point for top of pyramid: | Specify point 5. |
| Specify second corner point for top of pyramid: | Specify point 6. |
| Specify third corner point for top of pyramid: | Specify point 7. |
| Specify fourth corner point for top of pyramid: | Specify point 8. |

Note: To create the top of a pyramid, it is necessary to set the points in the same order as for the base.



When you create a truncated tetrahedron, the following prompts are displayed:

| | |
|---|------------------|
| Specify first corner point for base of pyramid: | Specify point 1. |
| Specify second corner point for base of | Specify point 2. |

pyramid:

Specify third corner point for base of pyramid:

Specify fourth corner point for base of pyramid or [**Tetrahedron**]:

Specify apex point of tetrahedron or [**Top**]:

Specify first corner point for top of tetrahedron:

Specify second corner point for top of tetrahedron:

Specify third corner point for top of tetrahedron:

Specify point 3.

Select the **Tetrahedron** option.

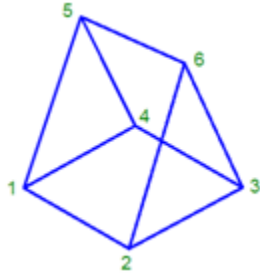
Select the **Top** option.

Specify point 4.

Specify point 5.

Specify point 6.

Note: To create the top of a tetrahedron it is necessary to set the points in the same order as for the base



When you create a pyramid with the top with the ridge form, the following prompts are displayed:

Specify first corner point for base of pyramid:

Specify second corner point for base of pyramid:

Specify third corner point for base of pyramid:

Specify fourth corner point for base of pyramid or [**Tetrahedron**]:

Specify apex point of pyramid or [**Ridge/Top**]:

Specify first ridge end point of pyramid:

Specify second ridge end point of pyramid:

Specify point 1.

Specify point 2.

Specify point 3.

Specify point 4.

Select the **Ridge** option.

Specify point 5.

Specify point 6.

Dish



Menu: **Draw – Meshes> Dish**

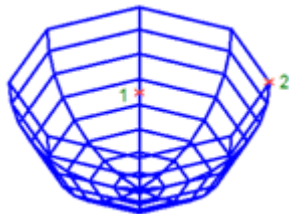


Command line: **3D**

The command creates the lower half of a spherical polygon mesh.

Option:

Diameter Switch to specify the diameter.



Command prompts:

Specify center point of dish:

Specify radius of dish or [**Diameter**]:

Enter number of longitudinal segments for surface of dish <16>:

Enter number of latitudinal segments for surface

Specify the centre point of the dish (point 1).

Specify the radius of the dish (point 2) or select the **Diameter** option.

Enter the number of longitudinal segments for the surface of the dish.

Enter the number of latitudinal segments for the

of dish <8>:

surface of the dish.

Dome



Menu: **Draw – Meshes > Dome**

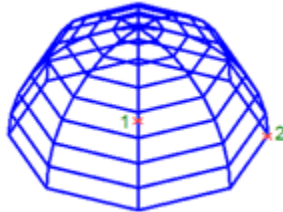


Command line: **3D**

The command creates the upper half of a spherical polygon mesh.

Option:

Diameter Switch to specify the diameter.



Command prompts:

Specify center point of dome:

Specify radius of dome or [**Diameter**]:

Enter number of longitudinal segments for surface of dome <16>:

Enter number of latitudinal segments for surface of dome <8>:

Specify the centre point of the dome (point 1).

*Specify the radius of the dome (point 2) or select the **Diameter** option.*

Enter the number of longitudinal segments for the surface of the dome.

Enter the number of latitudinal segments for the surface of the dome.

Mesh

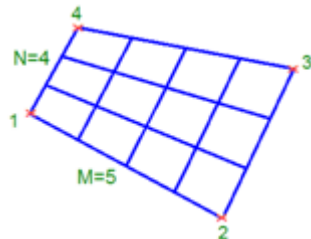


Menu: **Draw – Meshes> Mesh**



Command line: **3D**

The command creates a planar mesh.



Command prompts:

Specify first corner point of mesh:

Specify second corner point of mesh:

Specify third corner point of mesh:

Specify fourth corner point of mesh:

Enter mesh size in the M direction:

Enter mesh size in the n direction:

Specify point 1.

Specify point 2.

Specify point 3.

Specify point 4.

Enter a value between 2 and 256.

Enter a value between 2 and 256.

Note 1: The M and N directions are similar to the X and Y axes of an XY plane.

Note 2: M and N sizes determine the number of lines drawn in each direction along the mesh.

3D Mesh



Menu: **Draw – Meshes> 3D Mesh**

This command creates polygon meshes.

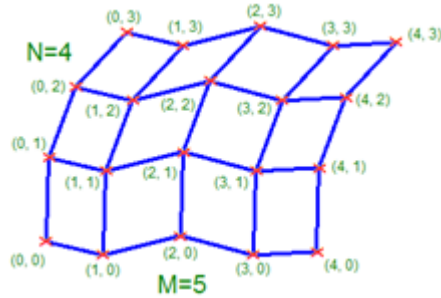
As you create the mesh, you specify the size of the mesh in the M and N directions.

A pair of **m** and **n** (row number and column number) defines the position of each vertex of the mesh. Specifying of the vertices starts with the vertex (0, 0). The value of n changes first.

The vertices can be located at any distance from each other.

You can close the meshes with **PEDIT**.

The following example shows the points corresponding to the vertices for a 3D Mesh with N=4 and M=5 sizes.



Command prompts:

Enter size of mesh in M direction: Enter a value between 2 and 256.

Enter size of mesh in N direction: Enter a value between 2 and 256.

Specify location for vertex (0, 0): Specify the location for the first vertex.

Specify location for vertex (0, 1): Specify the location for the second vertex.

...

...

Specify location for vertex (m, n): Specify the location for the last vertex.

Note 1: The M and N directions are similar to the X and Y axes of an XY plane.

Note 2: M and N sizes determine the number of lines drawn in each direction along the mesh.

Setting of the orthographic and isometric views

To quickly set the view, select the **View – Views** and select the required standard orthographic and isometric views.

Orthographic views

Set Top



Menu: **View – Views >**  **Set Top**



Command line: **VIEW1**

Set the top view .

Set Bottom



Menu: **View – Views >**  **Set bottom**



Command line: **VIEW2**

Set the bottom view .

Set Left



Menu: **View – Views >**  **Set Left**



Command line: **VIEW4**

Set the left view.

Set right



Menu: **View – Views >**  **Set Right**



Command line: **VIEW5**

Set the right view.

Set Front



Menu: **View – Views >**  **Set Front**



Command line: **VIEW3**

Set the front view .

Set back



Menu: **View – Views >**  **Set Back**



Command line: **VIEW6**

Set the back view.

Isometric views

SW Isometric



Menu: **View – Views >**  **SW Isometric**



Command line: **SWISO**

Set the southwest isometric view.

SE Isometric



Menu: **View – Views >**  **SE Isometric**



Command line: **SEISO**

Set the southeast isometric view.

NE Isometric



Menu: **View – Views >**  **NE Isometric**



Command line: **NEISO**

Set the northeast isometric view.

NW Isometric



Menu: **View – Views >**  **NW Isometric**

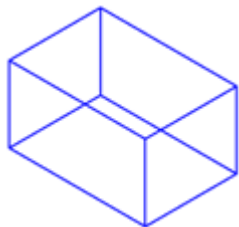
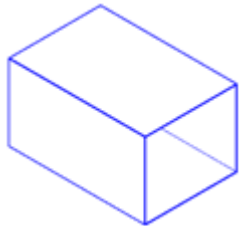


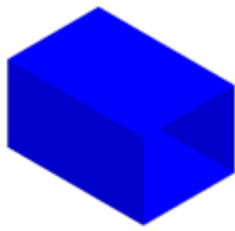
Command line: **NWISO**

Set the northwest isometric view.

Visual style

nanoCAD allows you to choose the methods (styles) to display the three-dimensional objects in the current viewport. The basic methods of object display:

| | |
|---|---|
|  | <p><i>Wireframe</i> displays the objects using lines and curves to represent the boundaries. The objects are visible.</p> |
|  | <p><i>Hidden</i> displays the objects using the 3D wireframe representation, but hides the lines representing the back faces.</p> |



Shade assigns a monochrome colour to the visible surface of the object and also makes the background invisible. Each object is shaded by the current colour. In the shaded visual styles, the faces are lit by one distant light that is located behind the left shoulder of the user. The object view depends on the selected shading method.

The command control styles are represented in the **View** menu – **Visual Style**.

2D Wireframe



Menu: **View – Visual style > 2D Wireframe**



Command line: **VSCURRENT0**

The objects are represented as transparent. The command displays the objects using lines and curves to represent the boundaries. Raster and OLE objects, linetypes, and lineweights are visible.

3D Wireframe



Menu: **View – Visual style > 3D Wireframe**



Command line: **VSCURRENT1**

The objects are represented as transparent. The command displays the objects using lines and curves to represent the boundaries. Raster objects are invisible, linetypes, and lineweights are ignored.

3D Hidden



Menu: **View – Visual style > 3D Hidden**



Command line: **VSCURRENT2**

The command displays the objects using the 3D wireframe representation and hides the lines representing the back faces.

Flat Shaded



Menu: **View – Visual style > Flat Shaded**



Command line: **VSCURRENT3**

Fills the surface with the colour which is determined by the colour of the object or layer. At the same time, a shadow is superimposed on the surface as if there is a light source in the same position as the observer. Curved surfaces have a multifaceted view.

Gouraud Shaded



Menu: **View – Visual style > Gouraud Shaded**



Command line: **VSCURRENT4**

The Gouraud method is used. The surface and colour transitions are displayed as smooth. If the object is not assigned a material, it will be shaded in the colour of an object or layer.

Flat Shaded with Edges



Menu: **View – Visual style > FlatShaded with Edges**



Command line: **VSCURRENT5**

The object will have simple filling of the faces and selected edges

Gouraud Shaded with Edges



Menu: **View – Visual style > GouraudShaded with Edges**



Command line: **VSCURRENT6**

In this case Gouraud shading is applied to objects, but the edges of their surfaces are visible.

The component layout and plot drawing

Model space and paper space

There are two distinct working environments, or “spaces,” in which you can create objects in a drawing. In nanoCAD they are represented by the *Model* and layout tabs.

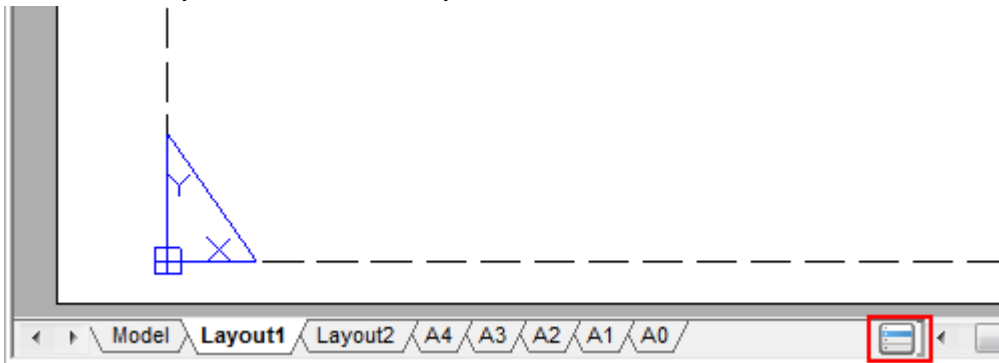
Typically, a model composed of geometric objects is created in a three-dimensional space called *model space*. A final layout with specific views and annotations of this model is created in a two-dimensional space called *paper space*. These spaces are accessible on two or more tabs near the bottom of the drawing area: the *Model* tab and one or more layout tabs.


Layout tabs access an area called *paper space*. In paper space, you place your title block, create layout viewports to display views, dimension your drawing, and add notes.

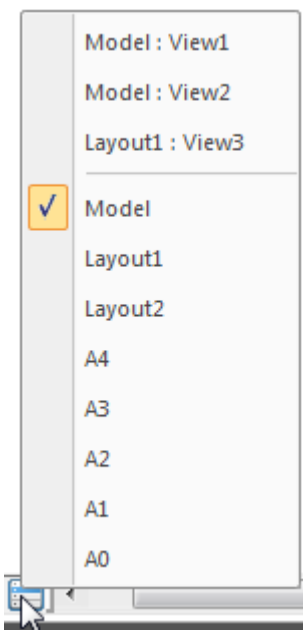
You can create a single layout viewport that fits the entire layout or create multiple layout viewports in the layout. Once you create the viewports, you can change their size, their properties, and also scale and move them as needed.


Layouts

It is possible to create multiple layouts in a drawing; each layout can contain different plot settings and paper sizes. The layout tabs are located in the bottom part of the document window, next to the *Model* tab. To switch between the *Model* tab and layouts, click the necessary tab:



To quick switch between the *Model* space and layouts, use the  button located at the end of tabs line. This button also allows you to switch between the named views available in the document. Left click to open a context menu that displays all the tabs and named views available in the document:



The top part of the menu displays the named views, the bottom part displays the *Model* and layouts tabs. The current space is marked with . To switch between tabs or named views, it is necessary to click on the corresponding name in the menu. When you switch to a named view, there is auto-panning of the view on the screen. You can rename, delete and add layout tabs unlike the *Model* tab.

Another important difference between *Model* space and *paper* space is that you create non-overlapping viewports in the *Model* space, i.e. snap-together at the boundaries. In *Model* space you can print only the current viewport. Viewports in the paper space are floating. They can be moved to any part of the layout. Their boundaries can be close to each other and overlapped or be located at some distance from each other. You can print all the viewports located on the layout at the same time.

To work with a layout, use the command from the **Insert** menu – **Layout** item, or from the **Layout toolbar**, or from the context menu that opens by right clicking the *Model* tab or a *Layout* tab.

To create a new layout, use the [Add Layout](#) command. To delete a layout, use the [Delete Layout](#) command. To rename a layout, use the [Rename Layout](#) command.

The format of displayed layout is set in the [Page Setup](#) dialogue box.

Note: The print area of the layout for the current format settings and printer is displayed by a dashed line.

You can change the layout colour in the [Options](#) dialogue box using the *Color settings* item – *Layout Paper*.

When you prepare a layout, you typically go through the following process:

- Create a model of your subject on the *Model* tab.
- Click a *Layout* tab and specify layout page settings, such as plotting device, paper size, plot area, plot scale and drawing orientation.
- Insert a title block into the layout.
- Create a new layer to be used for the layout viewports.
- Create the layout viewports and position them on the layout.
- Set the orientation, scale and layer visibility of the view in each layout viewport.
- Add dimensions and annotate in the layout as necessary.
- Turn off the layer containing the layout viewports.
- Execute the plot setting of the layout.
- Plot the layout.

Work with Layouts

To create a new layout



Menu: **Insert – Layout >  New Layout**



Command line: **ADDLAYOUT**

At the command prompt - **Input layout name <Layout3>**, enter the name of the new layout and press **ENTER**.

The layout name should be unique up to 255 characters.

Note: No more than 31 characters of the layout name are displayed on the layout tab.

To create a layout from a template



Menu: **Insert – Layout > Layout from template...**

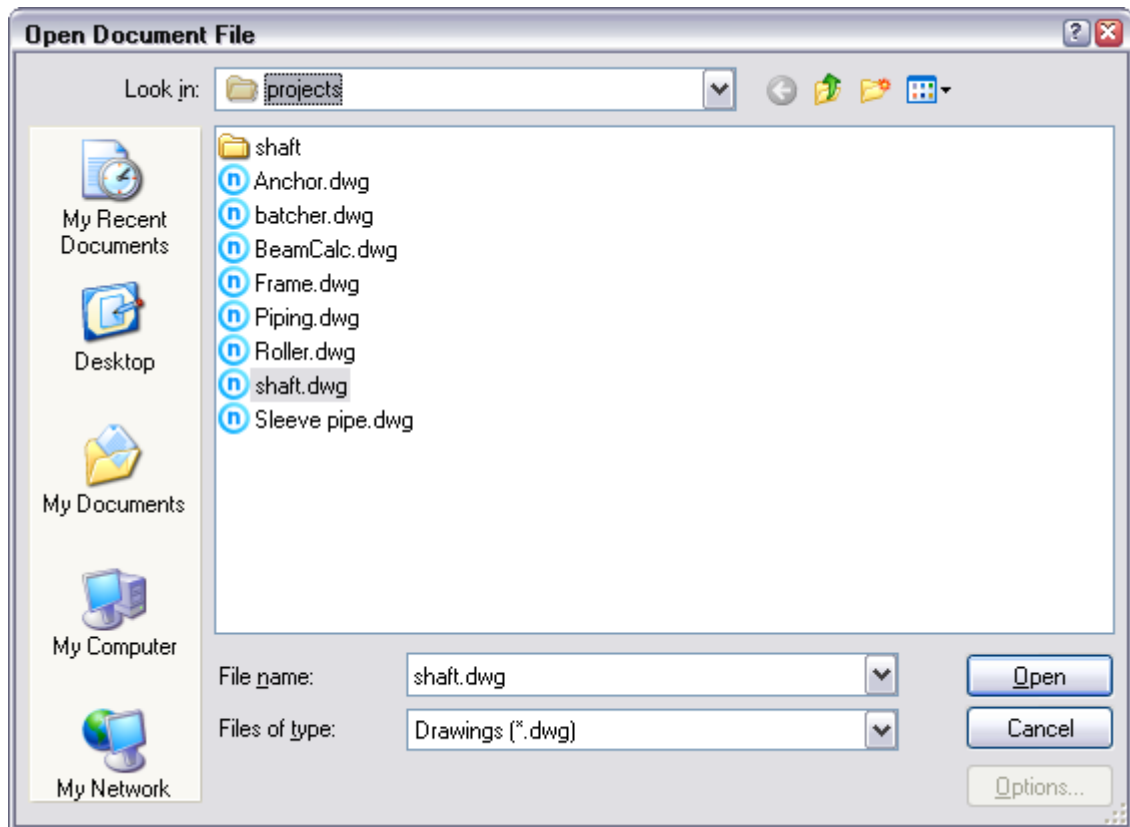


Command line: **LAYOUTFROMTEMPLATE**

The command creates a new layout in the document.

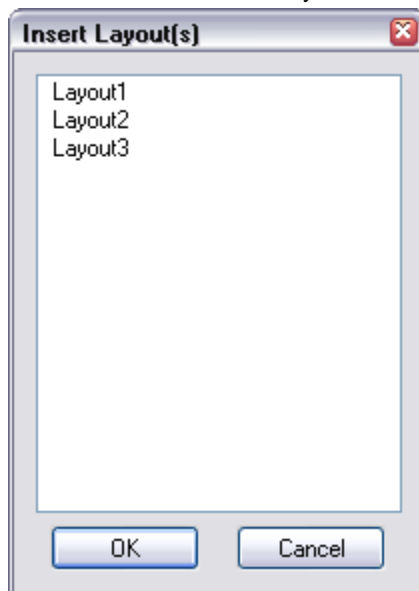
The command allows you to create one or several layouts from an existing template file (*.dwt) or drawing file (*.dwg).

Start the command to open the **Open Document file** dialogue box:



To select a template in the Open Document file dialogue box:

- Select the ***.dwt** format from the *Files of type* list;
- select the folder where the template file or drawing file is stored;
- select the file;
- click **Open**;
- select one or several layouts in the **Insert Layout(s)** dialogue box (use **SHIFT** and **CTRL** keys):



- click **OK**.

If the name of the inserted layout is the same as the name of a layout in the paper space, then the new layout will have a name consisting of the name of the existing layout, to which the name of the inserted layout is joined with an underscore character. For example, when you insert a layout called A4 into a document already having a layout with same name, a layout called A4_A4 will be created. When you next insert the layout called A4, it will be created with the name A4_A4_A4 and so on.

Save a layout as a template



Menu: **Insert – Layout > Save layout as...**



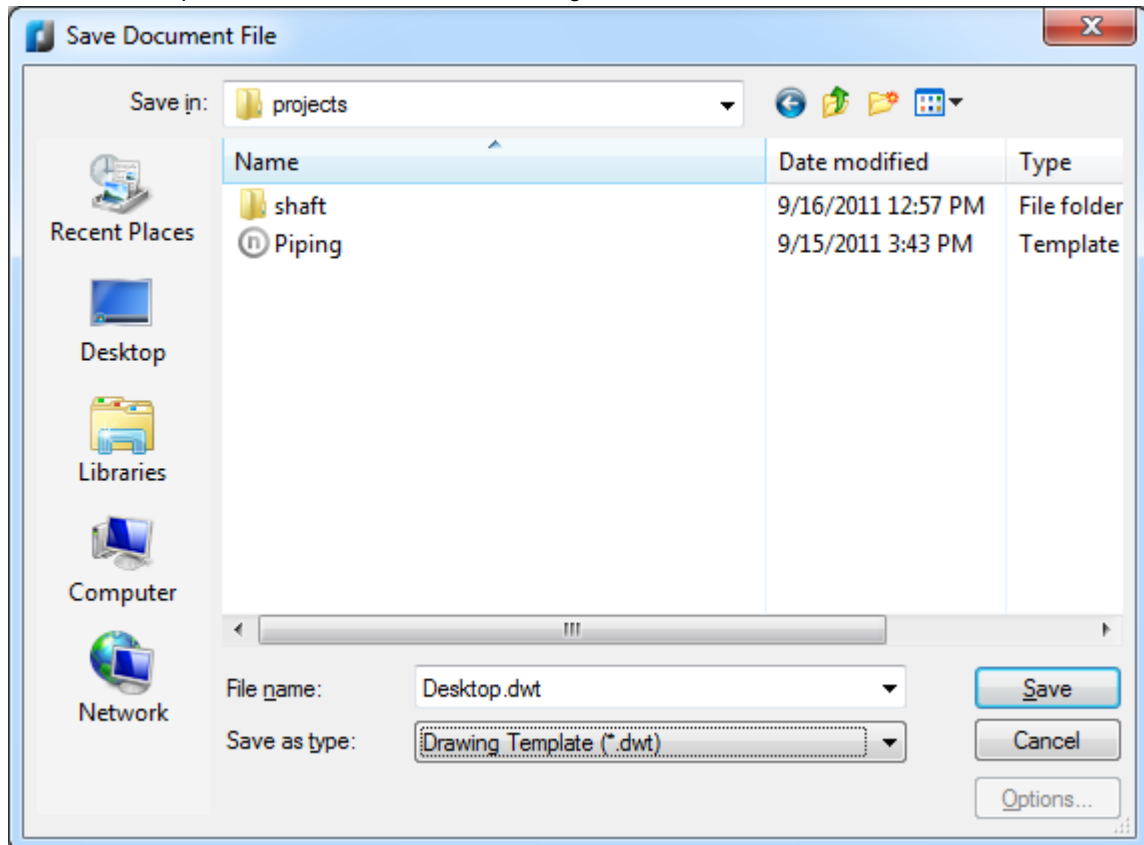
Command line: **LAYOUTTOTEMPLATE**

This command saves the created layout as a template with a *.dwt or *.dwg extension.

The command prompts, by default, to save the last active layout: **Enter layout to save <A1>:.**

Press **ENTER** to confirm. Enter the name in the command line and press **ENTER** to save a different layout.

The command opens the **Save Document File** dialogue box:




To save a template in the **Save Document File** dialogue box:

- to save the file, specify the path from the drop-down *Save in* list;
- enter a name into the *File name* field;
- select the *Drawing Template (*.dwt)* format from the drop-down **Save as type** list;
- Click **Save**.

Delete layout



Menu: **Insert – Layout >  Delete Layout**



Command line: **DELETETOCURLAYOUT**

This command prompts, by default, to delete the last active layout: **Enter layout to delete <A1>:.**

Press **ENTER** to confirm. Enter the name in the command line and press **ENTER**, to delete a different layout.

You can delete a layout by right clicking on the layout tab and selecting the **Delete Layout** command from the context menu.

Rename layout



Menu: **Insert – Layout > Rename Layout**



Command line: **RENAMECURLAYOUT**

This command prompts, by default, to rename the last active layout.

Press **ENTER** to confirm. Enter the name in the command line and press **ENTER**, to rename a different layout.

You can rename a layout by right clicking on the layout tab and selecting the **Rename Layout** command from the context menu.

Viewports

nanoCAD allows you to create layout viewports from the model space. You can create multiple layout viewports in the layout. Once you create the viewports, you can change their size, their properties and also scale and move them as needed.

It is important to create layout viewports on their own layer. When you are ready to plot, you can turn off the layer and plot the layout without plotting the boundaries of the layout viewports.

Create Layout Viewports

Create a rectangular viewport



Menu: **View – Viewports >**  **Rectangular**



Command line: **ADDRECTANGLEVIEWPORT**

nanoCAD allows use of this command on the current layout tab in which the viewport should be located, as well as directly from the model space.

Create a rectangular viewport from the model space

When you create a viewport from the model space, specify, with a frame, the drawing area (in the model space) that should be displayed in the viewport.

Command prompts:

Specify viewport first corner:

Specify the first corner of the rectangular viewport in the model space

Specify viewport second corner:

Specify the opposite corner of the rectangular viewport in the model space

Enter layout to make current <A4>:

*Enter the layout on which you want place the viewport.
By default, you will be offered the layout that was current (active) before the command start.*

Input or choose view scale
<Paper><1.000000>:

Input the viewport scale.

Specify Viewport Center Point:

Specify the viewport centre point.

Create a rectangular viewport from a layout

Options:

Extent

Display all objects from the model space in the viewport.

Active<0.000831>

Display the current view of the model space in the viewport.

Paper<1.000000>

Set the viewport scale corresponding to the layout scale.

Command prompts:

Specify viewport first corner:

Specify the first corner of the rectangular viewport in the layout tab.

Specify viewport second corner:

Specify the opposite corner of the rectangular viewport in the layout tab.

Input or choose view scale <Paper> or
[Extents/Active<0.000831>/Paper<1.000000>] :

Input the viewport scale or select the required option.

Create a polygonal viewport



Menu: **View – Viewports >**  **Polygonal**



Command line: **ADDPOLYGONVIEWPORT**

nanoCAD allows use of this command on the current layout tab in which the viewport should be located, as well as directly from the model space.

Create a polygonal viewport from the model space

When you create a viewport from the model space, specify the vertex points of the polygonal area of the drawing (in the model space) that should be displayed in the viewport.

Option:

Undo Consistently undo the specified points of the polygonal area.
The start point cannot be undone.

Command prompts:

Specify start point:

Specify the start point of the polygonal vertex.

Specify next point or [**Undo**]:

Specify the second point of the polygonal vertex.

...

...

Specify next point or [**Undo**]:

*Specify the end point of the polygonal vertex and press **ENTER**.*

Enter layout to make current <A4>:

*Enter the layout on which you want to place the viewport.
By default, you will be offered the layout that was current (active) before the command start.*

Input or choose view scale

<Paper><1.000000>:

Input the viewport scale.

Specify Viewport Center Point:

Specify the viewport centre point.

Create a polygonal viewport from a layout

Options:

Undo Consistently undo the specified points of the polygonal area.
The start point cannot be undone.

Extent Display all objects from the model space in the viewport.

Active<0.000831> Display the current view of the model space in the viewport.

Paper<1.000000> Set the viewport scale corresponding to the layout scale.

Command prompts:

Specify start point:

Specify the start point of the polygonal vertex.

Specify next point or [**Undo**]:

Specify the second point of the polygonal vertex.

...

...

Specify next point or [**Undo**]:

*Specify the end point of the polygonal vertex and press **ENTER**.*

Input or choose view scale <Paper> or
[**Extents**/**Active<0.000831>**/**Paper<1.000000>**]:

Input the viewport scale or select the required option.

Create a viewport by object



Menu: **View – Viewports >**  **by Object**



Command line: **DDOBJECTVIEWPORT**

This command allows you to convert a closed polyline, circle or ellipse into a viewport.

nanoCAD allows use of this command on the current layout tab in which the viewport should be located, as well as directly from the model space.

Create a viewport by object from the model space

Options:

Extent Display all objects from the model space in the viewport.

Active<0.000831> Display the current view of the model space in the viewport.

Paper<1.000000> Set the viewport scale corresponding to the layout scale.

Command prompts:

Enter layout to make current <A4>:

Enter the layout on which you want to place the viewport.

By default, you will be offered the layout that was

Select closed object to create viewport:
Input or choose view scale <Paper> or
[Extents/Active<0.000831>/Paper<1.000000>]:

current (active) before the command start.
Select the closed object.
Input the viewport scale or select the required option.

Create a viewport by object from a layout

Options:

- Extent Display all objects from the model space in the viewport.
- Active<0.000831> Display the current view of the model space in the viewport.
- Paper<1.000000> Set the viewport scale corresponding to the layout scale.

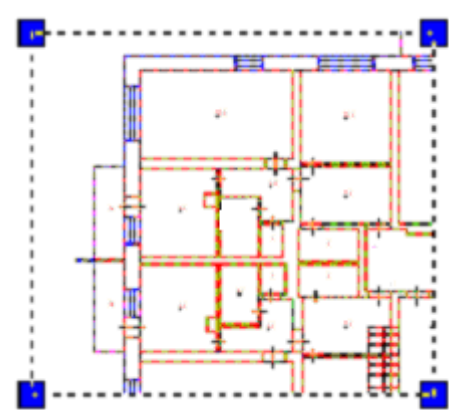
Command prompts:

Select closed object to create viewport:
Input or choose view scale <Paper> or
[Extents/Active<0.000831>/Paper<1.000000>]:

Select the closed object.
Input the viewport scale or select the required option.

Edit Layout Viewports

You can redefine the boundary of a layout viewport by using the grips. You can either use the multi-feature modes for grips editing of viewports which are used as the boundary of the closed polylines and splines (for more information, see «[Editing viewports of paper space](#)» («*EDITING OBJECTS*»-«*Ways to edit objects*»-«*Editing objects using multifunctional grips*»)). The type and location of grips depend on the type of viewport or the object from which it is converted:



The following editing commands can be used with the selected viewport: **Copy**, **Move**, **Scale**, etc. You can copy and move the viewports to the clipboard and paste them from the clipboard. The clipboard options can be modified in the **Inspector** window:

| Misc | |
|--------------------------|--------------|
| On | Yes |
| Display locked | No |
| Standart Scale | 1:1 |
| Scale | 1.0000 |
| Clipped | Yes |
| Layer property overrides | No |
| Shade plot | As displayed |

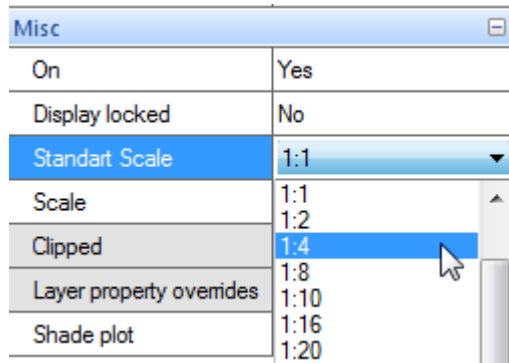
To turn on/off the content display of the viewport, use the **Yes** or **No** options from the drop-down list of the **On** option (or use the **ON/OFF** command from the **View** menu – **Viewports** item).

| Misc | |
|----------------|-----|
| On | Yes |
| Display locked | No |
| Standart Scale | 1:1 |



The objects on the turned off viewport are invisible. You cannot set as default the turned off viewport. A large number of active (with turned on visibility of content) viewports requires considerable time for regeneration. Turn off the visibility of some unused viewports to improve the regeneration of document. You can move and resize the deactivated viewports. The deactivated viewports are not printed.

The Display locked option allows you to block or unblock a viewport (**Yes** and **No** options are available in the drop-down list). A similar function is performed by the **Lock/Unlock** command from the **View** menu – **Viewports** item. Locking a viewport used to previously set the scale of a viewport remains unchanged (viewport zoom does not affect the viewport scale).

The **Standard Scale** and **Scale** option allows you to set or change the scale of the displayed view in a viewport. For example, if a drawing fragment is created in 1:1 scale and you want to specify 1:4 scale for it, it is necessary enter 0.25 value in the right column of the **Scale** option or select 1:4 from the drop-down list of the **Standard Scale** option.



The view scale of a viewport is not changed when you edit the boundaries of the viewport; for example, when you modify a boundary by grips. If you set the scale in the layout viewport before you access the model space, you can lock the scale to prevent changes. When the scale is locked, you cannot use zoom while you work in model space. You can clip the boundaries of the viewports (the **Modify** menu – **Clip** – **Viewport**).

The viewport allows you to edit the objects of the model space from the layout. If you are in paper space, double-click within a layout viewport. You are now in model space. The selected layout viewport becomes the current viewport, and you can pan the view and change the layer properties. If you are in model space in a layout viewport, double-click outside the viewport. To switch quickly from a viewport in layout, use the  button in the status bar. Located next, the  button performs switching from the layout back into the viewport.

Set show boundary for a viewport



Menu: **Modify** – **Clip** > **Viewport**

This command allows you to change the border of the created layout viewport.

Use the command to clip the border of the viewport and objects displayed on the viewport. The show boundary only defines the objects' visibility option in the viewport, the objects do not change on the drawing.

Options:

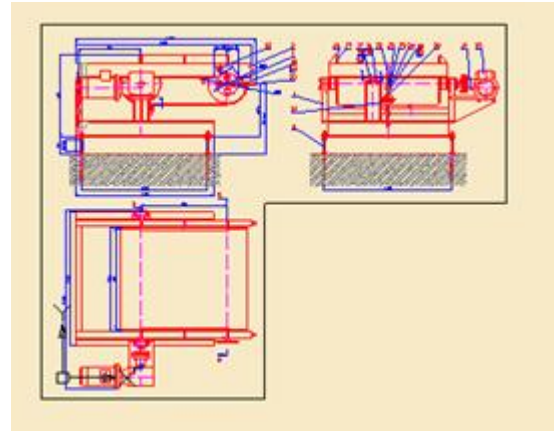
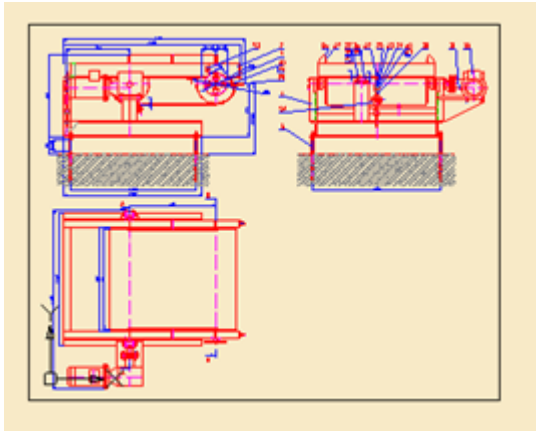
Polygonal Set the polygonal contour of the show boundary by specifying all the polygon vertices.

Undo Undo the specified points of the polygon vertices.
The start point cannot be undone.

Object Set the boundary contour by specifying the closed object.
The object should already be created.
You can use circles, ellipses, closed polylines and closed splines as objects,.

Viewport before setting the display boundary

Viewport display after setting a polygonal display boundary



Command prompts when you set polygonal boundary:

Select viewport to apply action:

Specify clipping option <Polygonal> или [**Polygonal**/**Object**/**Delete**]:

Specify start point:

Specify next point or [**Undo**]:

...

Specify next point or [**Undo**]:

Command prompts when you set boundary by object:

Select viewport to apply action:

Specify clipping option <Polygonal> или [**Polygonal**/**Object**/**Delete**]:

Select closed object to create viewport:

Select the viewport.

Select the **Polygonal** option.

Specify the start point.

Specify the second point.

...

Specify the end point and press **ENTER**.

Select the viewport.


Select the **Object** option.


Select the closed object.

Document plot

To print document:

- In the Layers dialogue box (the Format menu – Layers...), carry out the following for the layers that should be printed:

— turn on the visibility (),

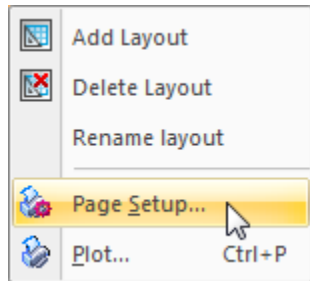
— turn on the availability to print ().

- If necessary, set the display order of the objects, using the **Display Order** commands from the **Tools** menu.

- Set the printer settings for each layout:

— right-click on the *Model* tab or corresponding tab in the *Layout*,

— in the opened context menu select the **Page Setup...** command:



— create a new, edit existing or import the parameters set of a page previously created in another document,

— when you create a new or edit an existing parameters set of a page, specify the required parameters in the **Page Setup Manager** dialogue box. Click the Apply to layout button to apply the parameters set to the current layout and close the dialogue box.

- Print out the task (the **File** menu - the **Plot** or **Batch plot** command).

Page Setup Manager



Menu: **File** –  **Page Setup...**



Toolbar: **Main** – 

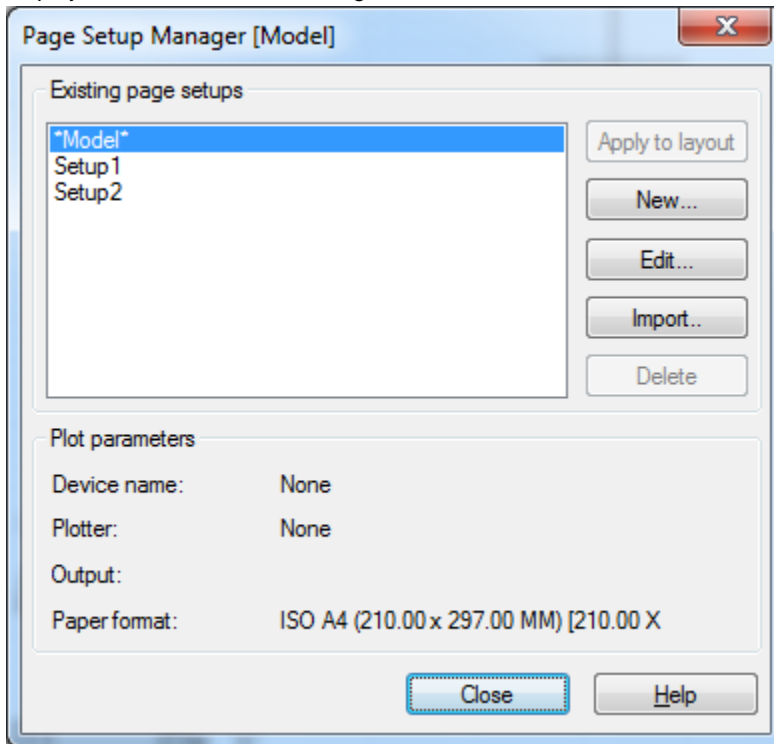


Command line: **PAGESETUP**

When you are preparing a document for printing, it is necessary to save the settings of the plot options in the *named page setups*. Use the *named page setups* to reduce the document preparation time for the plot by applying previously made plot settings to new documents layouts.

You can create named page setups, modify existing ones or import page setups from other drawings. The page setups are assigned to each page of the document and stored in a document file.

Start the command and the **Page Setup Manager** dialogue box will be opened. The name of the current layout is displayed in brackets in the dialogue box title bar:



Options:

Existing page setups

In the left part of the section, the list of page setups that are available to apply to the current layout is displayed. Layouts that have a named page setup applied to them are enclosed in asterisks (for example, **Model**, **Layout1**).

Apply to layout

Displays the page setup that is applied to the current layout. Layouts that have a named page setup applied to them are enclosed in asterisks, with the named page setup in parentheses; for example, **Model(Setup)**, **Layout(Setup)**, etc.

Note: When the current layout is selected, the **Apply to layout** button is not available.

New...

Opens the **New Plot Set** dialogue box, in which you can select a set created earlier as a template and enter a name for the new page setup.

Edit...

Opens the **Page Setup** dialogue box, in which you can edit the settings for the selected page setup.

Import..

Opens the **Open Document File** dialogue box (a standard file selection dialogue box), in which you can select a file from which to import one or more page setups.

Delete

Deletes the selected setup from the existing page setups list.

Plot parameters

Displays information about the selected page setup.

Device name:

Displays the name of the plot device.

Plotter:

Displays the type of plot device.

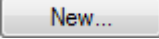
Output: Displays the physical location of the output device.

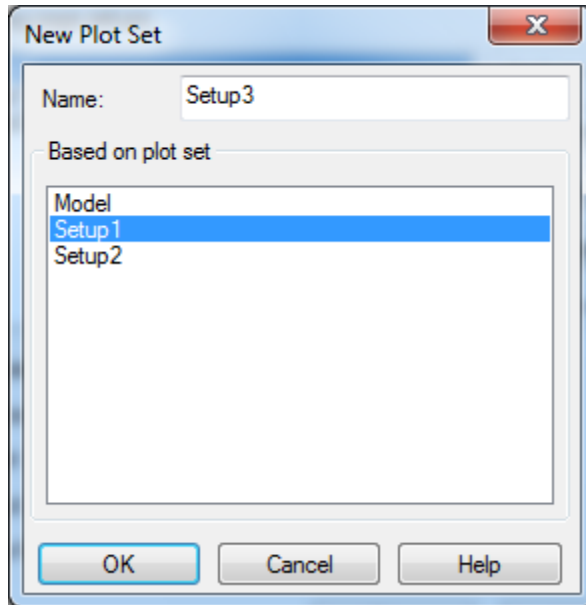
Paper format: Displays the name, paper size and orientation specified in the currently selected page setup. The paper size and orientation are displayed in square brackets [].

Note: The name of the paper format can include the paper size that is displayed in round brackets ().

Note: The page setups created for the layouts are not applied to the model space. Conversely, the page setups created for the model space are not applied to the layouts.

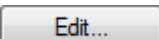
To create the page setup:

- Click the  button.
- In the opened **New Plot Set** dialogue box in the *Based on plot set* section, select an already created setup as the template.




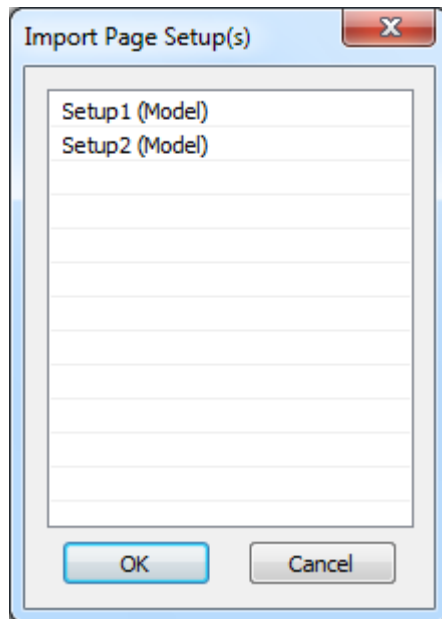
- In the *Name* field, enter the name of the new setup.
Note: By default, the created page setup will be named *SetupN*, where *N* is the number of the created setup. It is recommended to assign a user name to the new setup which will display the name and orientation, plotter name, etc. For example, *A4 (portrait) PDFCreator* or *A1 (landscape) CanonLargeFormat W7250*.
- Click **OK**.
- Specify the necessary parameters in the **Page setup** dialogue box and click **OK**.
The newly created setup is displayed in the *Existing page setup* list of the **Page Setup Manager** dialogue box.

To edit the page setup:

- Select the setup to edit from the *Existing page setup* list.
- Click the  button.
- Edit the required parameters in the opened **Page Setup** dialogue box.
- Click **OK**.

To import the page setup:

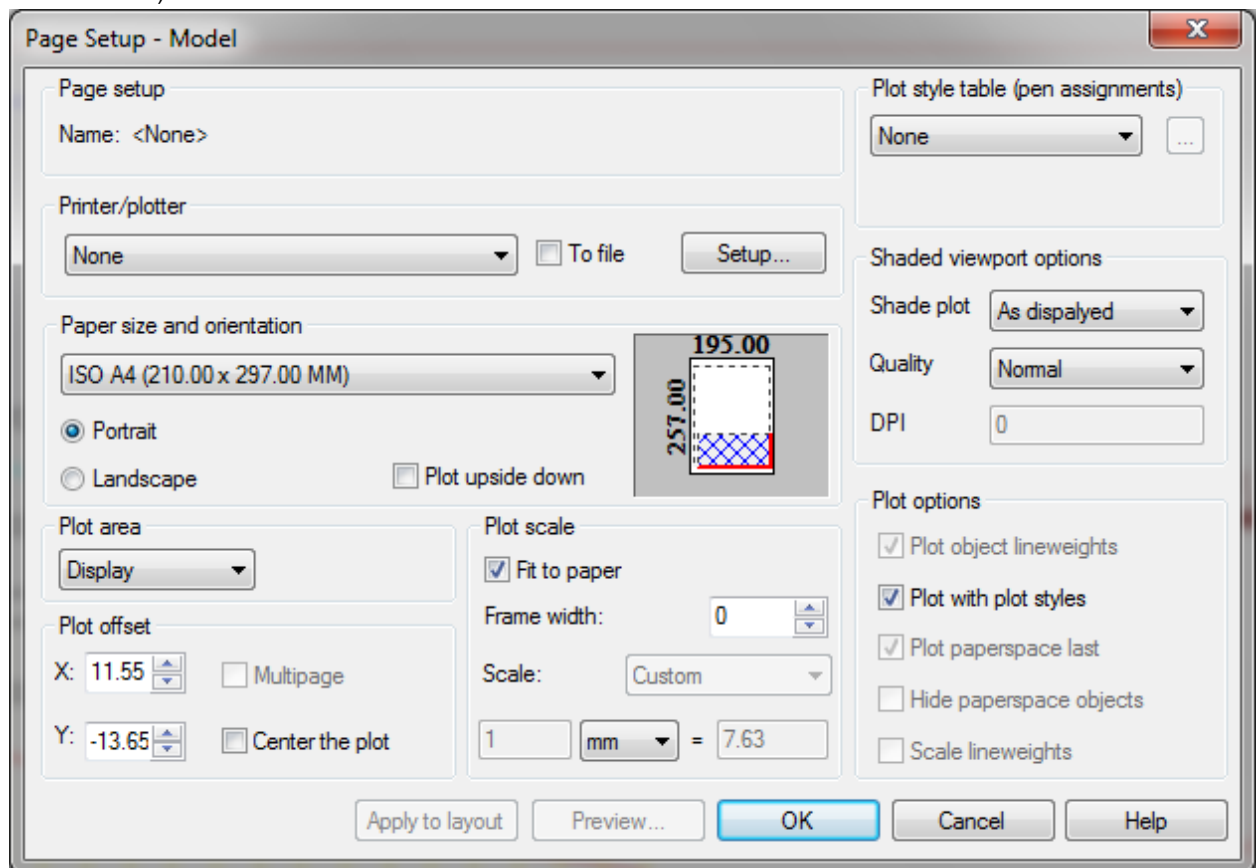
- Click the  button.
- In the opened standard file selection dialogue box, select the file or folder in which it is located.
Note: You can import page setups from the drawing format (DWG), drawing template (DWT) or drawing interchange format (DXF)
- Select one or more page setups in the Import page setup dialogue box (use the **SCHIFT** and **CTRL** keys).



- Click **OK**.

Page Setup

The **Page Setup** dialogue box is displayed when you create a new or edit an existing page setup (the **New** and **Create** button):



Options:

Printer/plotter

Name:



Drop-down list displays the available plotting devices.

Displays the dialogue box, where you can view or modify the current plotter configuration.

The dialogue box view and configuration settings are determined by the driver

of the current plotting device.

Paper size and orientation

ISO A4 ▼

Drop-down list to select the paper size for the current plotting device

Portrait

Sets the portrait paper orientation.

Landscape

Sets the landscape paper orientation.

Plot upside down

Turns round the print area.

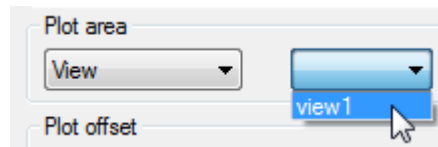
Plot area

Display ▼

Specifies the area of the drawing to be plotted.

From the drop-down list the following options are available:

- Display** Plots the view in the current viewport in the *Model* tab or in the current paper space view in a *Layout* tab.
- Extents** Plots the portion of the current space of the drawing that contains objects. All geometry in the current space is plotted.
- Limits** When plotting from the *Model* tab, plots the entire drawing area that is defined by the grid limits.
- Layout** When plotting a layout, plots everything within the printable area of the specified paper size.
- View** Plots a view that was previously saved with the *View* command. You can select a named view from the list:



The screenshot shows the 'Plot area' section of a dialog box. It contains a 'View' dropdown menu which is currently open, displaying a list with 'view1' selected. Below the dropdown is a 'Plot offset' checkbox, which is currently unchecked.

Note: If there are no saved views in the drawing, this option is unavailable.

Window

Plots any portion of the drawing that you specify.

When you select this option, the *Page Setup* dialogue box will be temporary closed and in the command line the prompt appears: *Specify First Corner or [Select]*: you can specify the two corners of the area to be plotted. The paper size and orientation in this case are defined by the parameters specified in the *Paper size and orientation* section.

Click the *Window* button to use the pointing device to specify the two corners of the area to be plotted, or enter coordinate values. The *Select* option allows you to specify the plot area by selecting one side of the frame that limits the paper size.

When you specify the first plot area, additional options are displayed in the dialogue box:



The screenshot shows the 'Plot area' section of a dialog box. It contains a 'Window' dropdown menu. To the right of the dropdown are three icons: a square with a plus sign, a square with a smaller square inside, and a square with an 'X' inside.



New window print area

Cancels all plot areas by setting a new area.



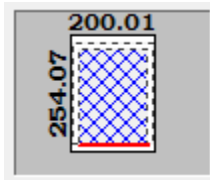
Add window print area

Adds the plot area to those already specified.



Delete last print area

Removes the specified plot areas in the reverse order of their assignment.



Shows an accurate representation of the effective plot area relative to the paper size and printable area.

The tooltips display the paper size and printable area:

Dashed line – defines the plot area for the specified paper size (numbers indicate the length and width of the plot area).

The blue hatch bounded by the dashed contour – defines the location of the specified plot area on the paper.

Red line – is a warning that the specified plot area exceeds the plot boundaries.

Plot offset

X:

Specifies the plot origin in the *X* direction relative to the setting of the *Plot Offset* option.

Y:

Specifies the plot origin in the *Y* direction relative to the setting of the *Plot Offset* option.

Multipage

Turn on/off the multipage plot mode. This option is available when the *Fit to paper* option is off.

Center the plot

Turn on/off the mode for automatic offset detection in the *X* and *Y* directions so that the drawing is placed in the centre of the paper.

Plot scale

Fit to paper

Scales the plot to fit within the selected paper size.

Frame width:

This option considers the weight of the frame line defining the drawing boundaries and coinciding with the plot area boundaries. When you set a parameter value equal to the weight of the frame lines, the frame prints without reducing its thickness.

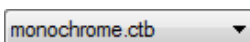
For example, if the parameter value is equal to zero, the frame created by lines of 1 mm, will be printed at 0.5 mm thick. Set the parameter to a 1 mm value to print the frame at 1 mm thick.

Note: The **Frame width** option is available when the **Fit to paper** mode is on.

Scale:

Defines the exact scale for the plot. You can create a custom scale by entering the number of inches (or millimetres) equal to the number of drawing units.

Plot style table (pen assignments)



Drop-down list to select the current plot style.



Opens the **Plot Style Table Editor** dialogue box to edit the plot styles for the currently assigned plot style table or create new.

Shaded viewport options

Shade plot:

Specifies how shaded and rendered viewports are plotted.

The following options are available:

As displayed - plots objects the way they are displayed on the screen.

Wireframe - plots objects in wireframe, regardless of the way they are displayed on the screen.

Hidden - plots objects with hidden lines removed, regardless of the way the objects are displayed on the screen.

Rendered - plots objects as rendered, regardless of the way they are displayed on the screen.

Quality:

Specifies the resolution at which shaded and rendered viewports are plotted.

The following options are available:

Draft –sets rendered and shaded model space views to be plotted as wireframe.

Preview - sets rendered and shaded model space views to be plotted at one quarter of the current device resolution, to a maximum of 150 dpi.

Normal - sets rendered and shaded model space views to be plotted at

one half of the current device resolution, to a maximum of 300 dpi.

Presentation - sets rendered and shaded model space views to be plotted at the current device resolution, to a maximum of 600 dpi.

Maximum - Sets rendered and shaded model space views to be plotted at the current device resolution with no maximum.

Custom - sets rendered and shaded model space views to be plotted at the resolution setting that you specify in the DPI box, up to the current device resolution.

DPI:

Specifies the resolution of the current plotting device.

This option is available if you select **Custom** in the **Quality** box.

Plot options

The options that allows you to switch between ready-configured plot styles (the *.ctb or *.stb files) and customised properties for the objects in the current document.

Plot object lineweights

Specifies whether lineweights assigned to objects and layers are plotted.

Plot with plot styles

Specifies whether plot styles applied to objects and layers are plotted.

When you select this option, **Plot Object Lineweights** is automatically selected also.

Plot paperspace last

Plots model space geometry first.

Hide paperspace objects

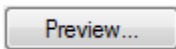
Specifies whether the **Hide** operation applies to objects in the paper space viewport.

This option is available only from a layout tab.

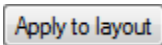
The effect of this setting is reflected in the plot preview, but not in the layout.

Scale lineweights

Scales lineweights in proportion to the plot scale. Lineweights normally specify the linewidth of plotted objects and are plotted with the linewidth size, regardless of the plot scale.



Displays the drawing as it will appear when plotted on paper by executing the **Preview** command.



Displays the page setup that is applied to the current layout.

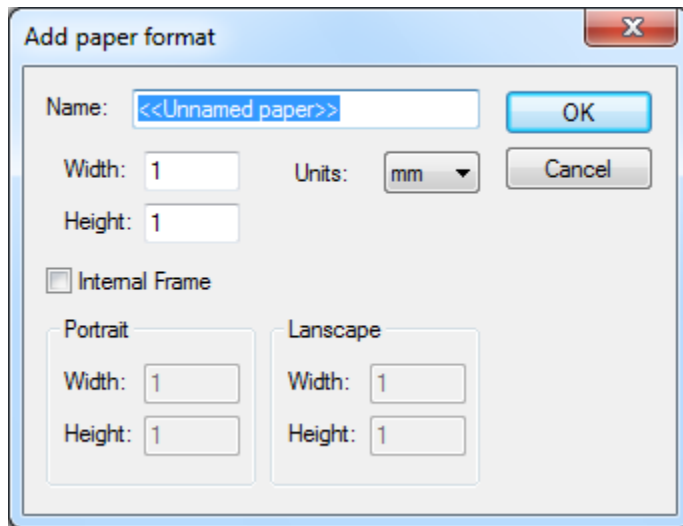
The named page setup applied to the layout is displayed in the title of the **Plot** dialogue box when you print this page.

Set the custom paper formats

If necessary, it is possible to set a paper format that is not in the drop-down list of the *Paper size and orientation* list in the **Plot** and **Page Setup** dialogue boxes. You can set new paper formats in addition to those available in nanoCAD in the *Papers* section of the **Options** dialogue box (the **Tools** menu – **Options**).

To set a new paper format:

- Select paper formats in the *Papers* section of the **Options** dialogue box.
- Click the **Add** button.
- In the opened **Add paper format** dialogue box specify the name, units, paper size and plot area size for the new paper format.



Note 1: The plot area sizes are set according to the plotting device used because these are determined by the manufacturer of the plotting device.

Note 2: When you set the paper size, the minimum and maximum sizes for the plotting device used should be considered. They are also determined by the manufacturer of the plotting device.

- Click **OK** to close the window.
- Click **OK** to close the **Options** dialogue box.

The specified paper format will be displayed in the *Paper size and orientation* list in the **Plot** and **Page Setup** dialogue boxes after you select a specific plotting device.

Note 3: If the paper format for which the sizes given are more or less acceptable for a specific plotting device, then the paper size and orientation list for this device are not displayed.

Plot Style Editor



Menu: **File** –  **Plot Style Manager...**



Toolbar: **Main** – 



Command line: **PLOTSTYLEMANAGER**

A plot style controls how an object or layer is plotted by determining the plotted properties such as lineweight, colour, and fill style. Plot style tables collect groups of plot styles. The same drawing can be printed in different ways, for example, in colour or monochrome, by assigning various plot styles to it.

The **Plot Style Manager** dialogue box is also used to change plot style settings for both *named* and *colour-dependent* plot style tables.

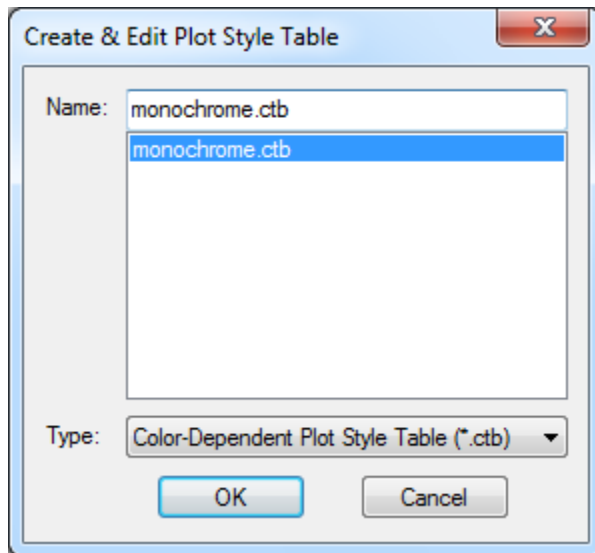
Colour-dependent plot style tables are stored in the *Plot Styles* folder and have a ***.ctb** extension. By using colour-dependent plot styles to control how objects are plotted, you ensure that all objects that share the same colour are plotted the same way.

All named plot style tables have an ***.stb** extension. You can only create, delete and apply plot styles in a named plot style table. You can define as many or as few plot styles as you need in a drawing.

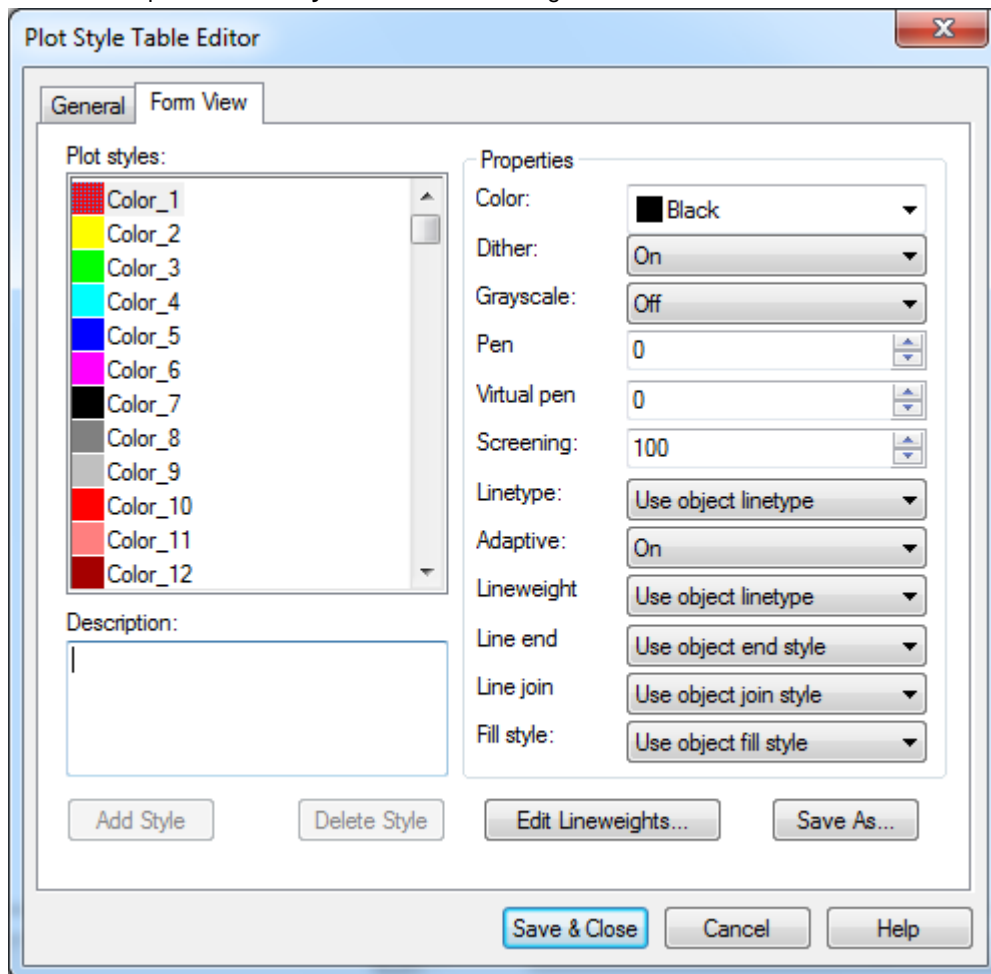
Colour-dependent (***.ctb** extension) and named (***.stb** extension) plot style tables are stored in the *Plot Styles* folder by default:

c:\Documents and Settings\User_name\Application Data\Nanosoft\NanocAD 3.0\PlotStyles.

You can add, delete, rename and copy plot styles in a named plot style table using the **Plot Style Manager** dialog:

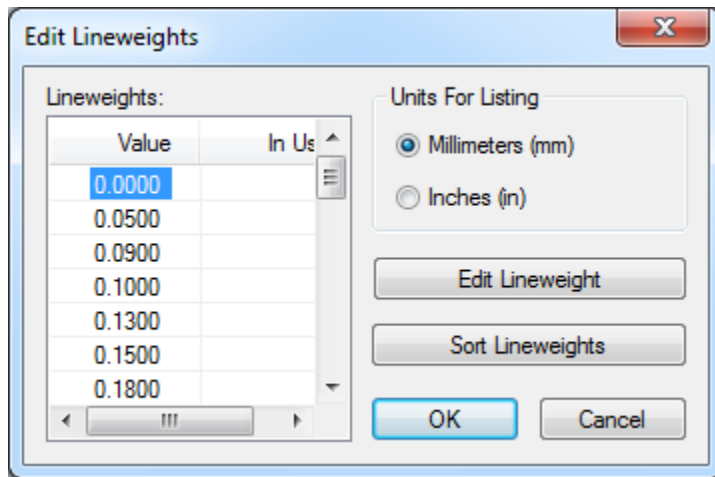


Click OK to open the **Plot Style Table Editor** dialogue box:



The **General** tab contains a description of the plot style table and general information about the file. The **Form View** tab lists all of the plot styles in the plot style table and their settings.

The **Edit Lineweights...** button opens the **Edit Lineweights** dialogue box:



The plot style editor can be opened by the  button from the **Page Setup** dialogue box.

Plot Preview



Menu: **File** –  **Plot Preview...**



Toolbar: **Main** – 

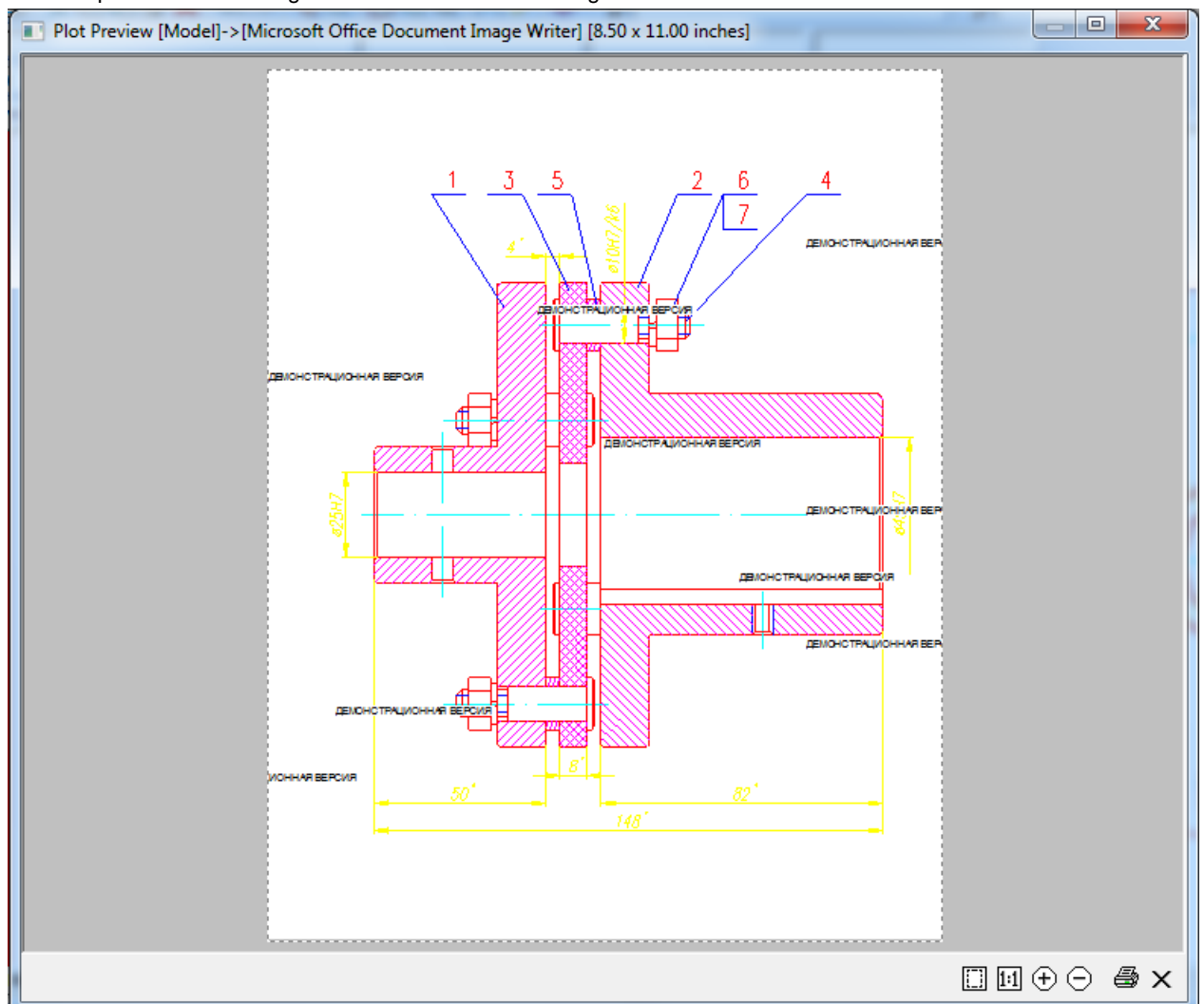


Key combination: **CTRL+F2**









Command line: **PREVIEW**

You can preview the drawing from the **Plot Preview** dialogue box:








Note: A preview is not available for a layout that does not have an assigned plotter.

Options:

| | | |
|---|-----------------|--|
|  | Zoom all | Display the whole plot area in the preview window. |
|  | Zoom 1:1 | Display the plot area at a scale of 1:1. |
|  | Zoom in | Zoom in to the image. |
|  | Zoom out | Zoom out from the image. |
|  | Print | Sends the drawing to the plotter. |
|  | Close | Closes the Plot Preview dialogue box. |

Additional options are displayed in the dialogue box for a multipage plot:

| | | |
|--|----------------------|---|
|  | First page | Display the first page of a plot set in the preview. |
|  | Previous page | Display the previous page of a plot set in the preview. |
|  | Next page | Display the next page of a plot set in the preview. |
|  | Last page | Display the last page of a plot set in the preview. |

| | |
|---|--|
|  | Display the number of the page you are viewing in the window, and the total number of pages of the plot set. |
|---|--|

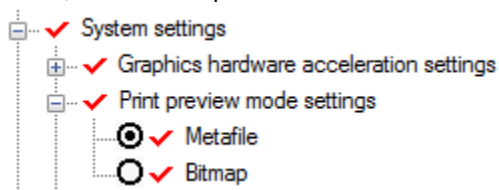
You can zoom and pan an image in the preview window by clicking inside the window:

- To zoom an image, rotate the mouse wheel.
- To pan an image, move the mouse with the left or right button pressed and also the mouse wheel.







You can open the **Plot Preview** dialogue box from the **Page Setup**, **Plot** and **Publish** dialogue boxes.

When printing files that have rich graphics, a large number of viewports, etc., the message “Not enough memory to create preview” can appear in the preview window. In this case it is necessary to change the *Print preview mode settings* in the *System setting* item of the **Options** dialogue box.

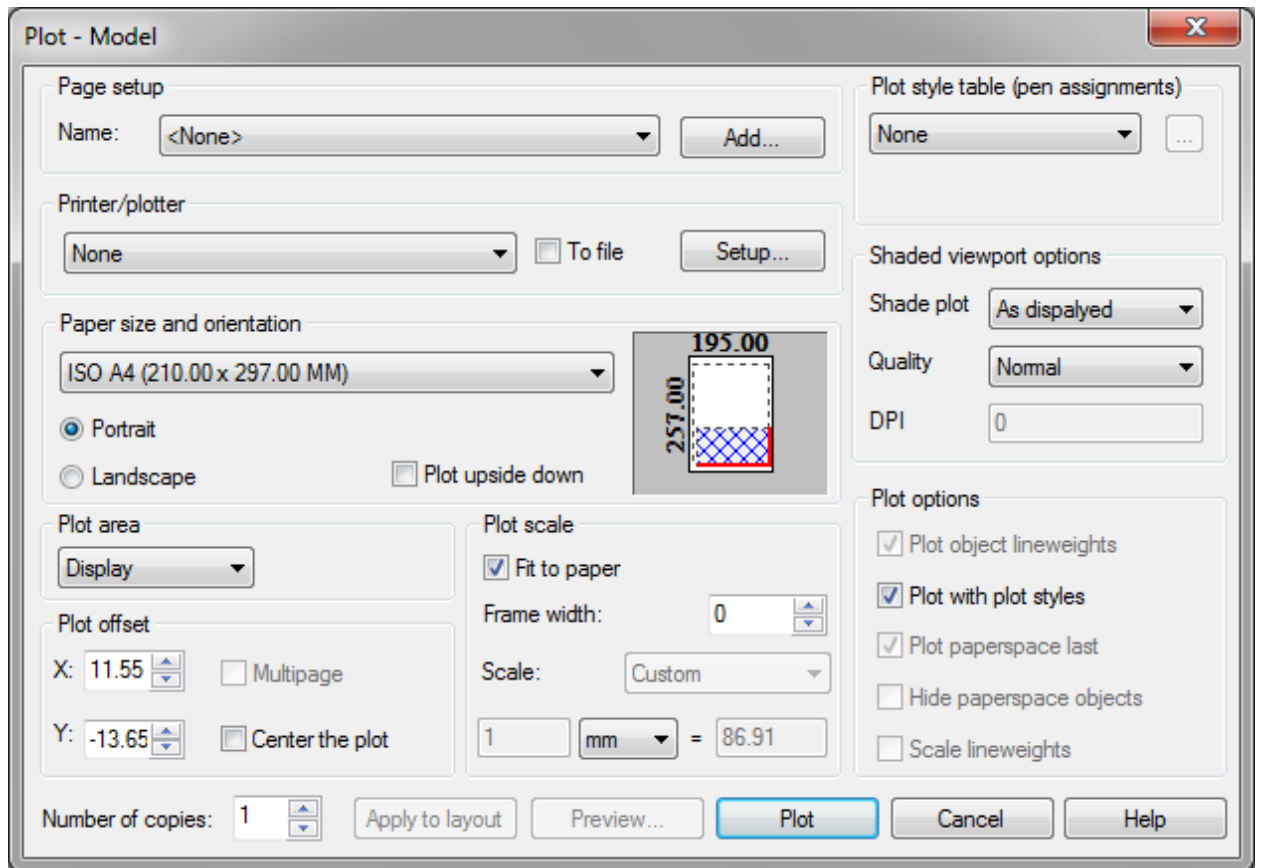
By default, the **Metafile** parameter is set:



Plot

-  Menu: **File** –  **Plot...**
-  Toolbar: **Main** – 
-  Key combination: **CTRL+P**
-  Command line: **DWFOUT, PLOT**

This command opens the **Plot** dialogue box that differs from the **Page Setup** dialogue box only in the *Page setup* section and it has a **Plot** button instead of **OK**.



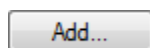
Options:

Page setup

Name:

Displays a list of available page setups in the drawing.

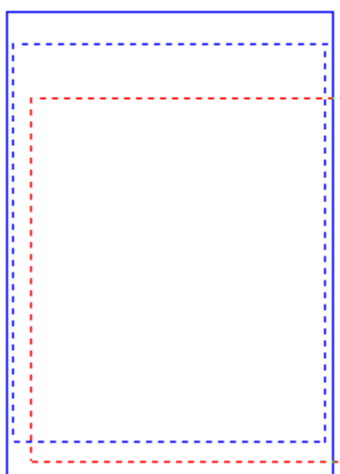
After the first drawing plot, the <Previous plot> option is available in the list. It stores the settings of the last plot.



Displays the **New Plot Set** dialogue box, in which you can save the current settings in the Plot dialogue box to a named page setup.

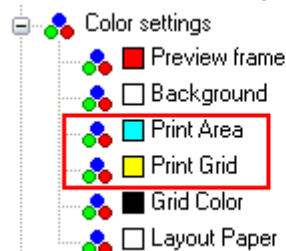
Description of other options see in the [Page Setup](#) section («*THE COMPONENT LAYOUT AND PLOT DRAWINGS*» - «*Document plot*» - «*Page Setup Manager*»).

When you set the plot area, the following parameters are displayed on the screen:





- size and orientation of the selected plot paper format as a frame which is displayed by a solid blue line,
- real plot area for the specified printer and selected paper format as a frame which is displayed by a dashed blue line,
- specified plot area of the drawing as a frame which is displayed by a dashed red line.

You can change the colour of the frame specified by default in the *Color settings* section of the **Options** dialogue box (the **Tools** menu – **Options**):




To set several plot areas:

- In the *Plot area* section, select the **Window** option from the drop-down menu.
- Specify the first and second corners of the first plot area on the screen.

- In the Plot dialogue box click the **Add window print area**  button and specify the second plot area.
- Repeat the process of specifying plot areas for other plot areas.
- To cancel the last plot area specified in error, click the **Delete last print area**  button. When you click this button again, the previous plot area will be deleted and so on.

Note 1: When you click the **Delete last print area**  button, the **Plot** dialogue box is not closed.

Note 2: Click the **New window print area**  button to cancel all specified plot areas and specify a new plot area.

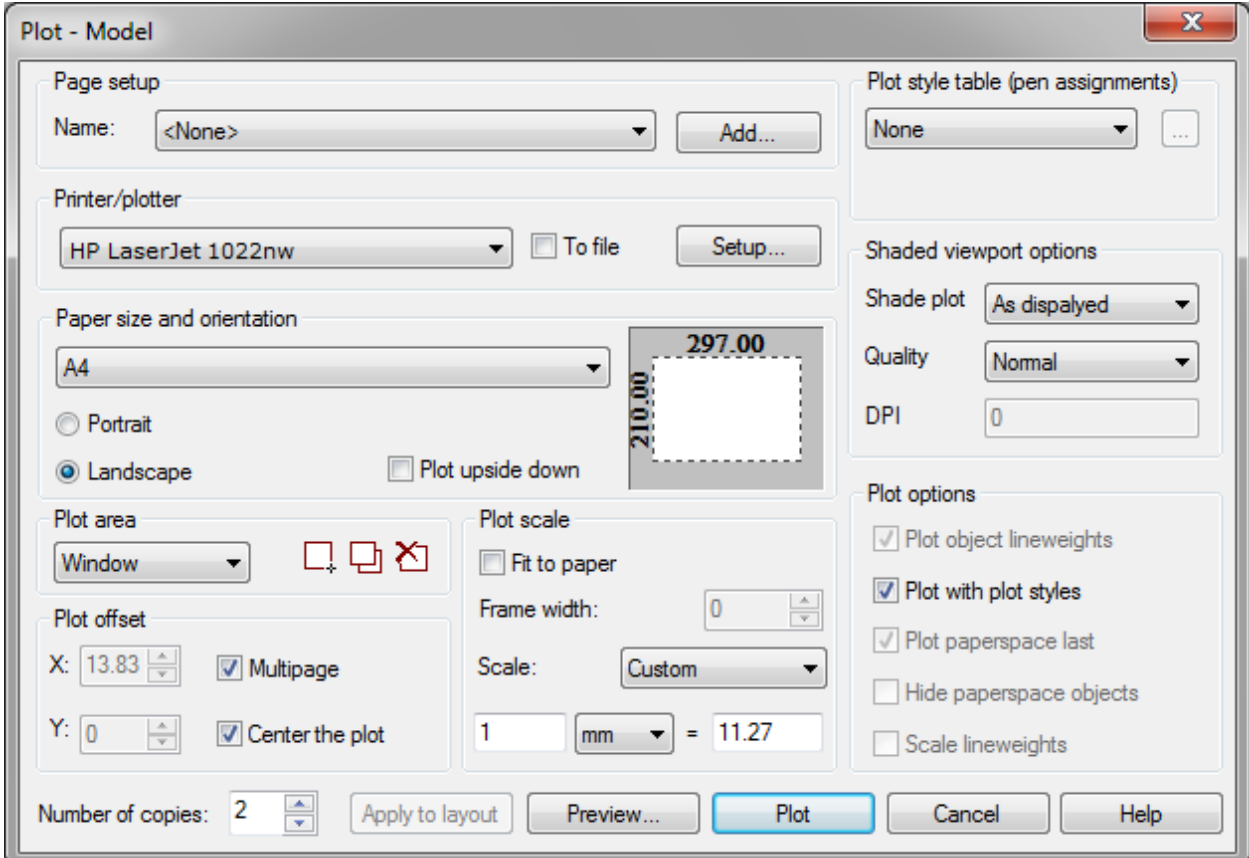
To set a multipage plot:

- Select a printer.
- Specify the paper size and orientation.
- Turn off the **Fit to paper** option.
- Specify the plot scale.
- Turn on the **Multipage** option.
- Specify the plot area.
- If necessary, set the plot offset or turn on the **Center the plot** option.

The multipage plot is useful to print large formats (A0, A1, etc.) on printers that do not support a plot of these formats.

Example: Plot of the A1 (landscape) format on A4 papers

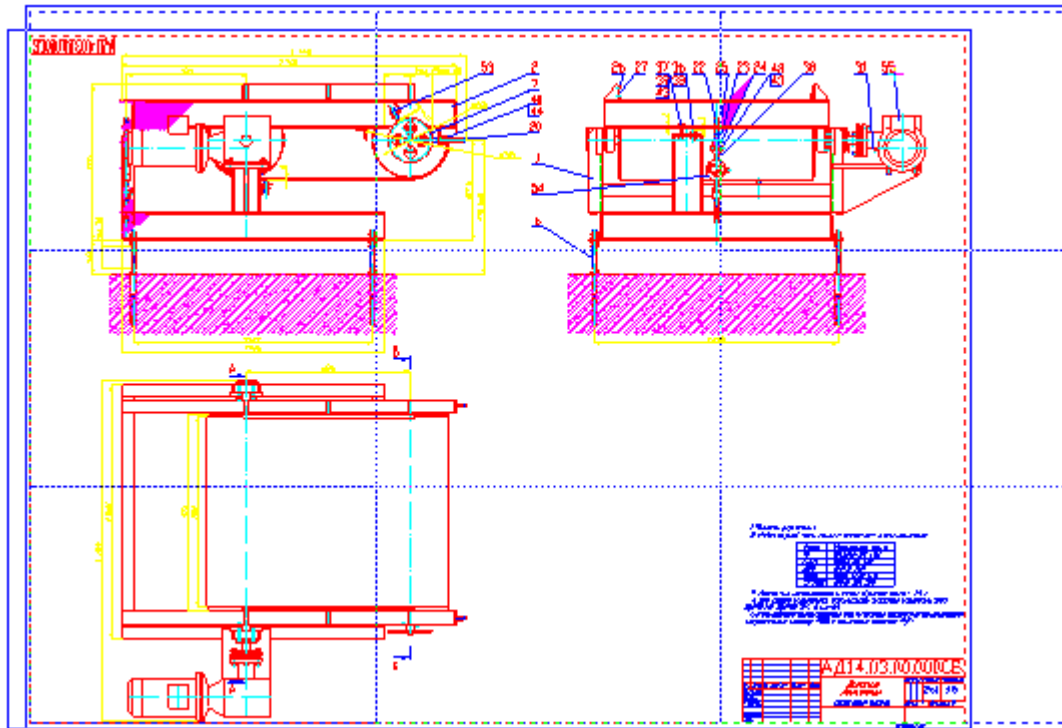
The Plot settings:



The screenshot shows the 'Plot - Model' dialog box with the following settings:

- Page setup:** Name: <None>, Add... button.
- Printer/plotter:** HP LaserJet 1022nw, To file checkbox, Setup... button.
- Paper size and orientation:** A4, Portrait/Landscape radio buttons (Landscape is selected), Plot upside down checkbox, a preview of the A1 layout on A4 paper with dimensions 297.00 and 210.00.
- Plot area:** Window dropdown, Add, Remove, and Delete buttons.
- Plot offset:** X: 13.83, Y: 0, Multipage and Center the plot checkboxes.
- Plot scale:** Fit to paper checkbox, Frame width: 0, Scale: Custom dropdown, 1 mm = 11.27.
- Plot style table (pen assignments):** None dropdown, ... button.
- Shaded viewport options:** Shade plot: As displayed dropdown, Quality: Normal dropdown, DPI: 0.
- Plot options:** Plot object lineweights, Plot with plot styles, Plot paperspace last, Hide paperspace objects, Scale lineweights checkboxes.
- Bottom bar:** Number of copies: 2, Apply to layout, Preview..., Plot, Cancel, Help buttons.

Display the A1 layout to A4 papers in the model space:



Batch Plot



Menu: **File** –  **Batch Plot...**



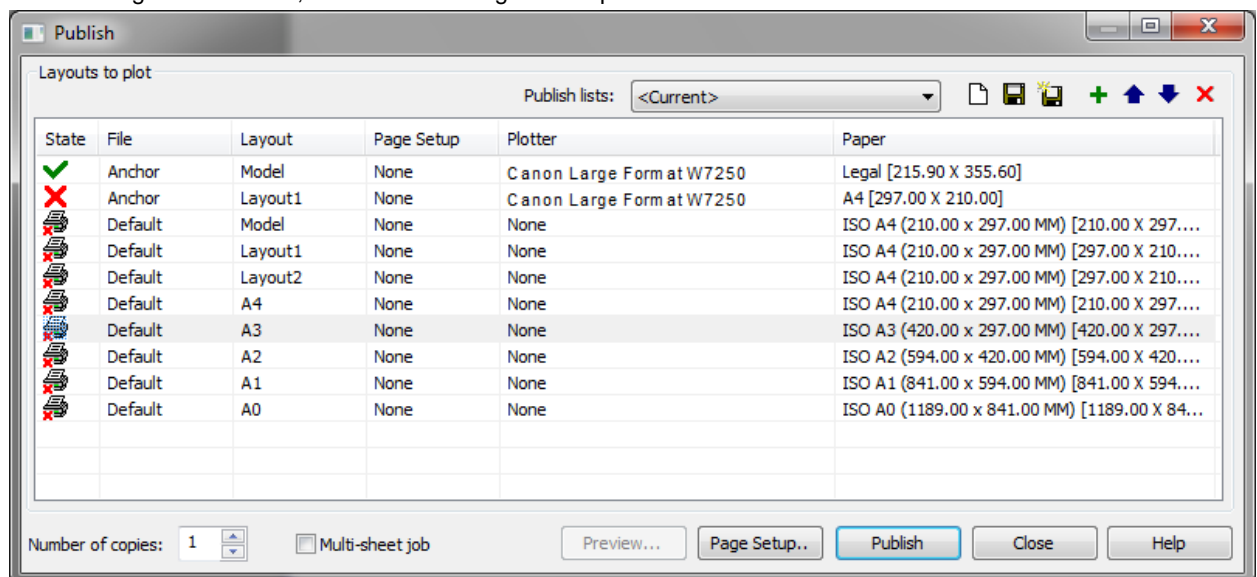
Toolbar: **Main** – 



Command line: **PUBLISH**

This command allows you to create and send to print a drawing set by publishing the sheet set to a single, multi-sheet file. You can create a paper drawing set by publishing the sheet set to the plotter named in each sheet's page setup. You can customise a collection of drawings for a specific user and you can add and remove sheets from the list. The **Multi-sheet job** option allows you to place one or more files in the list, for example, when you create PDF-files using a virtual printer. The order of the sheets in the file corresponds to the order of the sheets in list.

After starting the command, the **Publish** dialogue box opens:



Options:

Publish lists

Drop-down list containing the named publish lists available to plot.

The two lists are displayed by default:

Last - last layouts lists sent to plot.

Current – list of all layouts available in the opened documents.

Number of copies:

Specifies the number of copies to publish.

Multi-sheet job

Turn on/off the mode to plot sheets in the file, for example, when you print on a virtual printer. The sheets plot is carried out in order, as specified in the dialogue box. If you assign the same plotter for all sheets, they will be placed in one file. If you assign different plotters for different sheets, the plot will be carried in separates files for each plotter. If you change the plotter, a new file will be created.

Columns

State

Column displays the plot status:



Print sheet.



Don't print sheet.



The plot is forbidden (in the case of incorrect plot settings, for example, do not specify a plotter).

Double-click to switch the status from *Plot* to *Not plot*.

File

Displays the file name.

Layout

Displays the name of the layout tab in paper space. Displays the *Model* for the model space.

Page setup

Displays the named page setup for the sheet. If the sheet does not have an assigned page setup, it displays *None*.

Plotter

Indicates that the output devices given for each drawing sheet in the page setup will be used. If you don't assign an output device to a sheet, the *None* value will be displayed.

Paper

Displays the paper size and orientation.

Buttons



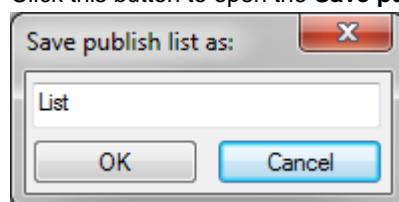
Deletes all layouts from the list to create a new publish list.



Кнопка сохранения списка листов.

Saves publish list.

Click this button to open the **Save publish list as** dialogue box:



The *List* name is displayed by default.

But you can set the user name list.

Click **OK** and the named list will be saved in the folder:

C:\Documents and Settings\Users\User name \Application Data\Nanosoft\nanoCAD\PlotConfigs

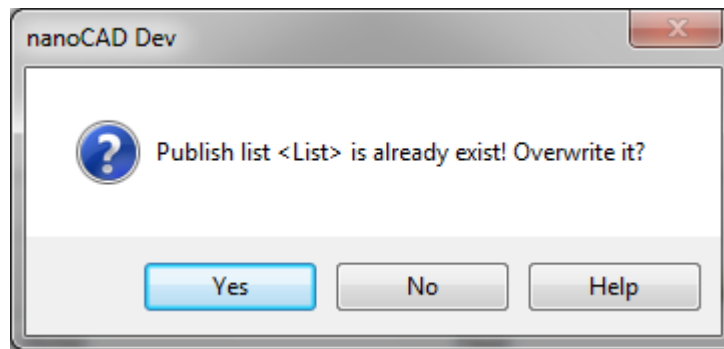
The named lists are saved in the files with ***.plst** extension.



Saves the publish lists with a new name.

Click this button to open the **Save publish list as** dialogue box. The *List* name is displayed by default.

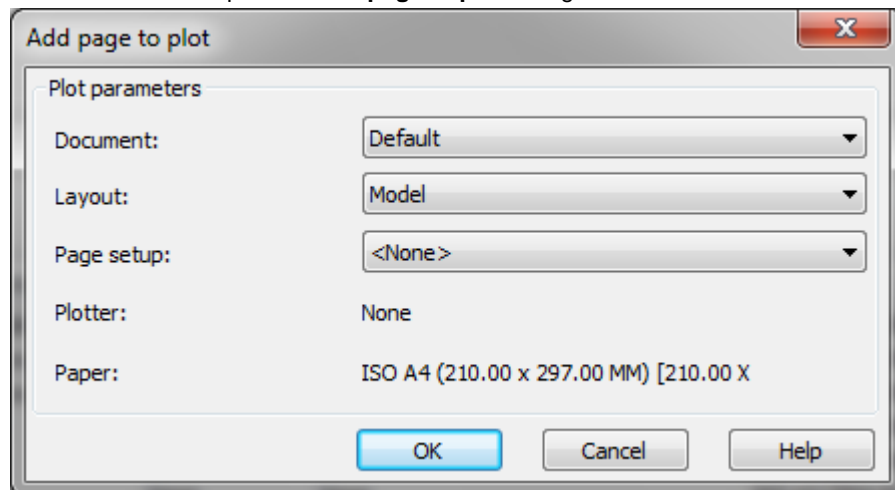
If a new name of publish list is same as already existing, after you click **OK** the following dialogue window appears:



Adds a new page to plot.

It is possible to select page only from the opened documents.

Click this button to open the **Add page to plot** dialogue box:



From the drop-down lists of the **Add page to plot** dialogue box you can select:

- document containing page to add to plot (**Document:**);
- Layout from selected document (**Layout:**);
- A named set of parameters list, if it exists in the document (**Page setup:**)



Moves the selected drawing sheets up one position in the list.



Moves the selected drawing sheets down one position in the list.

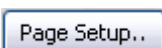


Deletes the selected drawing sheets from the list of sheets.



Opens the **Plot preview** dialogue box.

The preview of plot results is available for the selected sheet in the list. When you print from the preview mode, only the viewable sheet will be printed.

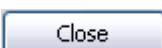


Opens the **Page Setup** dialogue box to specify the plot options.

Double-click on any column of the selected sheet, except the **State** column, to open the **Page Setup** dialogue box.







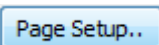

Starts the publishing operation. The sheets, with the state allowing printed output, will be printed.

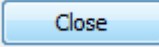


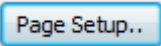
Closes the dialogue box.

To create named publish lists to plot:



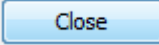
- Open one or more documents, the layouts of which it is necessary to plot.
- Start the  **Batch Plot** command.
- When you open the **Publish** dialogue box, the list of the layouts, available in the opened documents, is formed automatically. The following order of the layouts is set by default:

- the first in the list is a document from which you run the **Batch Plot** command. This is followed by the documents in order of their tabs in the graphic editor window (from left to right);
- the *Model* tab is first displayed in the document. Next there are tabs of *Layout1*, *Layout2*, *A4*, *A3*, etc.
- To edit list:
 - to remove unnecessary to plot layouts, select them and click  **Remove from list** button or press **DEL**;
 - to move the layouts, use the  **Move up** and the  **Move down** buttons.
- To set plot settings for each page, click the  button (double click on any selected layout column, except the *State* column, also opens the **Page Setup** dialogue box).
Use the previously created named sets of page setup, to reduce the time of parameters settings.
- Click the  **Save publish list** button.
- Enter a name of the list in the **Save publish list as** dialogue box.
- Click **OK** to save list.



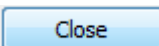
- Click  button, to close the **Publish** dialogue box.

Note: When you save a plot setting for each page, the name and location of the file are stored in the list. To select a named list of layouts to plot, double click on any column of selected layout, except for the Status column (or click the  button). File will be loaded automatically, provided that the path has not changed.


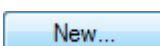


To edit the publish list to plot:

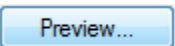
- Open the document.
- Start the  **Batch Plot** command.
- In the **Publish** dialogue box select necessary list from the drop-down **Publish lists** list.
- Edit list.
- Click the  **Save publish list** button.
- Click  button, to close the **Publish** dialogue box.

To rename the publish list to plot:

- Open the document.
- Start the  **Batch Plot** command.
- In the **Publish** dialogue box select necessary list from the drop-down **Publish lists** list.
- Edit list.
- Click the  **Save publish list as** button.
- Enter a new name of list.
- Click **OK** to save renamed list.
- Click  button, to close the **Publish** dialogue box.


Features of the batch plot from the model space

- Start the  **Page setup** command.
- In the opened **Page Setup Manager** dialogue box click the  button.
- In the **New Plot Set** dialogue box enter name of set, for example, *A4 Landscape*.
- Click **OK**.
- In the opened **Page Setup** dialogue box select printer, paper size – *A4* and orientation – *Landscape*, set scale and other plot parameters.
- Specify by **Window** first print area of A4 format, then second and all other area of A4 format (the  **Add window print area** button). To delete the wrong set of the last area, use the  **Delete last print area** button.

The  button allows you to see preview plot result of the selected areas. Preview can be used at any stage of set print areas.





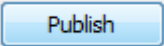
- Set the last print area of A4 format and to complete command, click **OK** in the **Page Setup** dialogue box.

Note: When you create a named set of the pages setup for the batch plot from the model space, it is necessary to include in the set all areas of the appropriate format that need to be printed.

- Click the  button to create the following parameters set of lists in a similar way (for example, *A3 Portrait*)
- Repeat these steps to create other sets of parameters (for example, to create a set of *A2 Portrait*, *A1 Landscape*, etc.).
- Specify the last parameters set and click the **Close** button in the **Page Setup Manager** dialogue box.

When you create all named sets of setup pages in model space, you can start the batch plot.

To run the batch plot of the created named sets of setup pages:

- Start the  **Batch Plot** command.
- In the **Publish** dialogue box, lick the  **New publish list** button to clear the current list.
- Click the  **Add page to plot** button.
- In the opened **Add page to plot** dialogue box:
 - in the drop-down **Document** list select document (if you plot multiple documents at once),
 - in the drop-down **Layout** list, select *Model* (displayed by default when you plot from model space),
 - in the drop-down **Page setup** list, select the *A4 Landscape* parameters set.
- Click **OK** to add selected named set into the layouts to plot list.
- Click the  **Add page to plot** button, to add next set (for example, *A3 Portrait*), etc.
- To start plot, click the  button.