User Guide nanoCAD Mechanica 20.1

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nanoCAD Mechanica

Intro

This manual describes work in nanoCAD Mechanica 20.1

Legend

Important!	Information on actions that may lead to work difficulties

Note:	Useful information on the topic

Example:	Link to the internal manual page or file

Link:	Link to a resource on the Internet

Video:	Link to the video instruction

Abbreviations

LBM - left button mouse.
RBM - right button mouse.

Main sections

Procedure - The section describes the user actions for constructing the object being described.
Edit dialog - This section describes the commands and parameters of the object editing dialog. As a rule, the editing dialog is invoked by double-clicking on the object.
Properties - the section describes the parameters on the functional panel "Properties", referring directly to the object being described.
Grips - the section describes the intelligent design handles of the object being described. Handles appear when selecting an object in the drawing.
Features of the team - the section describes special techniques for working with the called team.

Basic knowledge

About the program

nanoCAD Mechanica 20.1 – it is an integrating software solution for platforms nanoCAD, designed to speed up and simplify the design of components, the development, design and editing of design documentation, project data management, focused on the domestic user, made on the basis of regulatory recommendations.
nanoCAD Mechanica 20.1 provides everything for engineering design objects: design projections of drawings on ISO using algorithms automated control, unique design technology systems of hydropneumatic elements, gear parts, engineering analysis with mapping the result of the calculation on the model, the calculation of dimensional chains, gearing, shafts, creating custom libraries of intellectual objects and more. Powerful multivariate design tool unites planners of flat and three-dimensional design, users nanoCAD.
The tools nanoCAD Mechanica 20.1 complement the familiar environment nanoCAD, without disabling or reassigning its commands, toolbars, and menu items.
Technology work ***nanoCAD Mechanica 20.1*** is based on the use of object-dependent parts with assembly and parametric dependencies pre-installed on them.

Important!	Save drawings only in the format * .dwg. When resaving to other formats (for example * .dxf), objects of ***nanoCAD Mechanica*** will be permanently converted into proxy graphics and cannot be edited.

Copyrights and Licenses

No section of the documentation may be changed, adapted or translated into other languages without the prior written permission of the firm "Nanosoft". It is not permitted to create derivative documents based on material from this publication.
The nanoCAD Mechanicasoftware complex described in this manual is distributed in accordance with the terms set forth in the License Agreement and cannot be used, transferred or sold under any conditions other than those expressly specified in this agreement.
Licenses:
WpfTookit (c) Xceed Software Inc
Xbim Toolkit by Xbim

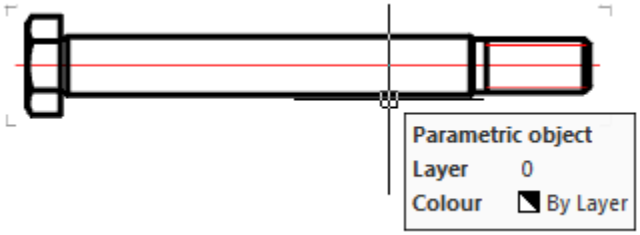
Compatibility

nanoCAD Mechanica20.1 platform support nanoCAD, as well as vertical solutions nanoCAD versions 2, 3, 4, 5, 6.
If the drawing is created ***nanoCAD Mechanica20.1***, it opens on a computer that does not have any one of the modules of the complex, the special objects nanoCAD Mechanica20.1 converted into so-called proxy-objects. To preserve the ability to edit them using "handles", as well as view and change the objects in the dialog box "Properties" on a computer, you must install a freeware distribution nanoCAD Mechanica20.1 Enabler, located on the CD with a licensed copy nanoCAD Mechanica20.1.
When transferring files to a user or customer drawings that do not have nanoCAD Mechanica20.1and not having the ability or desire to use nanoCAD Mechanica20.1 Enabler, you need to convert all the objects nanoCAD Mechanica20.1 in the standard system primitives nanoCAD.
To do this, use a special command ***"Break all objects"*** (the toolbar "Edit"). This command will automatically select all of the objects in the drawing, and divides them into primitives nanoCAD. For individual subdivision nanoCAD Mechanica20.1 - objects can use the standard command nanoCAD "EXPLODE".
In the dialog box ***nanoCAD Mechanica20.1 - Setting*** possible as a random location nanoCAD Mechanica20.1 - objects on any current layer and automatic fixing nanoCAD Mechanica20.1 - objects on said layer. In the second case, creating nanoCAD Mechanica20.1 - subject to pre-select the desired layer in the list and make it current is not required: the program will automatically execute it, regardless of the current layer.

Selected and allotted object

An object refers to an nanoCAD graphic item as well as an object created by the nanoCAD Mechanica 20.1.

Allotted object



An object is considered allotted (indicated) if additional graphic information appears on the screen when the mouse cursor is moved to it:

- the object is highlighted in color;
- the *"rectangle corners*" of the object appeared;
- the object information tooltip appeared.

Note: The mouse keys are not pressed in this case - you just hover the cursor over the object.

Scaling to the allotted object

- Move the mouse pointer over the nanoCAD Mechanica 20.1 object until *"rectangle corners"* appears, the object color changes, or an info tip appears;
- Hold down the SHIFT key and press RMB. It will auto scale to the center of the screen.

To scale to the selected object, the user coordinate system (UCS) must be set to * WORLD *.

Selected object

An object is considered selected after it has been clicked with LMB:

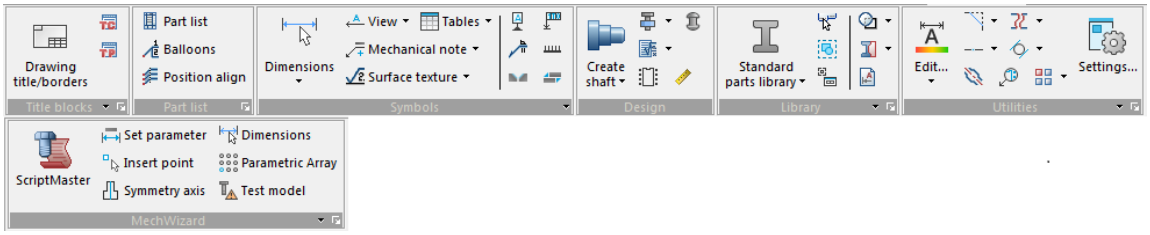
- the image becomes dotted or highlighted;
- the insertion point is displayed;
- *"grips"* are displayed.

Object context menus

To set additional options for the current command, right-click context menus are widely used when invoking commands. The content of the context menu depends on whether the editable object is selected or allotted.

Workspace

Menu



All tools nanoCAD Mechanica 20.1 located in the ribbon (in the main menu) tab *"Mechanica"* and *"MechWizard"*. Commands nanoCAD Mechanica 20.1 grouped by purpose and organized hierarchically. When you select a menu item marked with a black triangle, the corresponding drop-down menu is activated. This way you can access all the tools nanoCAD Mechanica 20.1.
Menu nanoCAD Mechanica 20.1 also serves for loading / unloading individual functional panels nanoCAD Mechanica 20.1, call dialogue "nanoCAD Mechanica 20.1 - Settings" and to access some common features nanoCAD Mechanica 20.1.

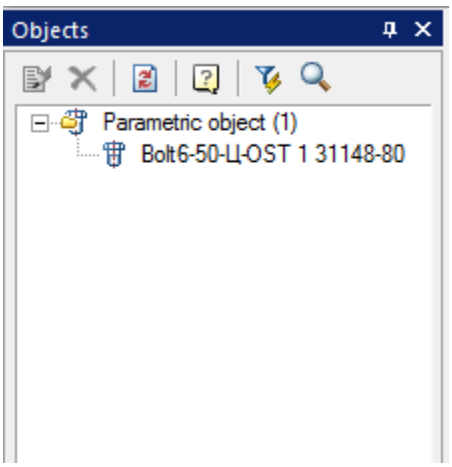
Toolbars

You can access the nanoCAD Mechanica 20.1 tools through the toolbars. Toolbars are divided by purpose. The user can place on the screen those panels that contain the most frequently used groups of tools.
There is also a universal toolbar "Mechanica Main" - instrumental two-level panel on which groups of tools are located for working with symbols and graphical constructions defined by standards nanoCAD Mechanica 20.1.

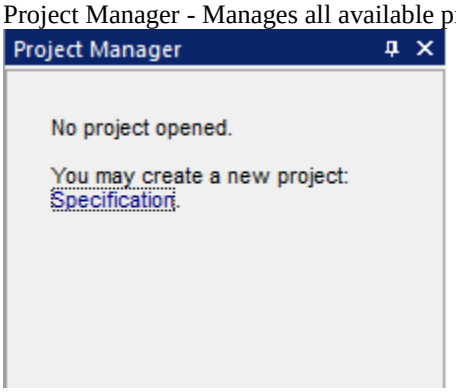
Tools are grouped by purpose (group of callouts, group of commands for setting level marks, etc.).
Click the LMB user to select the first in order tool of the group. If you click and hold the corresponding button on the panel with the mouse, a sub-panel opens, containing the rest of the tools in this group. Without releasing the LMB, you need to move the cursor to another command of this group, and then release the button. As a result, the selected command will be launched.

Functional panels

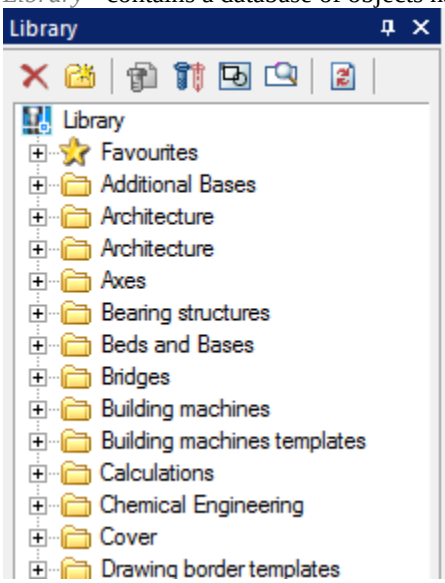
On the left side of the screen are added functional panels, such as:
Objects - displays all objects nanoCAD Mechanica 20.1 created in the drawing. Directly from this palette you can perform a series of actions with objects: edit, delete, hide and search for them.



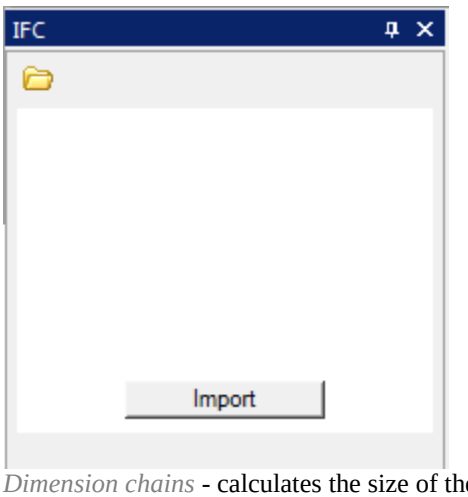
Project Manager - Manages all available projects nanoCAD Mechanica 20.1.



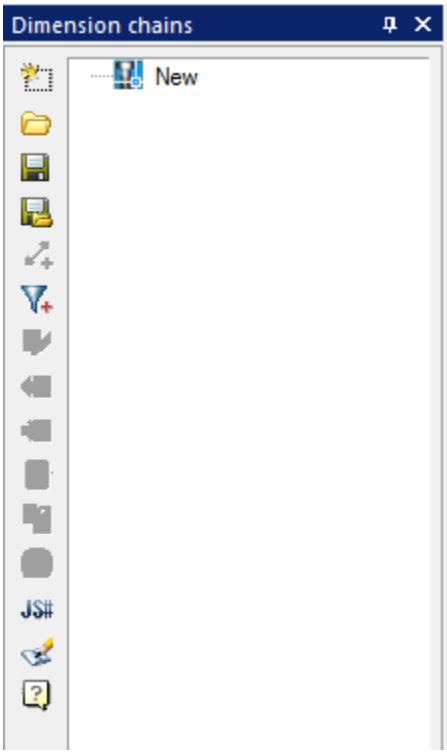
Library - contains a database of objects nanoCAD Mechanica 20.1 used for design.



IFC - works with IFC objects in the drawing.



Dimension chains - calculates the size of the chain.



Help system

Background information is available to the user at any time. The help system has two access modes - general and contextual.
General mode provides access to the help content in the absence of a command being executed.
Contextual mode automatically provides access to reference information that corresponds to the command being executed.

Scale

Option 1. Making projections in model space

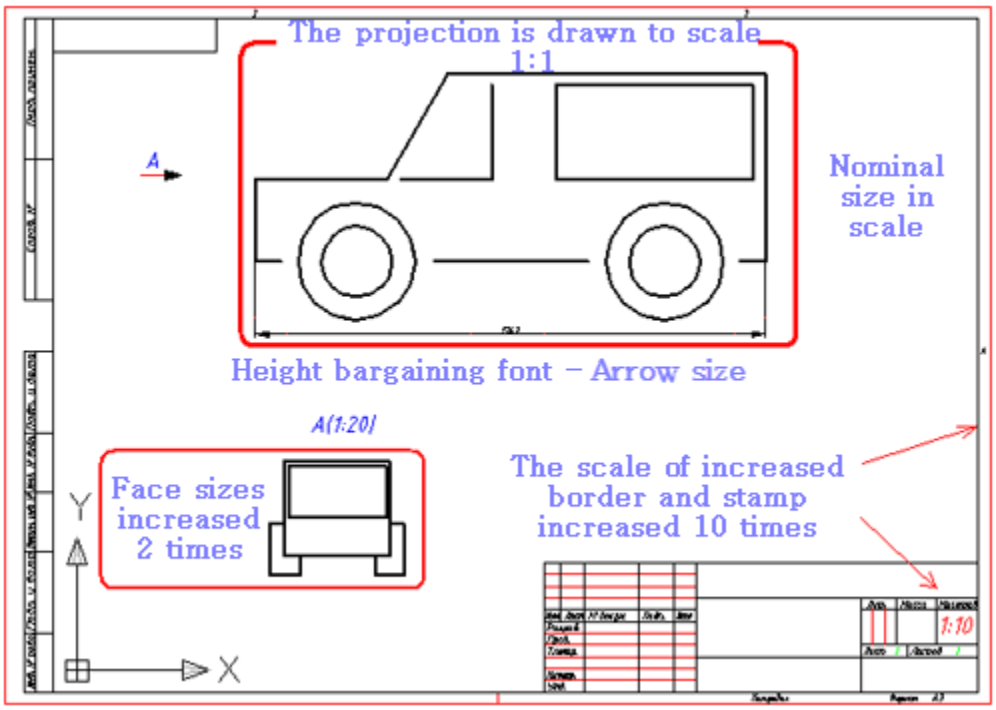
Suppose the principal species in the model space are performed one by one.
Additional types, sections are made with scale factors other than the main species.

Features insert frame

When placing a border with a title stamp in model space given its scale relative to the principal species. Frame increases or decreases.

Features design drawings

When dimensioning must take into account the scale factor drawing (the value of the dimension text, the value of the arrow, spacing, etc.) and numerical designations for each species.
Assume frame is increased with respect to the main sight 10 times. In this case the arrowheads, dimension text height and other text labels should be increased by 10 times. A nominal size of the text for the main species should be marked on a scale of 1:1.
When applied to the kinds of sizes, made in a different scale (in our example, 1:20), nominal size must be multiplied by a scaling factor.



Example of projection drawing frame, increased 10 times

Especially when printing

In this example, the sheet is printed at 1:10 in order to maintain a standard frame size, title block text and symbols. Such technology is not supported Construction Site 20.1.

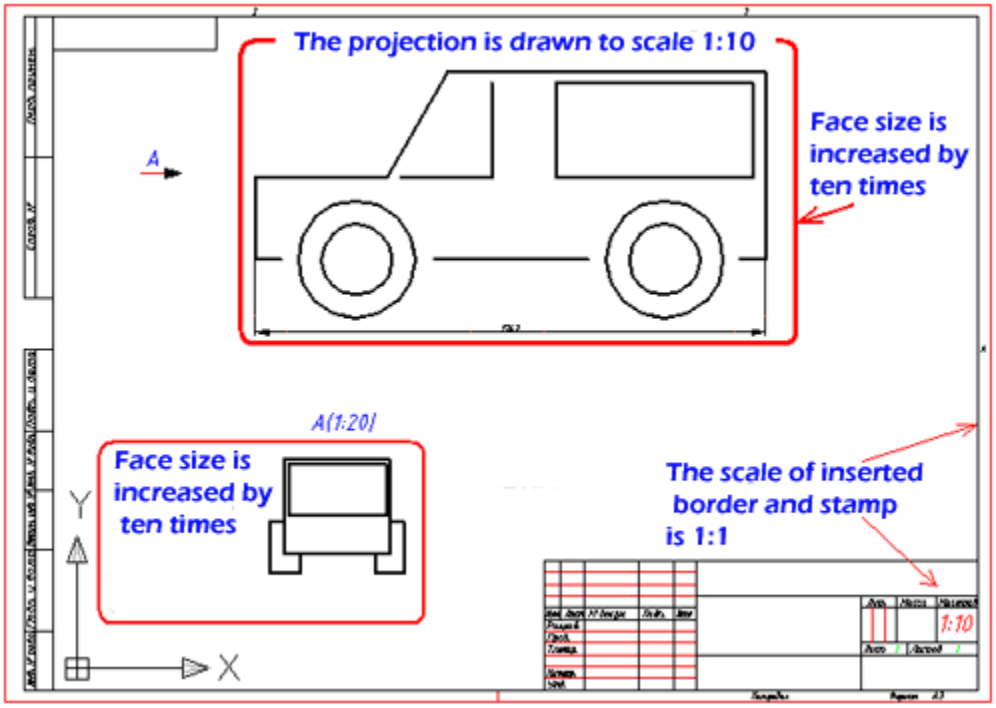
Option 2. Making projections in model space

Features insert formats

The frame is inserted into the space pattern in the scale of 1:1.
Main types, made a scale of one to one in the model space scale when placed on a frame with a title block.

Particulars of

The magnitude of arrowheads, dimension text height and other text refer to all types are specified in a scale of 1:1.
When dimensioning must take into account only the scale factor of the form in which they are applied.



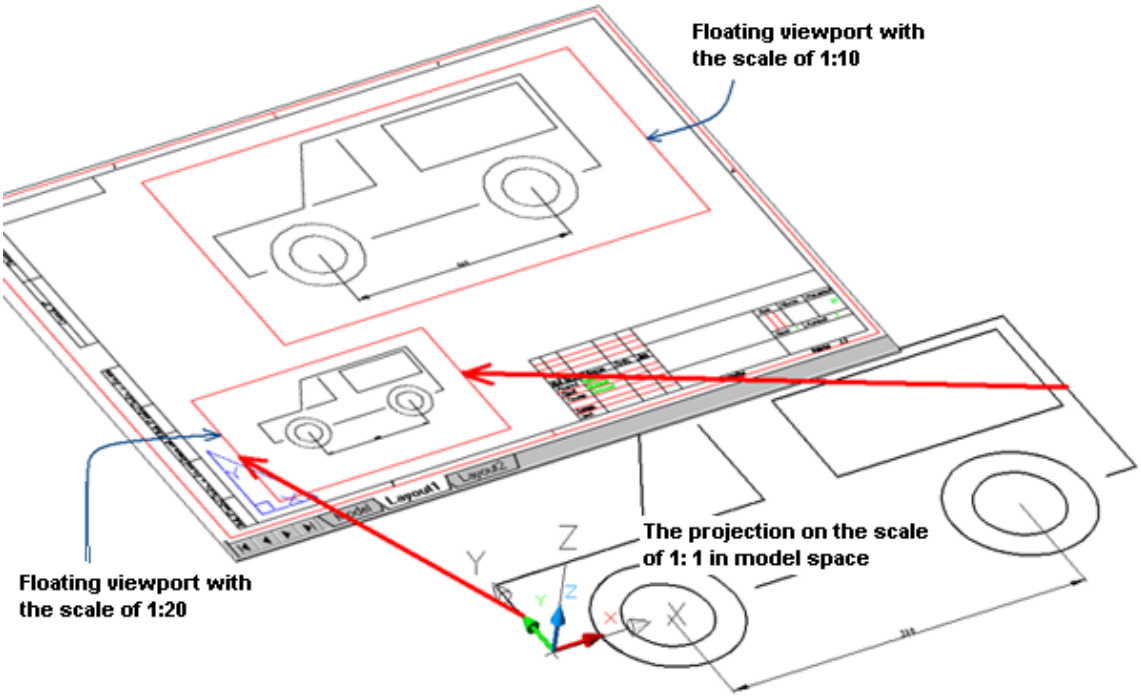
Example of drawing. Frame in a 1:1 projection in a given scale

Especially when printing

Under this option, registration in model space frame with projections printed at a scale of 1:1. This variant of design drawings in model space adopted in Construction Site 20.1.

Option 3. Making projections in paper space

The most convenient way to design drawings projections. All images of objects executed in one-to-one scale model space.
Border and title stamp inserted in 1:1 scale in paper space.
Scale for each species, cut or section is given in the floating viewport in paper space. No need to convert the nominal values of dimensional numbers. Floating viewport - a kind of lens with a predetermined magnification (scale) of the portion of the drawing.



Border and two floating viewports placed in paper space

Particulars of

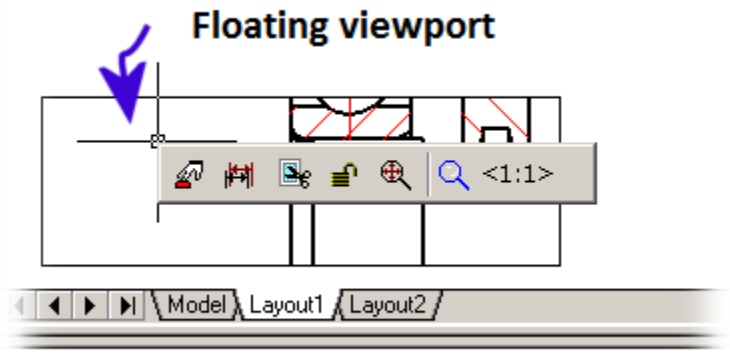
Scale factor when dimensioning for each species is taken into account automatically.

Especially when printing

Frame with projections printed in 1:1. scale in nanoCAD Mechanica 20.1 frame and title block inserted in model space or sheet at a scale of 1:1.

Paper space commands

When designing drawing projections in paper space, the context menu is available nanoCAD Mechanica 20.1. To call the menu, place the cursor in the area of the floating viewport and press RMB.



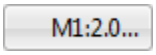
Activate viewport - command activates viewport for editing.

Refresh dimensions - command updates the dimensions in the viewport.

Cut viewport - command allows you to change the shape of the viewport.

Lock viewport - command fixes scales (including zoom commands) of viewport.

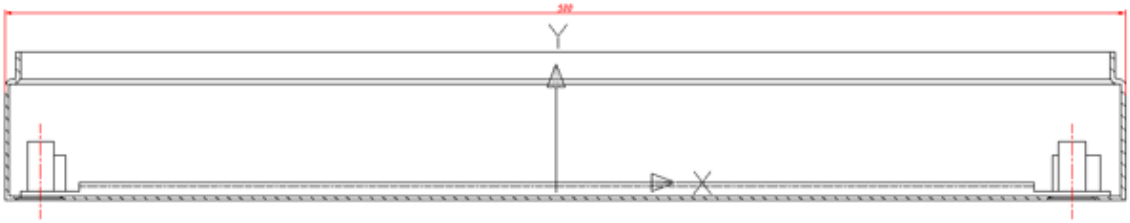
Zoom all - command inserts all the graphics from the model space into the viewport.



Scale - a drop-down spikok allows you to select a viewport scale.

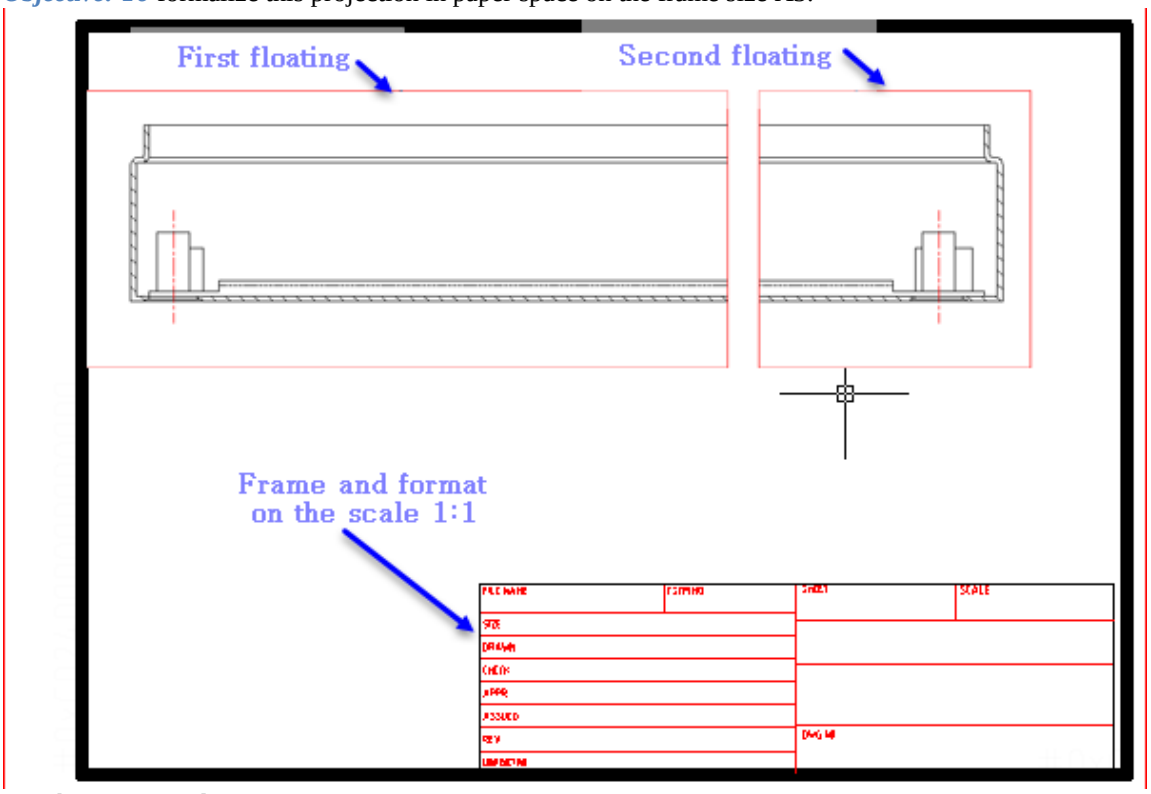
Example of design drawings of extended objects

Especially useful when you make a paper space drawings of extended objects. A small example. In model space, one to one (in real terms) has been drawn projection with dimensions 500 mm horizontally.



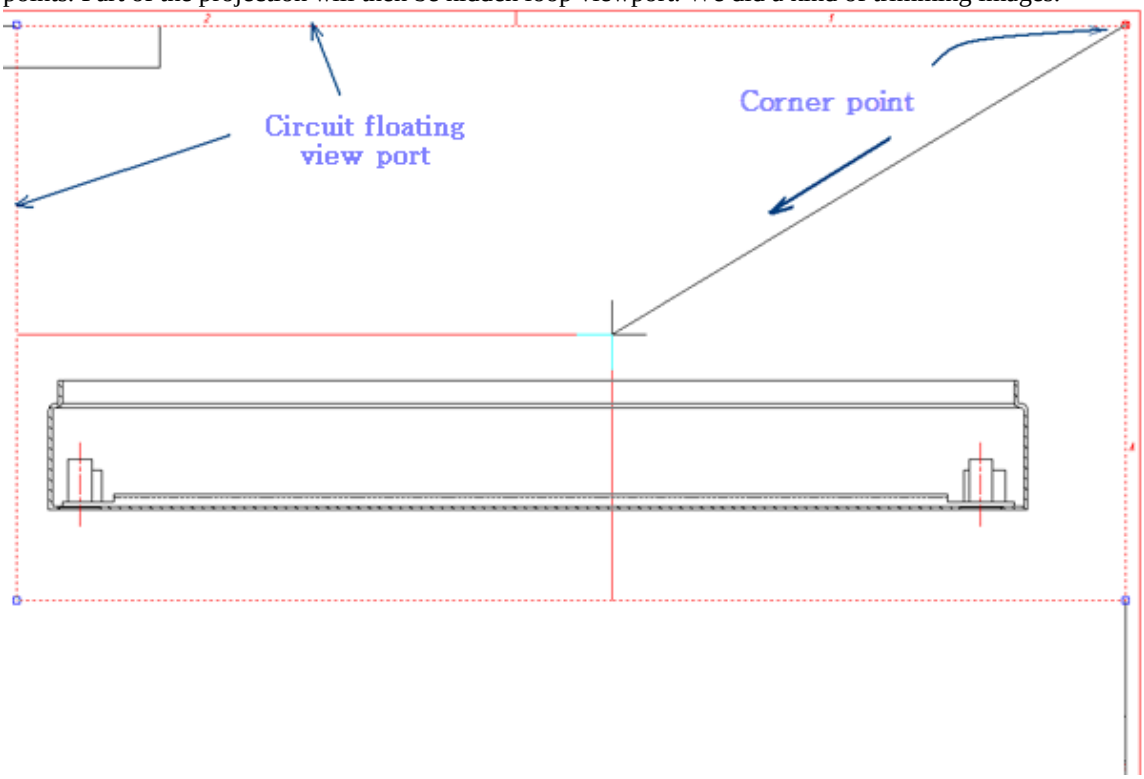
Projection with dimensions of 500 mm horizontally

Objective: To formalize this projection in paper space on the frame size A3.

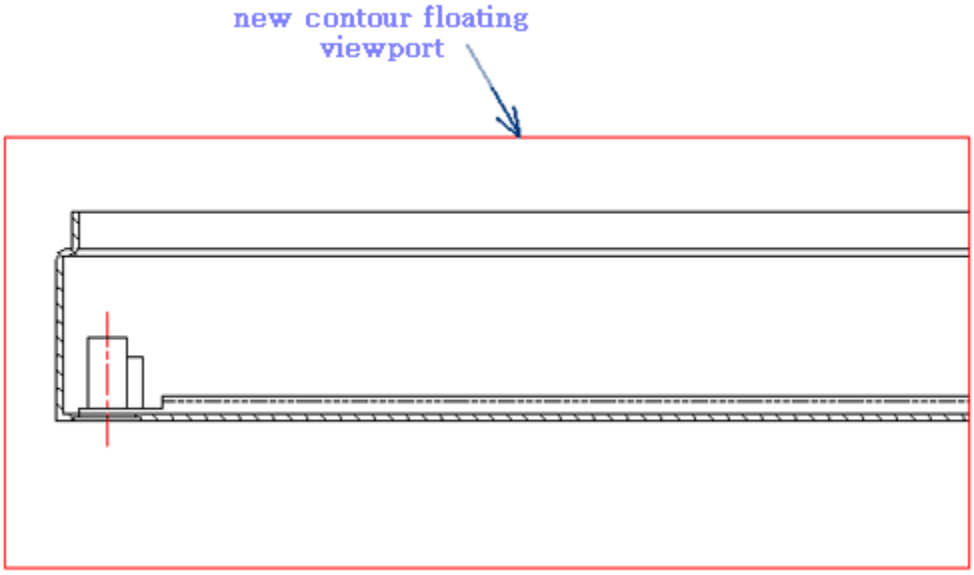


Result projection design in paper space

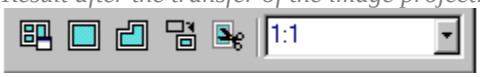
We turn to paper space by selecting a bookmark *Sheet1 (Layout1)*. Insert Frame A3 clicking the icon *format* in the main toolbar nanoCAD Mechanica 20.1.
After placing the frame in paper space is created by default floating viewport rectangular shape. Inside it displays all the objects in the model space. Zoom while arbitrary and requires further editing.
Resize a floating screen. To do this by clicking on its contour the left mouse button, move the highlight corner points. Part of the projection will then be hidden loop viewport. We did a kind of trimming images.



Move the corner points of the floating viewport

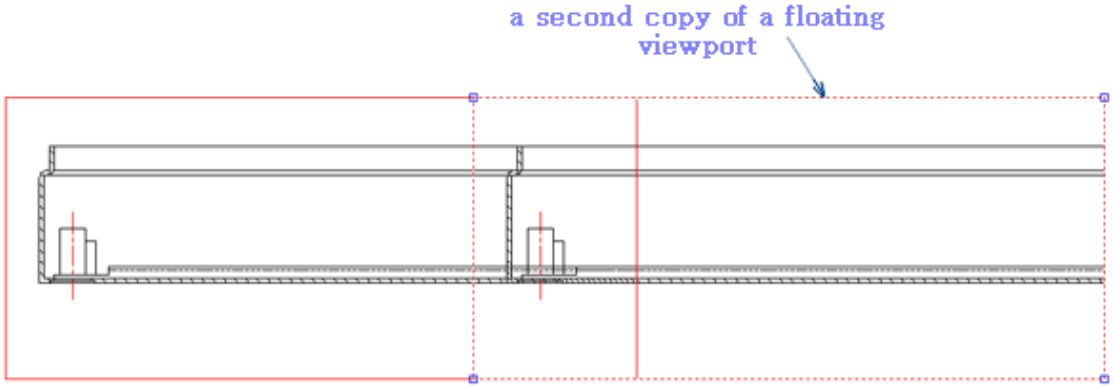


Result after the transfer of the image projection of the angular points of the contour floating viewport

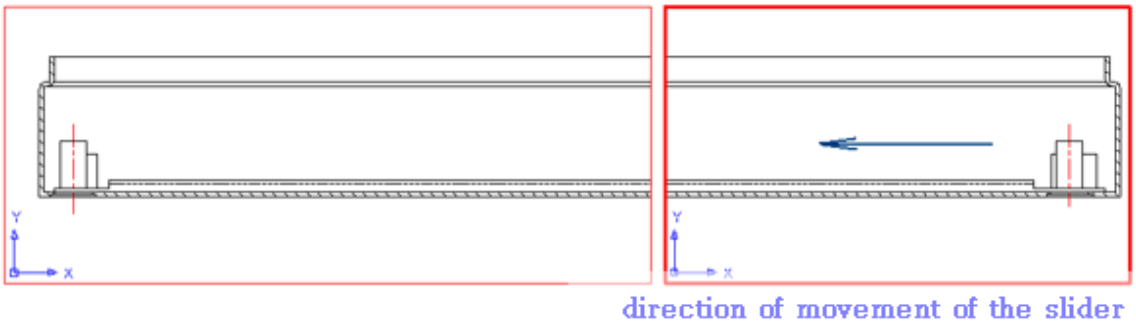


*Panel **Viewports (Viewports)***

Once the boundaries of the first viewport defined, define the scale of the image. Distinguish the border of the viewport. We define the scale of the image using the dashboard Viewports (Viewports) . To get the image on the right part of the projection make a copy of the floating screen and move it to the right.

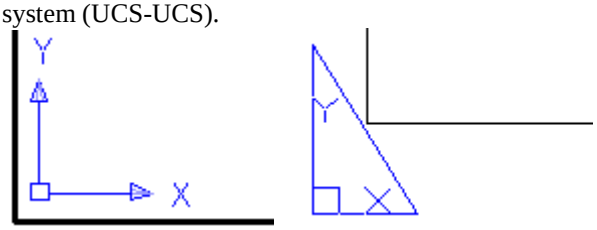


Move the copy right of the screen



Changed the boundary of the right floating viewport

With a team of pan define the right side projection view inside the right floating viewport. To do this, double-click the left mouse button within the contour of the viewport. Sign space layout changes to mark the user coordinate system (UCS-UCS).



Especially when printing

Before printing frame with the projections need to go out of the floating viewport to paper space. To do this, double-click the left mouse button outside the boundaries of the viewport. Sign appears in paper space.

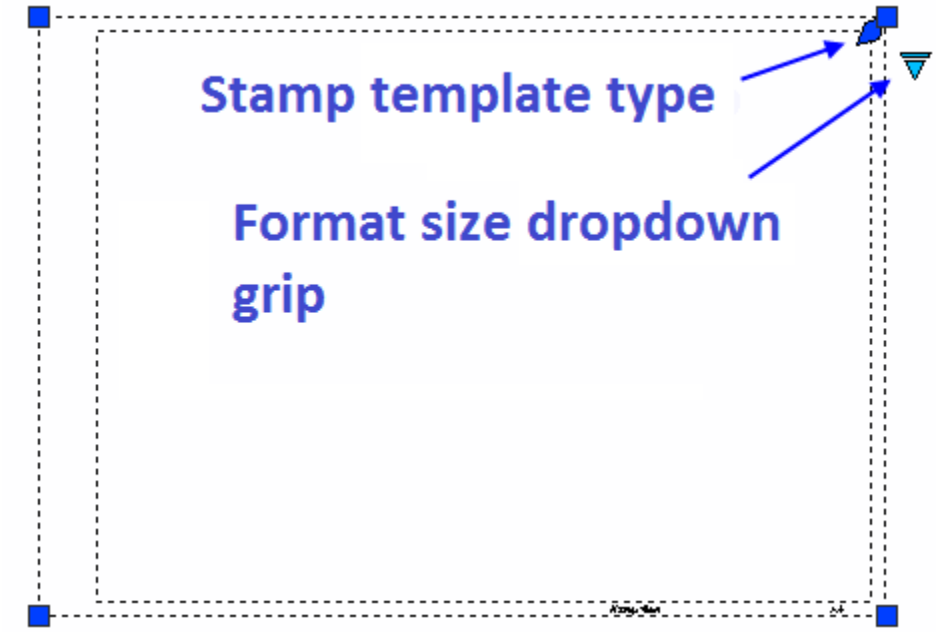
Controls features

Intelligent controls grips

Intelligent grips are intended for editing the properties of design elements inserted into the drawing without calling the editing dialog.

Intelligent grips are included in the settings nanoCAD Mechanica: *Main options - Edit - Enable enchanced grips*.

Descriptions of grip actions are found in the description of the corresponding design elements.



The appearance of the grip corresponds to its action (frequently used ones are shown):

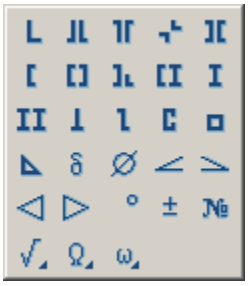
- Node move grip
- Graphics move grip
- Grip for adding a new node
- Grip for adding additional graphics
- Direction grip
- Tilt grip
- Text alignment grip
- Grip for selecting a value from a drop-down list

Special characters and fonts

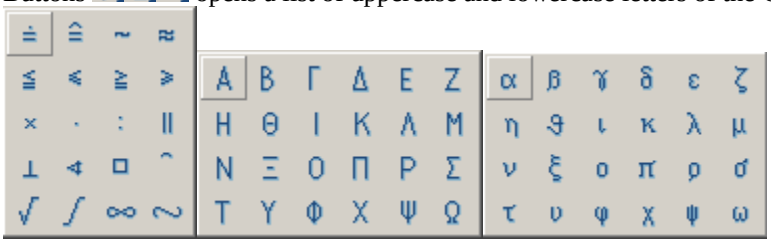
The dialogs remote printing and editing tables are buttons insert special characters by which a dialog box in which you presented the symbols used in architectural design.

For remote printing button *"Insert special character".*

To the Editor button table *"Insert Character".*
Select the character you want by clicking on it with the left mouse button, after which it is automatically inserted at the current cursor editing table.



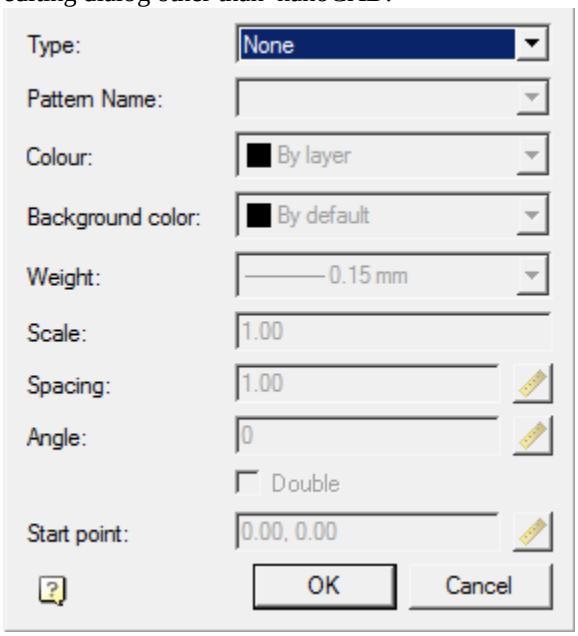
Buttons opens a list of uppercase and lowercase letters of the Greek alphabet.



Special characters are correctly displayed only text based on the fonts nanoCAD Mechanica (.shx). Displaying characters when pasting in text based on TrueType-font, the screen looks incorrect (replaced by question marks, boxes, etc., depending on the settings of TrueType-font).

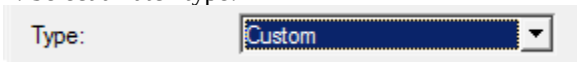
Parameters hatch

In the dialogs for editing nanoCAD Mechanica objects, there may be a *"Hatch"* button, which opens the *"Hatch"* editing dialog other than nanoCAD.



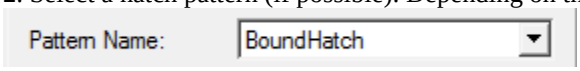
Procedure

1. Select a hatch type:

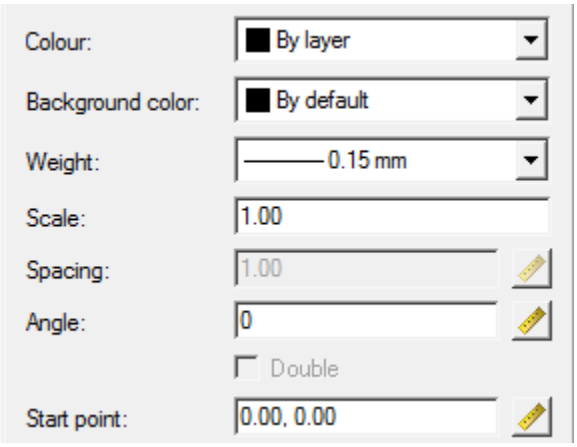


- *None* - hatch will not;
- *Predefined* - hatch is selected by abrasive from the list of standard hatches;
- *Custom* - hatch is selected by abrasive pattern from the list of custom hatches;
- *User-Defined* - hatching is formed in the form of parallel lines with a set interval and rotation angle. When you turn on the *"Double"* switch, the second row of lines will be drawn perpendicular to the first.

2. Select a hatch pattern (if possible). Depending on the selected sample, certain parameters will become available.



3. Configure the available options.



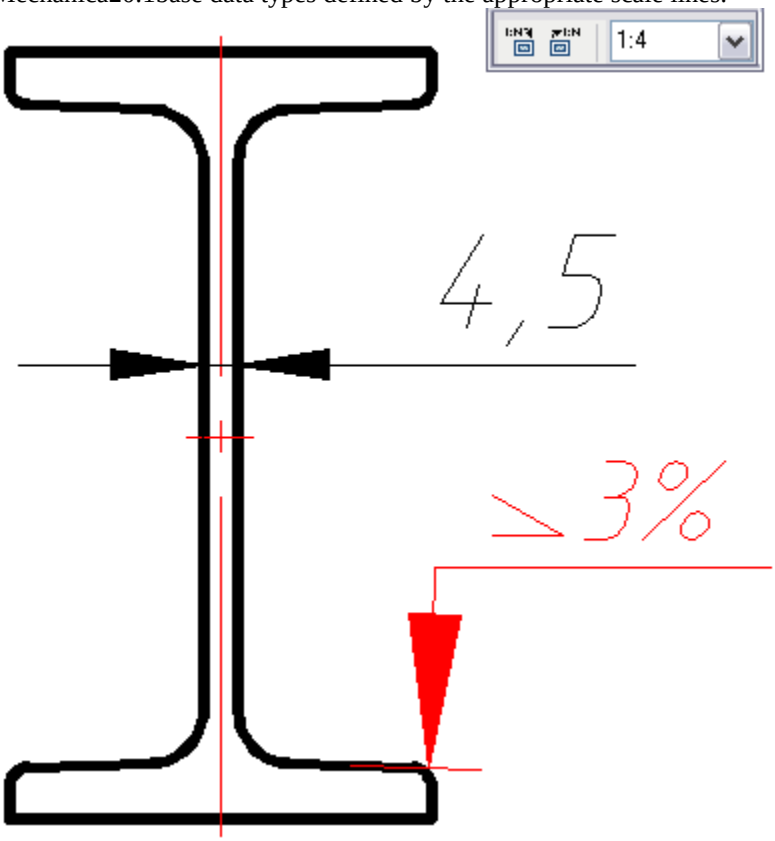
4. Click the *"OK"* button. The hatching will be adjusted and added to the appropriate location.

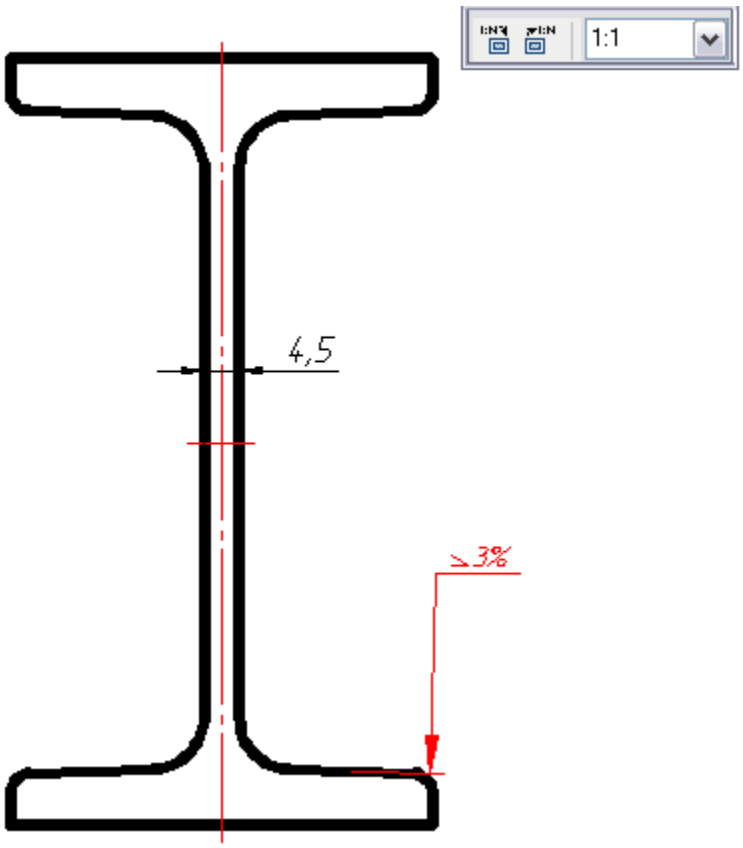
Jobs in the current scale

The nanoCAD Mechanica 20.1 uses the concept of the current scale. The current scale is different from the current global proportions, expressed in units (units).

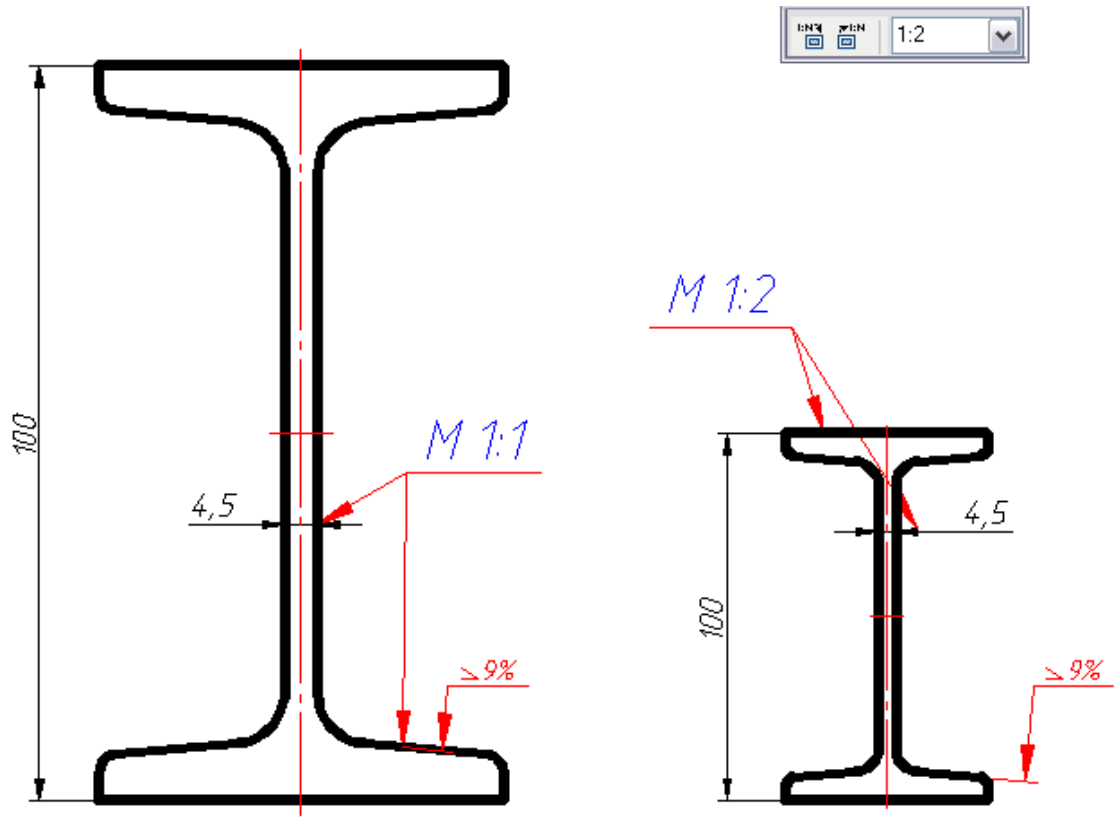
Scale objects made depending on the setting of "Use the global scope to design elements" (see. Customizing the appearance / General settings / Registration)

If the value of this setting - "Yes", the dimensions are scaled design elements and standard elements nanoCAD Mechanica20.1base data types defined by the appropriate scale lines.





This setting is useful when drawing in model space on the scale of 1: 1, followed by design drawings in paper space viewport on an appropriate scale. In this case, design elements will appear consistently in viewports with different scales.
If the "apply globally to design elements" is set to "no" to the size of standard parts varies in accordance with the scale and size of the design elements remain unchanged. In addition, there is a linear scaling sizes by setting a linear scale factor.
This method is used when plotting the initial scale model other than 1:1.

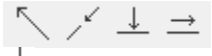


The scale of the database objects of standard products and design elements can be installed with the help of the scale.

In the absence of selected objects using the zoom panel zoom button or change the current zoom scale drawing.
If you have selected objects, the button and the panel scale display their scale. Zooming only affects the selected items without changing the current scale drawing.
The program features the ability to zoom the selected object nanoCAD Mechanica 20.1 center of the screen as a display frame around the object (the default frame is disabled). To do this, move the cursor to the object that appeared "Overall corners", hold down the SHIFT key, click on the right mouse button - will automatically increase the screen of the selected object.
If you press on an object at the same time the right mouse button and SHIFT key returns to the previous display with automatic centering of the object on the screen.

Selecting the direction of drawing

When drawing some graphics nanoCAD Mechanica20.1 may require special functions choice of direction of drawing. For example, when constructing Weld seam.
Auxiliary toolbar *"Destination"* is automatically activated when a call instruction nanoCAD Mechanica20.1, which is available for control of the direction of drawing.



Perpendicular. Construction perpendicular to the selected segment.

Free. The choice is arbitrary direction.

Along. Construction along the selected segment. The direction of the object is aligned to the segment, and will not indicate the direction of the opposite side.

Parallel. Parallel to the selected segment.
When constructing select the desired command and specify the length of the drawing, to be followed by leveling (except commands available).

Special keys

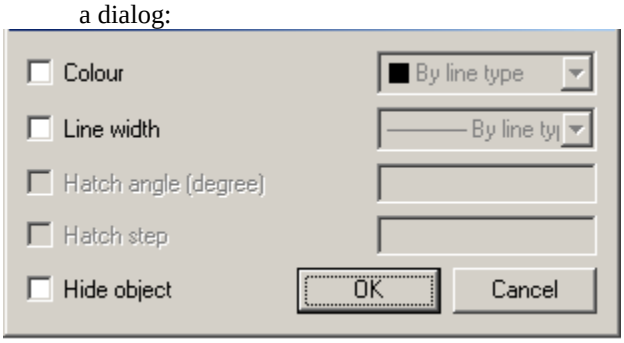
Key SHIFT

Key Shift used:

- When placing the dimension number. To accommodate the size of close to any external dimension line, move the mouse pointer, hold down the SHIFT key. To complete the team, click on the left mouse button.

Key Alt

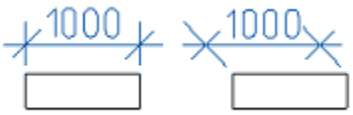
- To override ***entities included in the standard components*** a combination of Alt + right-click, which brings up a dialog:



Allowed to override the following properties: color, line thickness, slope and pitch hatch of hidden objects.

Key Ctrl

- Used to set the angle of the extension lines when placing dimensions.



- Used in the form editor to select the type of control when you drag a variable in a custom form.
- Editing a table at the place called the keyboard shortcut Ctrl + left click.

Shortcuts

Ctrl+5 - strikethrough text works in text input fields (in callouts, sizes, etc.), table cells, TT and TX.
Ctrl+U - text underline, works in input text fields (in callouts, sizes, etc.).
Ctrl+O - overline text, works in input text fields (in callouts, sizes, etc.).

Geometric constructions

Tools nanoCAD Mechanica20.1 for the geometric constructions are specific for mechanical design. All objects created with their help, made up of primitives nanoCAD, and does not require special tools for editing.
Editing inserted objects in a drawing can be performed using a universal object editor nanoCAD Mechanica20.1, called button Editing. After selecting the object being edited is automatically recognized by the type and loads the appropriate dialog box.
Furthermore, editing objects nanoCAD Mechanica20.1 it can be made by other means:

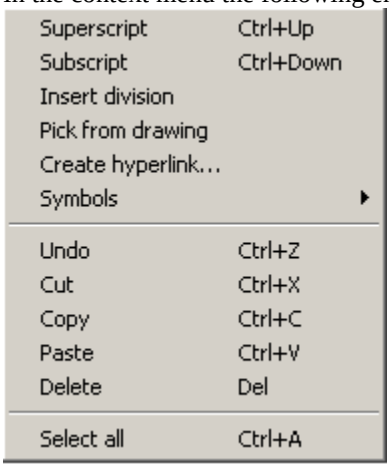
- clicking the right mouse button and then selecting the command "Edit in the context menu";
- double-clicking the left mouse button on the object nanoCAD Mechanica20.1. Adjust settings registration (Customizing the appearance / General settings / edit / double click);
- use grips;
- in window"Properties";
- standard command prompt ERASE, MOVE, COPY, ROTATE, STRETCH.

Object-dependent parts

Object-specific details - the details of this database nanoCAD Mechanica20.1 with the described properties of behavior when they are placed in the context of the assembly. With the change in the parameters of one such detail all the related object-specific details are automatically changed in accordance with their values in the database.
Using this technology works, it is important to take into account the order of application of object-dependent items.
When applying standardized object-dependent parts of the image cursor-pointer signals the automatic recognition of the insertion point and the direction of the applied rendering details.

Context menu entry fields

In the context menu the following entry is available.



History - recall last entry.
Recent - recall the most frequently used values.
Superscript - insert superscript (a combination of *Ctrl+Up*).
Subscript - insert a subscript (a combination of *Ctrl+ Down*).
Insert division - insert shot.
Pick from drawing - the command allows you to take data from the drawing and the properties of a standard part. When you select this command, the panel *Select the value* apperas :

Measure distance (Z) - to take drawing diamensionally linear or geometric size. Command can be selected by pressing *Z.*

Measure angle (X) - take the angular size of the drawing. Command can be selected by pressing *X.*

Measure Perimeter (C) - to take drawing the perimeter of a closed polygonal line or length. Command can be selected by pressing *C.*

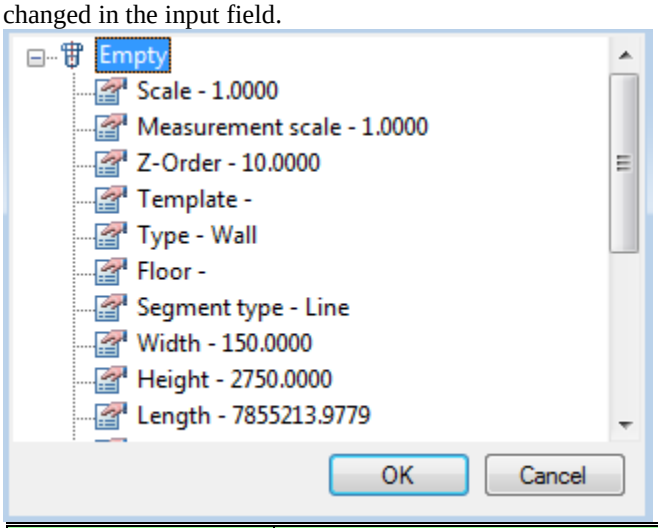
Measure area (V) - take the value of the drawing area of the closed loop. Command can be called by pressing *V.*

Complex area(Shift+V) - take a few drawing value areas closed loop.

Calculator (M) - Calculate the number with built-in calculator in nanoCAD Mechanica 20.1.

Take from notes (,) - insert text information from the notebook.

Take from property (B) - allows you to take command of the sub-parameters database in Construction Site 20.1. Parameter is inserted preserving the dynamic link with the object. When you change a parameter line items are changed in the input field.

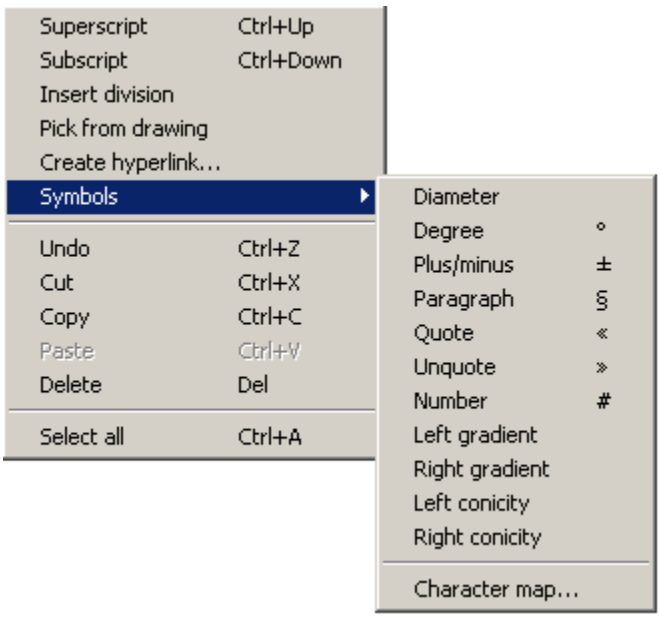


Note:	To insert static text, hold the Ctrl key while selecting parameter.

Mark text object (N) - to take the text from the drawing.

Cancel - stop the command and return to the Previous menu.

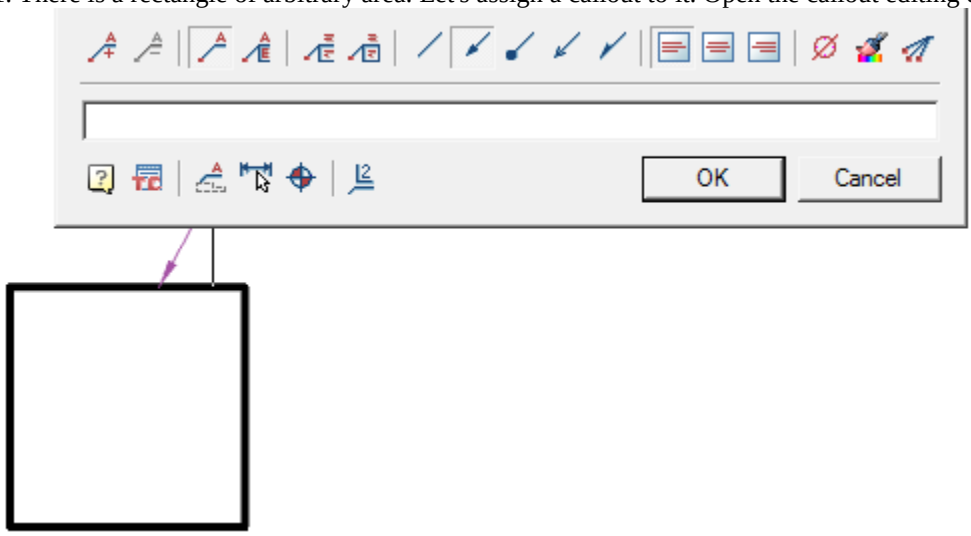
Symbols - inserting special characters



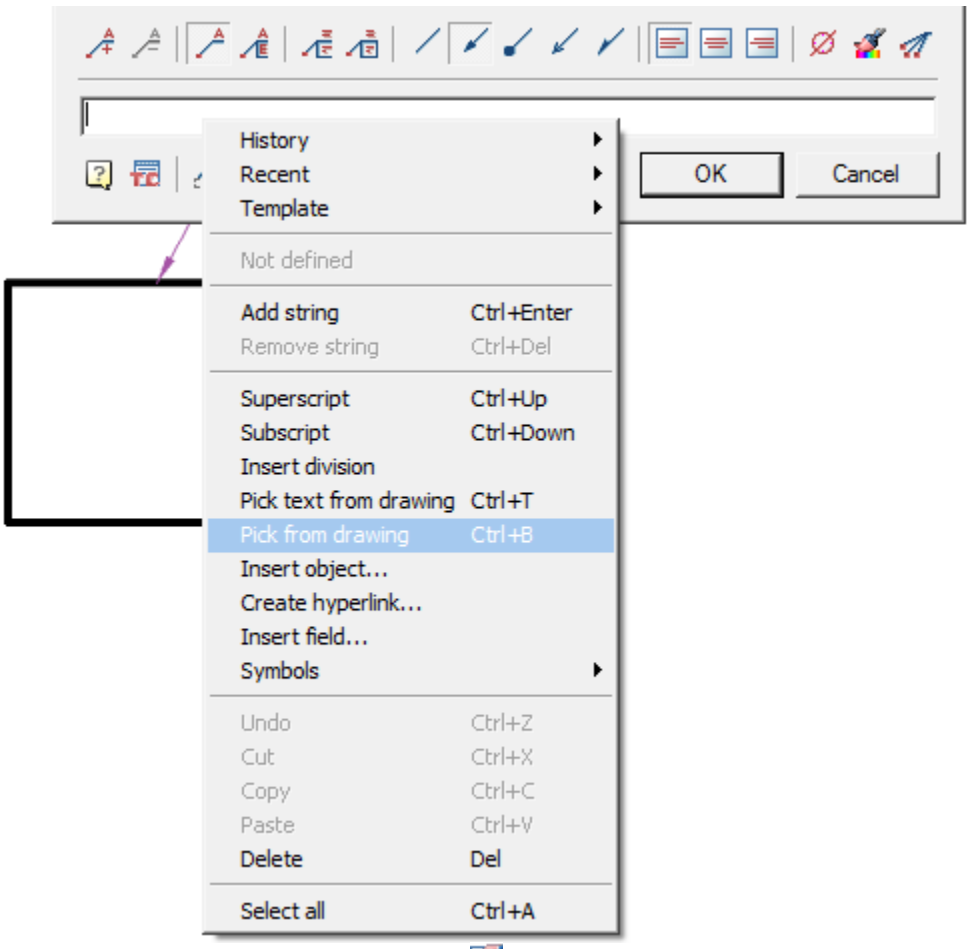
Undo - Undo the last command (the combination of *Ctrl+Z*).
Clipboard operations
Cut (a combination of *Ctrl+X*).
Copy (combination *Ctrl+C*).
Paste (a combination of *Ctrl+V*).
Delete - Delete the entry field.
Select all - highlight the input field.

Template

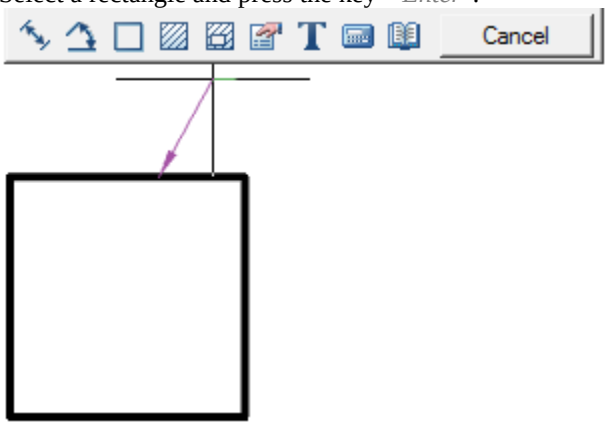
Templates are used to create objects of the same type with the same properties.
An example of creating and applying a template based on callouts.
1. There is a rectangle of arbitrary area. Let's assign a callout to it. Open the callout editing dialog.



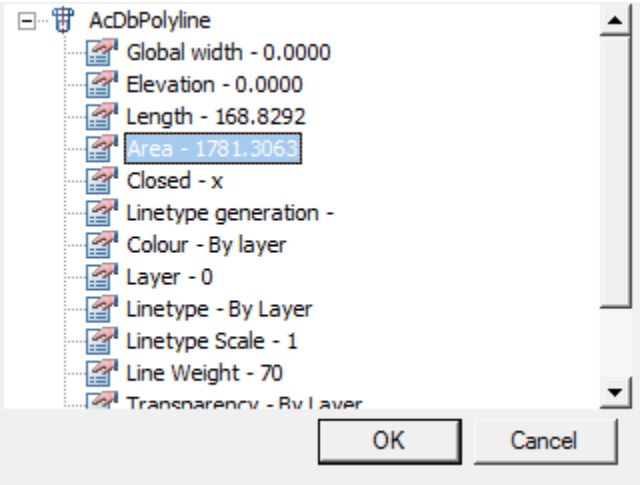
2. From the entry field callouts call the context menu and select the command "Pick from drawing." The "Value picker" dialog opens.



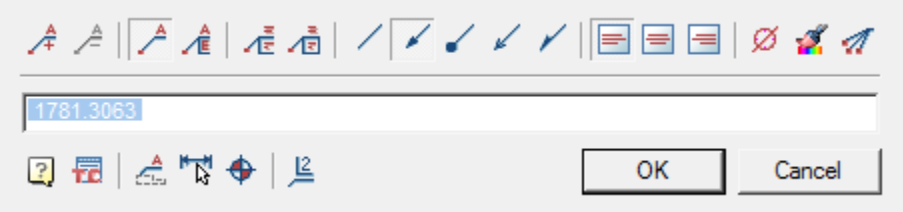
3. In dialogue *"Value picker"* call command *"Take from property*.
4. Select a rectangle and press the key *"Enter"*.



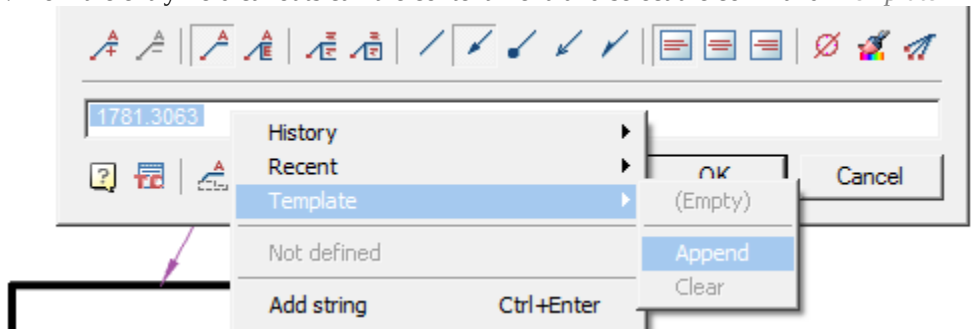
5. In dialogue *"Properties"* selecting *"Area"* and press *"OK"*.



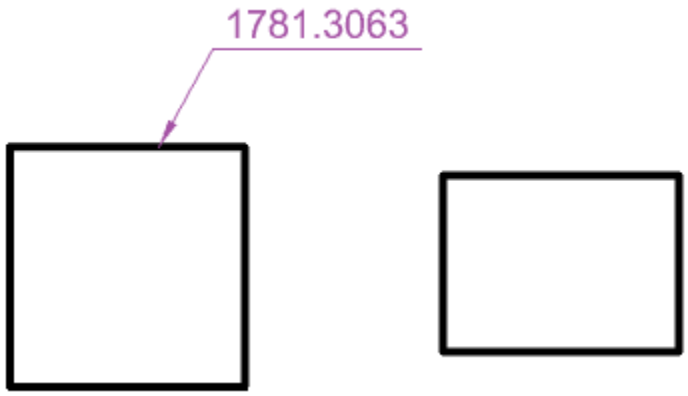
6. The link to the property appeared in the text of the callout, in this case - the area of the figure.



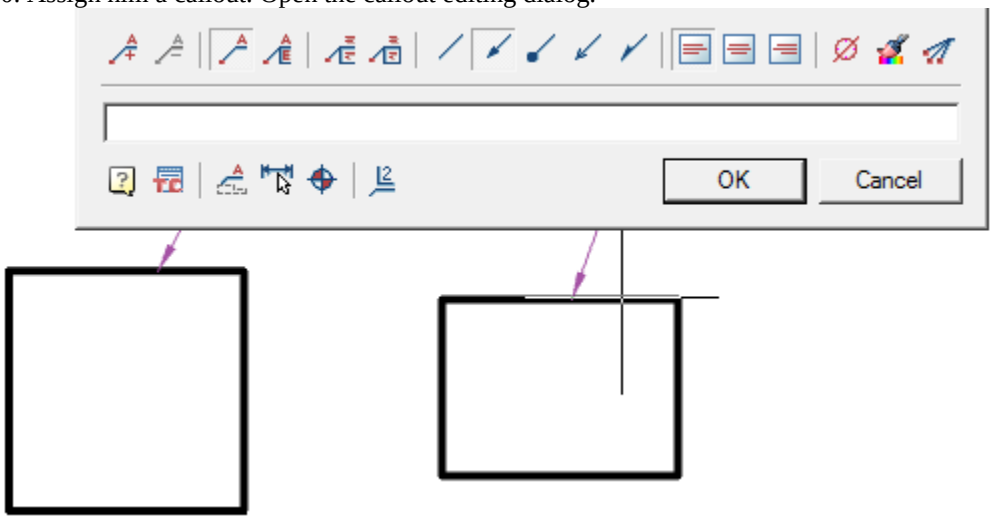
7. From the entry field callouts call the context menu and select the command *"Template - Append"*.



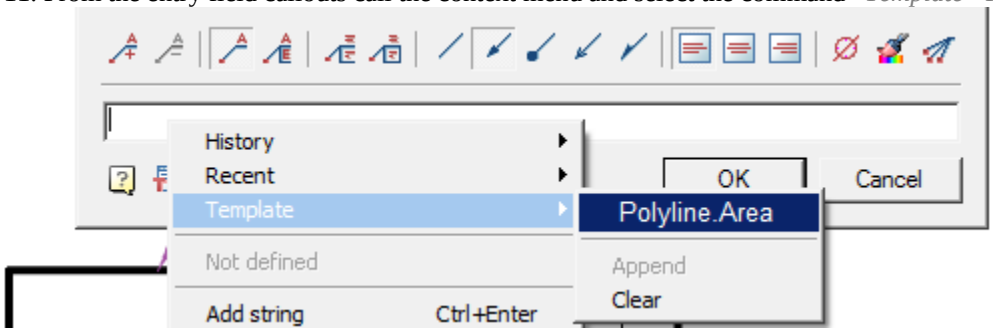
8. Close the callout editing dialog.
9. We build another rectangle, different in size from the first.



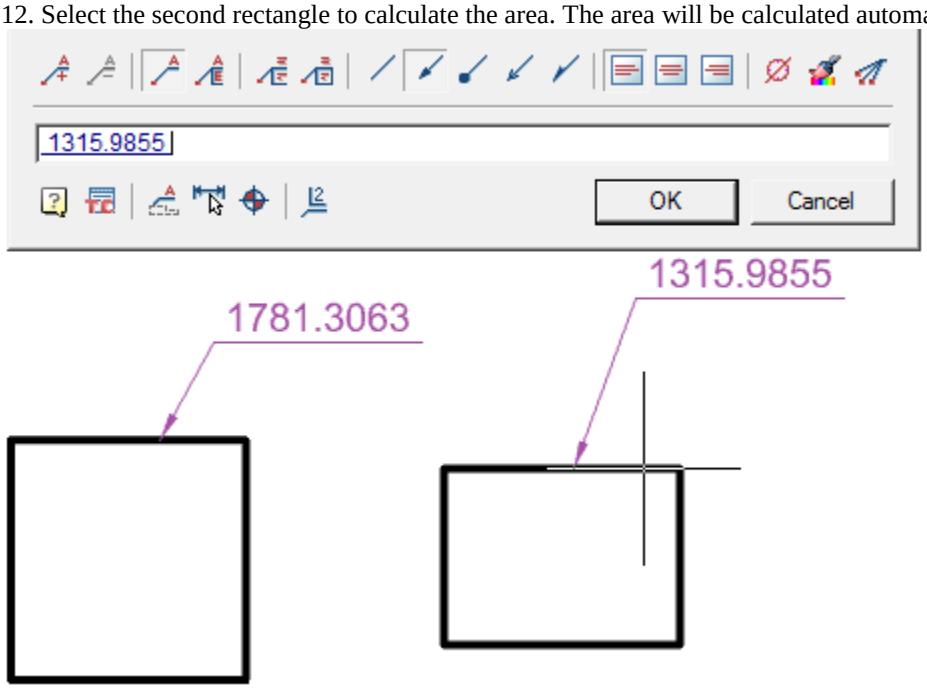
10. Assign him a callout. Open the callout editing dialog.



11. From the entry field callouts call the context menu and select the command *"Template - Polyline.Scale"*.



12. Select the second rectangle to calculate the area. The area will be calculated automatically.



Such templates can be used not only for callouts, but also for other elements, for example:

- position designation;
- node designation;
- note level field;
- e.t.c.

Regen

Command line: ***MCREGEN, RE, REGEN***.

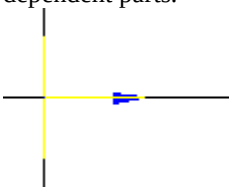
- Update objects nanoCAD Mechanica
- It recalculates table
- Updates overlapping objects
- Updates automatically calculates the attribute values

The cursor-pointer

Depending on the graphics platform (AutoCAD or Autodesk Inventor) and executes commands mouse pointer takes different forms.

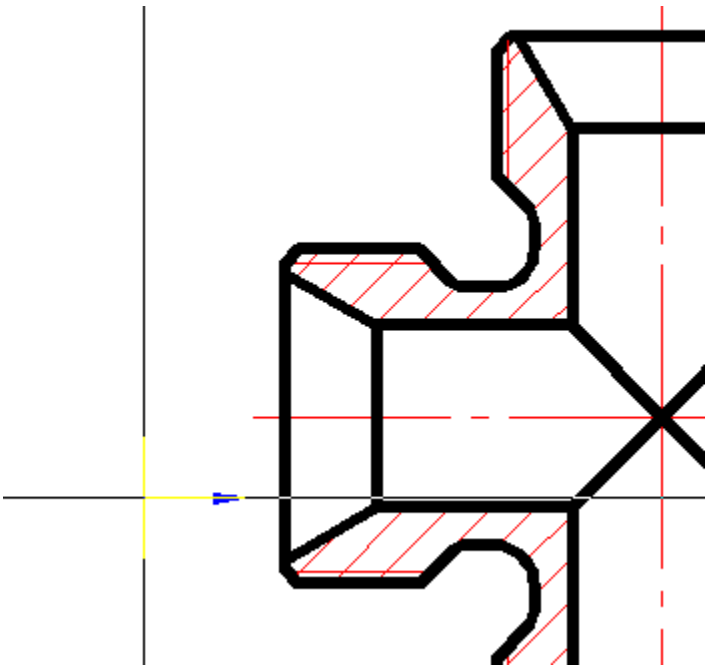
The application of a single object

Under a single object we understand insert parts from a database without automatic reference to other object-dependent parts.

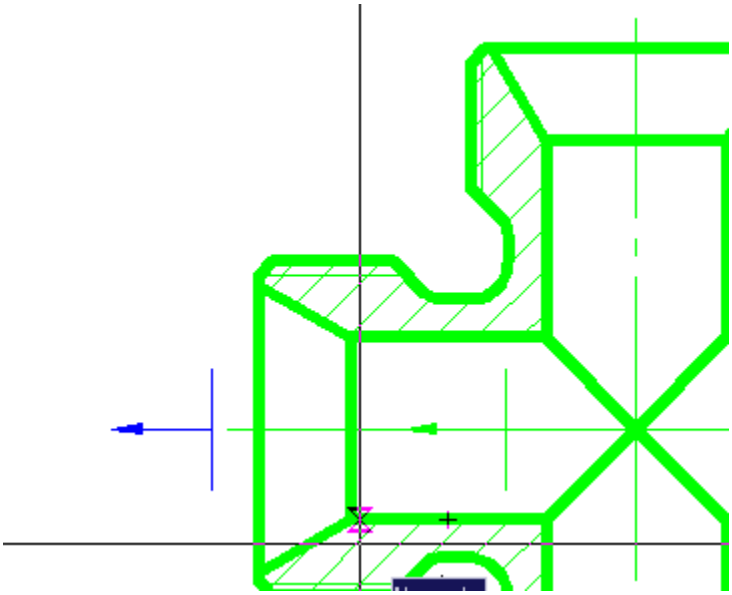


In the nanoCAD mouse pointer is displayed as a standard cross-hair, blue arrows and if there is a preview of the items - the item itself displayed.
Blue arrow indicates the insertion point and the direction of the applied drawing object.
After you specify the insertion point of the applied object of the cursor, the mouse pointer changes, the user is prompted to set the direction of drawing an object, it will coincide with the direction of the blue arrow. To confirm the direction you need to click the left mouse button.

Recognition of object-dependent parts



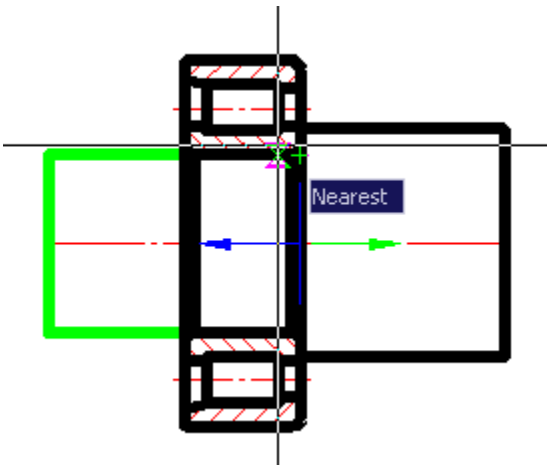
If you insert a part in an assembly before the cursor passes over the inserted workpiece, with inserted part can be installed according to the parts over which the cursor is, the original item is highlighted, and the position is determined automatically inserted details. To confirm the installation of dependencies inserted details, just click the left mouse button.



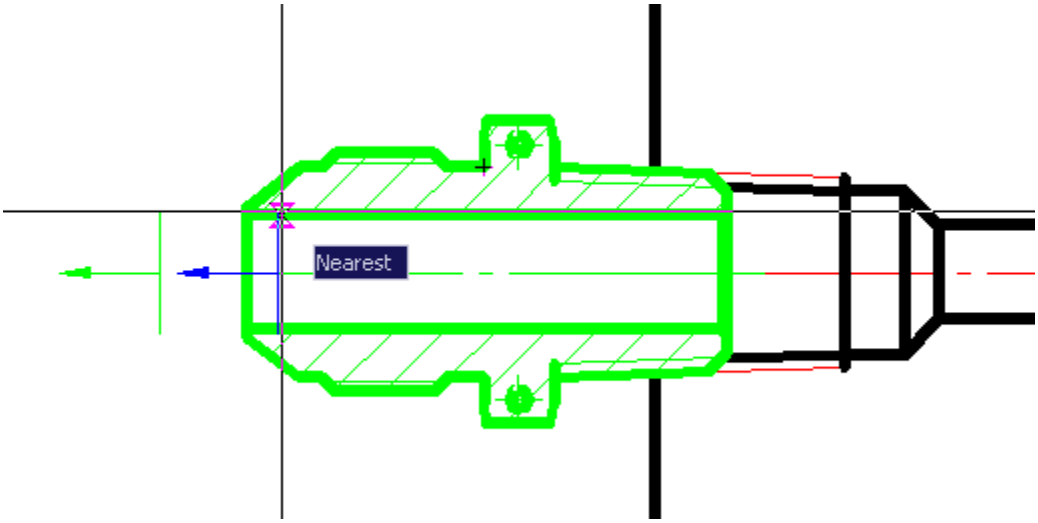
After this, a dialog box appears with parameters inserted parts.

Application of the bearing on the shaft neck

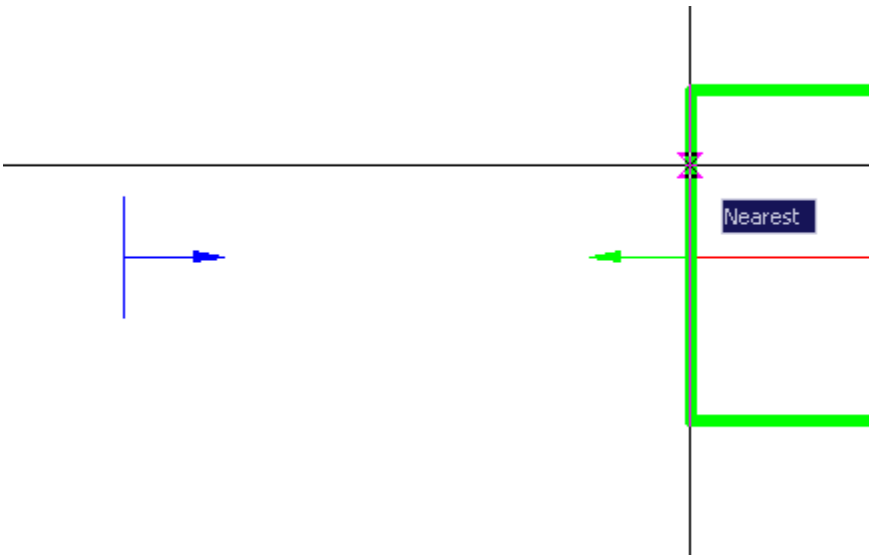
The cursor-pointer indicates that as the insertion point of the bearing end of the bead is recognized and given the direction of its rendering (left).



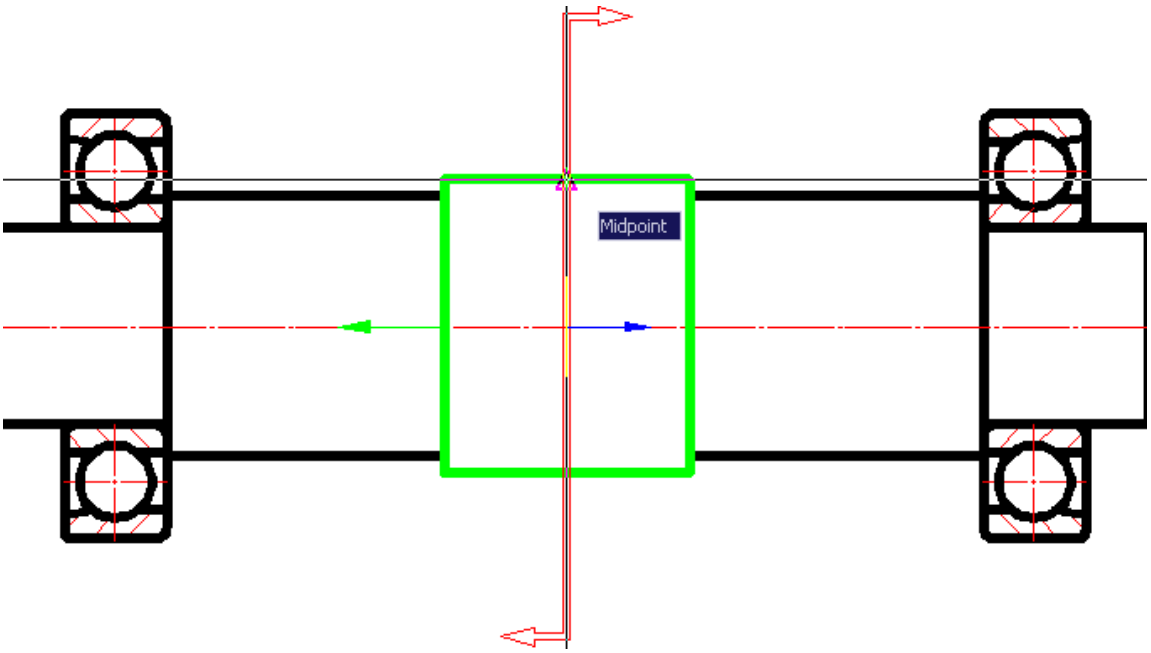
Recognition of the insertion point in the tapered portion of the pipe bushings



Application of standard shaft end



Drawing sign of torque



Settings

Settings

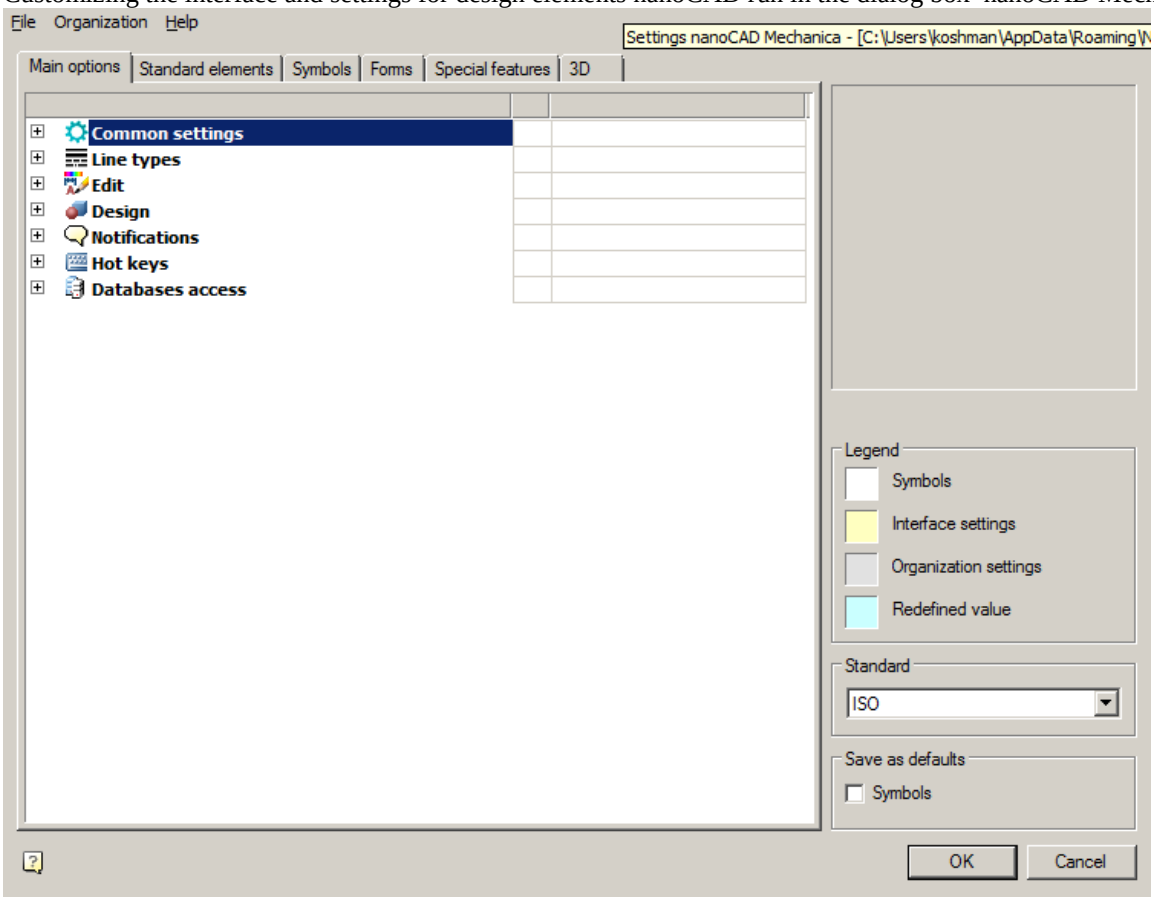
Main menu: *Tools -* *Design Settings...*.

Main menu: *Mechanical -* *Settings...*.

Toolbar: ***Settings... (*** "Edit"***)***.

Command line: ***MCPARAMS, PARAMS***.

Customizing the interface and settings for design elements nanoCAD run in the dialog box nanoCAD Mechanica:



The header of the dialog displays the path to the settings file.

Main menu

Menu "File"

Menu *"File"* contains controls:
Save settings - saves changes to the current settings file.
Save settings as... - saves changes in settings to a new settings file.
Load settings - loads settings from the specified file.
Types of uploaded files:

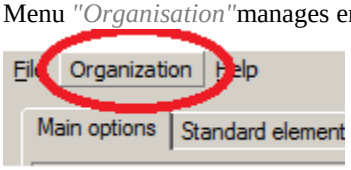
- Configuration files (*.xml)
- Skin Settings (*.cfg)
- Interface Settings (*.icf)

Configuration files *.cfg and *.icf used in nanoCAD Mechanica lower version 5 and contain settings that are currently stored in AppOptions.xml.
Restore start settings - loads settings preinstalled in nanoCAD Mechanica.

Important!	The function of saving the settings, when the saved file is the same as the one already existing, does not overwrite the old file, but complements it. When changing the standard, for example, from the ESKD to ISO, both standards will be available in the settings file.

Menu "Organisation"

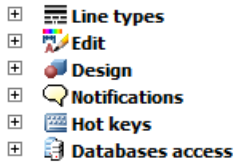
Menu *"Organisation"*manages enterprise standard settings.



Enterprise Standard - a single configuration file (parameters, layers, profiles) for the enterprise. All settings are stored in one file.
In the settings dialog, the profiles table and the *"Organisation"* settings layers are highlighted in light gray.
Menu *"Organisation"* contains controls:

- *Create organisation settings* - command allows you to create a corporate settings file for subsequent transfer to other machines or for location on a shared server.

Creating the settings file:
1. After the command is called, a selection field appears next to each parameter and section.



2. You must specify (check) which settings will be included in organisation settings. Then click the *"OK"* button.
The dialog box prompts you to enter the location and name of the corporate settings saving.
3. Indicate the path, name and confirm. A file with corporate settings will be created.

- *Set settings file...* - the assigned organisation settings settings file overrides the custom settings file AppOptions.xml. New documents will be created in accordance with the values in organisation settings settings file.
- *Flush* - command to refuse to use the organisation settings. In this case, the control file becomes the user settings file AppOptions.xml.
- *Apply organisation settings to document* - the command applies corporate settings to files created earlier or in another place that do not have these settings.

Important!	The user can not delete layers and profiles of organization settings from the corresponding tables in the settings dialog. When saving settings with new values, they are saved only in the current document. The configuration settings in the organization settings file and the standards within the configuration must strictly correspond to the downloaded settings in the application. If there is a mismatch, the override of the settings will not work!

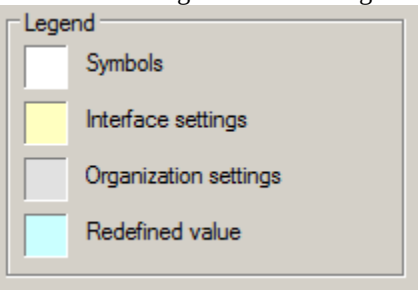
Work area

Contains:
1. Tabs with settings settings:

- Main options
- Standard elements
- Symbols

- Forms
- Special settings

2. Graphic window for displaying custom parameters (not editable).
3. Container "Legend" with the legend of the kind of settings (not editable).

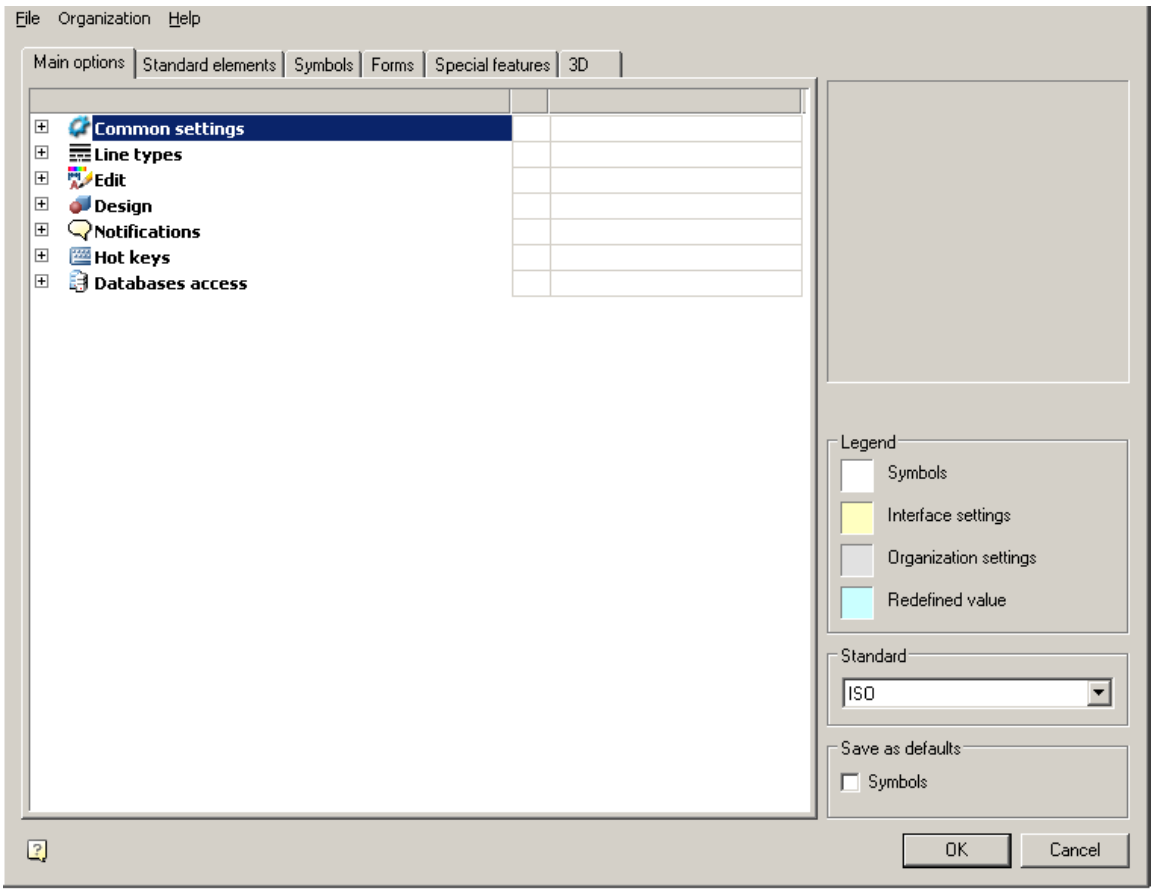


- *Symbols* - parameters with this background refer to the design parameters. In the nanoCAD Mechanica low version 5 settings are stored in the format files *.cfg
- Interface settings - parameters with this background refer to the interface parameters. In the nanoCAD Mechanica low version 5 settings are stored in the format files *.icf
- Organisation settings - parameters with this background indicate that they were downloaded from the Organization settings.
- Redefined value- parameters with this background indicate that they have been changed by the user.

4. The *"Standard"* container is intended for displaying the default development system of the design documentation (the choice is possible if there are more than one standards).
5. The *"Save as defaults"* container contains configuration items that indicate what will be saved by default when you click the *"OK"* button. Contains the selection field *"Symbols"*.

Main options

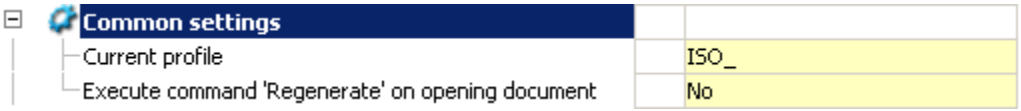
The tab is used to configure the general parameters of the interface nanoCAD Mechanica.



It contains sections:

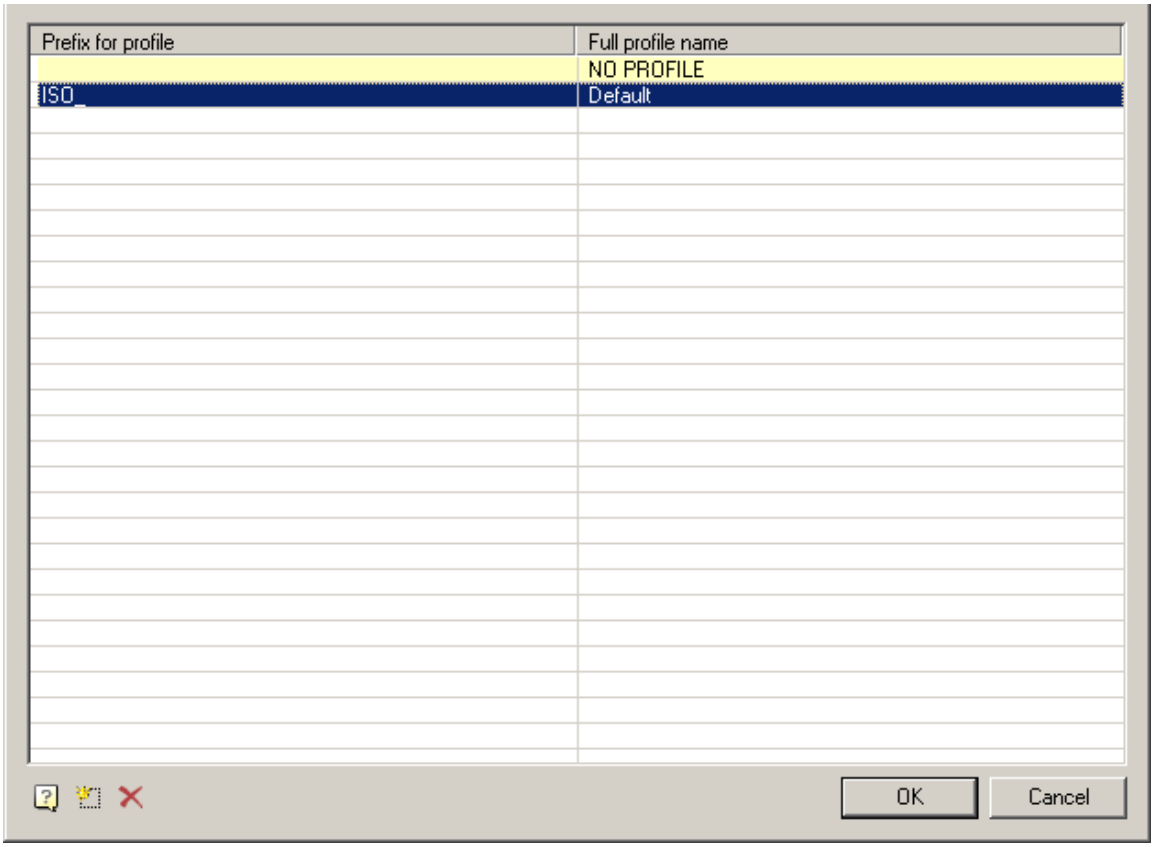
- Common settings
- Line types
- Edit
- Design
- Notifications
- Hot keys
- Database access

Common settings

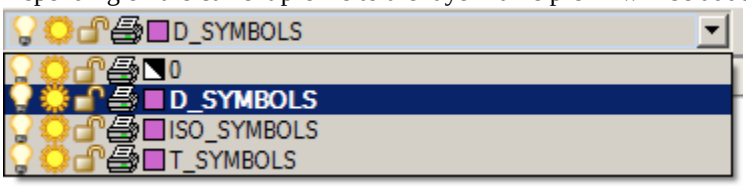


Current profile

Layer profiles are designed to organize the work of different departments of the enterprise over a single drawing file. Each user at the same time works with their group of layers, controlling their visibility by means *nanoCAD*.



Since the design of the drawing depends on the settings of the deployment of registration of the layers and the current profile of the drawing, then for each object type you want to specify in the settings option to be placed on the appropriate layer (for example, callouts given layer "Leaders")
Depending on the current profile to the layer name prefix will be added.

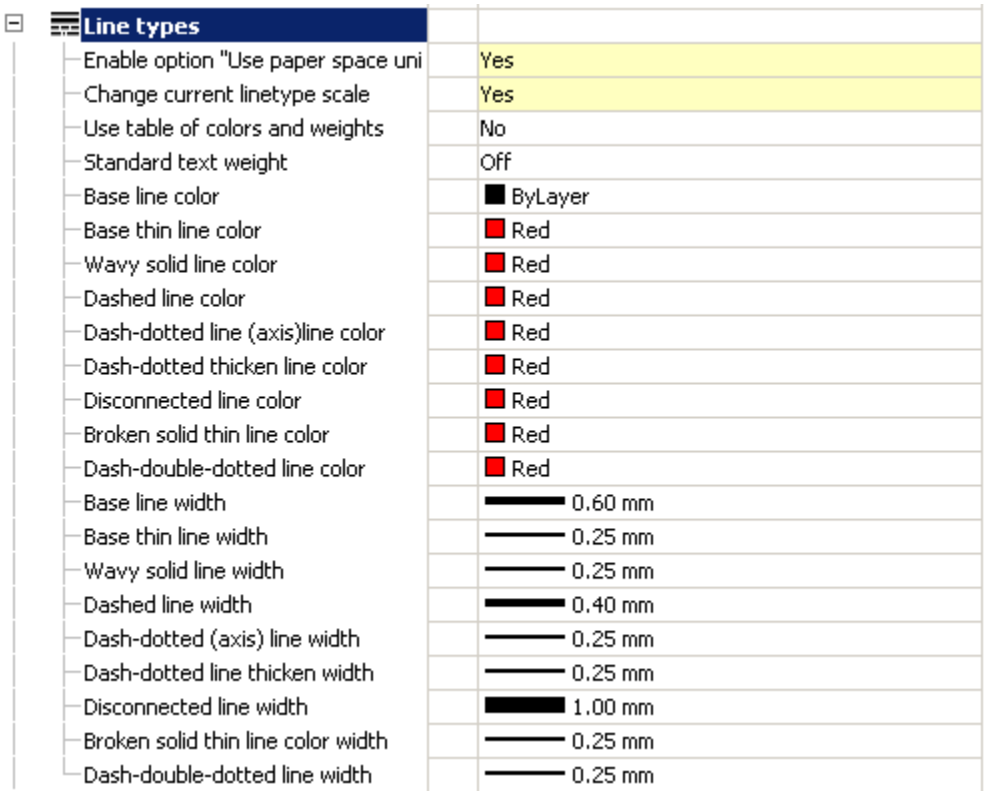


Thus, it is possible to group layers created by users with similar profiles (for example, the objects created by the designer - prefix "D_", and technologists - prefix "T_").
When adapting the distribution in the setup.ini file, you can assign a prefix in the parameter by default APP_OPTIONS_PROFILE.

Execute command "Regenerate" on opening document

The command "Regenerate" whenever you open a document

Line Types



Enable option "Use paper space units fo scaling" automatically

Automatically changes the setting nanoCAD Mechanica "Scale to paper space units" ("Use paper space units for scaling") for the types of lines.

Change current linetype scale

It enables or disables the scaling of line types in the standard objects when you change their size.

Use table of colors and weights

It enables or disables the use of the table of colors and weights. If enabled, you can use the standard colors from the table. "Compatibility Mode" - is used when working with documents created in early versions ofnanoCAD Mechanica.

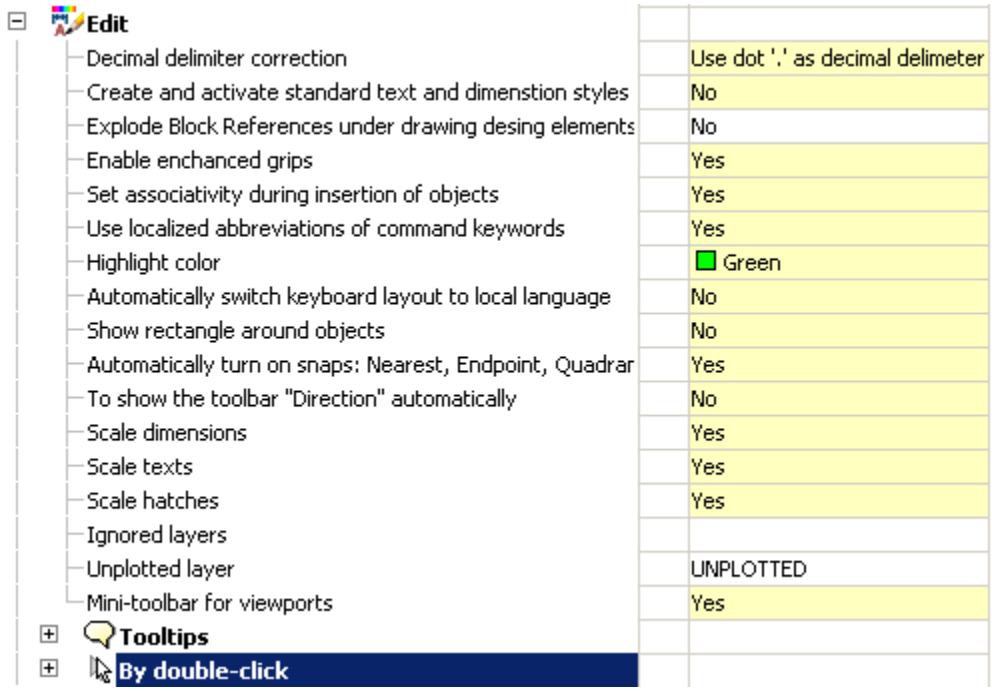
Standard text weight

It standardizes the value of the weight of the text for all text in the document.

Color and line width

Is to set the standard design elements.

Edit



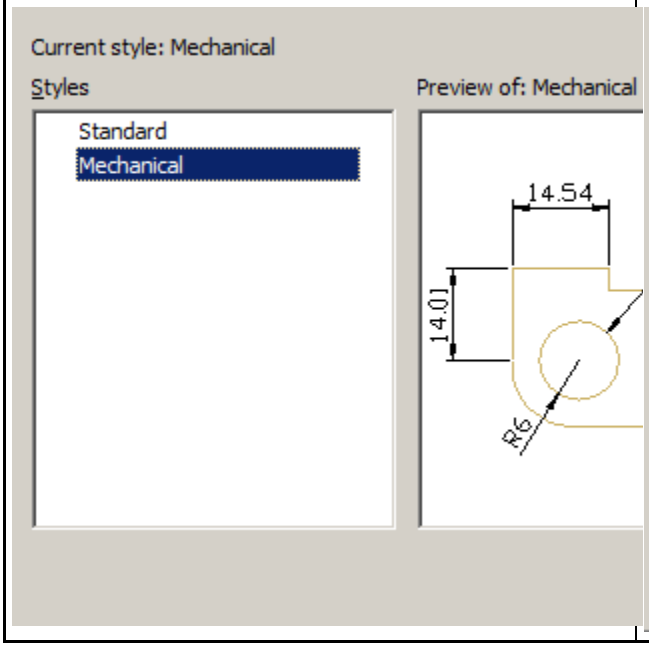
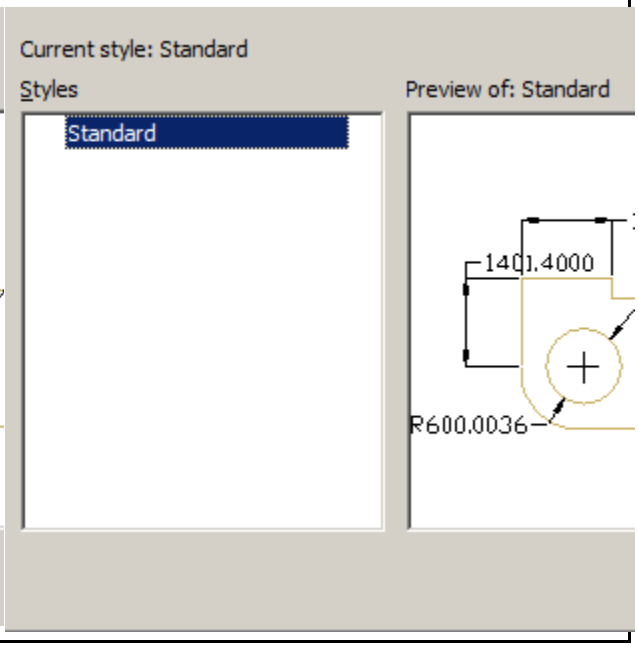
Decimal delimeter correction

- *Do not correct decimal delimeter* - does not perform any actions with a separator.
- *Use dot '.' as decimal delimeter* - automatically replaces the delimiter with a dot.
- *Use system locale decimal delimeter* -in this case, the separator will be replaced with the symbol set in the system settings.

Create an activate standard text and deimension styles in new documents

Managing the creation of standard styles. The default setting is "Yes". Specifying the value "No" allows you to create new documents without predefined standard dimensional styles.

Yes	No

Yes	No
	

Explode Block References under drawing design elements

Yes - when overlapping objects nanoCAD Mechanica blocks nanoCAD break up.

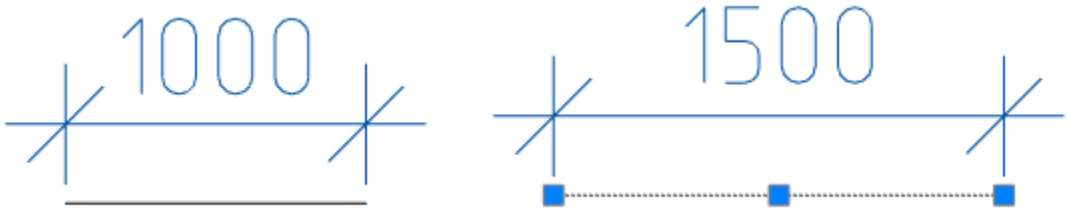
No - when overlapping objects nanoCAD Mechanica blocks nanoCAD masked with the command *wipeout*.

Enable enchanced grips

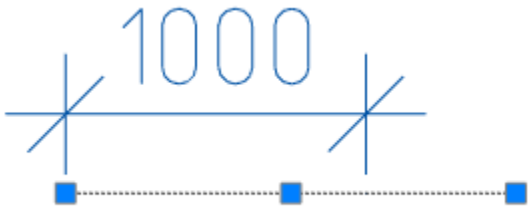
When this function is enabled, additional handles are displayed on the objects nanoCAD Mechanica.

Set associativity during insertion of object

By default, the setting is enabled. When you turn off, the binding of the inserted objects to the primitives is inactive. Example with dimensions. If the setting is enabled, then, when setting the dimensions on the line, the size will be associated with the line. When you change the line, the size changes.



If the setting is turned off, only the line will change.



Use localized abbreviations of command keywords

Abbreviations for the keywords of the commands are described in the file with the extension .pgp and are located in the installation location nanoCAD Mechanica

Highlight color

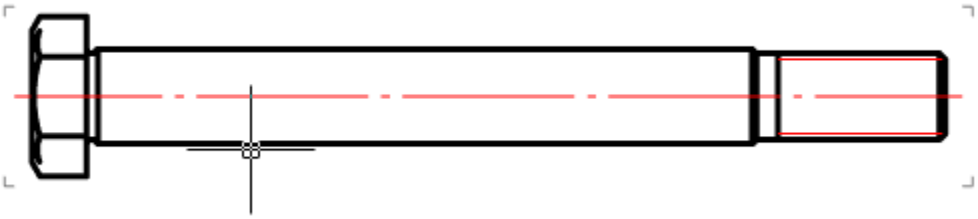
The color of the highlighting of primitives, when their indication is required. For example, the line's highlight when specifying a size for the placement.

Automatically switch keyboard layout to local language

When you call dialogs with fields nanoCAD Mechanica automatically turns on the layout set in the system settings.

Show rectangle around objects

Display control of the frame around objects nanoCAD Mechanica and blocks nanoCAD.



When this option is enabled, scaling is turned off Shift+RBM.

Automatically turn of snaps: Nearest, Endpoint, Quadrant, Center

Temporarily includes bindings when inserting objects nanoCAD Mechanica.
With the option disabled, the "Direction" toolbar may not work correctly.

To show the toolbar "Direction" automatically

The "Direction" toolbar control that appears when you insert database objects and a number of drawing design teams.

Scale dimensions

Yes value:
1. The measurement scale specified in the "Dimensional Styles" on the "Primary units" tab is replaced by the measurement scale specified in the Design Settings - Main Options - Design.
2. The global scale specified in the "Dimension Styles" on the "Fit" tab is replaced by the design scale specified in the Design Settings - Main Options - Design.
The value is "No". The scale is not replaced.

Scale texts

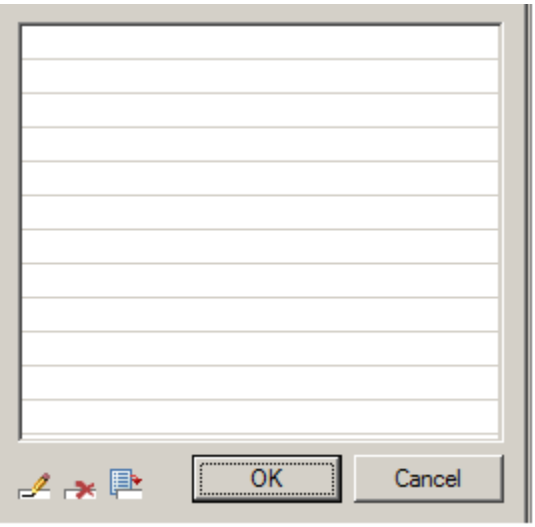
The value is "Yes". When you change the scale using the Scale toolbar, the "Height" parameter in the "Text Format" dialog for the multi-line text and the command line for the single-line text proportionally changes.
The value is "No". When you zoom in using the Zoom toolbar, the Height option does not change.

Scale hatches

The value is "Yes". When zooming with the Scale toolbar, the Scale parameter in the Hatch dialog is proportionally changed.
The value is "No". When zooming with the Scale toolbar, the Scale option in the Hatch dialog does not change.

Ignored layers

Specifies layers nanoCAD, on which primitives will not overlap with objects nanoCAD Mechanica.
When you click on the ellipsis, the "Excluded Layers" dialog opens to edit the list of layers.



The dialog menu allows you to add a layer and enter it manually, delete a row with a layer or select a layer from the ones available in the drawing.

Unplotted layer

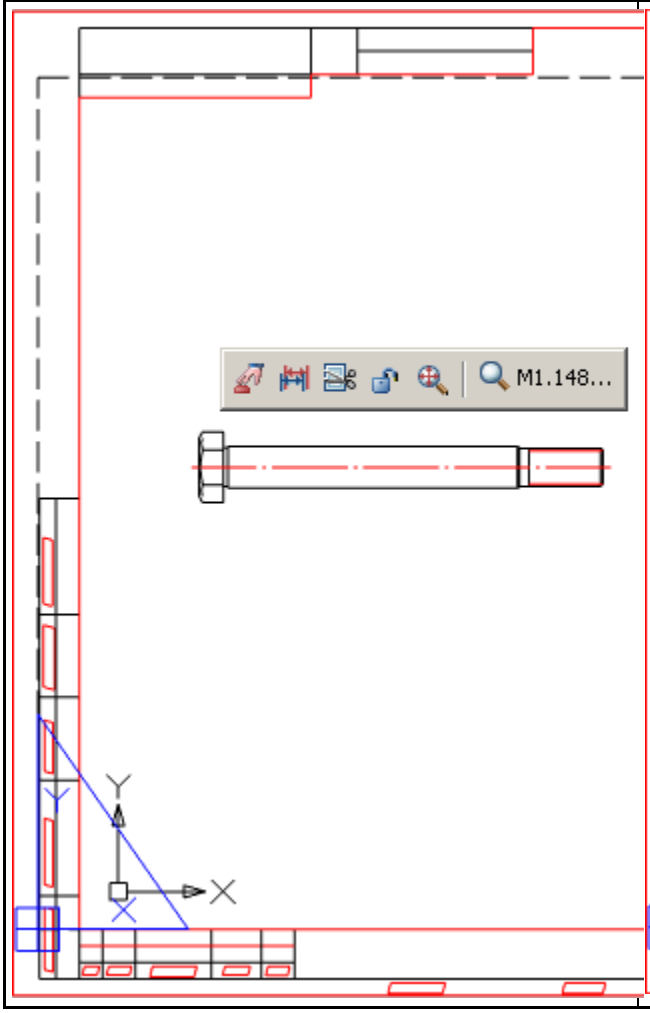
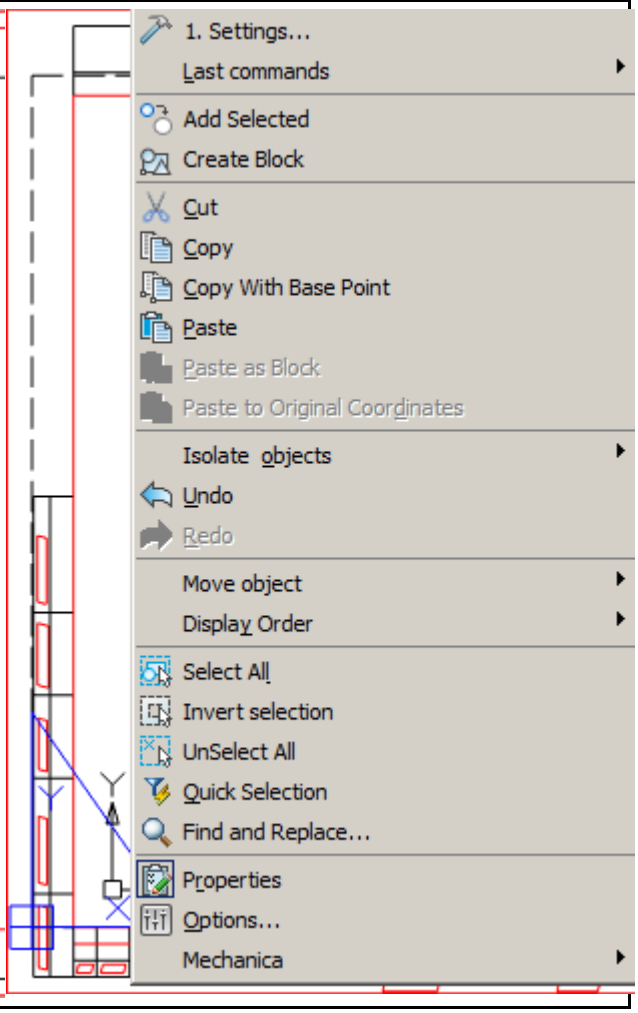
Specifies which layer the objects will be placed on nanoCAD Mechanica, which will not be printed.

Mini-toolbar for viewports

The nanoCAD Mechanicatoolbar control that appears when you right-click inside the viewport.

If the option is off, then when you right-click inside the viewport nanoCAD the standard menu will be called up nanoCAD.

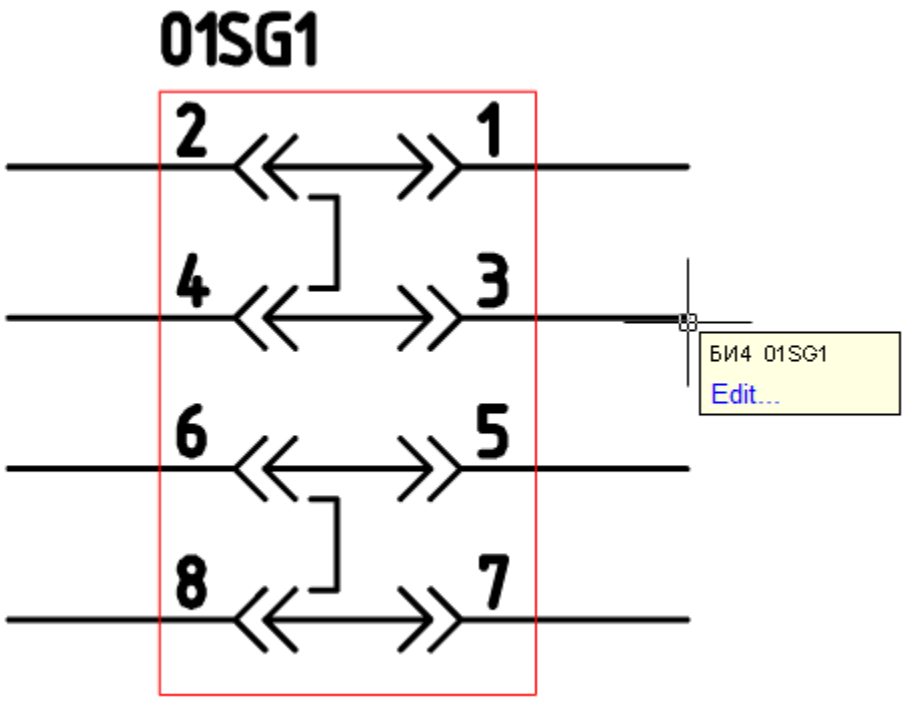
Yes	No

Yes	No
	

Tooltips

Show tooltips

Advanced Tooltip Management Tool (with hyperlink *"Edit..."*) for objects nanoCAD Mechanica and blocks nanoCAD.



Show delay, ms
Tool to control the speed of the appearance of tooltips.
Hide delay, ms
Tool for managing the duration of the tooltip display.

By double-click

Program objects
Sets the rule that when you double-click on object nanoCAD Mechanica, the editing dialog of this object will be called.
With the option disabled, the properties of the object are called.
Dimensions, Regular texts, Multi-texts
With the option enabled, double-clicking the left mouse button on the size brings up the editing dialog nanoCAD Mechanica.
With the option disabled, the properties of the object nanoCAD or edit dialog nanoCAD.

Design

Use symbol scale

Changes the type of scaling:

- *Yes* - Drawing scale.
- *On* - Model scale.

More details in the description of work with the current scale.

Drawing scale

Sets the default scale of design elements and the scale of the types of geometry lines.

Note: The scale type of the dimension lines is reserved and is always 1 by default.

Model scale

Sets the default measurement scale. The displayed value, which is set when dimensioning, increases in direct proportion to the scale value. For example, if you set the size of a 10 mm section and the measurement scale is 1:10, the displayed value is 100.

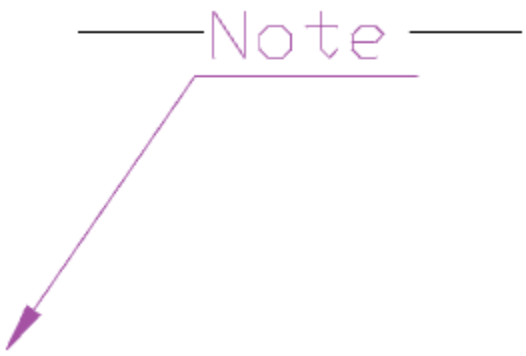
Text wipeout offset

Sets the standard indentation of the geometry marking from the text.
Using the example of a size marker. With a larger size, the square background overlaps the geometry.

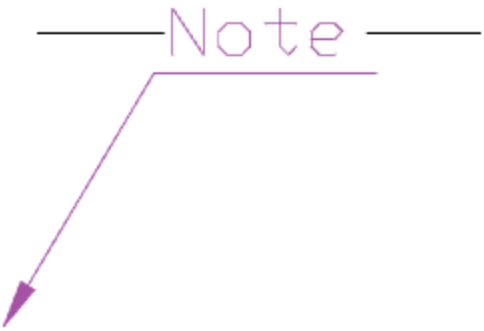
0.2	1

Cut CAD-platform elements

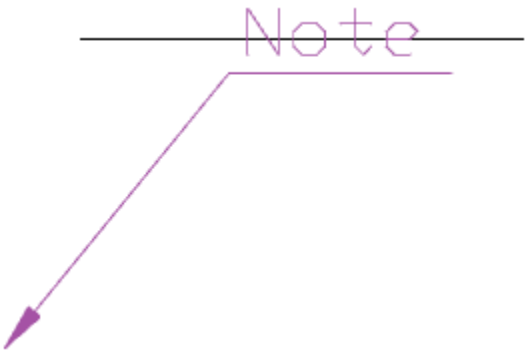
"Wipeout" - closes the primitive.



"Cut" - cut the portion of the primitive. If you delete or move carved primitive returns its status.



"No" - It does not cover the primitive.



Notifications

Enable hints

Tool tips display control.

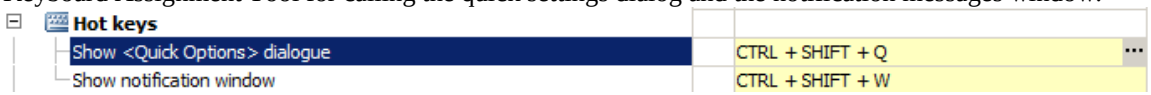
This option does not apply to messages with the status "Error".

Avoid mouse pointer

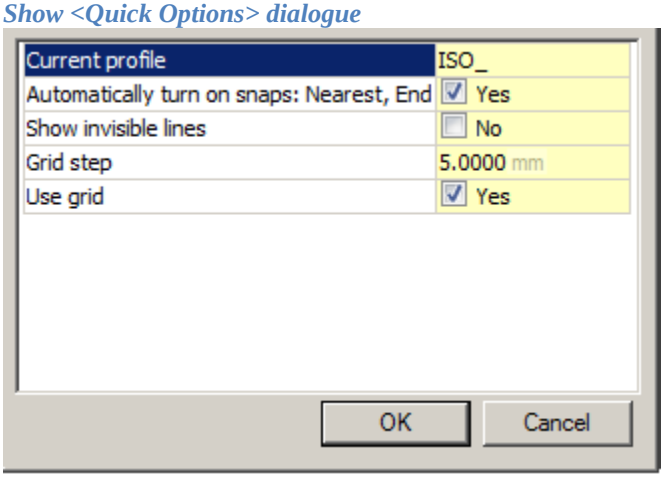
When this option is enabled the message will be moved around the screen so as not to obstruct the selection.

Hot keys

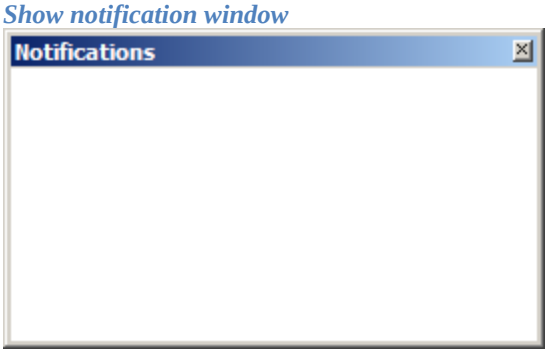
Keyboard Assignment Tool for calling the quick settings dialog and the notification messages window.



Show <Quick Options> dialogue

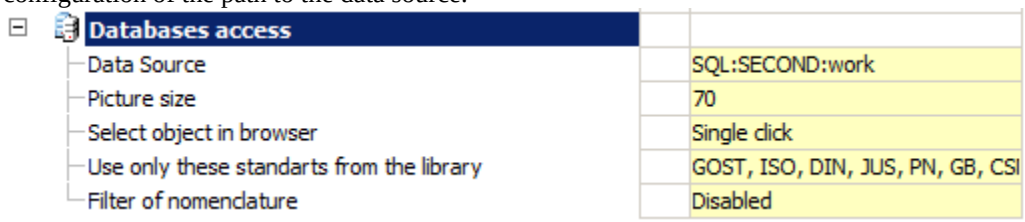


Show notification window



Databases access

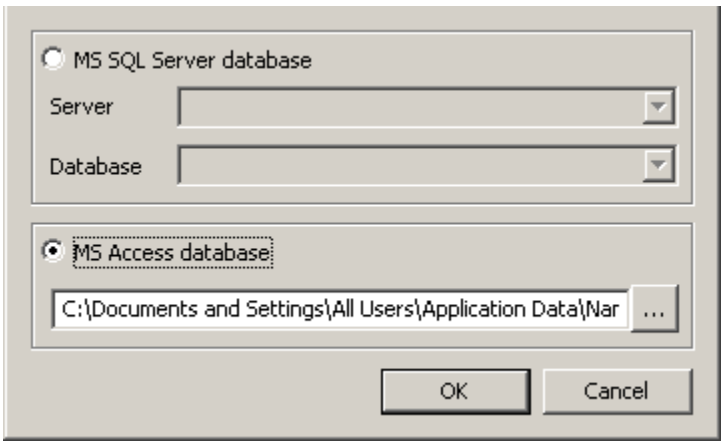
The database includes, in addition to standard elements, table and format templates, bolt assembly templates, groups and markers, as well as examples and other user elements. The choice of a specific database is determined by the configuration of the path to the data source.



Data Source

The database nanoCAD Mechanica 20.1 includes, in addition to standard elements, table and format templates, bolt assembly templates, groups and markers, as well as examples and other user elements. The choice of a specific database is determined by the configuration of the path to the data source.

When you click on the button in the "Data source" field, the dialog for specifying the data source appears.

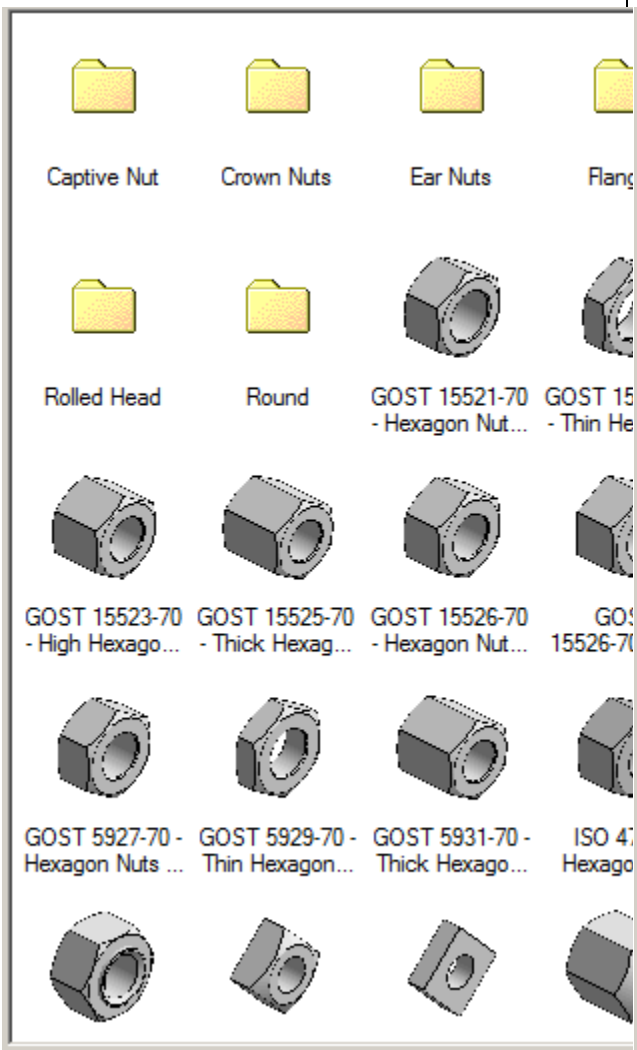
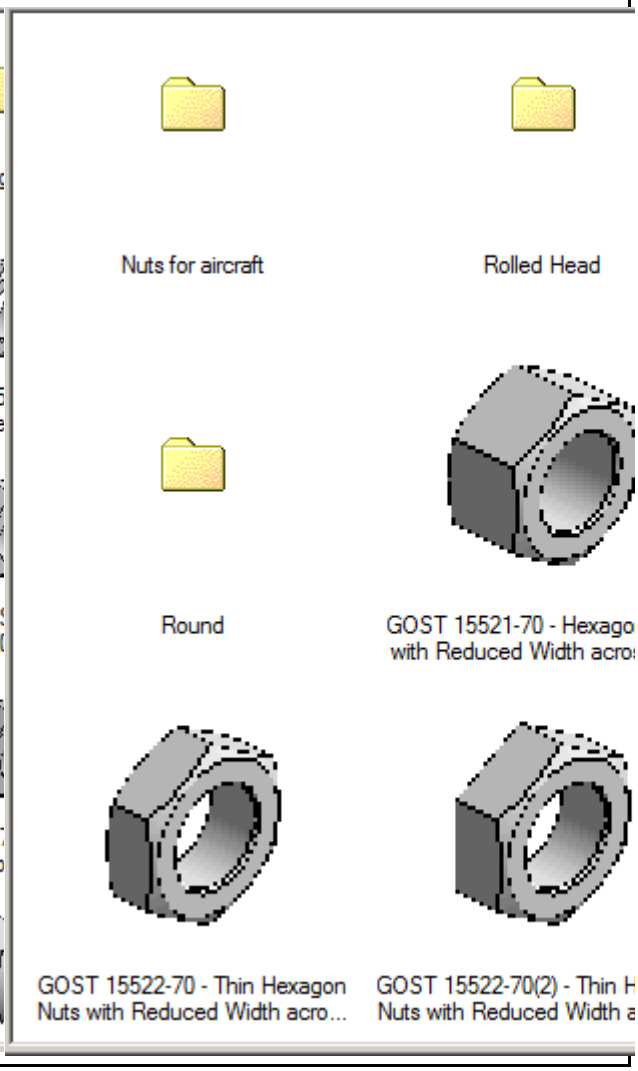


nanoCAD Mechanica 20.1 provides work, both with local databases, and with the database on the MSSQL server.
If you are using an MS SQL database, you must select the "Data Source - MSSQL" radio button. In the "Server" field, specify the server name (along with the name of the SQL Database Server Instance). For example: SERVER, or SERVER\SQLEXPRESS. In the "Database" field, you must specify the name of the database to which you are connecting.
If you are using a local database,you must select the "Data Source - Local Database" radio button, and then specify the path to the database file.
As a local database for computers with the x32 bit capacity, format files *.mcs (MS Access).
As a local database for computers with the x64 bit capacity, format files *.mdf (LocalDB, for Windows Vista and higher).
Required drivers are installed automatically when installed nanoCAD Mechanica 20.1.
By default, with a local installation, the database is located in the folder %programdata%\nanosoft*nanoCAD Mechanica20.1*7.0\DataRW\.

Picture size

The size of the preview image for the database elements in pixels.

70	**140**

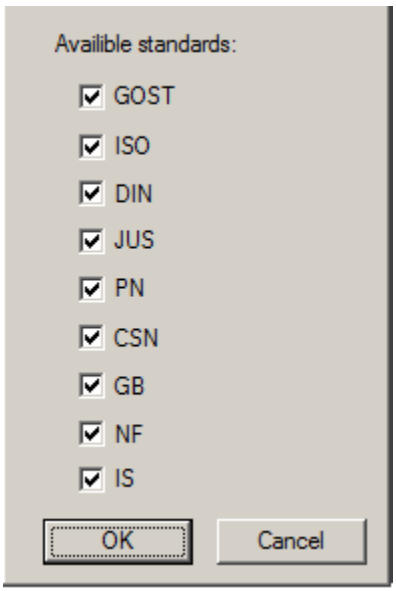
70	140
	

Select object in browser

- Single click
- Dowble click

Use only these standarts from the library

Restricting the used standards filter.

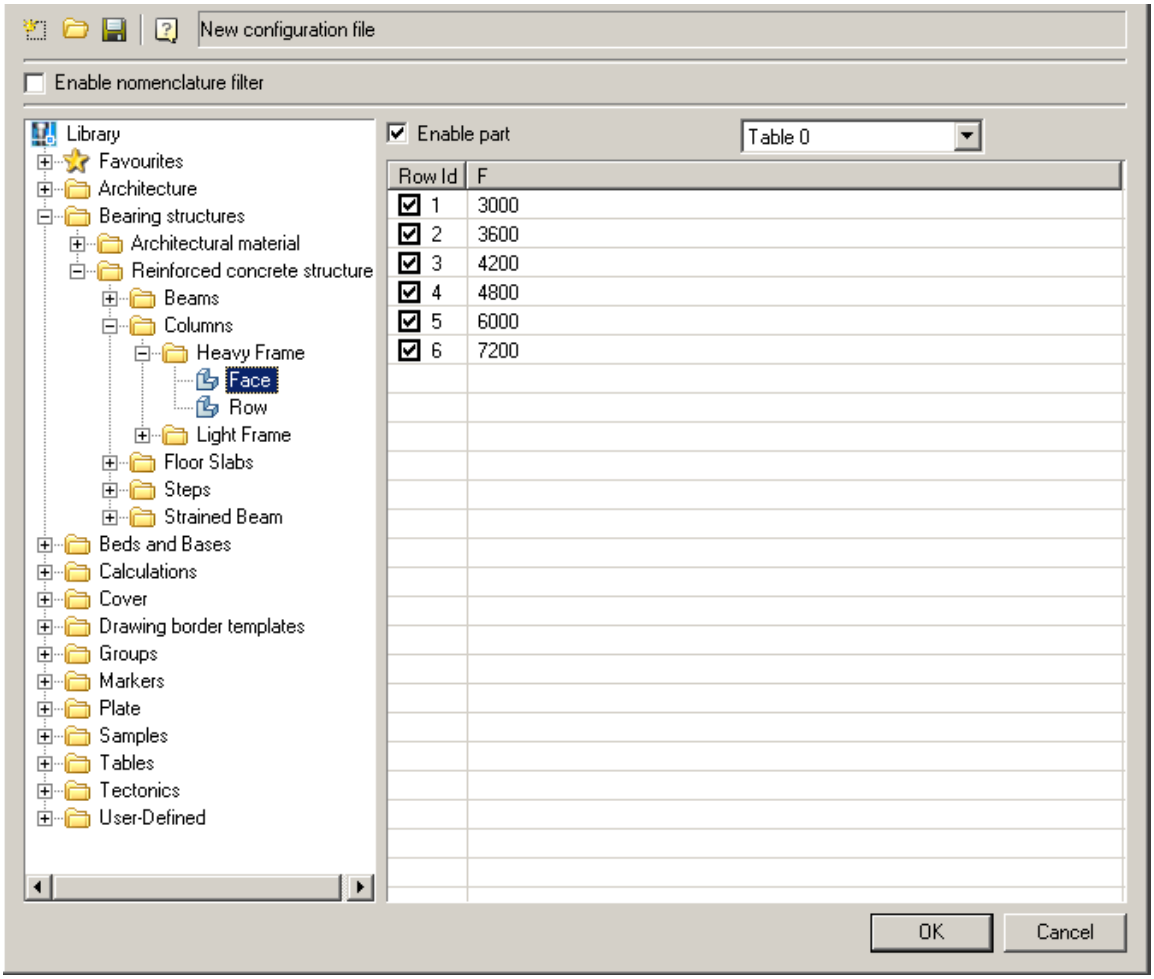


Filter of nomenclature

To limit the selection of parts from the catalog of standard and standardized products nanoCAD Mechanica 20.1 it is possible to automatically filter the database using nomenclature filters.

To enable and configure a nomenclature filter, you must enable this feature in the settings nanoCAD Mechanica 20.1.

The filter editor is called by pressing the button with the ellipsis. As a result, the settings window of the nomenclature filter appears:



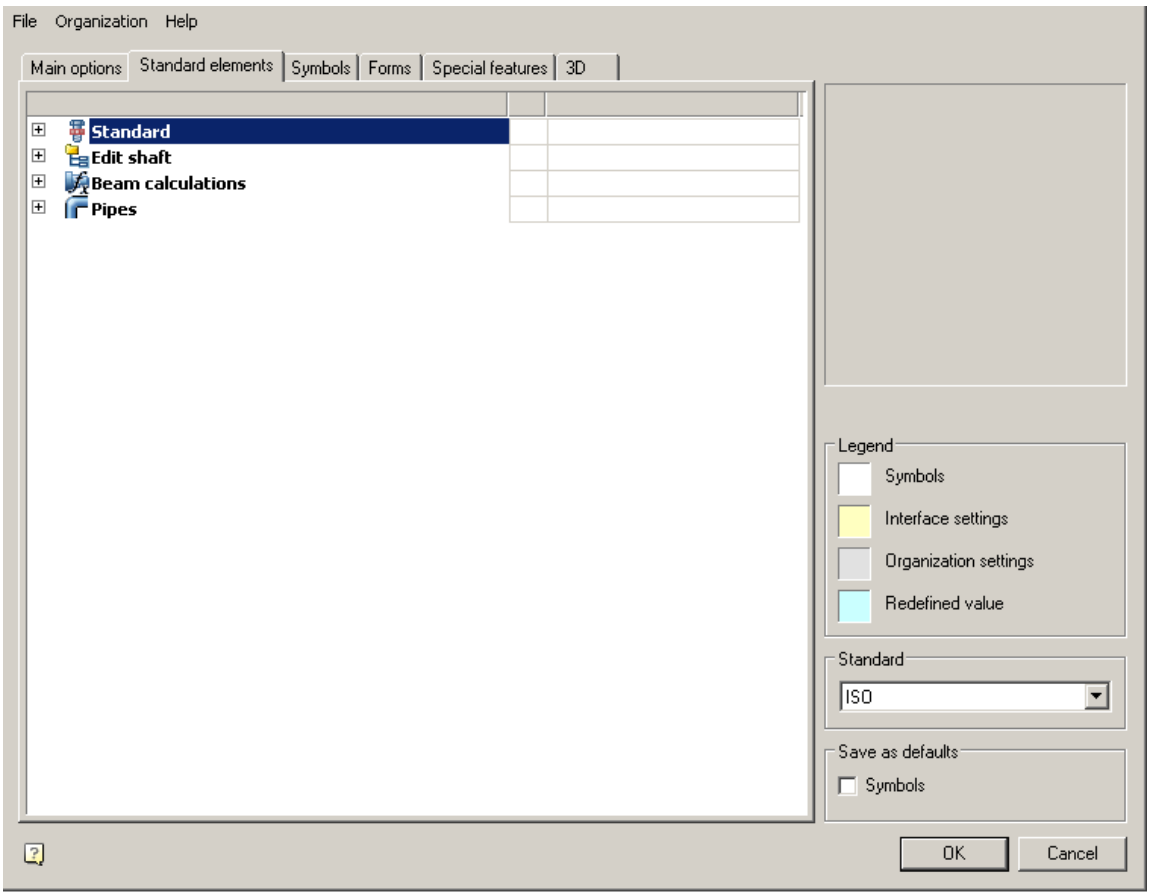
In the upper part of the window there are buttons for creating, downloading and saving files of the nomenclature filters, as well as a switch for enabling / disabling database filtering according to the selected filter. nanoCAD Mechanica 20.1 allows you to switch between different filters and temporarily turn them off.
The structure of the database is displayed in the left part of the window nanoCAD Mechanica 20.1, the icons of which show the current state of each element of the database:
- the standard icon of the database element. Indicates that all sizes and the element itself are allowed to be used.
- prohibition to use icon. Indicates that this element is not allowed to be used.
- partial use icon. Indicates that this element is allowed for use with a limited set of standard sizes.

To enable / disable the use of the entire element, use the "Allow part use" switch. To disable a specific part size, you must uncheck the column ID of a line of this size in the part dimension table.

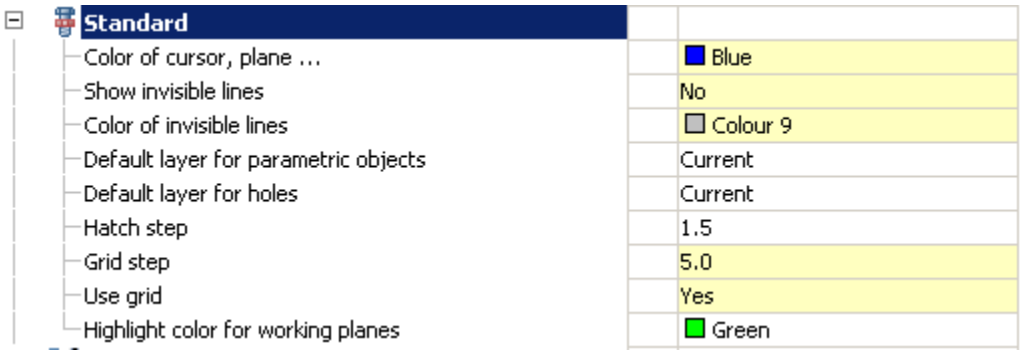
Note:	Database access settings are interface settings. If you use data compression on a disk, you may experience problems connecting to local databases (for x64 versions).

Standard elements

The tab is used to configure standard elementsnanoCAD Mechanica.



Standard



Color of cursor, plane...

The control display color selection standard elements when they are selected, the automatic installation of dependencies.
The color display can be selected from a set and the built-in color picker.

Show invisible lines

Control the display of invisible (hidden) lines of standard graphic elements.

Color of invisible lines

The control display color selection invisible (hidden) lines of standard graphic elements.
The color display can be selected from a set and the built-in color picker.

Default layer for parametric objects

The control layer selection, which by default will be placed standard parts database.
A layer can be selected from the existing (previously created) or create a new layer in the same window.

Default layer for holes

The control layer selection, which by default will be placed holes created by the operation "Bolting".
A layer can be selected from the existing (previously created) or create a new layer in the same window.

Hatch step

How to set the hatching step, is used by default.

Grid step

The control is used for the step of asking the values of the diameter and length of the shaft sections created using the "Spindles".
When you insert a section of the shaft the value of its length and diameter will be rounded up to a multiple grid spacing.

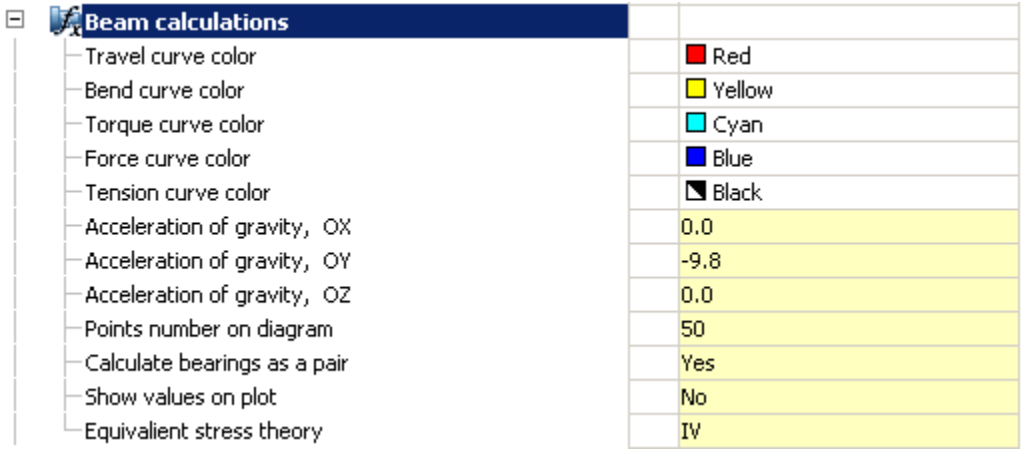
Use grid

Control Using the "spacing".
If the grid is not used, the values of the diameter and length of the shaft portions are not rounded.

Highlight color for working planes

Highlight color for working planes

Beam calculations



Curve colors travel, bend, torque, force, tension

Color control diagrams displayed in the calculation of gross.
The selected color is preserved when exporting diagrams in the drawing.
The color display can be selected from a standard set and set of built-in your palette.

Acceleration of gravity OX, OY, OZ

Set design parameters for the respective axes of acceleration of the shaft.

Points number on diagram

Control increments partition diagram plotted in the entire length.

Calculate bearings as a pair

Complementing the strength calculation of shaft reactions bearings.

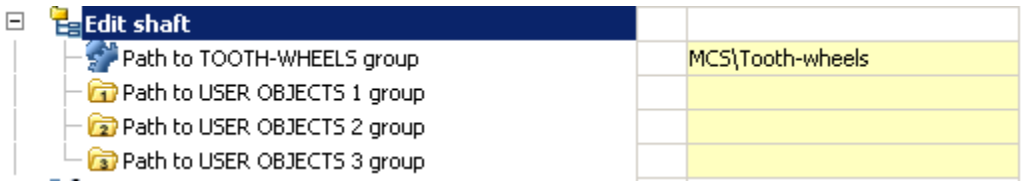
Show values on plot

Control the display of numerical values on the diagram exported to the drawing.
The values are displayed at the critical points of the shaft.

Equivalient stress theory

Sets the strength of the theory to calculate the equivalent stresses shafts.

Edit shaft



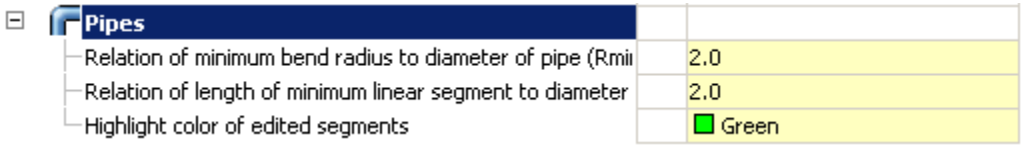
Path to TOOTH-WHEELS group

It specifies the path to the database that contains the elements of the bookmark "Gears" dialogue "Editing shafts".

Path to USER OBJECTS ...

It specifies the path to the folder Users. After you specify the folder elements of it will be displayed in the dialog "Edit shafts".

Pipes



Relation of minimum bend radius to diameter of pipe (Rmin/Dn)

Set the default value of the bending radius of the pipe.

Relation of length of minimum linear segment to diameter of pipe (Lmin/Dn)

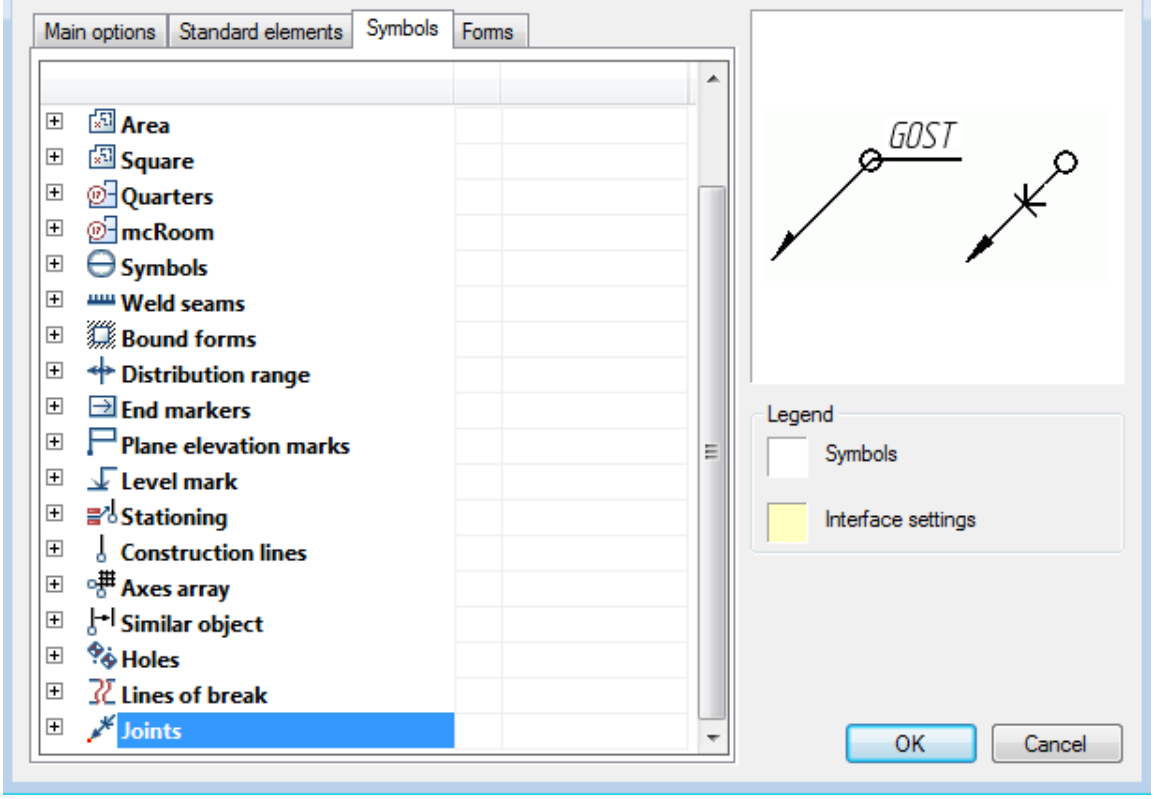
Set the default value of the minimum length of straight pipe.

Highlight color of edited segments

Set the backlight color workplanes valves when connecting the pipe to them.

Symbols

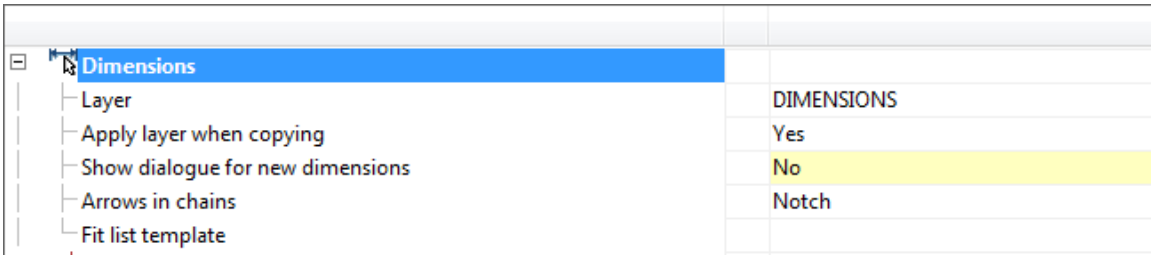
Tab "Symbols". Designed to customize the display of drawing symbols of nanoCAD Mechanica.



- Dimensions
- Notes
- Views, sections, cuts etc
- Welds seams
- Holes
- Lines of break
- Joints

Dimensions

Setting allows you to change the size of the layer, which is automatically updated with new dimensions and copied settings dialogs, etc.



Layer

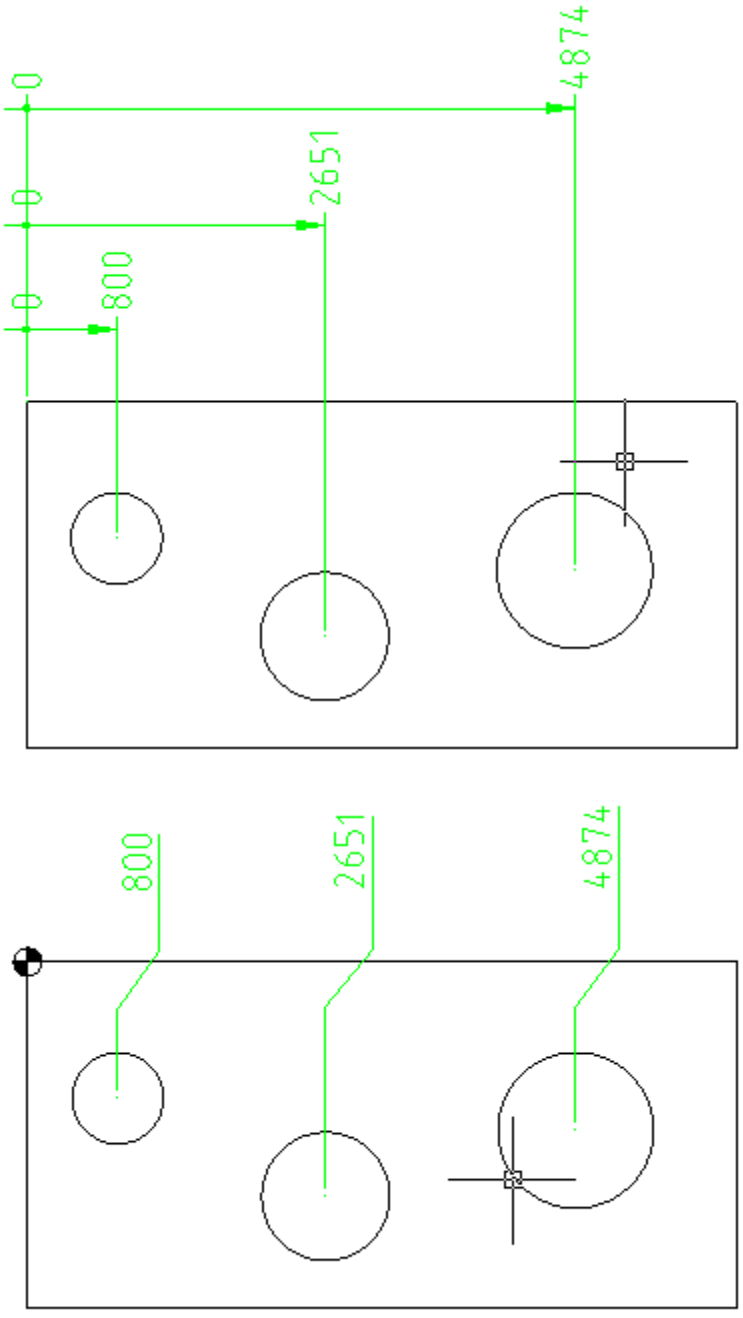
Allows you to choose the layer,which by default will include new dimensions and copied.

Apply layer when copying

When this option is copied dimensions will be placed on the drawing layer selected for size. When off, dimensions placed on the active layer.

Show dialog for new dimension

Control built ordinate dimensions. If the option is used ordinate dimensions standard ISO, built-in platform nanoCAD. When this option is used ordinate dimensions nanoCAD Mechanica.



Open a dialog to automatically

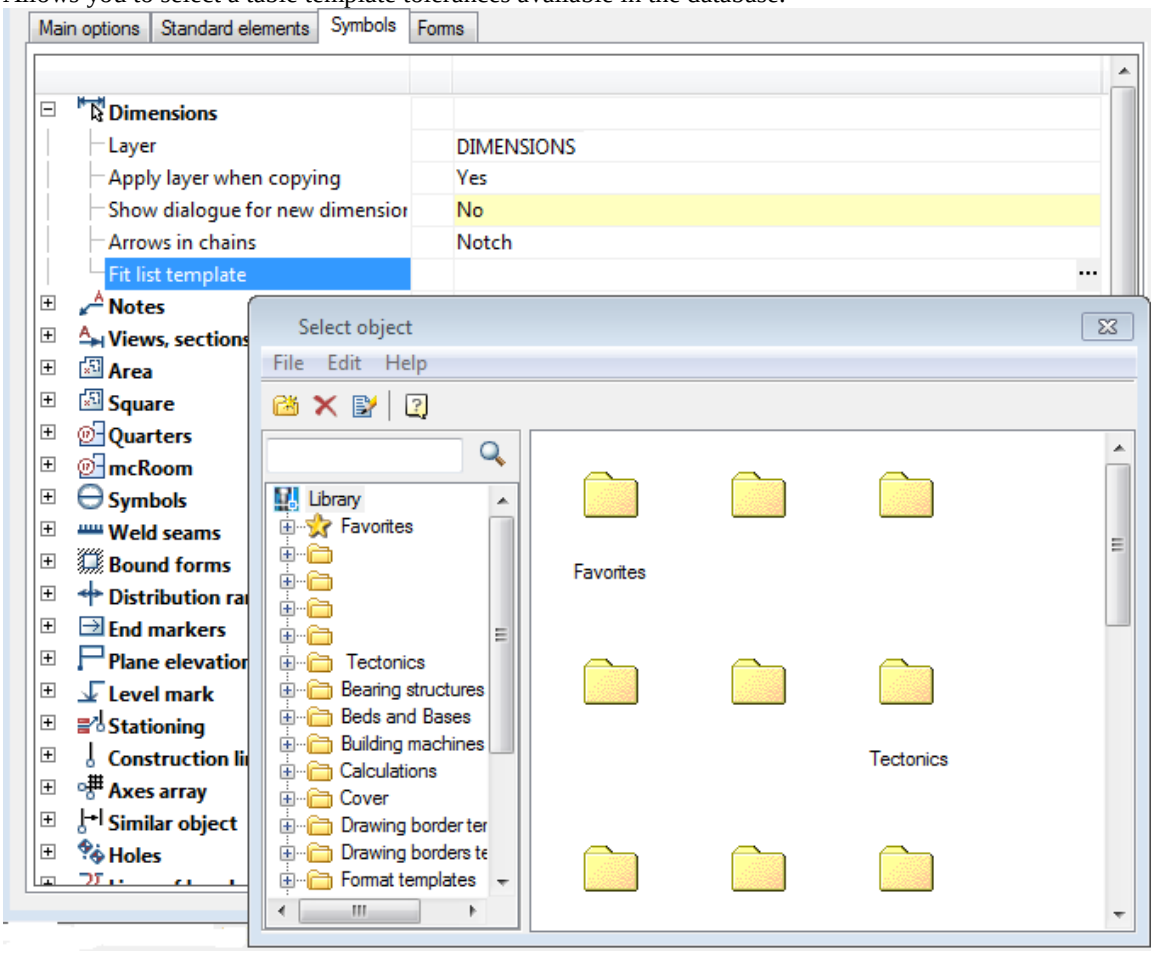
Control automatic opening dialogue sizes.

Arrows in chains

Replaces used in nanoCAD arrows in dimensional chains. There is a possibility to replace the serifs, or replace point at all.

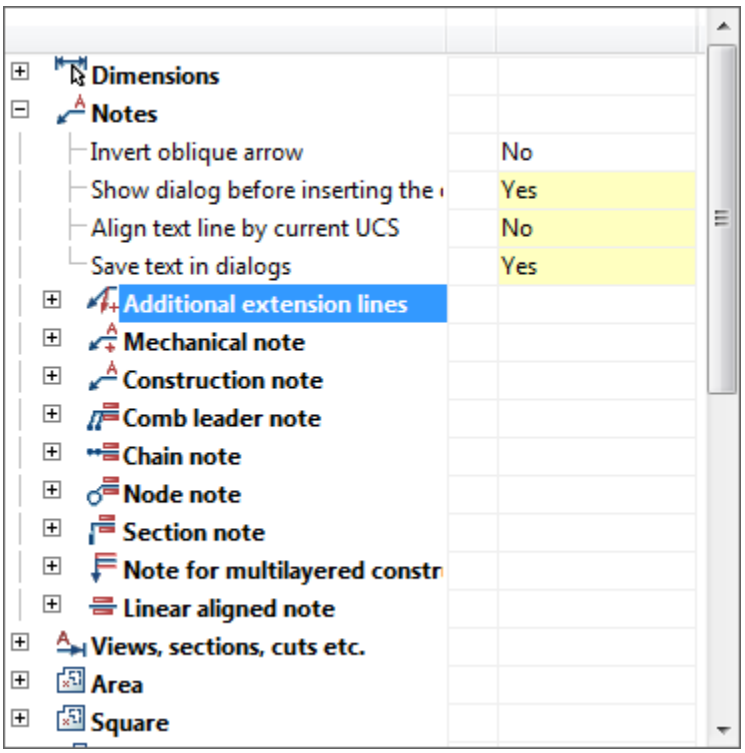
Spreadsheet template tolerances

Allows you to select a table template tolerances available in the database.



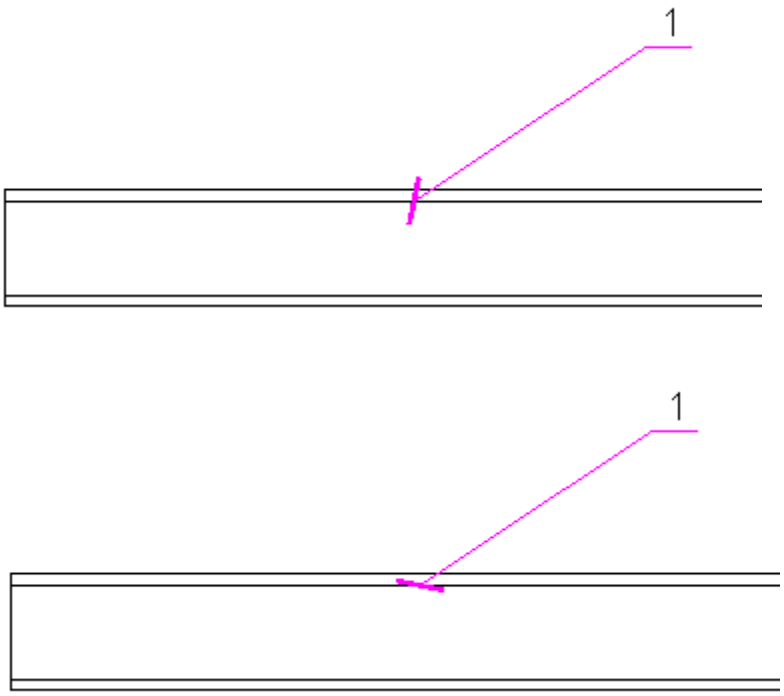
Notes

Settings of notes, such as text, arrows, pointers.



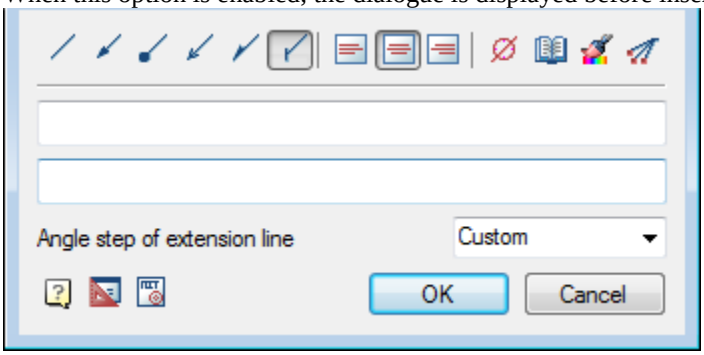
Invert oblique arrow

Allows you to choose the direction of oblique arrow in leader.



Show dialog before inserting the object

When this option is enabled, the dialogue is displayed before inserting note.

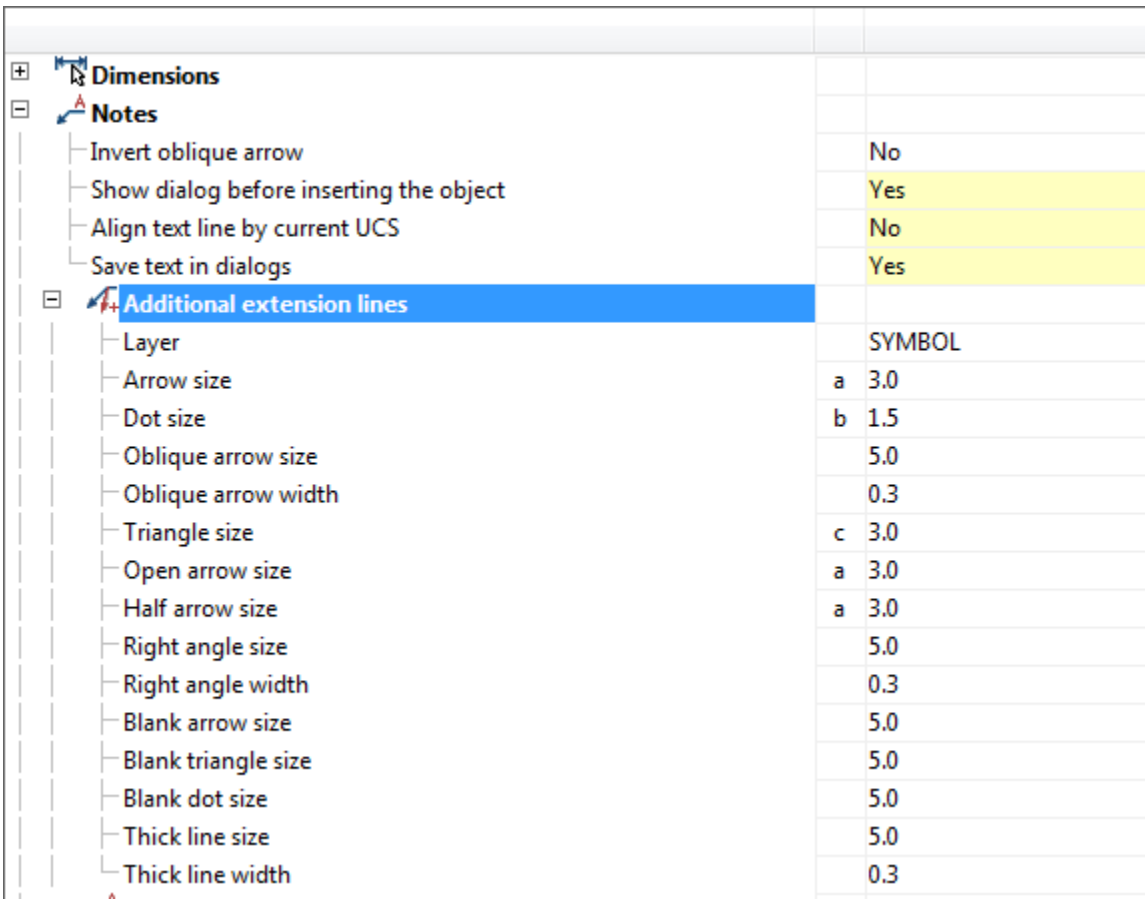


Align text line by current UCS

When this option is on, shelves of leaders are rotated by the User coordinate system.

Save text in dialogs

When this option is enabled on inserting the following notes previously entered text remains in the input field. The remaining options allow you to change the size of text, indexes, thickness, layer placement, color, etc.



Dimensions		
Notes		
Invert oblique arrow		No
Show dialog before inserting the object		Yes
Align text line by current UCS		No
Save text in dialogs		Yes
Additional extension lines		
Layer		SYMBOL
Arrow size	a	3.0
Dot size	b	1.5
Oblique arrow size		5.0
Oblique arrow width		0.3
Triangle size	c	3.0
Open arrow size	a	3.0
Half arrow size	a	3.0
Right angle size		5.0
Right angle width		0.3
Blank arrow size		5.0
Blank triangle size		5.0
Blank dot size		5.0
Thick line size		5.0
Thick line width		0.3

Additional extension lines

Additional extension lines		
Layer		SYMBOLS
Arrow size	a	3.0
Dot size	b	1.5
Oblique arrow size		3.0
Oblique arrow width		0.0
Triangle size	c	3.0
Open arrow size	a	3.0
Half arrow size	a	3.0
Right angle size		3.0
Right angle width		0.0
Blank arrow size		3.0
Blank triangle size		3.0
Blank dot size		3.0
Thick line size		3.0
Thick line width		0.3

Mechanical note

Mechanical note		
Layer		SYMBOLS
Colour		ByObject
Line thickness		ByLayer
Designation font size	c	7.0
Arrow size	a	3.0
Open arrow size	a	3.0
Half arrow size	a	3.0
Dot size	b	1.5
Oblique arrow size		3.0
Oblique arrow width		0.0
Text		
Text style		Standard
Text color		Black
Text height	c	2.5
Text weight		ByObject
Horizontal text offset from the end of line	d	1.5
Vertical text offset from line	e	1.0
Width factor		0.0

Construction note

Construction note		
Layer		SYMBOLS
Colour		ByLayer
Line thickness		0.25 mm
Arrow size	e	3.0
Dot size	f	1.5
Open arrow size	g	3.0
Half arrow size	h	3.0
Oblique arrow size	i	3.0
Oblique arrow width	j	0.0
Text		
Text style		Standard
Text color		ByLayer
Text height	a	2.5
Text weight		ByObject
Small text size	b	2.5
Horizontal text offset from the end of line	c	1.5
Vertical text offset from line	d	1.0
Width factor		0.0

Comb leader note

Comb leader note		
Layer		SYMBOLS
Colour		ByLayer
Line thickness		0.25 mm
Arrow size	d	3.0
Dot size	e	1.5
Open arrow size	f	3.0
Half arrow size	g	3.0
Oblique arrow size	h	3.0
Oblique arrow width	i	0.0
Text		
Text style		Standard
Text color		Black
Text height	a	2.5
Text weight		ByObject
Horizontal text offset from the end of line	b	1.5
Vertical text offset from line	c	1.0
Small text height		2.5
Width factor		0.0

Chain note

Chain note		
Layer		SYMBOLS
Colour		ByLayer
Line thickness		0.25 mm
Arrow size	d	3.0
Dot size	e	1.5
Open arrow size	f	3.0
Half arrow size	g	3.0
Oblique arrow size	h	3.0
Oblique arrow width	i	0.0
Text		
Text style		Standard
Text color		ByLayer
Text height	a	2.5
Text weight		ByObject
Horizontal text offset from the end of line	b	1.5
Vertical text offset from line	c	1.0
Small text height		2.5
Width factor		0.0

Node note

Node note		
Layer		SYMBOLS
Colour		ByLayer
Line thickness		0.25 mm
Space between knot and sheet	c	1.0
Linetype of mark`s line		ByObject
Color of mark`s line		ByObject
Lineweight of mark`s line		ByObject
Text		
Text style		Standard
Text color		ByLayer
Text height	a	3.5
Text weight		ByObject
Small text size	b	2.5
Horizontal text offset from the end of line	d	1.5
Vertical text offset from line	e	1.0
Width factor		0.0

Section note

Section note		
Layer		SYMBOLS
Colour		ByLayer
Line thickness		0.25 mm
Space between knot and sheet	c	1.0
Stroke length	f	10.0
Output through the blank line	g	3.0
Distance between a stroke and the beginning of a ext	h	2.0
Stroke weight	i	1.0
Text		
Text style		Standard
Text color		ByLayer
Text height	a	3.5
Text weight		ByObject
Small text size	b	2.5
Horizontal text offset from the end of line	d	1.5
Vertical text offset from line	e	1.0
Width factor		0.0

Note for multilayered constructions

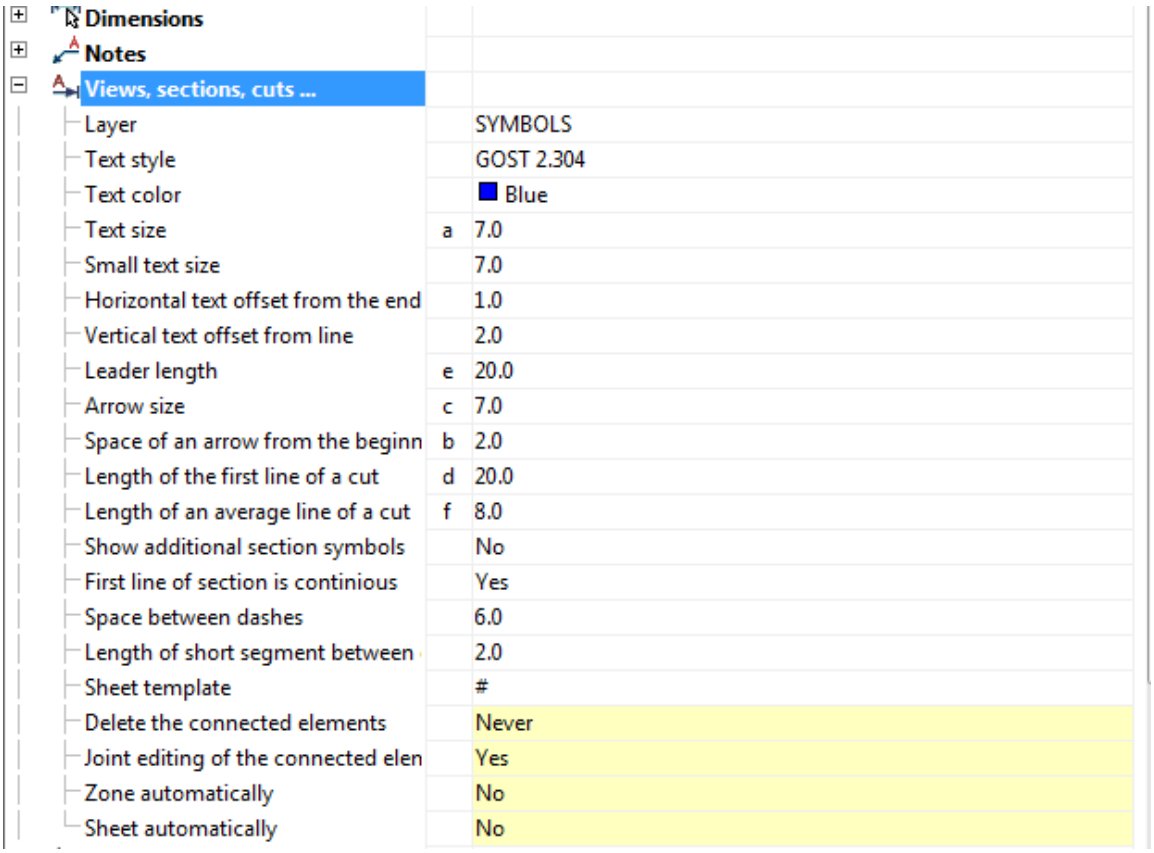
Note for multilayered constructions		
Layer		SYMBOLS
Colour		ByLayer
Line thickness		0.25 mm
Arrow size	d	3.0
Dot size	e	1.5
Open arrow size	f	3.0
Half arrow size	g	3.0
Oblique arrow size	h	3.0
Oblique arrow width	i	0.0
Right angle size		2.0
Right angle width		0.0
Text		
Text style		Standard
Text color		ByLayer
Text height	a	2.5
Text weight		ByObject
Horizontal text offset from the end of line	b	1.5
Vertical text offset from line	c	1.0
Width factor		0.0

Linear aligned note

Linear aligned note		
Layer		SYMBOLS
Colour		ByLayer
Line thickness		0.25 mm
Text		
Text style		Standard
Text color		ByLayer
Text height	a	2.5
Text weight		ByObject
Horizontal text offset from the end of line	b	1.5
Vertical text offset from line	c	1.0
Small text height		2.5
Width factor		0.0

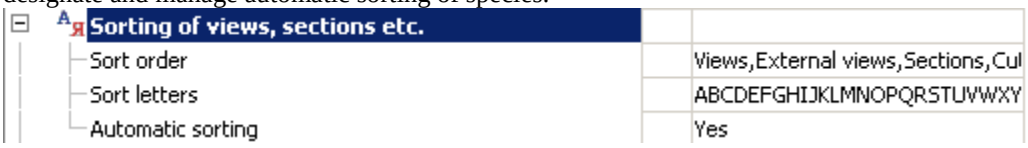
Views, sections

Settings of views, sections, detail views. It allow you to choose the layout for placing cuts and sections, color of symbols, text, size of text, lines, arrows.



Sort kinds of cuts ...

Settings menu sorting species profiles, detail views, cross sections. It allows you to specify a set of letters to designate and manage automatic sorting of species.



Hole tables

Settings menu to set the table all sorts of patterns, as well as a way to refer to sets of holes.

Holes tables	
List of coordinates template	List of coordinates
Hole table template	Hole table
Designation method	Numerical
Coordinate system	
Layer	SYMBOLS
Text style	GOST 2.304
Text height	3.5
Text color	Blue
Arrow size	3.5
Axis offset	1.0
Origin terminator	None
Origin terminator size	3.0
NC (Genius) sign size	6.0
Show coordinate system	Yes
Show 0 origin indicator	Yes

Holes

Settings of holes that allow you to change the layer for label placement, text style, size and color of the text, etc.

Holes		
Layer		SYMBOL
Text style		GOST 2.304
Text size		5.0
Text color		ByLayer
Length of a ledge of axial lines	a	2.0
Center identifier size		5.0

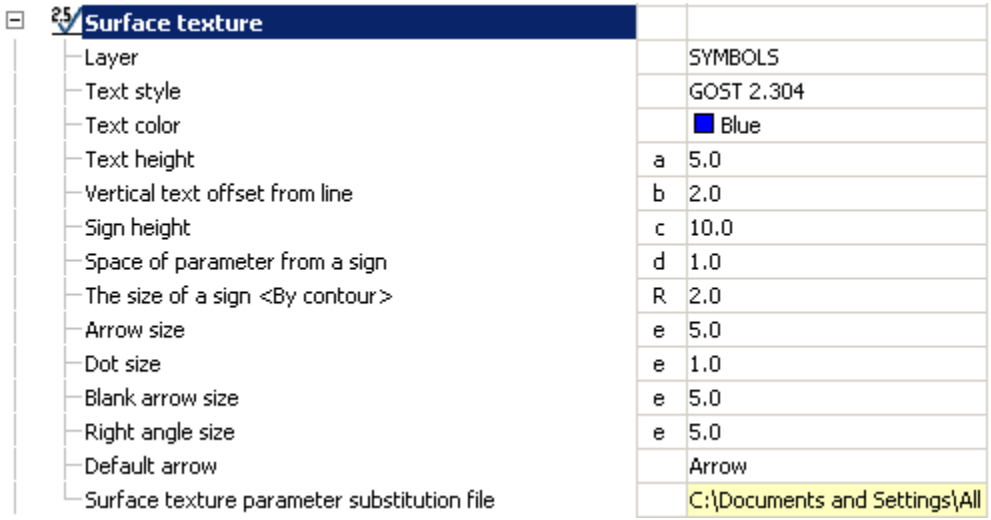
Marking and branding

Settings menu markers and brands.

Marking and branding		
Layer		SYMBOLS
Text style		GOST 2.304
Text color		Blue
Arrow size	a	7.0
Dot size	b	1.0
Text size on the extension line	c	5.0
Text size inside of a sign	d	5.0
Circle diameter	e	15.0
Triangle height	f	15.0
Coeff. of compression of TR item text		0.6

Surface texture

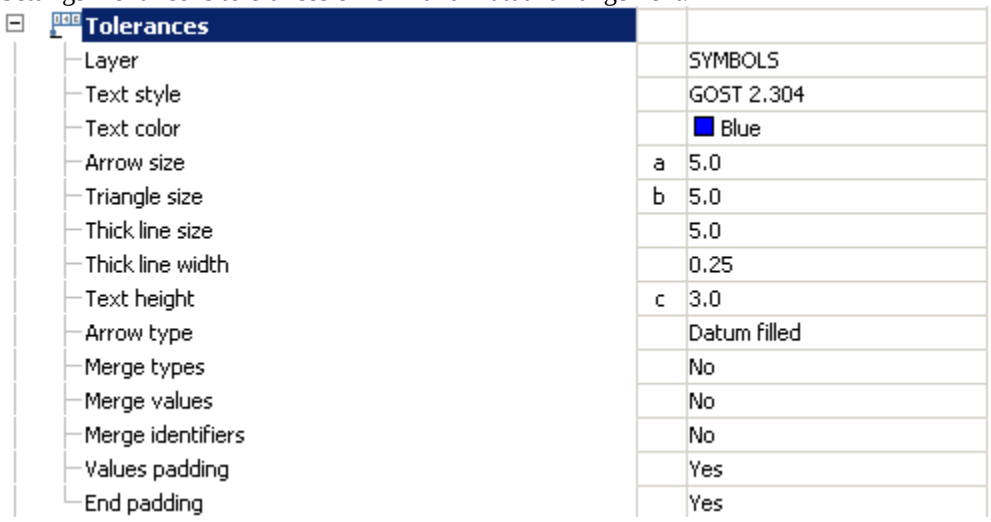
Setup menu icon, surface roughness, and allows you to specify the path to the directory roughness.



Surface texture		
Layer		SYMBOLS
Text style		GOST 2.304
Text color		Blue
Text height	a	5.0
Vertical text offset from line	b	2.0
Sign height	c	10.0
Space of parameter from a sign	d	1.0
The size of a sign <By contour>	R	2.0
Arrow size	e	5.0
Dot size	e	1.0
Blank arrow size	e	5.0
Right angle size	e	5.0
Default arrow		Arrow
Surface texture parameter substitution file		C:\Documents and Settings\All

Tolerances

Settings menu icons tolerances of form and mutual arrangement.



Tolerances		
Layer		SYMBOLS
Text style		GOST 2.304
Text color		Blue
Arrow size	a	5.0
Triangle size	b	5.0
Thick line size		5.0
Thick line width		0.25
Text height	c	3.0
Arrow type		Datum filled
Merge types		No
Merge values		No
Merge identifiers		No
Values padding		Yes
End padding		Yes

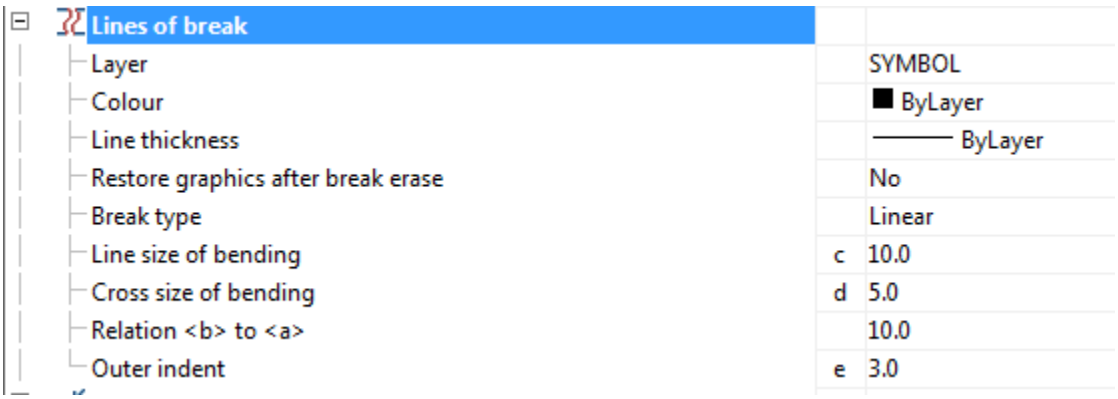
Construction lines

Setup Menu display of auxiliary lines.

Construction lines		
Layer		CONSTRUCTION_LINES
Color		Red
Draw ray		Yes
Line thickness		0.25 mm

Line of break

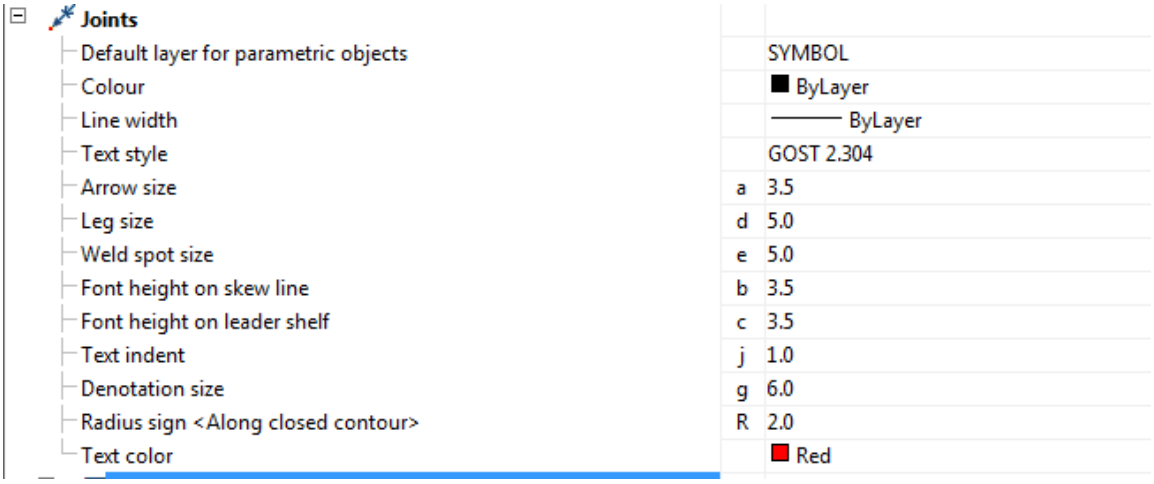
Settings of break lines,that allow you to change the layer for label placement, color, and line sizes, type of fracture lines, etc.



Lines of break		
Layer		SYMBOL
Colour		ByLayer
Line thickness		ByLayer
Restore graphics after break erase		No
Break type		Linear
Line size of bending	c	10.0
Cross size of bending	d	5.0
Relation <b> to <a>		10.0
Outer indent	e	3.0

Fixed Joints

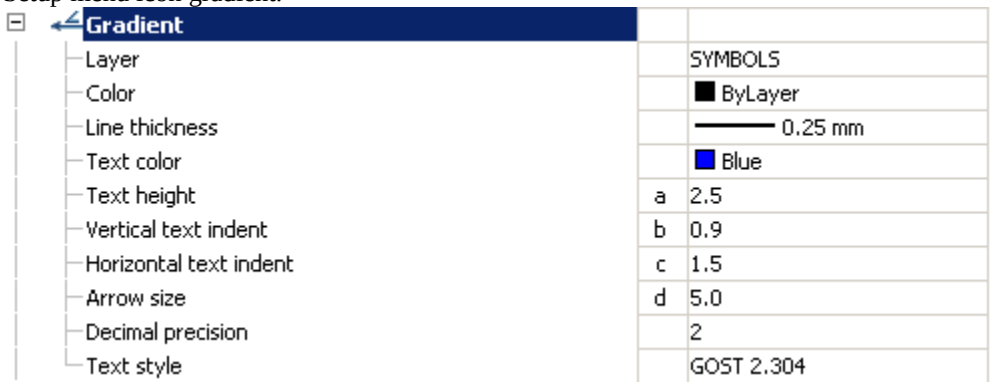
Settings of symbols of permanent connections, that allow you to change the layer, label placement, color, size, style, text and lines.



Joints		
Default layer for parametric objects		SYMBOL
Colour		ByLayer
Line width		ByLayer
Text style		GOST 2.304
Arrow size	a	3.5
Leg size	d	5.0
Weld spot size	e	5.0
Font height on skew line	b	3.5
Font height on leader shelf	c	3.5
Text indent	j	1.0
Denotation size	g	6.0
Radius sign <Along closed contour>	R	2.0
Text color		Red

Gradient

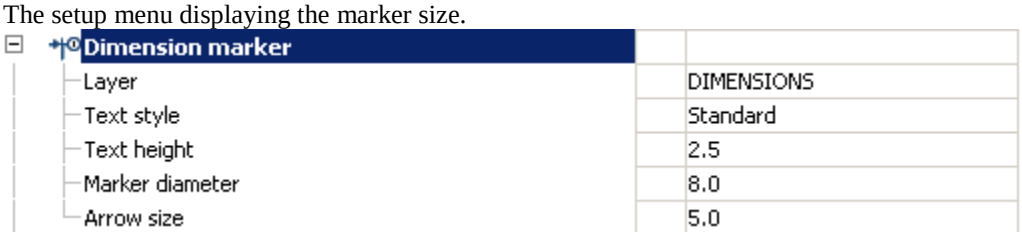
Setup menu icon gradient.



Gradient		
Layer		SYMBOLS
Color		ByLayer
Line thickness		0.25 mm
Text color		Blue
Text height	a	2.5
Vertical text indent	b	0.9
Horizontal text indent	c	1.5
Arrow size	d	5.0
Decimal precision		2
Text style		GOST 2.304

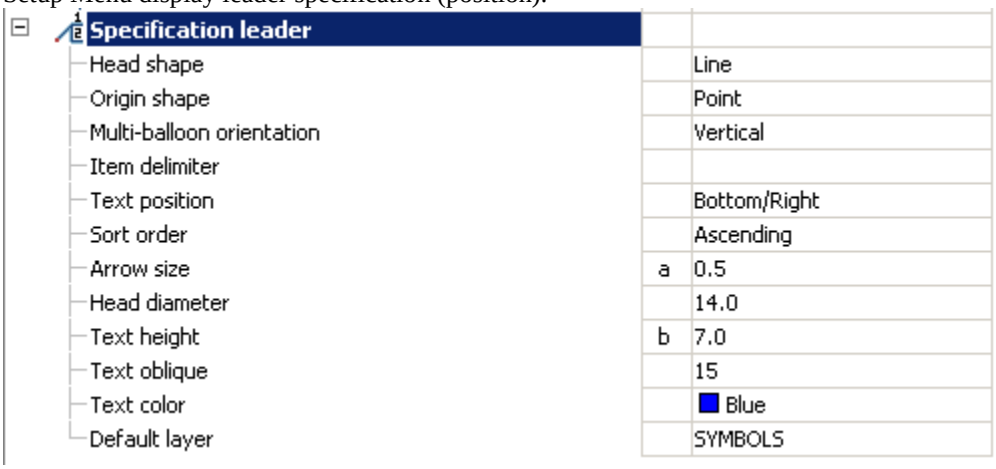
Dimension marker

The setup menu displaying the marker size.



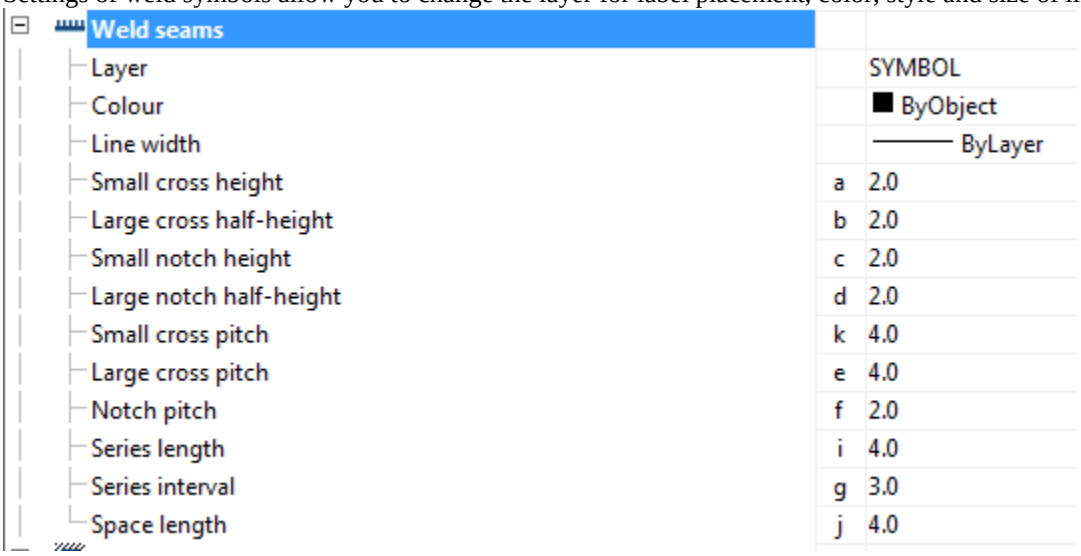
Specification leader

Setup Menu display leader specification (position).



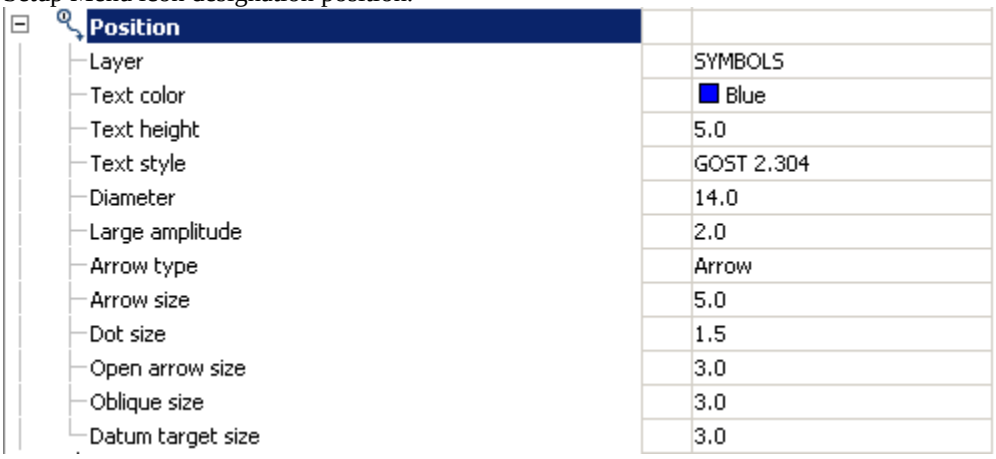
Welds

Settings of weld symbols allow you to change the layer for label placement, color, style and size of lines and text.



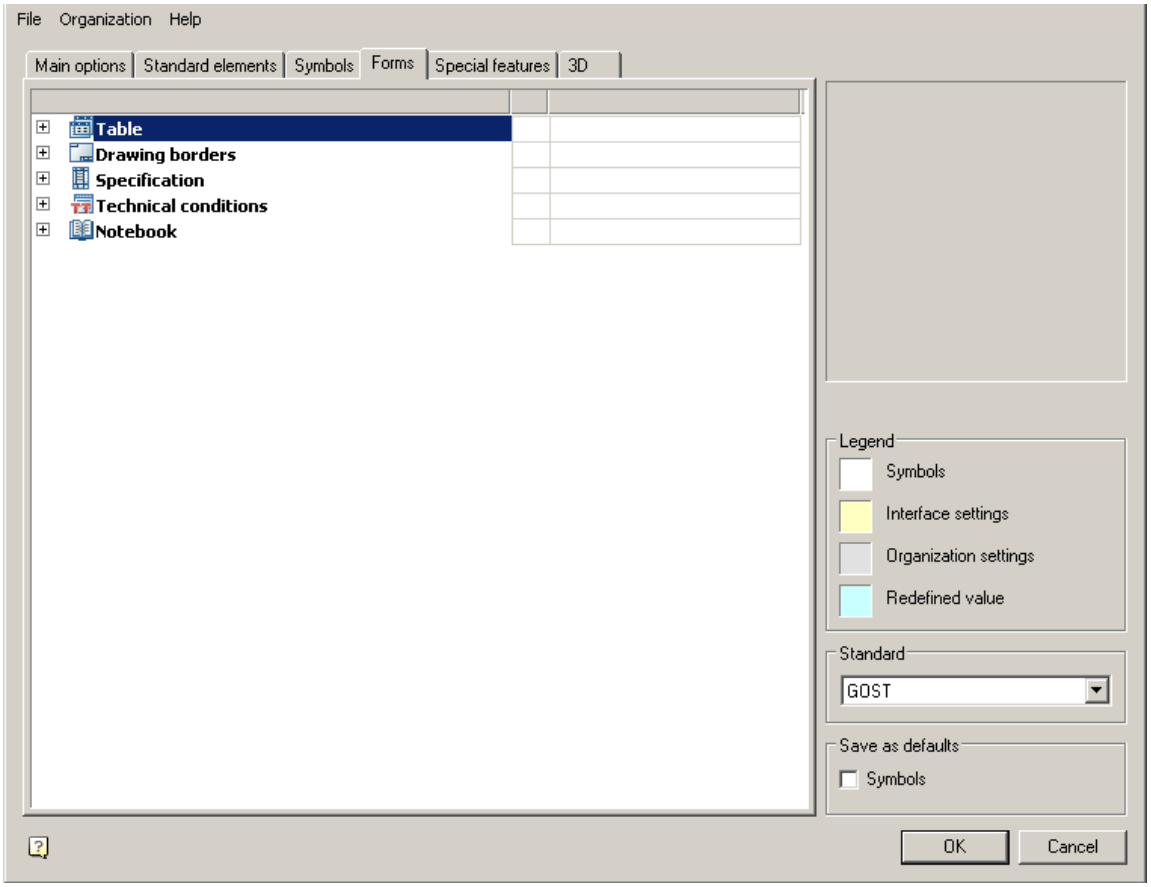
Position

Setup Menu icon designation position.



Forms

Menu tab "Forms". Designed for setting up the parameters of parts of the text tools to create a standard layout elements.

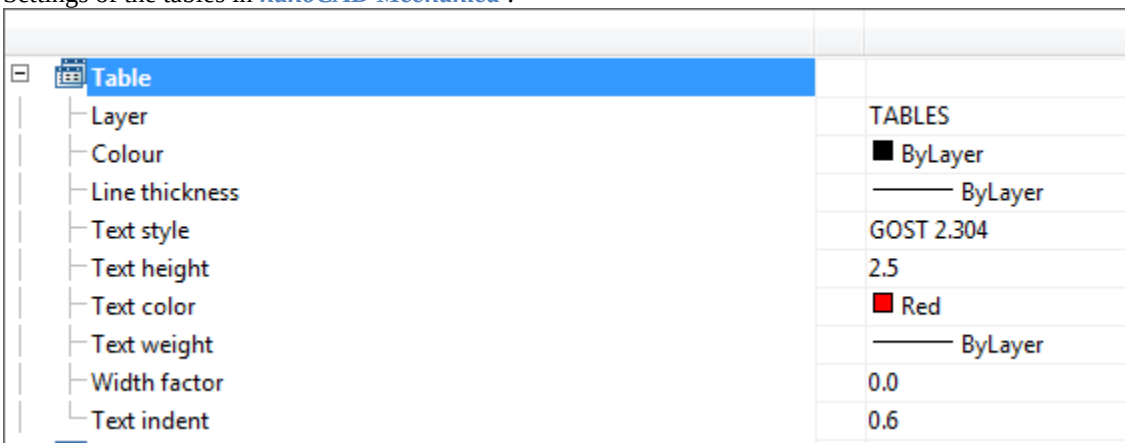


It contains the following sections:

- Table
- Drawing borders
- Specification
- Technical condition
- Notebook
- Export specification to MS Office

Table

Settings of the tables in ***nanoCAD Mechanica*** .



Layer

Specifies layer for placing tables.

Color

Sets color of table elements.

Line thickness

Allows you to choose the thickness of the lines of text in the table.
Option "by layer" sets the line width from the layer it is placed (default value).
Option "By block" sets the line width for by the block in which it is included.
Option "By object" sets the line width individually using properties palette.
Selecting "Default" sets the default line thickness for the table.
You can also set the line width of a standard set of values.

Text style

Sets the default text style in the table

Text Height

Allows you to choose the standard text height.

Text Color

Control choice of color of text in tables.

Text weight

Allows you to choose the thickness of the lines of text in the table.
Option "by layer" sets the line width from the layer it is placed (default value).
Option "By block" sets the line width for by the block in which it is included.
Option "By object" sets the line width individually using properties palette.
Selecting "Default" sets the default line thickness for the table.
You can also set the line width of a standard set of values.

Width factor

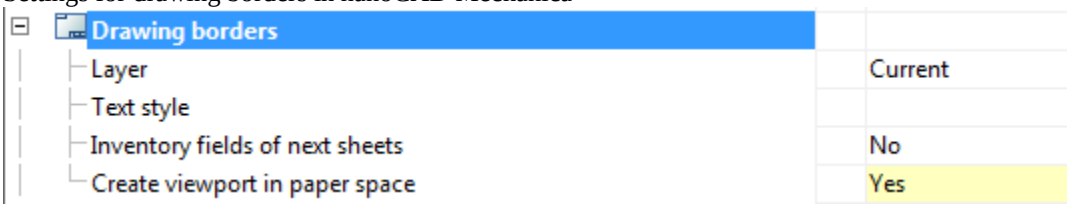
Sets width factor for text in table cells.

Text Indent

Sets text indent in table cells.

Drawing borders

Settings for drawing borders in nanoCAD Mechanica



Layer

Allows you to select the layer for the drawing border.

Text style

Sets the default text style within the drawing border.

Inventory fields of the next sheets

Enables placing of inventory title blocks on drawing border.

Create viewport in paper space

Creates a viewport when drawing border is inserted on the sheet.

Specification

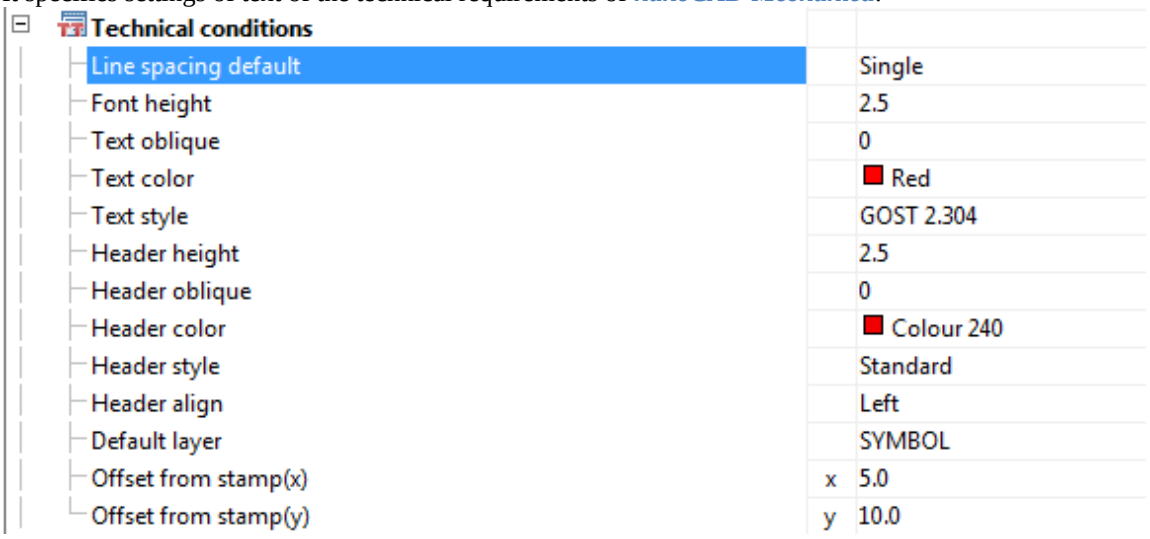
Set of tools for configuring the module specifications ***nanoCAD Mechanica 20.1***.

Default style

Allows you to choose the format specification: a simple, group type A (after the constant data placed separate sections for each performance), group type B (for each AC-used "tion use a separate column of the specification).
For more information on setting specifications here.

Technical conditions

It specifies settings of text of the technical requirements of ***nanoCAD Mechanica***.



Line spacing default

Sets the default line spacing in the text of the technical requirements.

Font Height

Sets a default text height.

Text oblique

Sets a default value of the slope of the text in degrees.

Text Color

Sets color of text in technical requirements

Text style

Sets the default text style specifications.

Header height

Sets a default value of header height in technical requirements.

Header oblique

Sets a default value of the header oblique angle in degrees.

Header Color

Sets color of header

Header Style

Sets the default header style.

Header align

Aligns the heading of technical requirements for the title block.

Default Layer

Sets the standard layer to accommodate the technical requirements.

Designations indicate areas near to the position

Automatically inserts lettering area next to the item number.

Offset from title block(x)

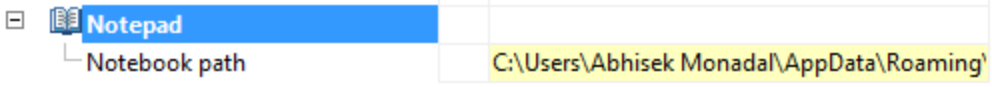
Sets the value of the indentation from the left edge of the title block horizontally.

Offset from title block(y)

Sets the offset value from the main vertical lettering.

Notepad

Settings for notebook in ***nanoCAD Mechanica 20.1***.



Path to the notebook

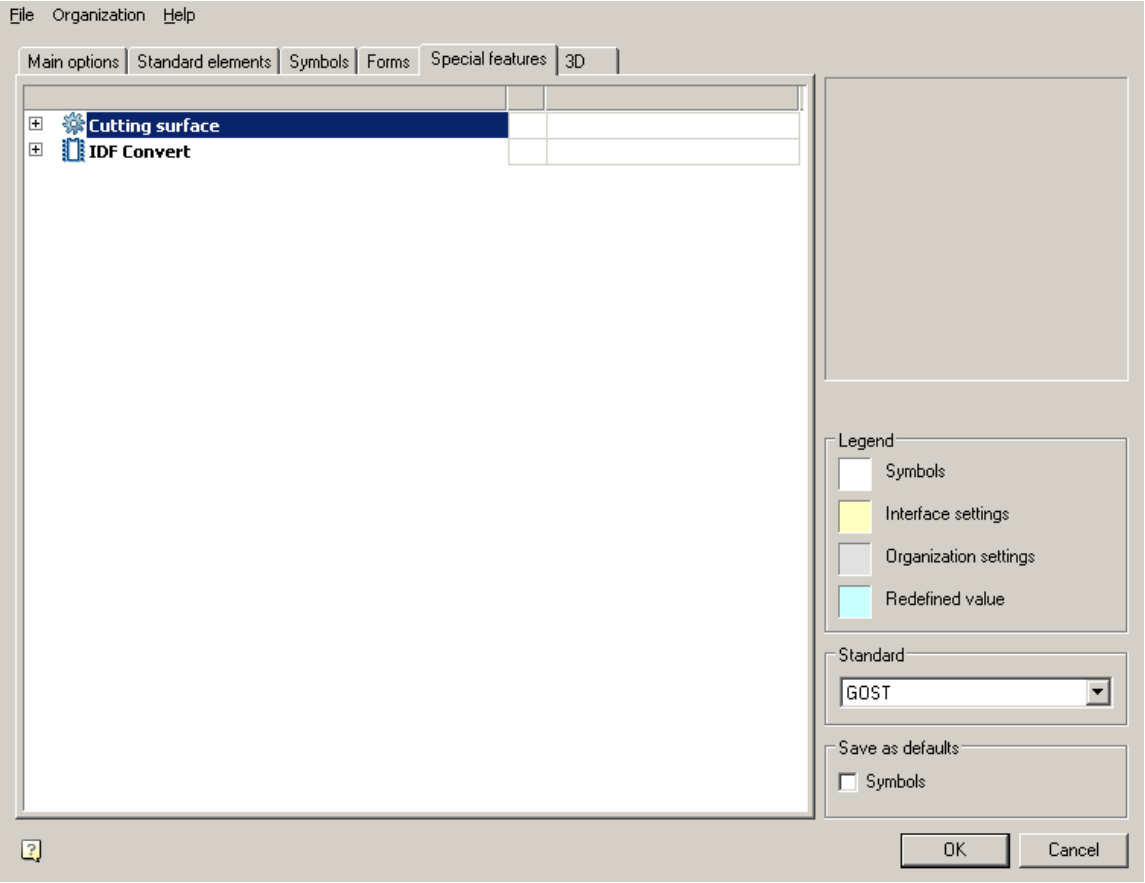
Specifies the path to the notebook.
Additional information on configuring the notebook here.

Export specification to MS Office

Setting the path to xml which will be exported specification

Special settings

The tab "Settings" menu. Designed for setting up the parameters of the auxiliary functions ***nanoCAD Mechanica 20.1***.



It contains the following sections:

- Cutting surface
- IDF Convert

Cutting surface

Complex tools to customize the display surface to be treated in the drawings *nanoCAD Mechanica 20.1*.

Color

Tool selection control line color, indicating the surface.
The color display can be selected from a standard set and set of built-in your palette.

Line thickness

Sets the standard thickness of the line marking the surface. Typically, twice the thickness of the base line.

Layer

Set the layer to accommodate the line marking the surface.

IDF Convert

Set of tools for setting the inserted printed circuit boards in the IDF format *nanoCAD Mechanica 20.1*.

Capacitor, Default, Resistor colors

Tool selection control color of individual elements of printed circuit boards.
The color display can be selected from a standard set and set of built-in your palette.

Draw drilled holes

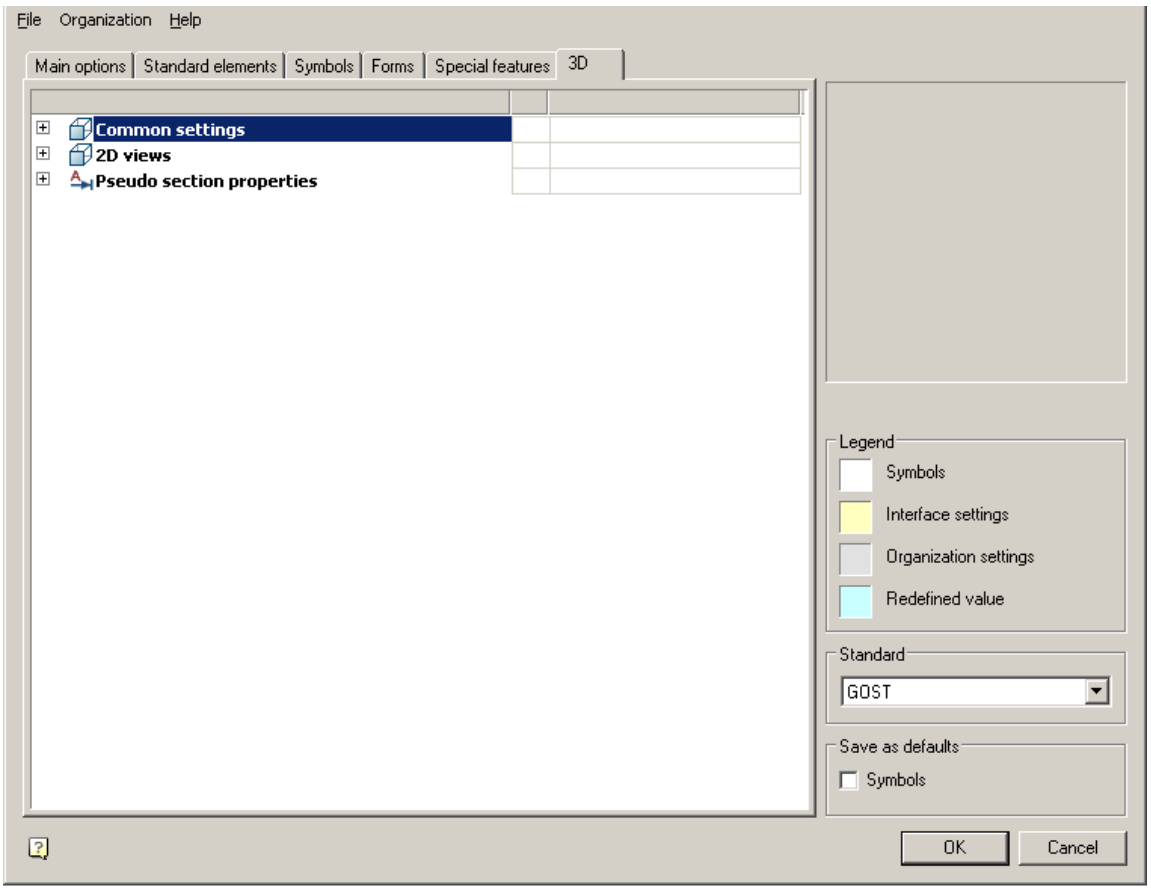
Shows or suppresses the holes in the printed circuit boards.

Convert type

It allows you to choose the type of the inserted printed circuit board that is displayed in the tree object manager (part, subassembly, assembly).

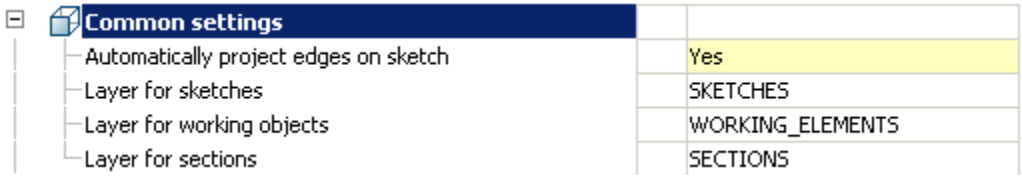
3D

The tab is used to configure 3D settings nanoCAD Mechanica. The tab is available with a license for 3D.



- Common settings
- 2D views
- Pseudo section properties

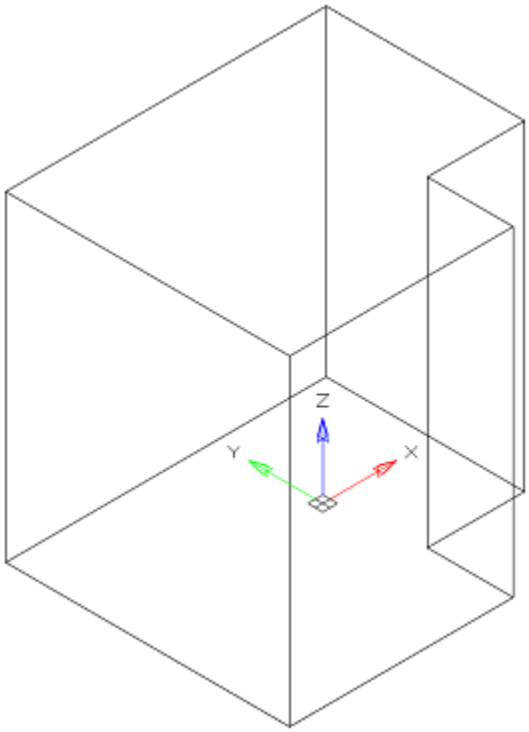
Common settings



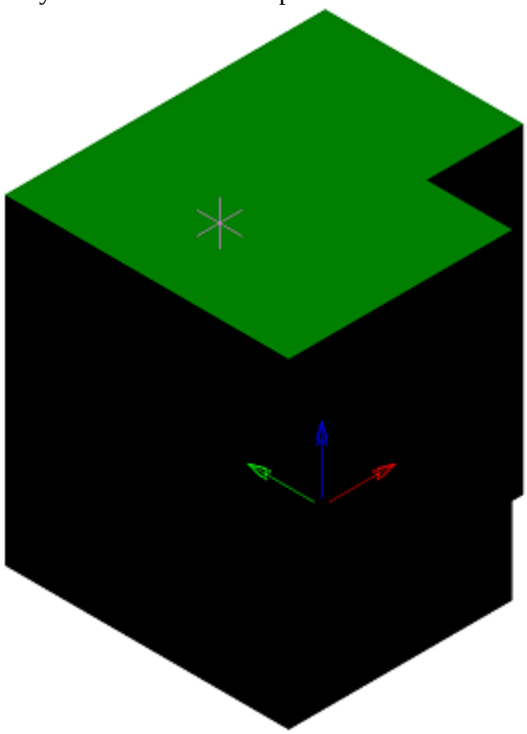
Automatically project edges on sketch

The parameter when adding a new sketch adjusts the display of the projection of the edges of a flat face taken as the working plane for the sketch.

Call command *"Add planar sketch"*.

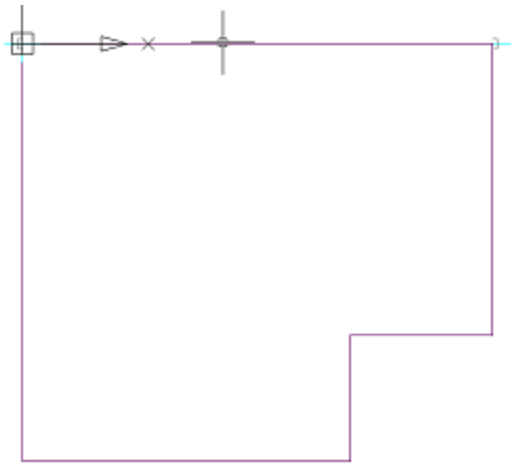
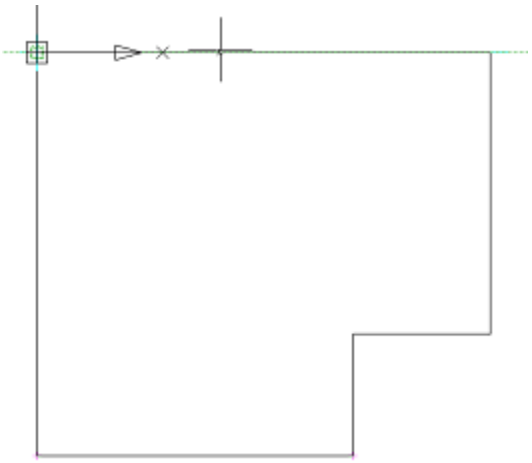


Specify a flat face as a work plane.



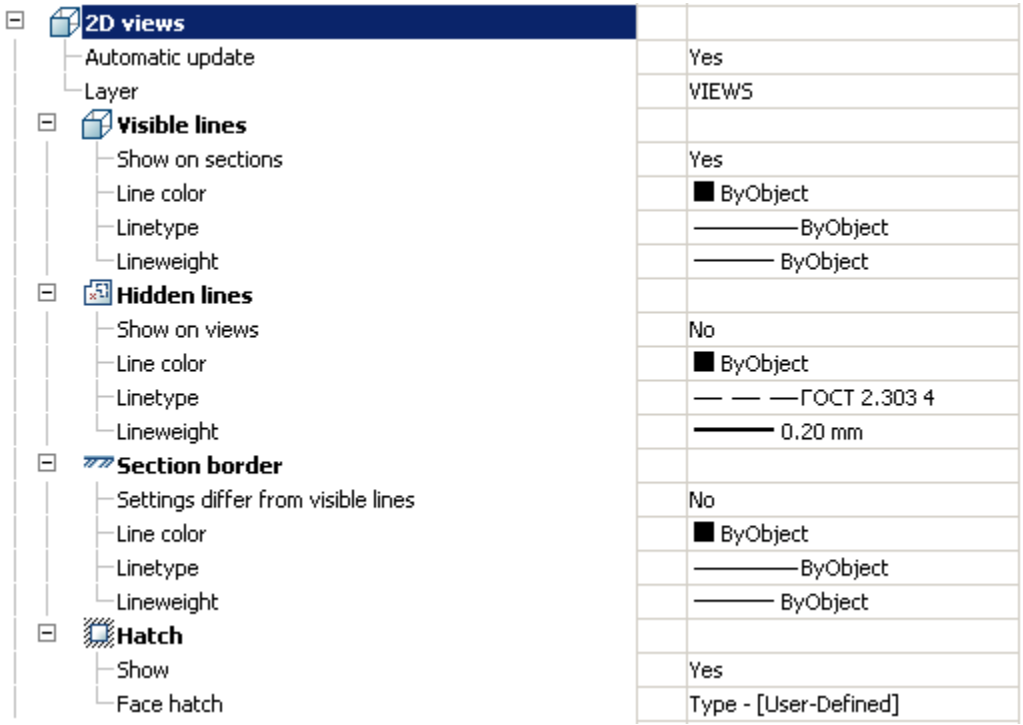
Depending on the setting, a projection will be added to the sketch.

Yes	No

Yes	No
	

Layer for sketches
It allows you to customize the name of the layer on which will be placed flat sketches.
Layer for working objects
It allows you to customize the name of the layer on which the objects will be located.
Layer for sections
It allows you to customize the name of the layer on which section will be located.

2D views



Automatic update

Sets the update mode 2D views
Layer
It defines the layer which will be located 2D views

Visible lines

Show on sections
Adjusts the image visible lines on sections
Line color
Specifies the color of visible lines
Linetype
Specify the type of visible lines
Lineweight
It determines the weight of visible lines

Hidden lines

Show on sections
It adjusts the display of invisible lines on 2D views
Line color
Specifies the color of hidden lines
Linetype
Specify the type of hidden lines
Lineweight
It determines the weight of invisible lines

Section border

Settings differ from visible
It determines whether the parameters are different boundary lines of the section visible lines
If not, the next line settings are not valid.
Line color
Specifies the color of the boundary line section
Linetype
Specifies the type of the boundary line section
Lineweight
It determines the weight of the boundary line section

Hatch

Show
It controls the display of hatching
Face hatch
Settings such as shading

3D. Pseudo section properties

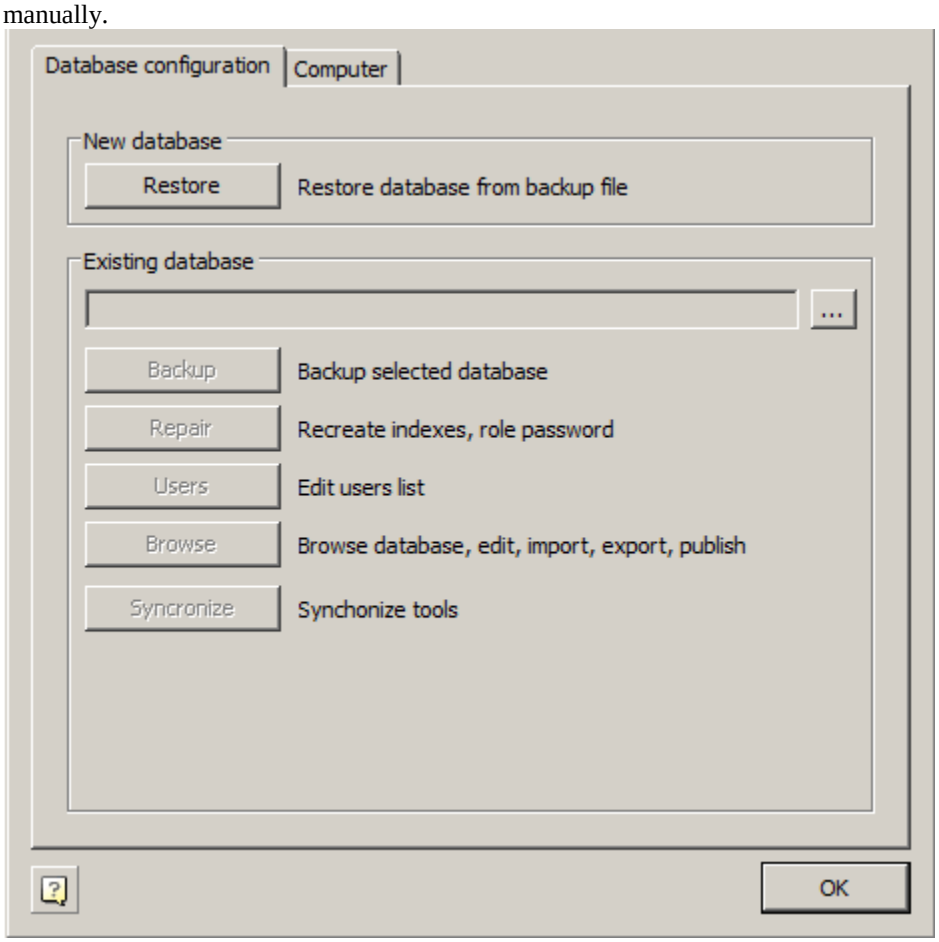
Hatch

Show
It controls the display of hatching.
Face hatch
Settings such as shading.
Surface transparency
Settings such as shading. Default 0 - full transparency.

Configuration utility

Configuration Utility

C:\Program Files\Nanosoft\nanoCAD Mechanica 20.1\mg11\McConfiguration.exe
The utility is designed to manage a database of parametric objects of nanoCAD Mechanica 20.1.
The choice of a particular database depends on the size of the organization and the number of licenses used in it. If the application is used in a large organization that uses a lot of licenses nanoCAD Mechanica, it is advisable to store the database on one server(MSSQL). Any change to the database will be immediately available to all users. If you have multiple licenses in your organization, or you can not maintain the server, it is advisable to use local databases without using a server. The configuration utility allows you to connect files *.mcs - for win32 and *.mdf - for win64 (on Windows Vista and higher). Updating the database on different machines will have to be done manually.



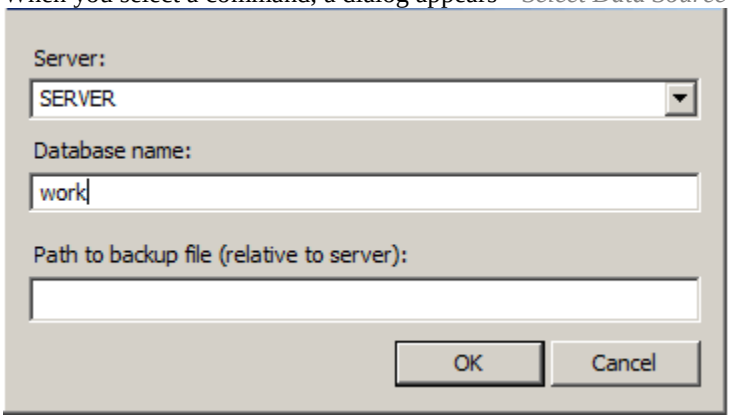
Creating and restoring the database

Restore

Creates the database MS SQL Server from a backup created earlier. File *"std.mdf"* from the distribution as an archive does not fit.

For initial creation of a database it is possible to use a file of a backup of a database published on ***spds.ru***.

When you select a command, a dialog appears *"Select Data Source"*.



In this dialog, specify the machine name and the name of the Instance of SQL Server on this machine,which is supposed to restore the database. The desired values can be entered manually, or choose from a list of SQL Server present on the local subnet.

Important!	To restore a database, you have access to it and have sufficient rights established by the administration of SQL Server

In the *"Database"* must specify a new database name standard nanoCAD Mechanica 20.1, and then click *"OK"*.

In field *"Path to backup file (relative to server):"* appears enter the relative path to the database. It must be a local path (UNC is not supported).

Note that when restoring a database on the server (different machine from the one that runs the configuration utility), database backup file must also reside on the server. This is necessary to ensure that SQL Server account have the right to recover the database from the specified file.

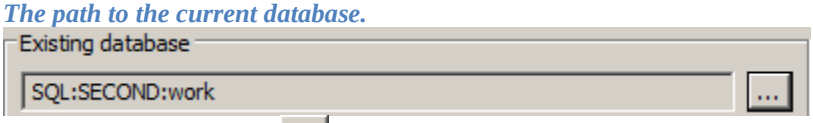
When you click on *"OK"* to start the recovery process (restore) the database.

In the menu of the current database, use the following commands:

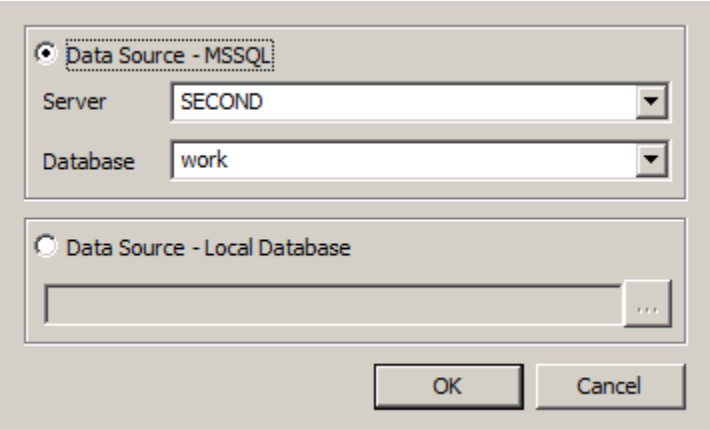
Important!	You should not use administrative tools SQL Server (Enterprise Management or SQL Management Studio) to restore the database from a file archive created by archiving of the configuration utility

Current datebase

The path to the current database.



By clicking on the button opens a dialog *"Select Data Source"* to specify the server and database.



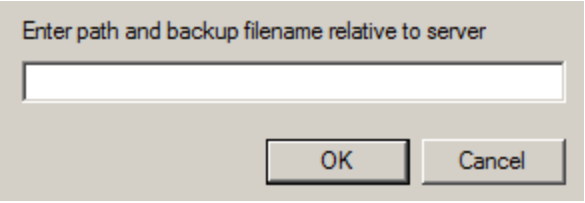
In the dialog box, you can select the type of data source:

- MSSQL - The server and database fields are filled in.
- Local base - The file on the disk is selected. The format of the local database files can be mcs(Access) and mdf(LocalDB).

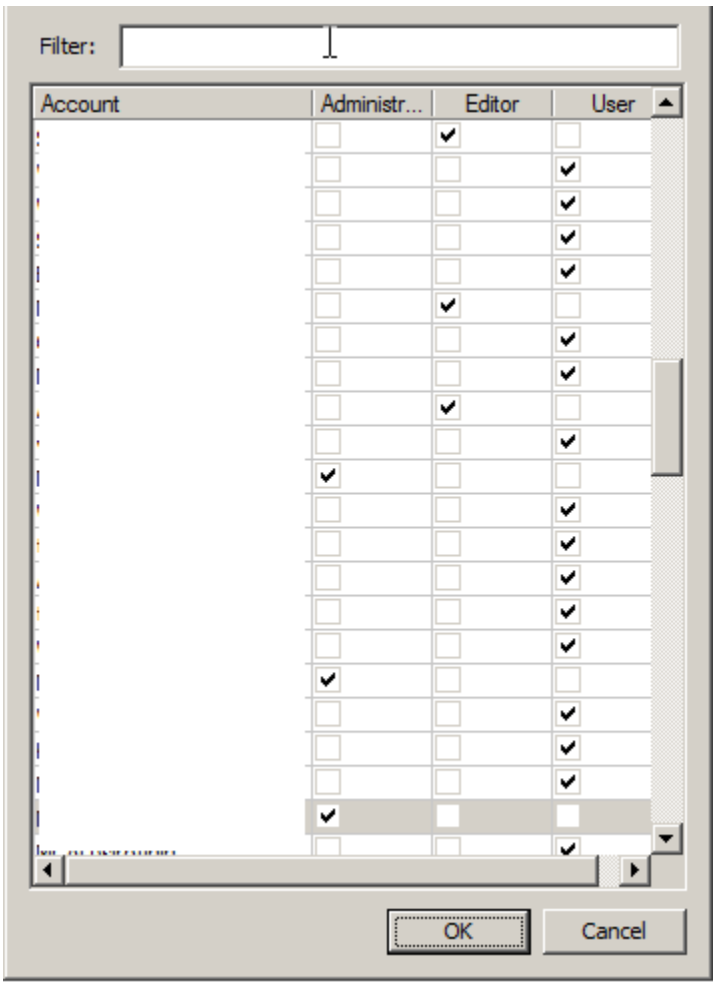
This field contains the path to the current database in the form of:
SQL:<Server Name \ Name Instance>:<database name>
The field to the local database contains the absolute path to the database file.

Important!	After specifying the database, the base control buttons will be available, depending on the type of database and access rights.

Backup
Creates a file - a copy of the current SQL Server database for subsequent recovery. By pressing the button, a dialog appears to specify a relative path to the file archive database.



Clicking on *"OK"* will create an archive file.
Repair
Fixes corrupted database.
Users
Allows you to edit user rights (to run, you need to have the appropriate rights).



The following templates are right:
User - sees only system and its objects. Edits only its objects.
Editor - sees only system and its objects. Edits only its objects. Has the right to publish its objects.
Administrator - sees all the objects in the database. Has the rights to edit all objects and publish them.
Filter selects from the list of users on the typed letters contained in the user name. The list can be sorted by attributes, for this click on the column headings.

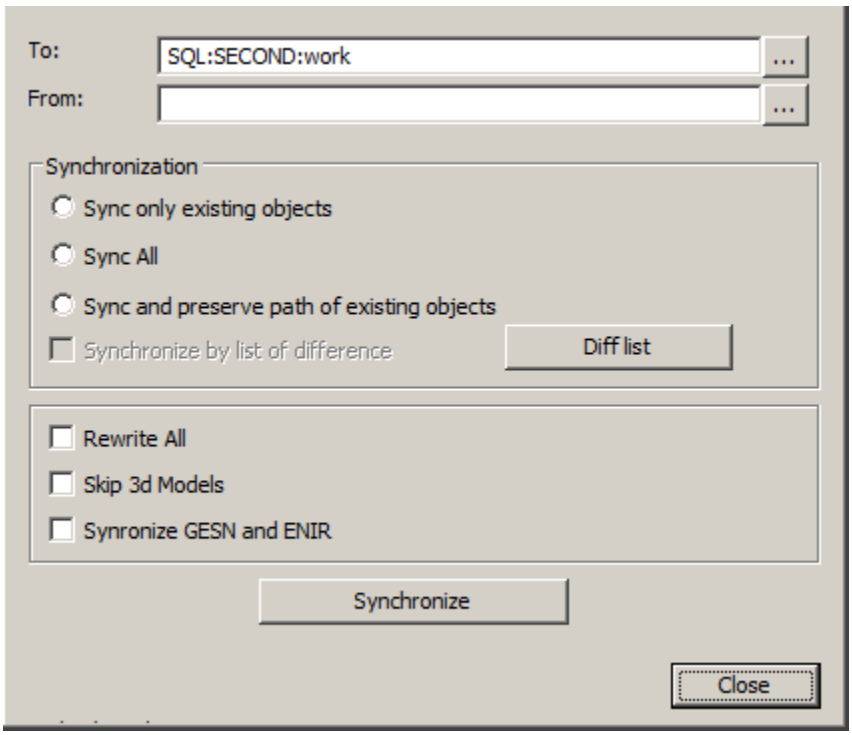
View

Starts window browser base, where you can modify the database structure:

- Delete and move folders and objects;
- Import and export objects (it is possible to import multiple files at once);
- Publication and editing objects;
- Create shortcuts;
- Renaming, editing notes, setting the preview pictures.

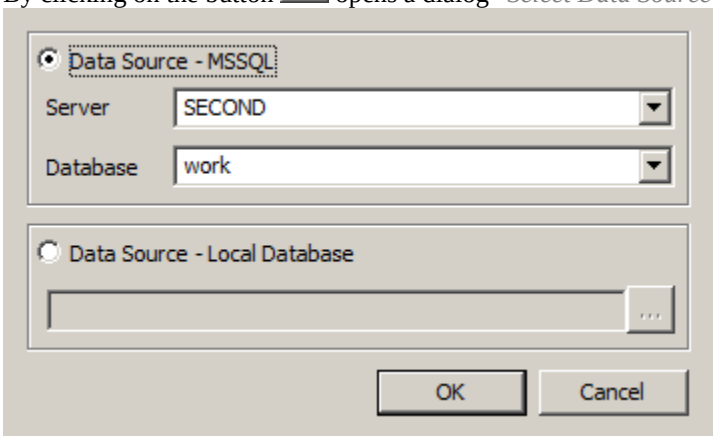
Synchronize

Open dialog *"Database Management"*.



This utility is designed to provide an update target databases from the source. Through it may be, for example, to update the local database sections or individual users with respect to a single central database Company (located, for example, on a server).
To synchronize the database to choose:
Target database - this database to be updated;
Source database - this update source.

By clicking on the button ... opens a dialog *"Select Data Source"* to specify the server and database.



In the dialog box, you can select the type of data source:

- MSSQL - The server and database fields are filled in.
- Local base - The file on the disk is selected. The format of the local database files can be mcs(Access) and mdf(LocalDB).

This field contains the path to the current database in the form of:
SQL:<Server Name \ Name Instance>:<database name>

The field to the local database contains the absolute path to the database file.
Database Synchronization:

- *Synchronize only existing objects* -in the target database will be updated only those objects that are present in the target database.
- *Synchronize all* - in the target database will be updated all the objects present in the original database (if the target object does not present, it will be created), object paths are set as in the original database.
- *Synchronize and preserve path of existing objects* - in the target database will be updated all the objects present in the original database (if the target object does not present, it will be created), the path to the objects that exist in the target database will be stored for no longer exist - will be created.

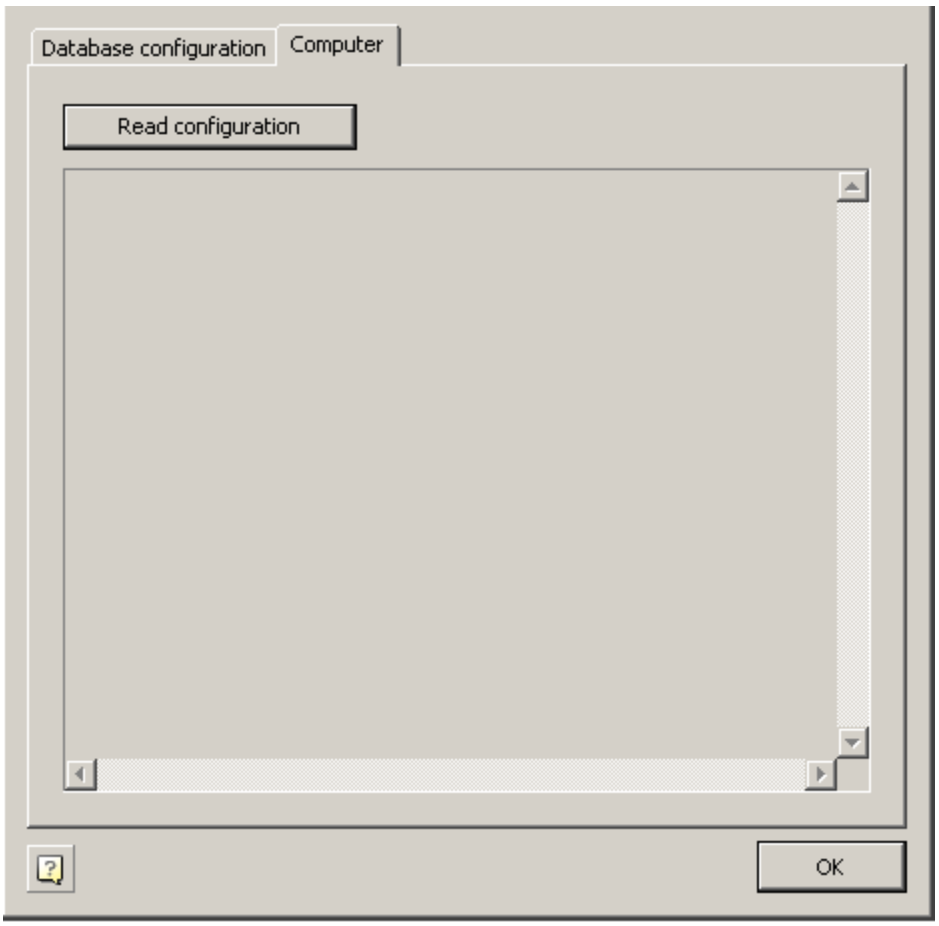
Synchronize by list of difference - synchronizes the database objects selected in the difference list.
Rewrite All - overwrites all database objects from the source database to the target database.
Skip 3d Models - do not modify the 3d model found in the database.
Synchronize GESN and ENIR - for the product "Construction Site": synchronizes the tables of the GESN and ENIR.
Button *"Synchronize"* synchronizes to the previously configured parameters. When synchronizing, it imports from the target database to the source database based on the specified synchronization parameters. When you import, only those objects will be updated, the modification date of which in the source database is later than the modification date in the target database.

Procedure for updating nanoCAD Mechanica.

On the SQL-server there is a database of enterprise details containing user-defined elements. The company received an update nanoCAD Mechanica, including an updated backup of the database.
To update the database correctly, you must:
1. Run the Configuration Utility.
2. Restore the update database nanoCAD Mechanica from a backup.
3. Press button *"Synchronize"*, a dialog will open. Analyze the list of differences of objects in the current database and in the update, for which in the *"Synchronize"* dialog it is necessary to click on the button *"Diff list"*. All necessary facilities (concerns standard objects nanoCAD Mechanica, change of which is not required) export, and after synchronization - import into an already updated database.
4. In the dialogue *"Synchronize"* select the type of synchronization *"Synchronize all"* and press button *"Synchronize"*:
4.1. User objects created in the database will be saved when they are synchronized.
4.2. Modified standard objects nanoCAD Mechanica will be merged. When synchronizing, each different item will be analyzed - if a new view is added - it will be saved, if the script is changed, for example - the date of changes in the current database and in the update will be compared, a newer script will remain. Thus, these details merge, and if they are present and different in both bases, the new ones are taken.

Configuration Utility (system data)

Tab "Computer" is designed to collect data about the hardware of the computer running the utility.



To display information simply click *"Read configuration"*.

Transfer settings to another computer

Custom settings files are located in the folder
%APPDATA%\Nanosoft\nanoCAD Mechanica 20.1\ru-RU\AllOptions.xml - configuration file nanoCAD
To transfer settings, you must copy these files to the appropriate folders, and then in the settings nanoCAD Mechanica 20.1 specify the path to the settings for the design elements.
The application database is located along the following path:
%programdata%\Nanosoft\nanoCAD Mechanica 20.1\DataRW\std.mcs
User objects stored in the local database must be exported or transferred as part of the entire database.

Important!	The function of saving the settings, when the saved file is the same as the one already existing, does not overwrite the old file, but complements it. I.e., when changing the standard, for example, from the ESKD to ISO, both standards will be available in the settings file. All changes to the profile table must be saved to the configuration file, otherwise these changes will only be displayed in the current work session.

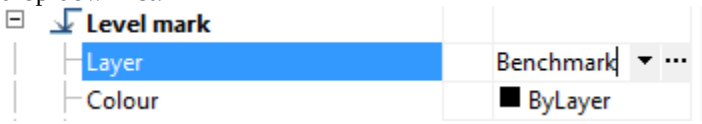
Layers and profiles

Objects nanoCAD Mechanica 20.1 when inserted, can be placed on a special layer.In this case, you do not need to pre-create the desired layer or make it the current one: the program will automatically do this regardless of the current layer.

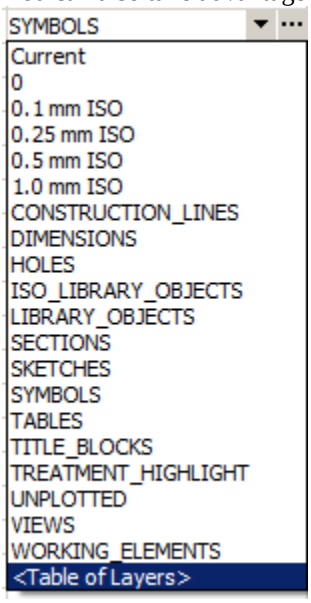
In the settings of design elements, some objects have a parameter ***Layer***.

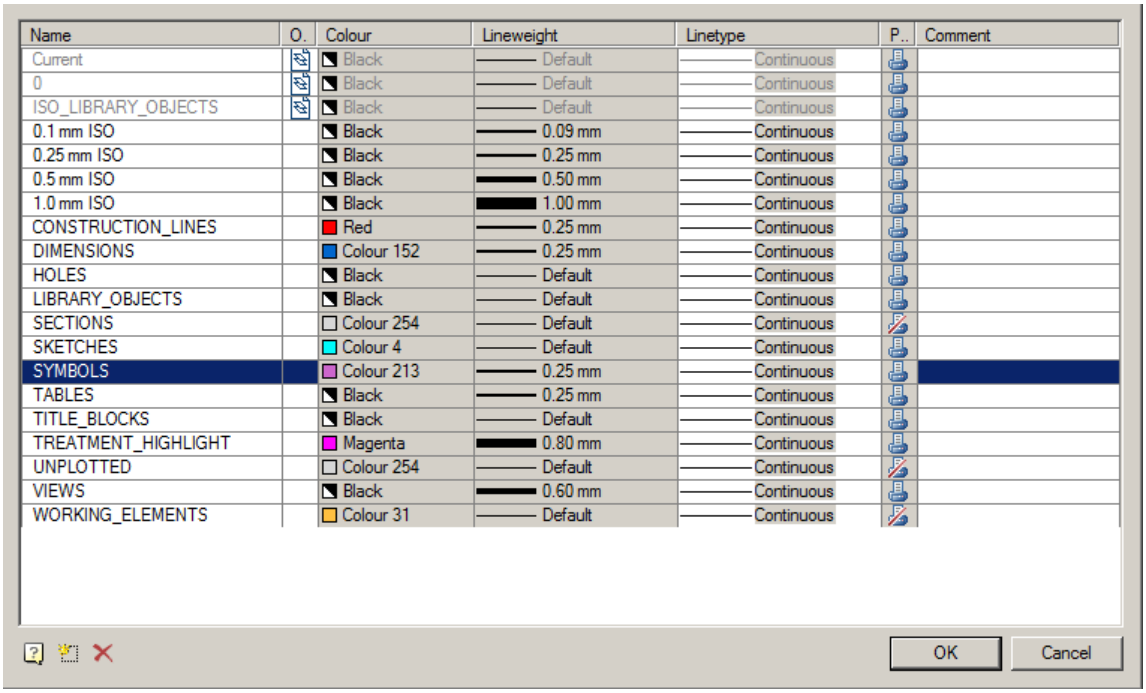
Each time an insert is inserted, the object is placed on the current layer nanoCAD.

To change the object's insertion layer, select the desired layer in the list or write its name in the input field of the drop-down list:



You can also take advantage of ***Table of Layers***, selecting an item in the layer selection drop-down list.





In the dialog box "Table of Layers" a list of the layers available for use is given.

the symbol in the "On driwing" column indicates the layers that are present in the current drawing.

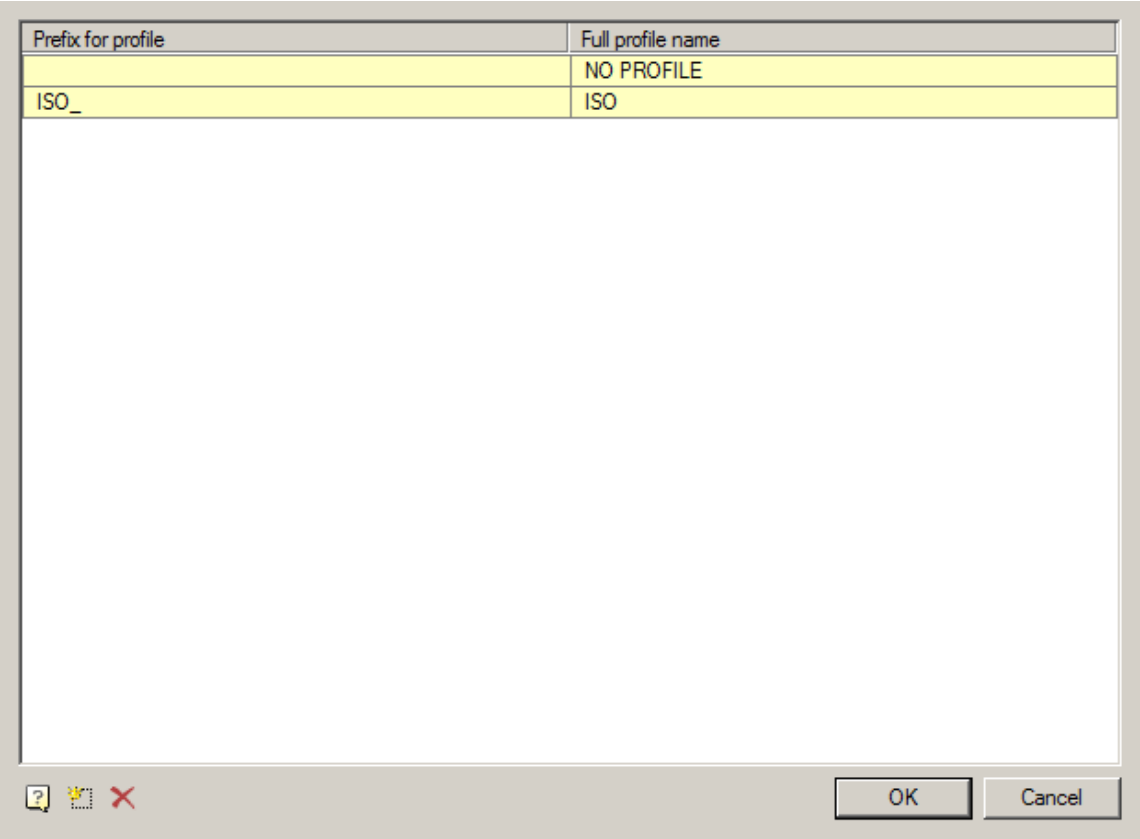
Button "*New Layer*" adds a new layer to the table.

Button "*Delete Layer*" removes the selected layer from the table.

Layer Profiles

Layer Profiles used when working together on drawings. Profiles allow you to automatically add a prefix to the names of the layers used to insert objects nanoCAD Mechanica 20.1.

To add and edit profiles, use the button ... in the parameter line *"Current profile"* dialog box *"Settings"*, tab *"Main options"*.



In the dialog box ***Table of profiles*** lists available profiles.

Button "***Add profile***" adds a new profile to the table.

Button "***Delete profile***" removes the selected profile from the table.

Add a profile, select it in the table and close the dialog with the OK button. Close the dialog box *"Settings"*.

When inserting objects nanoCAD Mechanica 20.1to the special layer, a prefix corresponding to the current profile will be added to the layer name:

Important!	The function of saving the settings, when the saved file is the same as the one already existing, does not overwrite the old file, but complements it. I.e., when changing the standard, for example, from the ESKD to ISO, both standards will be available in the settings file. All changes to the profile table must be saved to the configuration file, otherwise these changes will only be displayed in the current work session.

Overriding styles

Command line: ***MCSETGOST***.

This command is required to create (override) dimension style, text style and line types in accordance with the standards of environmentnanoCAD Mechanica for further work correctly.

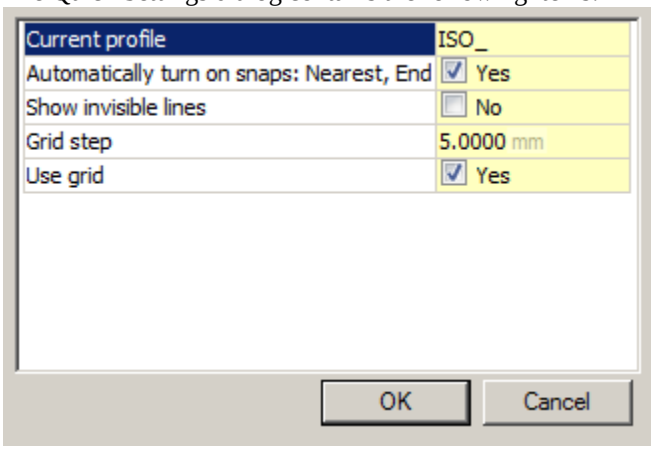
Using this command can take in two cases:

1. If the file was not created in the environment nanoCAD Mechanica
2. Dimension styles, text styles, or types of lines have been changed by hand and requires their recovery.

Quick options

Shortcuts for calling quick settings are set by setting hotkeys:

The Quick Settings dialog contains the following items:



- ***Current profile*** - indicates the current profile.
- ***Automatically turn on snaps: Nearest, Endpoint, Quadrant, Center*** - when you insert objects from the binding database, they automatically trigger.
- ***Show invisible lines*** - displays and hides invisible lines of standard parts hidden with the *"Alt+RBM"*.
- ***Grid step*** - step of changing the diameter and length of a section of the shaft with its dynamic rendering.
- ***Use grid*** - to use or not to use a grid snap when dynamically rendering a portion of a shaft.

Working with standard elements

Working with Database Objects

Parts Library nanoCAD Mechanica20.1 is a database and stored in a file with the extension * .mcs. By default, after installing nanoCAD Mechanica20.1 used base recorded in a file in the folder std.mcs data, housed in the installation directory of the program nanoCAD Mechanica20.1.
Select another database possible, specifying the desired file in the ***"Database access"***option in the settingsnanoCAD Mechanica20.1 ***"The path to the local database"***.
Parts of the database can be exported as individual files, or import from external files into the current database of standard elements.

The basic tools for working with database objects

The basic tools for working with database objects are:
"Main menu - Mechanica - Standard parts"
or on the toolbar ***"Standard parts"***

- Tab manager
- Standard parts library
- Reinsert object

- *Markers, Groups*
  - Insert group
  - Group objects
  - Add marker
  - Template wizard
- *Cover*
  - Move up
  - Move down
  - Change hiden mode
  - Regenerate supression contours
- *Part trim*
  - Add part trim
  - Remove part trim
- Add view
- Change representation
- Mirrow part
- Regenerate Std Parts
- User defined object
- Add constraint
- Connect objects

Tab Manager

Main menu: ***Mechanical - Standard parts -*** ***Tab manager***.

Ribbon: ***Mechanical - Standard parts -*** ***Tab manager***.

Toolbar: ***Tab manager (on toolbar*** "Standard parts"***)***.

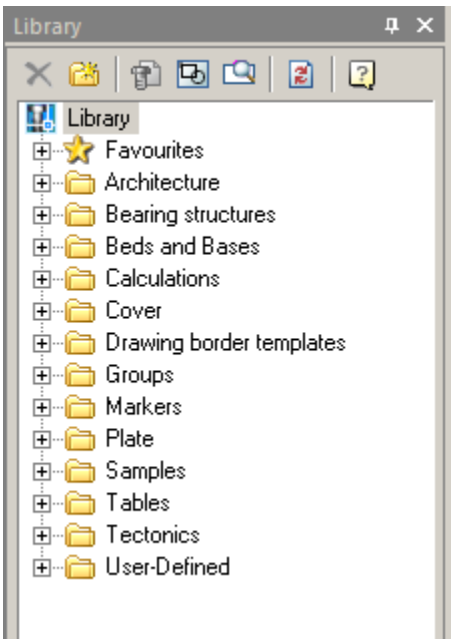
Command line: ***MCTABS***.

Object Manager nanoCAD Mechanica 20.1 includes the following tools:

- Manager database of standard objects
- Objects
- Pruning of standard parts
- Finding and replacing text
- Copying properties

Tab "Library"

Tab "Library" object manager is a universal means of access to the standard database objectsnanoCAD Mechanica20.1.
Details in the database are classified according to their design purpose.



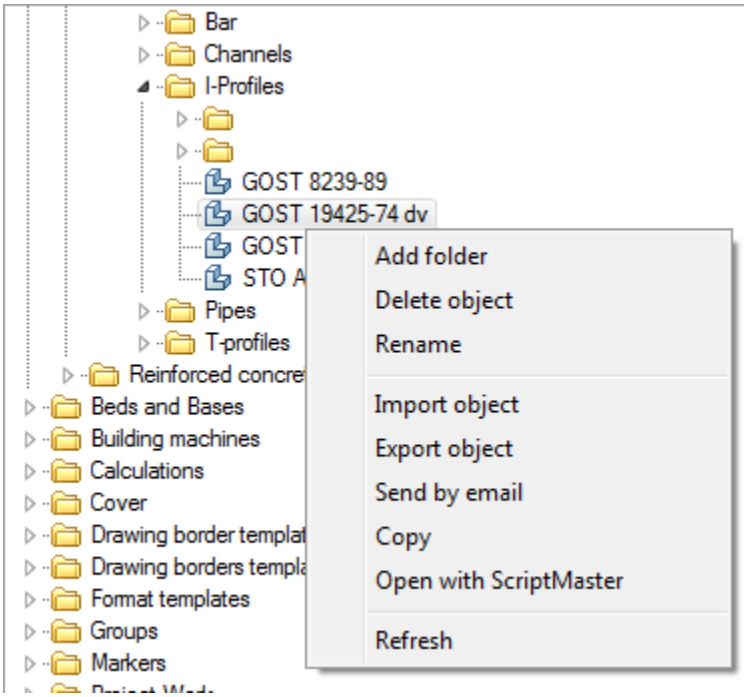
Management and structure the database by using the toolbar and the context menu that includes the ability to add, delete, or rename folders.

Toolbar

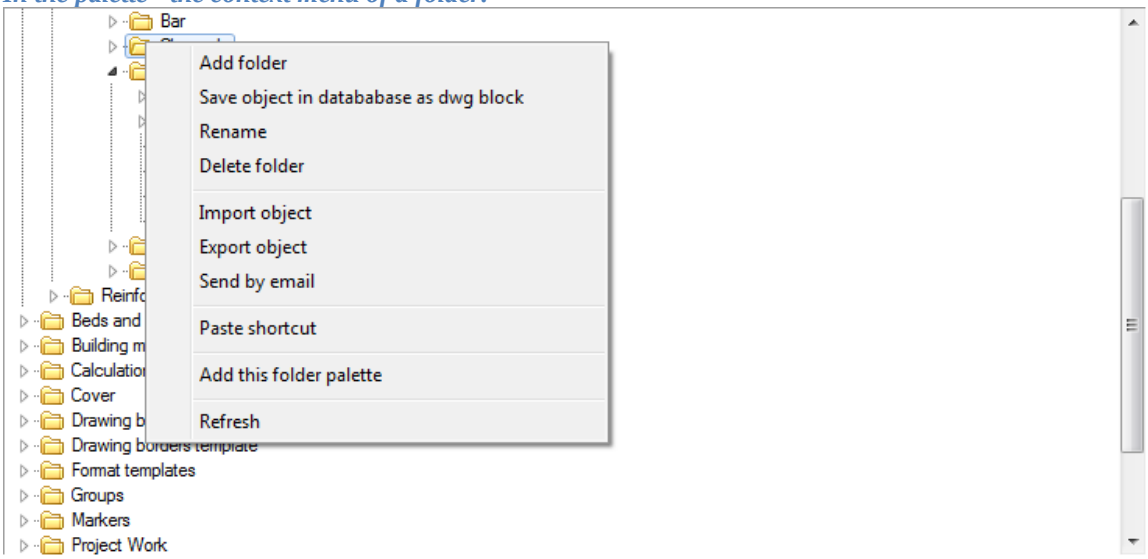
- ***Del object.*** Removes the selected item or folder from the database
- ***Add folder.*** Adds a folder to the database
- ***Insert object.*** Performs insert the selected database object in the drawing
- ***View.*** Includes preview of database objects
- ***Search panel.*** It includes search bar facilities on base
- ***Refresh.*** Updates the tree base in the object manager. Press this button after making changes to the database

Context menu

The context menu has a different structure depending on call.
In the palette - the object context menu:



In the palette - the context menu of a folder:



"Add Folder" - adds the folder to a dedicated subsidiary. If the command was run on the selected object, the folder will be added to the parent folder object.

"Save object in database as dwf block" **- This utility allows you to save a single database of the drawings in format dwg, including standard items** nanoCAD Mechanica20.1**, design elements and primitives** nanoCAD.

"Delete object" ***- This command deletes the selected object.***

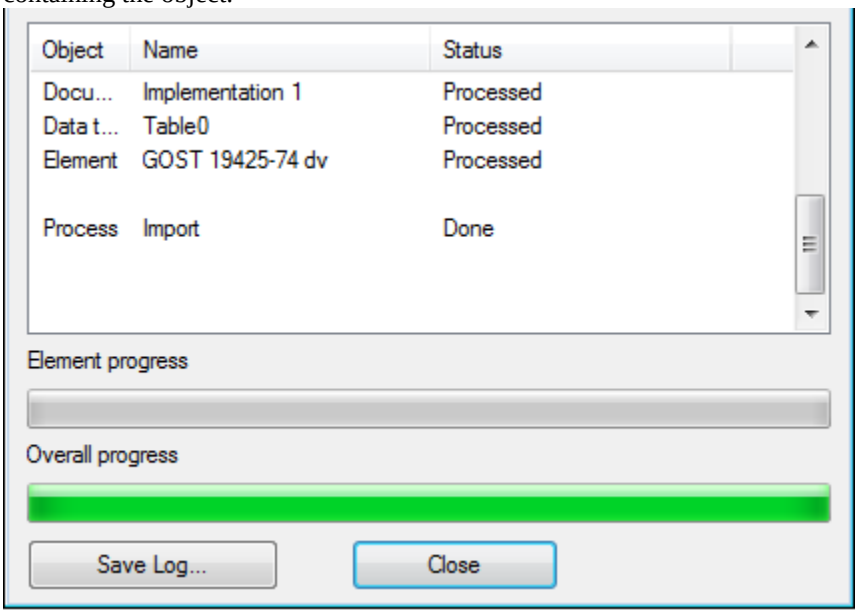
"Delete folder" - This command deletes the selected folder with all its contents.

"Rename" - The command allows you to rename an object (folder).

"Import object" - Adding to our previously exported file or folder object database DB.

To import a database from an external file, use the command "Import object" in the context menu invoked by a right-click on any folder or object in the section "Library" object manager. This will bring up a dialog box in which you specify the path to the file.

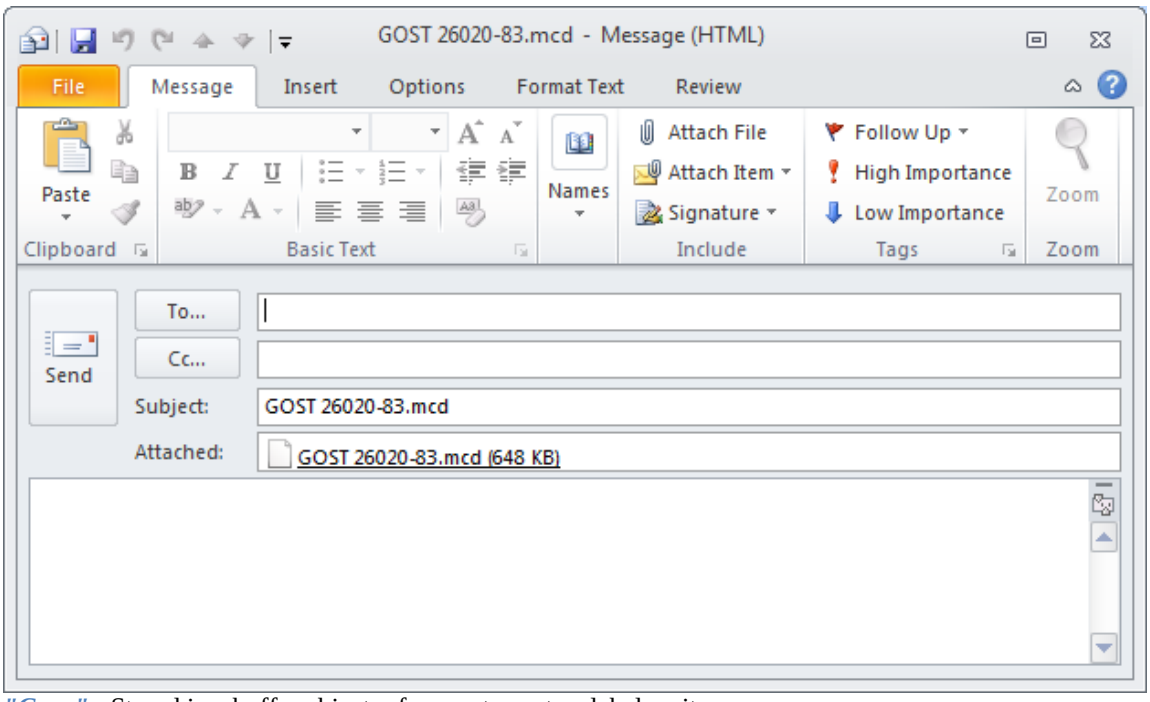
When importing compares the last modification date of the original (stored in the database) and the imported object. At concurrence of the imported content from content database objects will be updated only if they have more recent modification date.
If the command "Import object" was called for a standard lens, the outer fragment is imported to the folder containing the object.



"Export object" - Save the file to disk or folder object database DB.
To export any portion of the database you should use the command "Export object". In the resulting dialog box, then you need to specify the path and file name for saving the captured designated section base.

Note:	Import operation into the database or export from the database can be made as separate objects, and entire folders with libraries of parts.

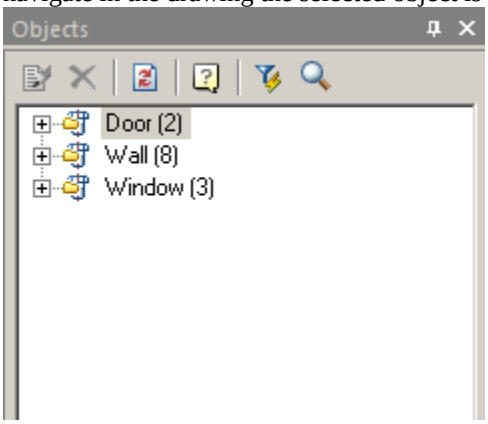
"Send by email" - It creates an e-mail with an attachment - Export objects.



"Copy" - Stored in a buffer object reference to paste a label on it.
"Paste shortcut" - Inserts a shortcut to a previously copied object
"Open with ScriptMaster" - Opens the object in the Object Wizard to edit.
"Add this folder palette" create a bookmark in the palette manager of the objects of the selected folder.
"Refresh" - Updating of the database (for example, after editing).

Tab "Objects"

Tab "Objects" object manager is a universal means of reviewing, monitoring, editing facilities nanoCAD Mechanica20.1, are on the drawing.
Places on this tab are grouped into sections - for example, standard parts, callout, bolted connections, etc. In order to navigate in the drawing the selected object is highlighted.



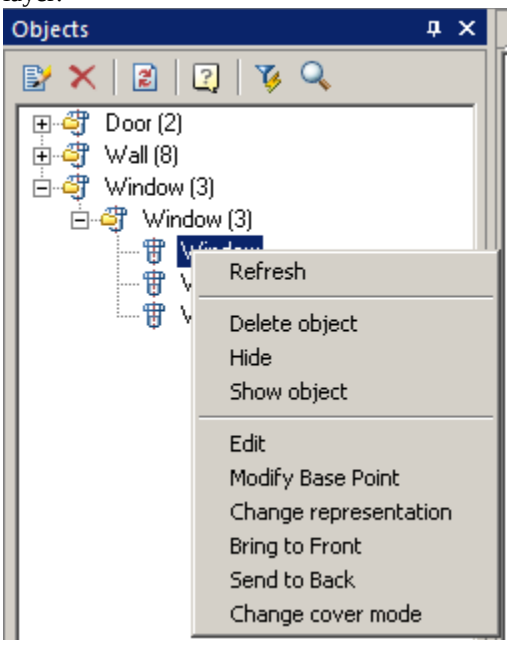
Toolbar

- *Edit object.*It calls the dialog for editing the selected object

- ***Del object.*** Deletes the selected object
- ***Refresh.*** Updates the list of objects. The button is used when adding or removing objects from the drawing
- s***Quick select.*** Search Tool nanoCAD Mechanica20.1 objects in the drawing on the set parameters, run the command "Quick Pick"
- ***Find and Replace***. Open dialog box*Find and Replace*.

Context menu

In addition to the teams that are in the toolbar, the context menu of a selected object contains the following items depending on the selected object, and includes a team of repeating the context menu of the object on the drawing. Thus the tab "Objects" allows you to edit objects without having to search for them saturated drawing regardless of layer.



For example, for sub-database nanoCAD Mechanica20.1, further include items "Change representation", "Bring to Front", "Send to Back", and "Change cover mode".
Option "Hide" makes the selected object invisible in the drawing.

Standard parts library

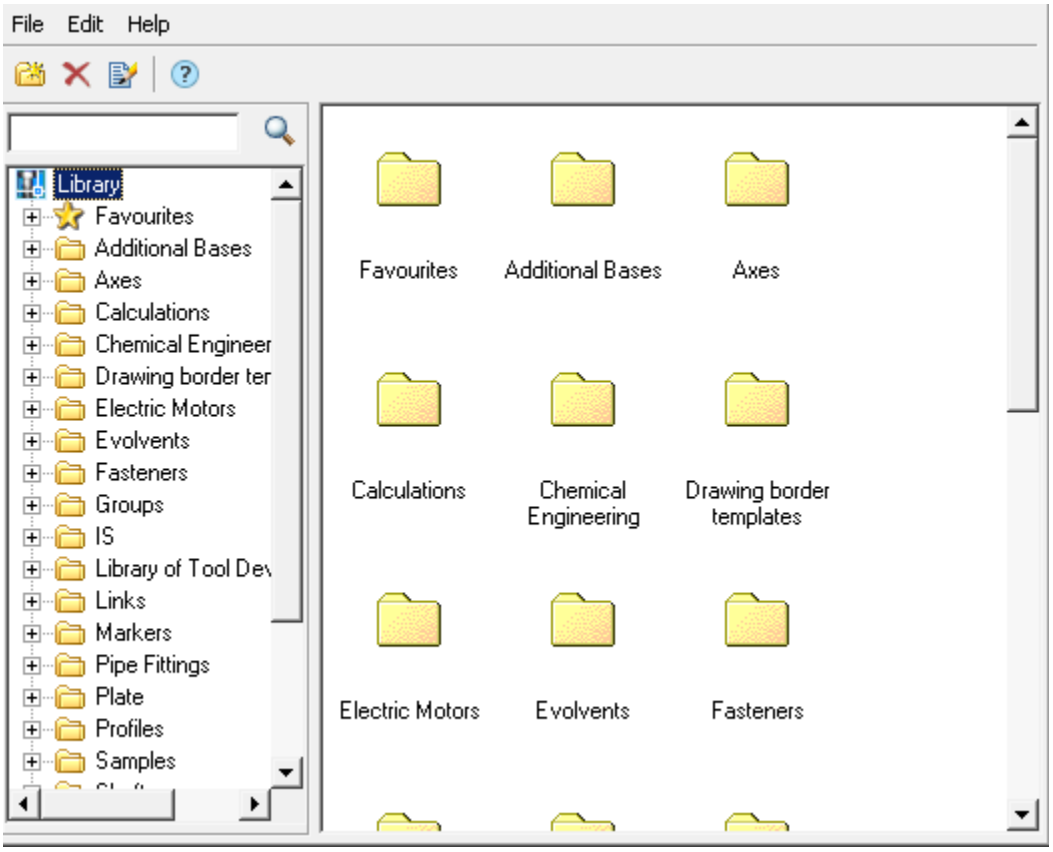
Main menu: ***Mechanical - Standard objects - Standard parts library***.

Ribbon: ***Mechanical - Library - Standard parts library***.

Toolbar: "Standard objects"***) - Standard parts library*** .

Command line: ***MCSTD***.

Inserting standard parts from the nanoCAD Mechanica 20.1 database is carried out using the database manager, which can be presented in full (dialog) form and in the form of a tool palette.



Inserting standard parts

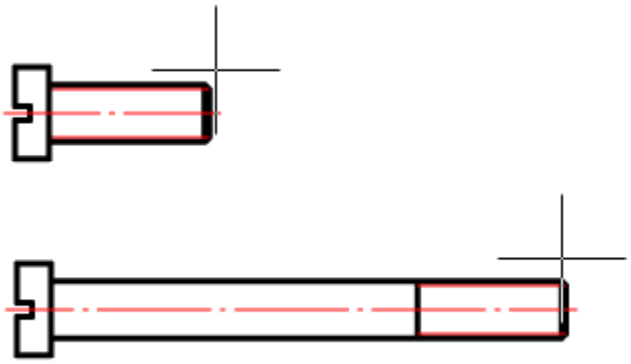
1. Select the insert object.
2. After selecting the object, specify the insertion point for the object.
3. Select the insertion direction (not all parts need to specify a direction vector). The dialog box for selecting object parameters will open.
4. Adjust the options in the dialog box.

The standard dialog contains the following sections (sections are divided into tabs):

- Table parameters
- Properties
- Dependences
- Calculations

Important!	The composition of the sections (tabs of the insert dialog) is determined by the class of the part, its parameters and the environment of the insert.

The button *"Dynamic input"* turns on the mode of visual definition of parameters when inserting.



As you move the mouse cursor, follow the change in the size of the object. LMB click to confirm the selection of the parameter value.
The *"Apply"* button allows you to apply the selected parameter values to the object without closing the dialog box. This deactivates the *"Dynamic input"* mode.

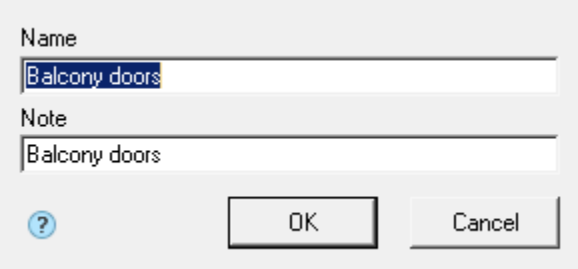
Important!	The description given here refers to the standard dialog for selecting parameters. Some objects of the nanoCAD Mechanica 20.1 base use special dialog boxes. In special dialog boxes, the location of the controls may vary.

Note:	If the path to the custom help is specified in the *"Help"* parameter of the standard part form, it will be called if the focus is on the form control. If the focus is on the *"Cancel"* or *"OK"* buttons, the help for standard items opens.

5. Click *"OK"* to complete the dialog.

Toolbar Commands

- *Add folder* - Adds a folder at the specified location in the object tree.
- *Delete* - Deletes the selected object or folder.
- *Edit note* - Changes the name and note of the selected object.



Context menu commands

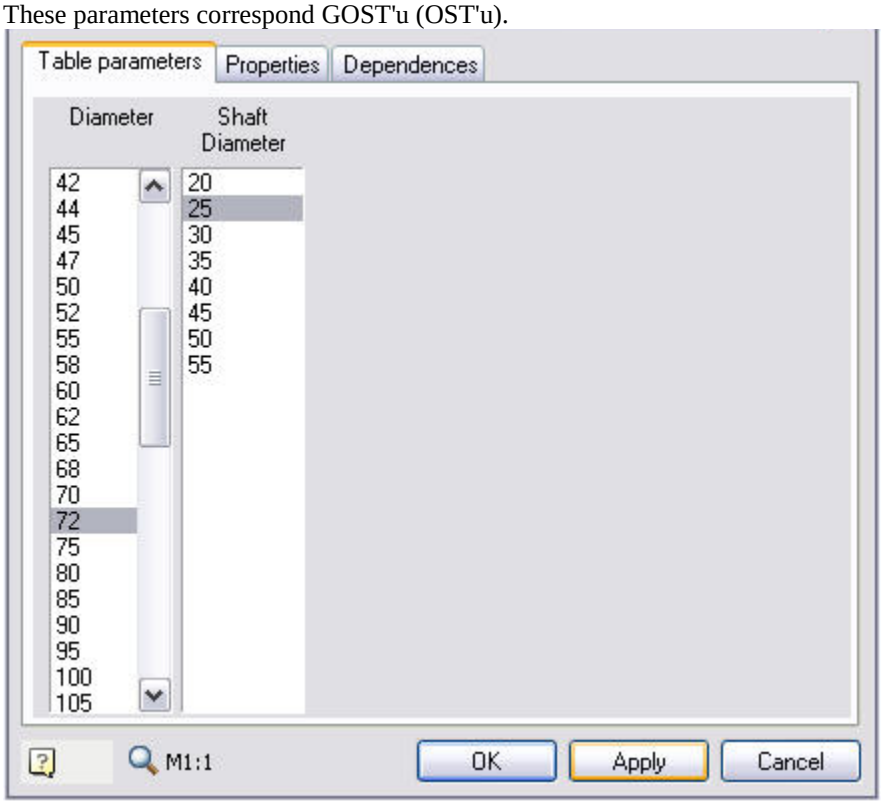
- *Import* - Imports a base object.
- *Export* - Exports a base object.
- *Send by email* - Exports the object and sends it to the specified mail address.
- *Cut* - Cuts an object, folder.

- *Paste* - Inserts a previously cut object, folder.
- *Paste shortcut* - Inserts a shortcut to a previously cut object, folder.
- *Add folder* - Adds a folder at the specified location in the object tree.
- *Delete* - Deletes the selected object or folder.
- *Edit note* - Changes the name and note of the selected object.
- *Publish*- Makes the object visible to other users of the network database. Objects can be published by administrators and editors (editors only their own objects).
- *Remove publish* - Makes the object invisible to other users of the network database. Objects can be unpublished byadministrators and editors (editors only their own objects).

Note: The *"Publish"* and *"Remove publish"* commands are only available to editors and database administrators.

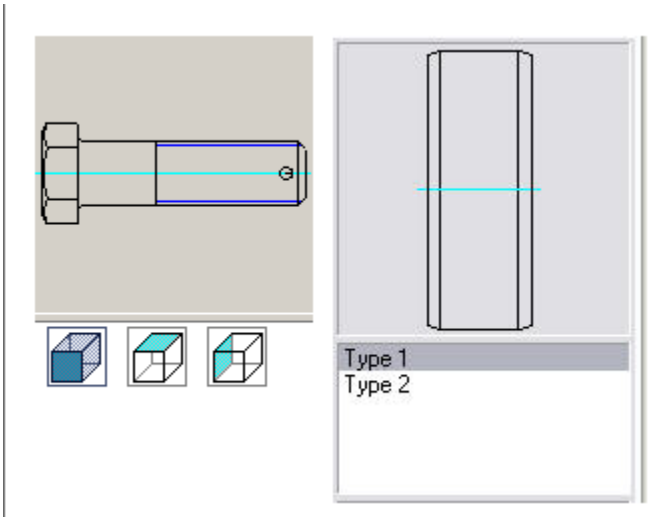
The table settings

On the "table settings" dialog insert standard parts displayed parameters responsible for the geometry of the object. These parameters correspond GOST'u (OST'u).

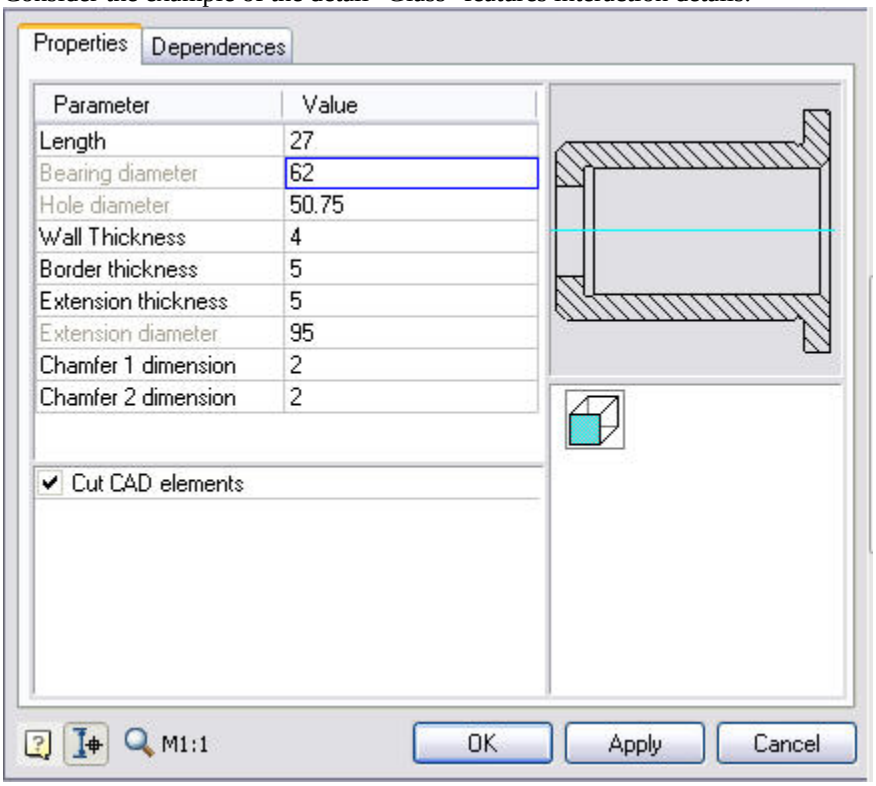


Properties

Properties of the parts that can be changed and described in the script are displayed on the "Properties". On this tab located custom and reference parameters, switches species details and details of performances.



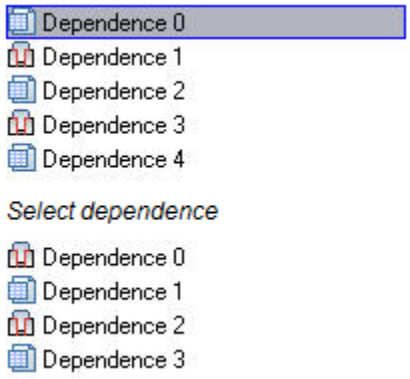
In the process of inserting parts "Glass" automatically set the relationship between the items "Bearing", "End caps" and "Glass" (with parts falling into the aperture).
For items that have parameters available for manual entry or dynamically change tab "Properties" presents additional opportunities.
Consider the example of the detail "Glass" features interaction details.



The "Bearing diameter" - gray. This says that this option is not available for editing and dependence associated with the diameter of the bearing.

Bearing diameter	62
Hole diameter	50.75

To enable editing of this option, you must go to the tab "Dependencies" and remove the dependency between the diameter of the items "Bearing" and an inner diameter of workpiece "glass.".



Remove Dependency pressing the "Delete"
After this option is available for editing.

Parameter	Value
Length	27
Bearing diameter	70
Hole diameter	50.75
Wall Thickness	4

Dependences

The "Dependences" tab allows editing of dependences set automatically during the details allocation on the drawing, and dependences set with the "Dependences control dialog" .

Types of dependencies:

Parametric - settable parameter object equality arithmetic expression object parameters, which occurs accession.

Geometrical dependences - defines the object position in relation to the object to which there is an addition.

To edit parametric dependence it is necessary to select it and set parameter names or arithmetic expressions. The Bidirectional dependence switch defines whether changes in the child object will be transferred to the parent.

To edit parametric dependence it is necessary to select it and set the value of numerical parameter dependence, view - Contradirectional or CoDirectional dependence, dependence type, and axial and surface alignment.

Calculations

For rod attachment parts provided estimation during insertion. The result of the calculation is the recommended thread diameter for given loading conditions and strength class details.
For some types of rolling bearings estimation is also provided. The result of the calculation is the life of the bearing for the given loading conditions.

Reinsert object

Main menu: ***Mechanical - Standard parts - Reinsert object***.

Ribbon: ***Mechanical - Standard parts - Reinsert object***.

Toolbar: ***Reinsert object (on toolbar*** "Standard parts"***)***.

Command line: ***MCPICKPOINT***.

Serves to re-insert the object in the drawing, combines editing commands properties and movement of item.

1. Click ***reinsert object***.
2. Select the drawing or pasted object from the database.
3. Specify the new position of the object and insertion point.
4. Click *OK*.

Markers, groups

User Groups

Working with groups in Construction 20.1 is performed using the following commands:

- Create custom groups
- Inserting User Groups

Create custom groups

Main menu: ***Mechanical - Standard parts - Markers, Groups - Group objects***.

Ribbon: ***Mechanical - Library - Group objects***.

Toolbar: ***Group objects (*** "Standard parts"***)***.

Command line: ***MCGROUPOBJECTS***.

Constraints describing the interaction of two or more objects, It is advisable to store these in the objects database in a related group. *Group nanoCAD Mechanica 20.1* has the following features:

- Group is stored in the object database, has a pattern preview before pasting.
- Components of the group in addition to objects nanoCAD Mechanica 20.1 can be primitives and blocks in nanoCAD.
- Group may be transferred as part of the database file to other users.
- Group may have additional dependencies with other groups and objects.
- Components of the group remain together parametric dependence.
- Components in the group can be specified both individually and in groups.

Thus, the group is a powerful way of organizing and structuring graphics data in a separate local file drawing document.

Create a Group

Button ***Group objects***

Gather and create database objects in the design position in the drawing. On the toolbar, ***Library objects***,***click Group objects***.

Select the required objects in the drawing AutoCAD specify the origin of the group.

In the dialog box ***New group*** on the left side is a list of objects in the group. Name objects in the list is taken from the string value for the specifications set in the properties of objects.

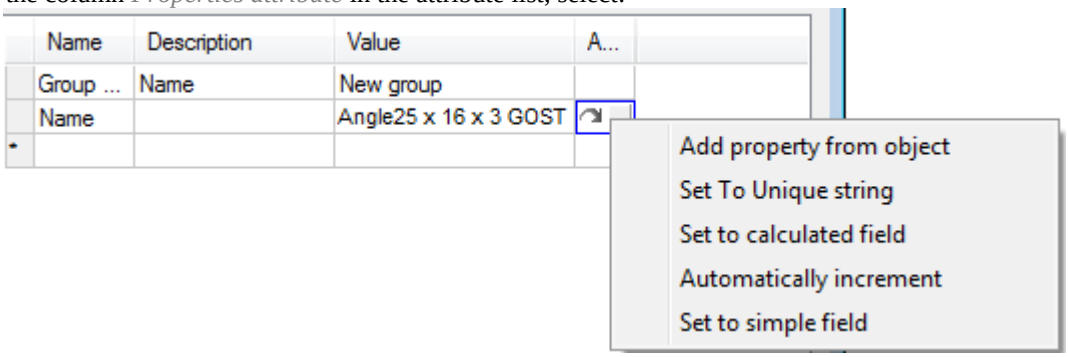
Button ***Highlight group objects*** highlights and centers on the screen objects in the group.
Button ***Set parameter table***. Opens the editor table settings. Table of parameters used to select the group attribute values from a predefined list of values.
Button ***Show form editor***. Opens *Form Editor* to create a special dialog box.
Button ***save changes to database***. Saves the base group Construction Site 20.1.
In the right part of the window picture preview and a list of attributes for the group.
Editing attributes produced in the cells of the table.

	Name	Description	Value	A...
	Group ...	Name	New group	
	massa	mass	0.05005	m.
	Name		Angle25 x 16 x 3 GOST	N.
*				

Adding attributes produced from the parameter list of objects in the group.
Expand the selected entry in the list (with the "+" next to the name of the object) and double click on the parameter, add an attribute in the table.
You can install a bidirectional relationship between the group and attribute parameter object that is associated with this attribute. The list of attributes in the column Attribute Properties , click the left mouse button on the icon .
Depending on the mode change bidirectional, it will indicate the icon . Thus, when the value of the attribute group, and the parameter of the object is changed.
Save the group of objects in the database by pressing the button ***save changes to database***. Choose a location and name for the group (the default group name matches the specified value attribute Group Name). Click ***OK***.
Group created and saved to the database. Now you can insert multiple grouped objects in a drawing, resizing and positioning of objects, while preserving established parametric and geometric constraints.

Important!	Group objects only retains information about dependencies between objects and attributes set. For insertion to be carried out correctly, you must have a database of all the objects in this group.

When working with a group of attributes of the various available methods for selecting values. In the context menu of the column *Properties attribute* in the attribute list, select:



- ***Add property from object*** - allows you to set the attribute value by linking it with an attribute value of another object in the drawing.
- ***Set to unique string*** - allows you to set the attribute-counter, which will increase its value by 1 for each subsequent inserted marker. In the Value column for the visible attribute counter must specify a character string - prefix. The drawing will be appended to the prefix value of the counter.
- ***Set to calculated field*** - allows you to set the attribute value using a mathematical expression, using references to other attributes of the marker and mathematical functions. In reference to another attribute marker use the name of the attribute recorded in braces.

- ***Automatically increment*** - allows you to set the attribute-counter, which will increase its value to 1, compared with the same attribute of the previous insertion marker. This attribute allows the counter-repetition of identical values for several markers that distinguishes it from the attribute set to a unique value.
- ***Set to simple field*** - sets the attribute type by default.

Insert group

Insert group may be carried out as the *object manager* , and with a tool ***insert group***.

Button ***Insert Group*** toolbar ***Library objects*** opens a dialog ***Box group***.
In the list of the group and press ***OK***. If you insert a group of objects , you must specify the insertion point of the group. If you insert a group of dependencies , you need to select the drawing objects that you want to associate with the dependencies.
In the dialog box ***Insert group*** commands are available:

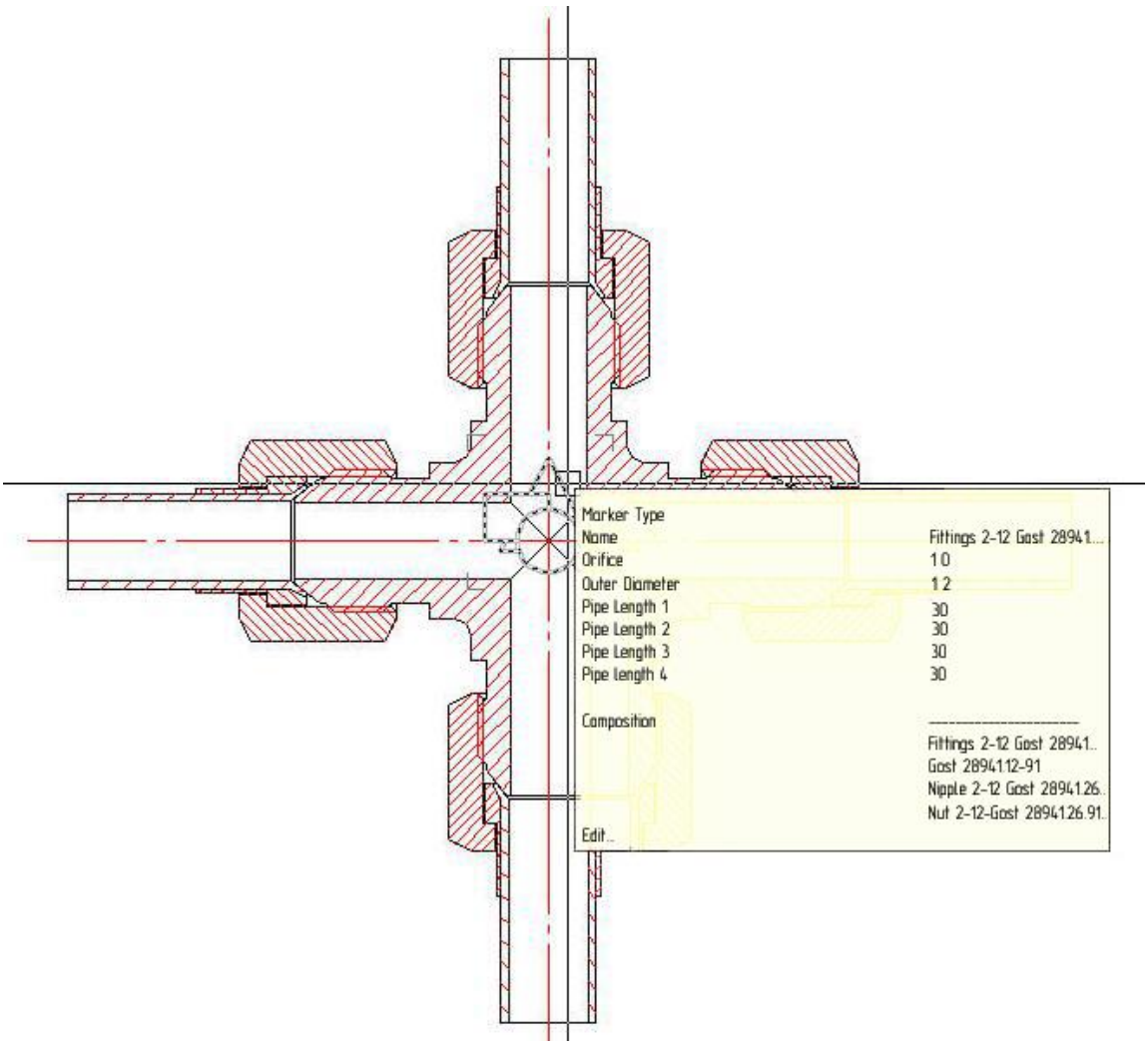
Create group. Creates a new group.

Delete group. Deletes the selected group from the database.

Select items from the drawing for the group. Used to edit an existing or new group.

Detail groups. Opens a list of groups of items.

Dependence group. Opens a list of *groups dependencies*.
Switch ***multiple*** . Enables insertion of multiple groups in the drawing.
Switch ***Add group marker***. If installed, together with a group of drawing a marker is added to the group.
Marker group is designed for group editing and data to other objects in the drawing (see *Binding arbitrary graphical and tabular data*).



Editing a Group

After you insert a group of objects in the drawing, or when the command to edit the marker group opens the edit dialog group.

Highlight group objects. Highlights and centers on the screen objects in the group.

Edit marker Properties. Includes editing mode, where you can change the set of objects in the group, add or remove attributes.

Object tree. Enables and disables the object tree in the left pane.

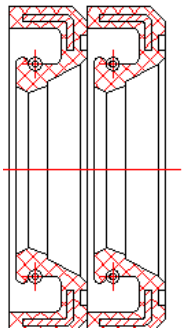
Set parameters table. This button is available if the group contains a table of parameters. In the dialog box, select the row attribute values, and click ***OK***.

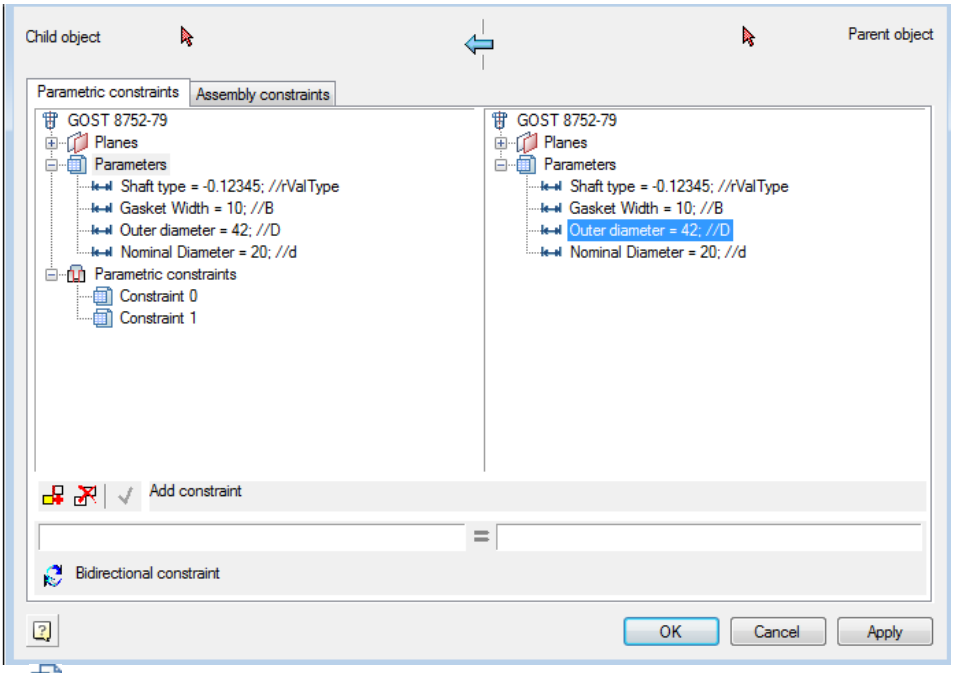
Group dependencies

Group dependencies remain parametric description and assembly relationships between objects. When you insert such groups they are not added to the drawing objects, but only established the relationship between existing objects. Consider creating and maintaining relationships between the two cuffs.

Important!	Before you create a group, you must place the drawing objects to be included in the group and set the dependencies between objects.

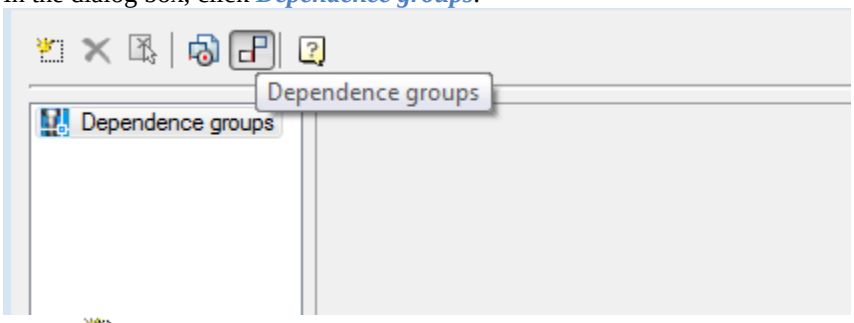
Insert the base of the two objects Cuff GOST 8752-79. Set between the geometric co-directional dependence ***Combining axis and the plane*** at a distance obj.B (see Constraints between two objects) and the parametric dependence of the outer diameter.



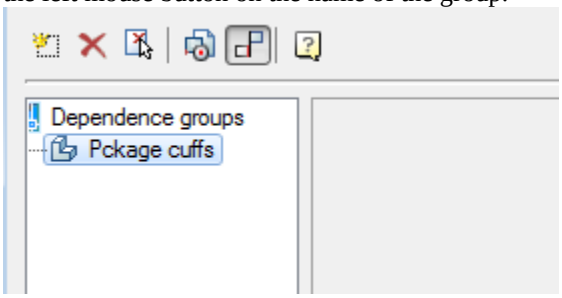


Select ***Insert group***.

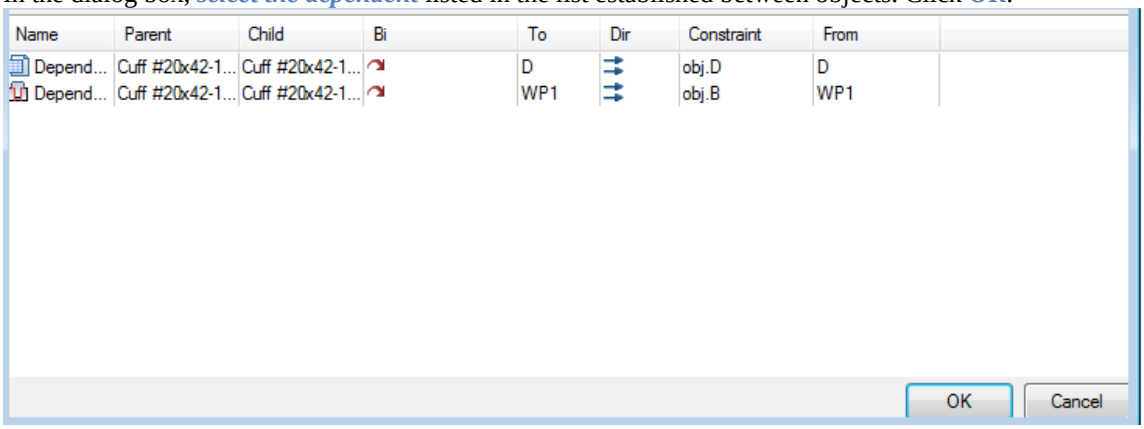
In the dialog box, click ***Dependence groups***.



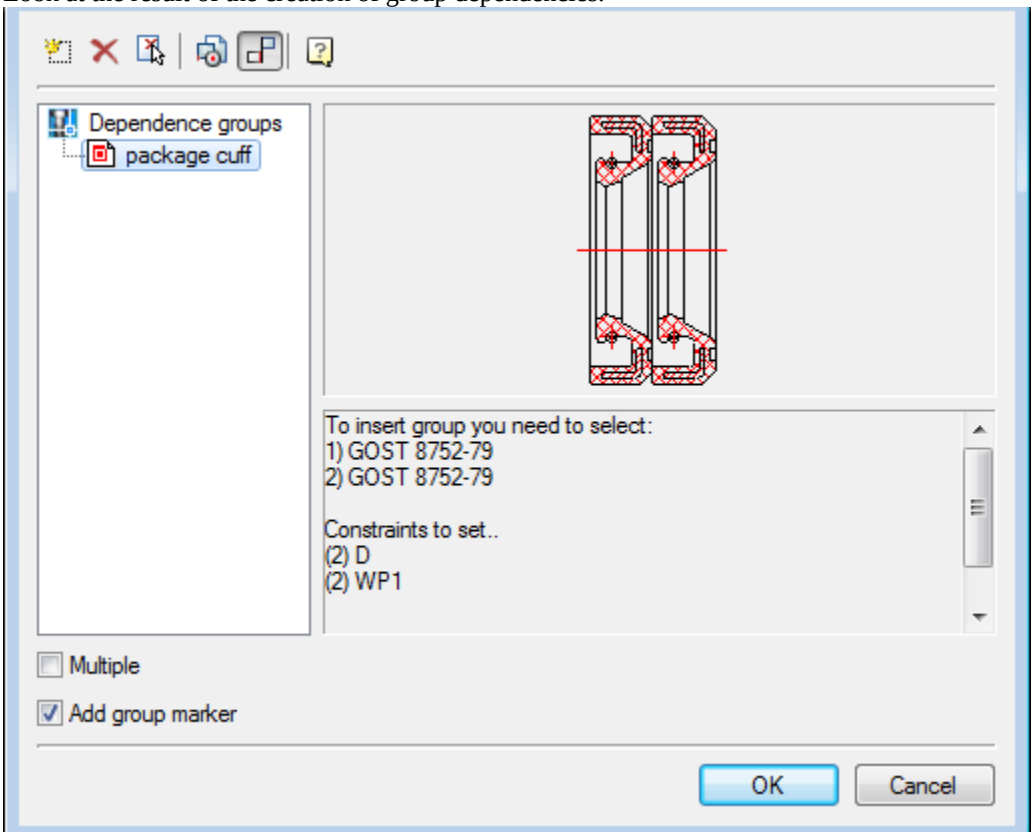
Button Creates group. In the groups list, change the name of the new group (eg, "Package cuffs"). To edit, click the left mouse button on the name of the group.



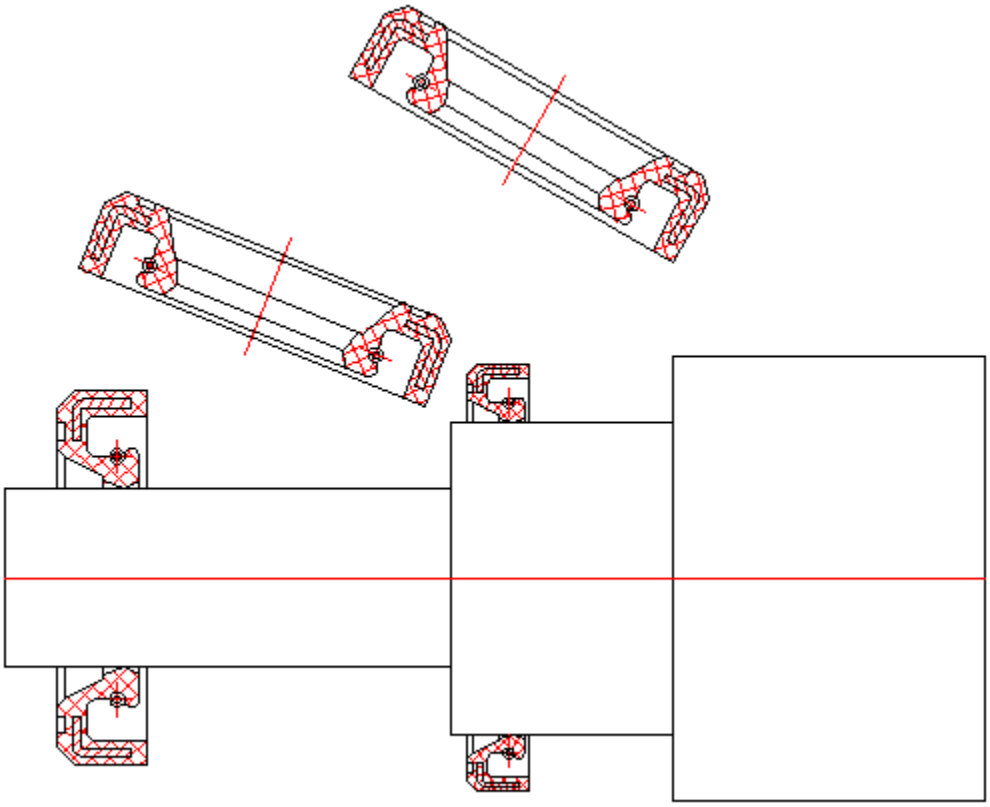
Click Select group details from drawing and select the drawing both cuffs. Press ***Enter***.
In the dialog box, ***select the dependent*** listed in the list established between objects. Click ***OK***.



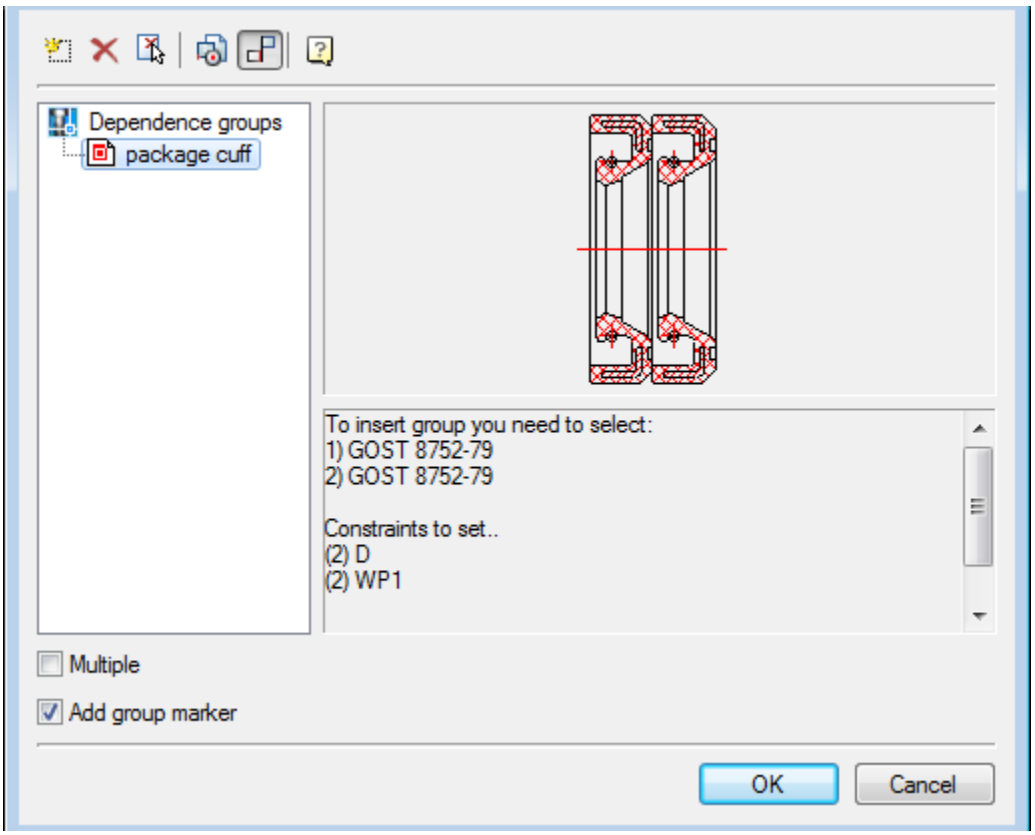
Look at the result of the creation of group dependencies.



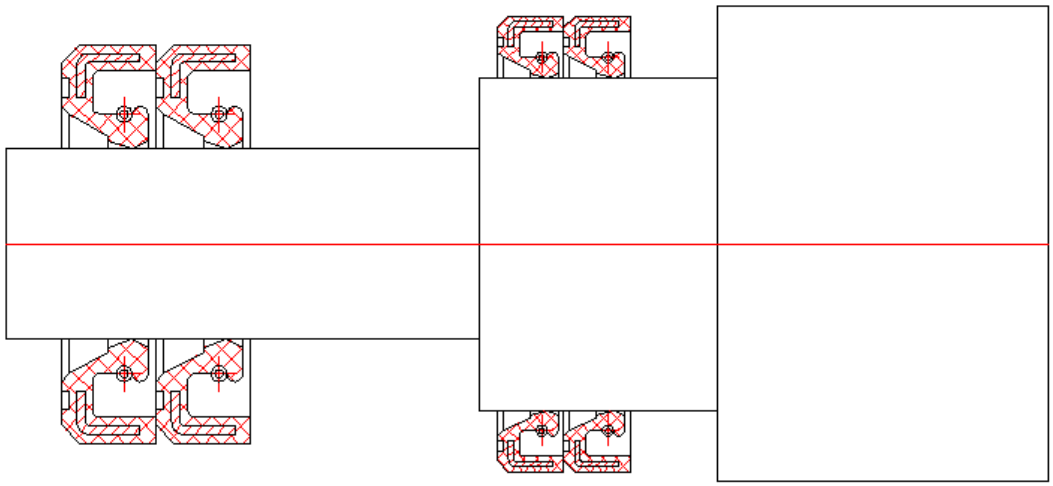
In the description of the group under the picture preview Set the selection order when inserting objects group dependencies.
Group dependencies are stored in the database automatically. Click Cancel to complete the group.
Insert a drawing of the base of the cuff, which must be applied to create a dependency.



Select ***Insert group*** , click ***Group Dependency*** . In the list of dependence " ***Package cuffs***. " Click ***OK.***



Select the drawing cuffs to be connected sequentially (in the example need to perform steps 10-11 twice).



Automatically set dependence stored in the description of the group.

Inserting Groups

Main menu: ***Mechanical - Standard parts - Markers, Groups - Insert group***.

Ribbon: ***Mechanical - Library - Insert group***.

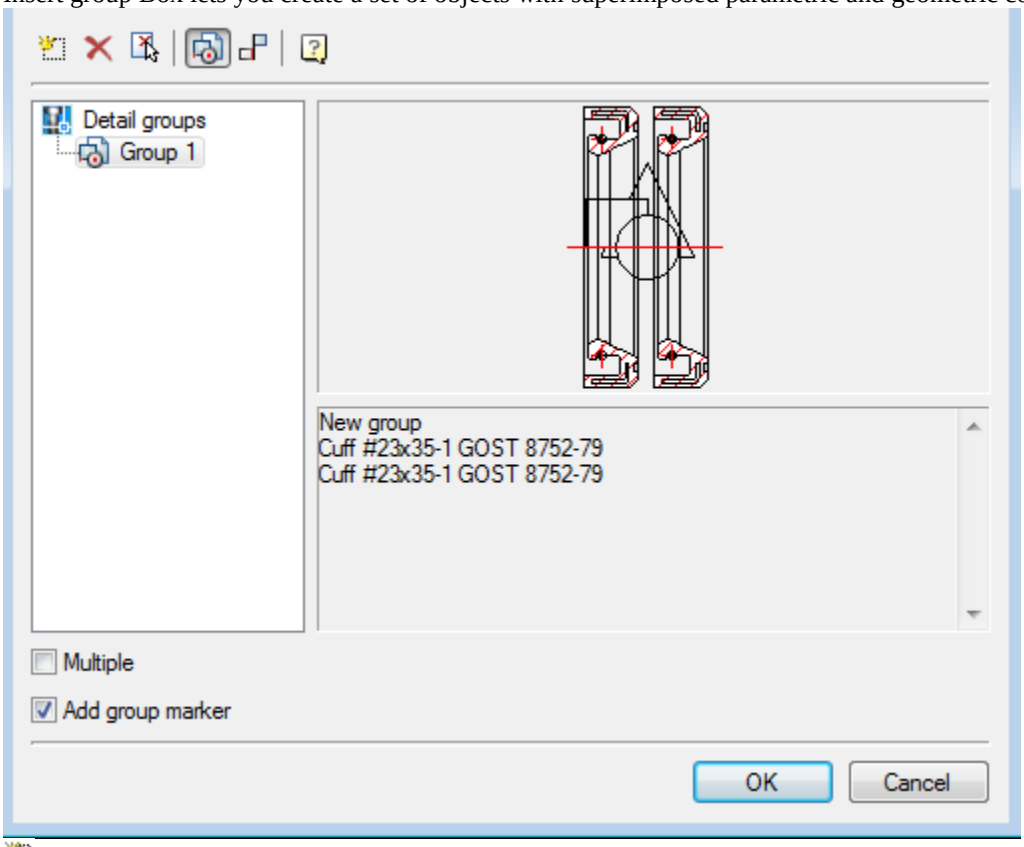
Toolbar: ***Insert group (*** "Standard parts"***)***.

Command line: ***MCINSERTGROUP***.

Library: ***Groups***.

Command Options

Insert group Box lets you create a set of objects with superimposed parametric and geometric constraints.



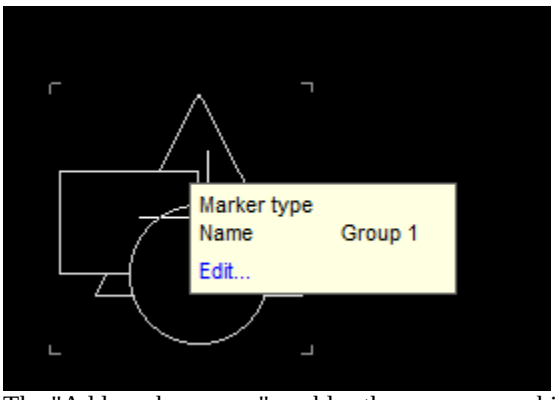
Create group

Deleting Group

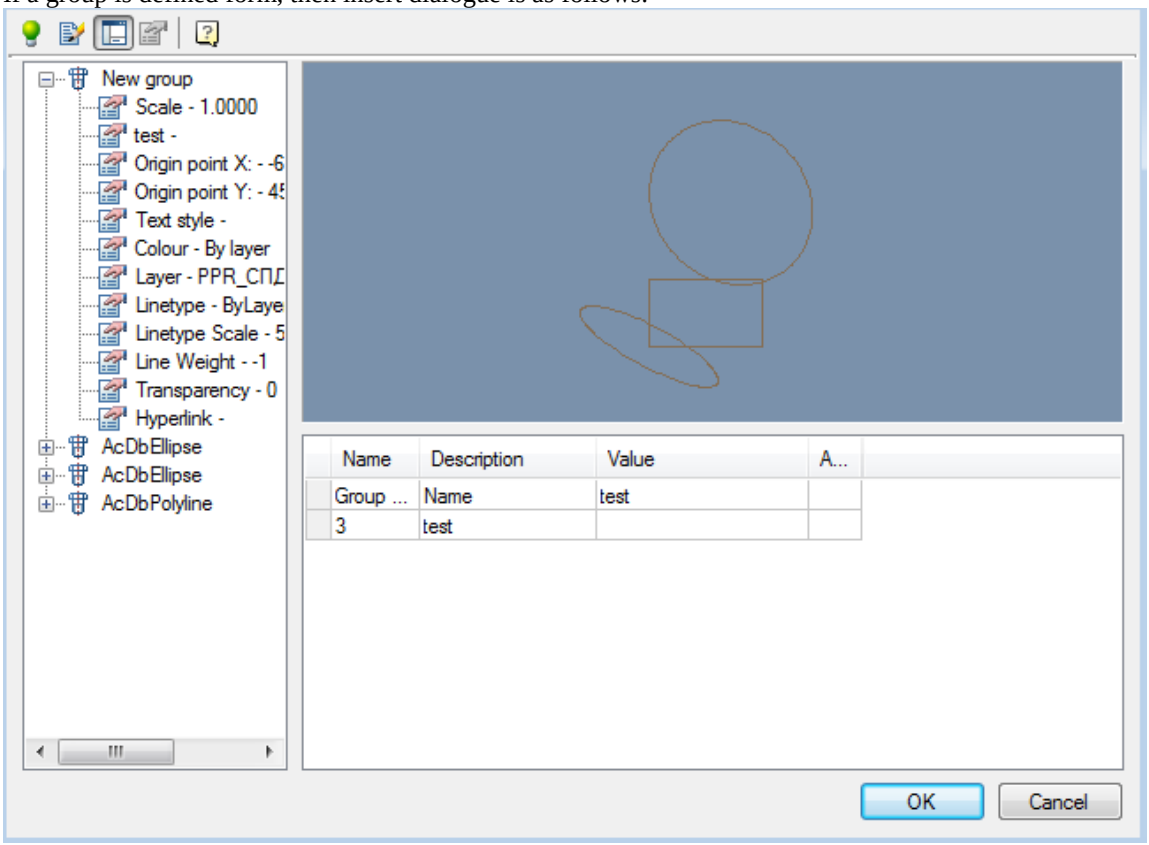
Select group details from drawing

Switch between groups of objects and dependencies

The mechanism used in the *team group insert* , you can insert a group of objects



The "Add marker group" enables the group a graphic element - a marker located at the insertion point of the group. In marker may include parameters of individual parts group that can help you when pasting automatically change the geometry of the group of related components.
"Multiple insert" allows you to create multiple copies of the selected group.
If a user group has been maintained by the standard in the folder $\MCS\Groups, then insert it in the tree by choosing the object manager.
If a group is defined form, then insert dialogue is as follows:



- The dialog box includes a tree of groups - a list of objects and their parameters group. This tree can disable button
- The preview window group
- Form input values group

When you click on you can switch to edit mode group settings.

Universal marker

Main menu: ***Mechanical - Standard objects - Markers, groups -*** ***Add marker***.

Ribbon: ***Mechanical - Library -*** ***Add marker***.

Toolbar: "Standard objects" - ***Add marker***.

Command line: ***MCCREATEUMARKER***.

The *"Marker"* tool is designed specifically to create a connection between an arbitrary graphic object (primitive) nanoCAD and a spreadsheet-specification. A universal marker is a translator of data from drawing objects into the nanoCAD Mechanica 20.1table by means of special means - marker attributes.

Marker attributes can be visible or hidden. The values of the visible attributes are displayed in the drawing as text strings.

The marker, as an object nanoCAD Mechanica 20.1, can be saved in the object library and used repeatedly. The marker can also be included in the groups of objects nanoCAD Mechanica 20.1, providing the ability to specify the entire functional groups.

Create marker

Let's consider creating a marker in the form of a positional leader.

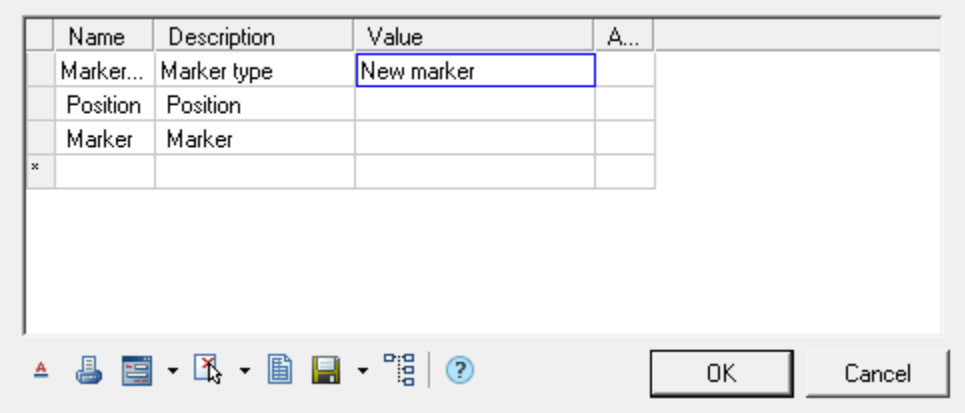
1. Set the required current scale.
2. Insert two text lines into the drawing and draw a horizontal line between them.

These objects will form a leader shelf template with two text labels. To use text strings as marker attributes, the first character must be *"$"*.

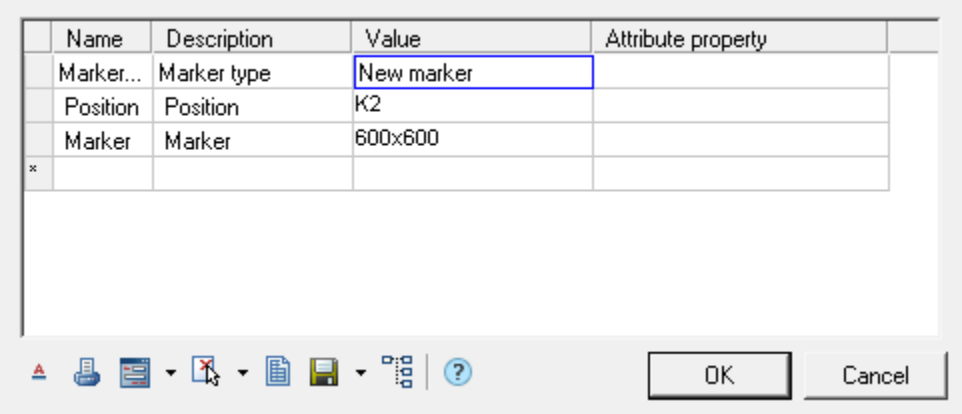
3. Call the command *"Add marker"*, use a box or crossing box to select the objects that made up the marker.

4. Snap the base point of the marker.

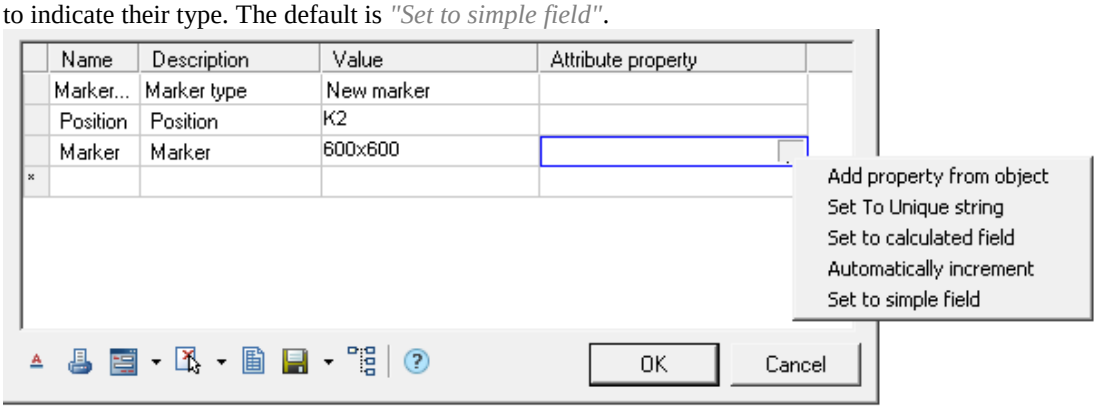
5. The *"Creation of a marker"* dialog box appears. The properties table will list the attributes entered in the callout text boxes:



Column *"Name"* contains the working name of the attribute.
Column *"Description"* describes the attribute.
In column *"Value"* the values of the attribute are entered. Values can be either numeric or text. Auxiliary commands are available in the context menu of the *"Value"* column.



Column *"Attribute property"* is used to indicate the type of the input attribute. Various attribute labels are provided to indicate their type. The default is *"Set to simple field"*.

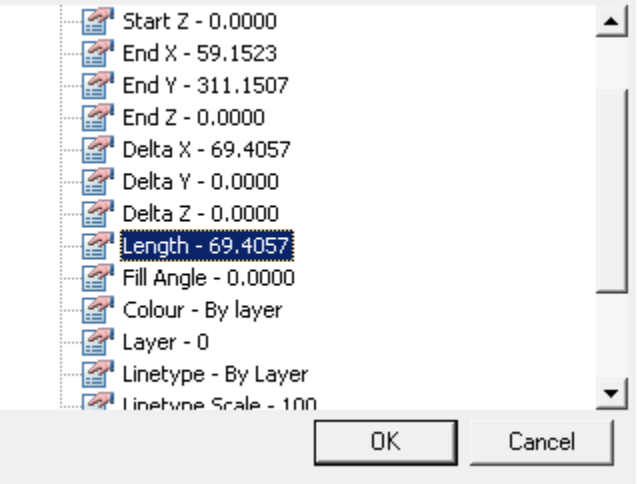


Attribute types:

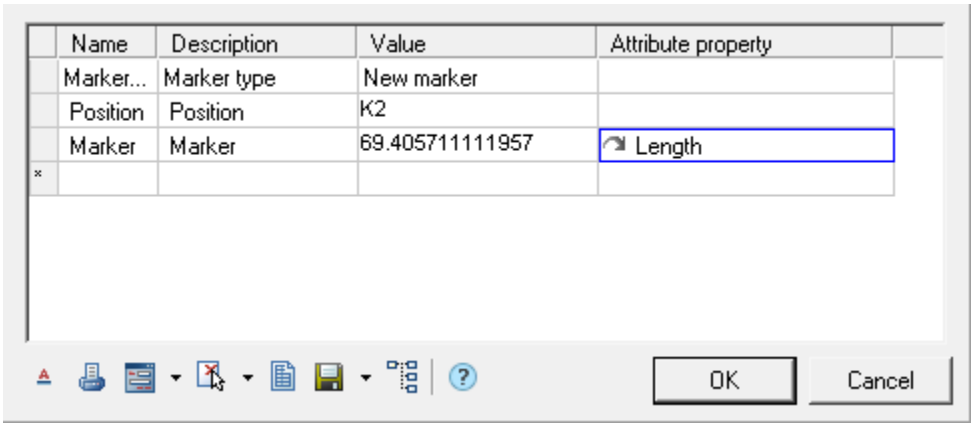
- *Add property from object* - allows you to set an attribute value by associating it with the attribute value of another object in the drawing.

Important! An object is understood as an object from the base of standard products.

After choosing the type, it is suggested to select the object and the object attribute.



After selecting an attribute, its value falls into the *"Value"* column, and in the *"Attribute property"* column the name of the selected attribute.



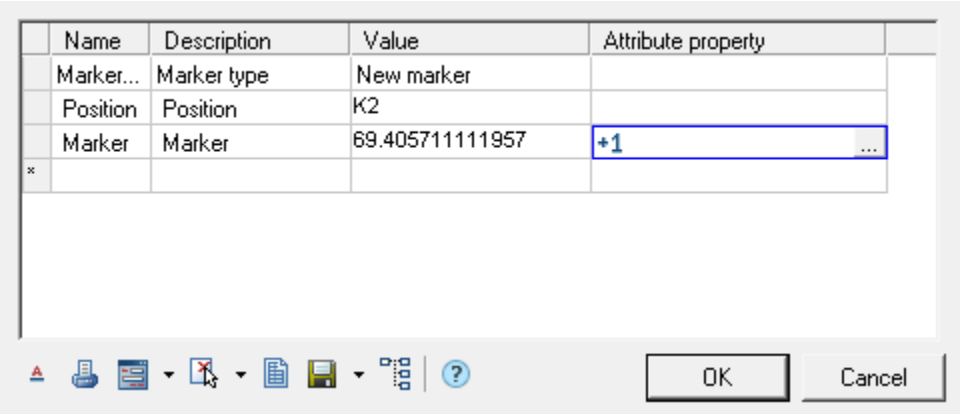
After assigning an attribute, you can change the type of relationship with an object attribute:

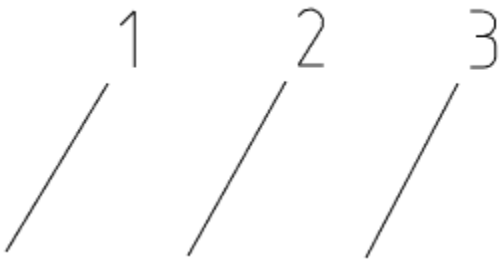
One-way communication - changing the value in an attribute of an object affects the value of the marker.

Two way communication - changing the value in the marker affects the attribute of the object, and vice versa.

Basic communication - an attribute of an object can only be changed from a marker. If the attribute is a table value in a basic relationship, the closest value from the table is displayed. The entered value will be displayed in brackets.

- *Set To Unique string* - allows you to set a counter attribute that will increase its value by 1 for each subsequent inserted marker. In the *"Value"* column for the visible counter attribute, you can specify a character string - a prefix. In the drawing, the counter value will be appended to the prefix.



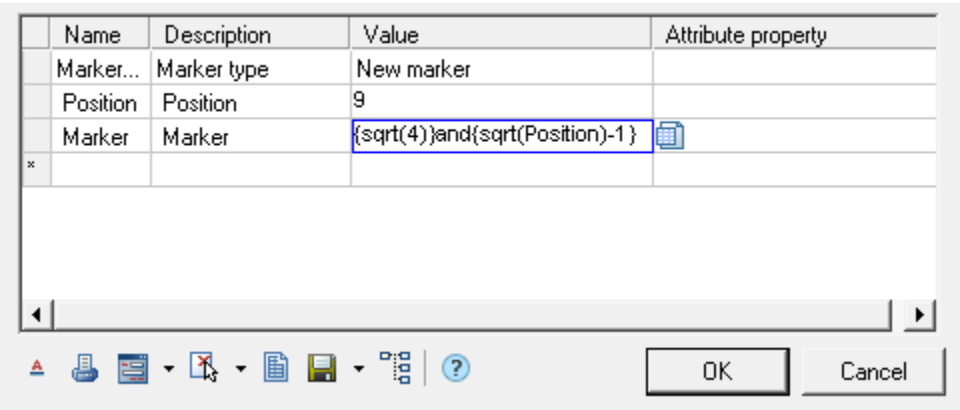


- *Set to calculated field* - allows you to set the value of an attribute using a mathematical expression, using references to other marker attributes, operators, and mathematical functions. The name of the desired attribute is used as a reference to another marker attribute.

Note: Computed attributes are defined and calculated from top to bottom.

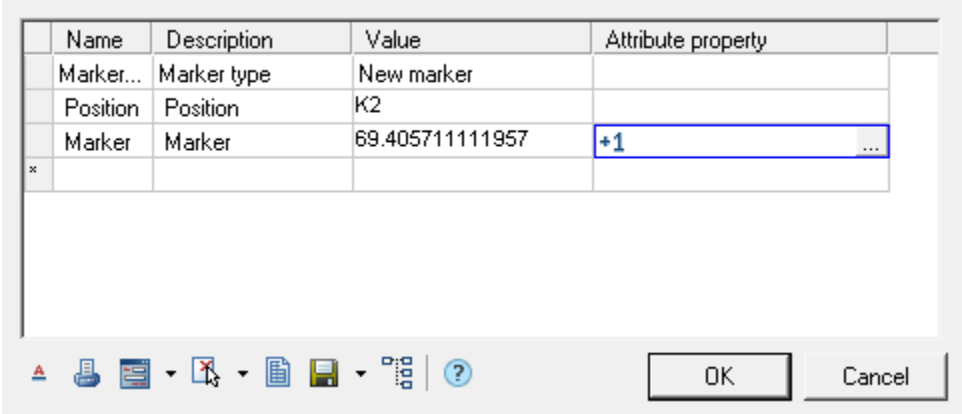
Important! Calculations are performed in curly braces.

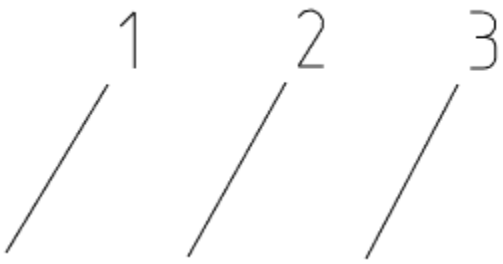
Function:



Result:

- *Automatically increment* - allows you to set a counter attribute that will increase its value by 1 compared to the same attribute of the previous inserted marker. Such a counter attribute allows repetition of the same values for several markers, which distinguishes it from an attribute set to a unique value.

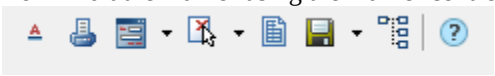




- *Set to simple field* - sets the default attribute type to plain text.

Note: Attributes included in the block can be included in the marker attribute table nanoCAD!

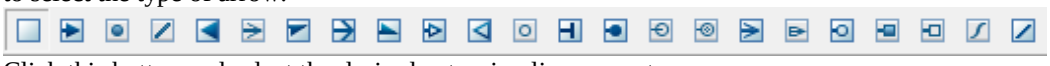
6. Edit the marker using the marker controls at the bottom of the dialog box.



Include:

- *"Marker draw mode"*. When you click on this button, a drop-down list of rendering modes appears:
  - Simple marker.
  - Draw position line.
  - Linear align.

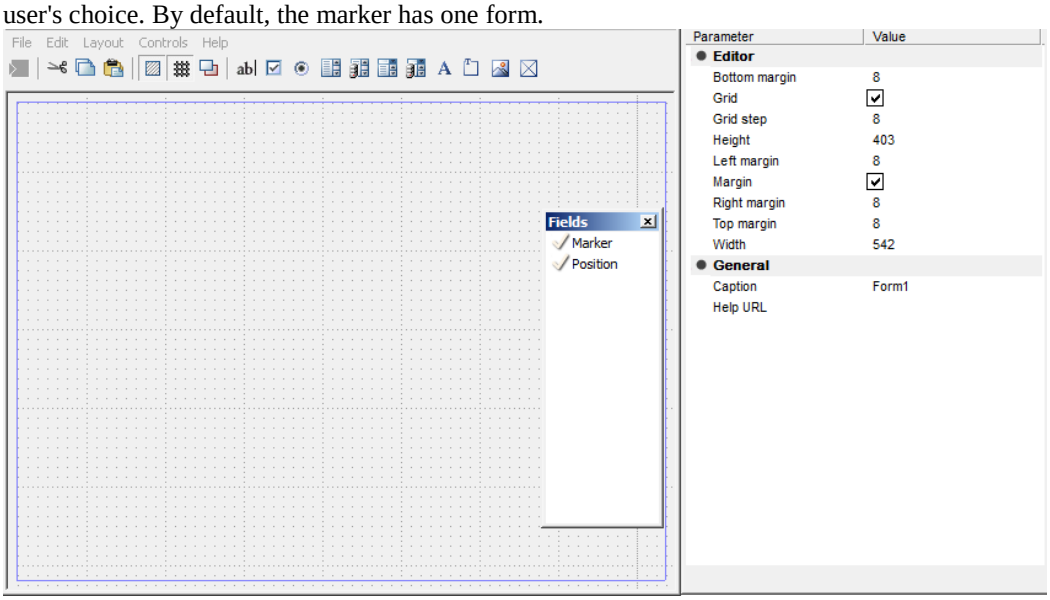
When you select the display method with the addition of a leader line, an additional button *"Arrow Type"* appears to select the type of arrow:



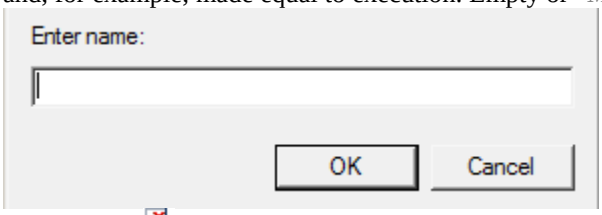
Click this button and select the desired extension line arrow type.
Arrow display parameters are defined in the "Settings - Symbols - Extension - Additional extension lines" settings.

- *"Unploted"*. When this button is enabled, the marker is placed on a special layer that is not displayed when printing. When this mode is activated, the icon on the ( ) button changes and the marker is displayed in gray on the screen (in accordance with the color of the non-printed layer).
- *"Show form editor"*- calls the custom form editor to create a dialog for working with marker attributes.

The following commands are available in the drop-down menu:
"Edit Form". Open Form Editor.
"Add Form. Adds additional forms, while the marker can change the appearance of the form, by condition or by the user's choice. By default, the marker has one form.



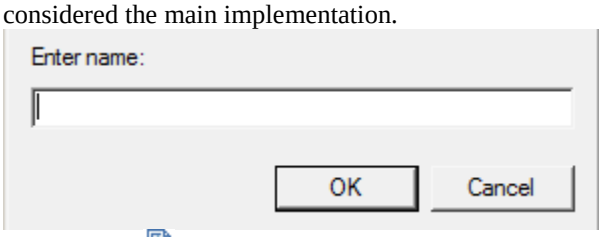
The form editor uses standard Windows form generation technology. The toolkit consists of three blocks - the graphic editor area, the toolbar with form objects and the properties window of the inserted elements. For inserted fields, the *"Field Name" ("Variable")* can be selected from the list of existing marker description fields. In this case, the default field value is equal to the field value in the marker description.
When adding a second form, you are prompted to enter a name for the form. The variable *Tag - mcFormName* is added to the properties table. This variable is responsible for the name of the form, it can be set in a calculated field and, for example, made equal to execution. Empty or *"Main"* is considered the main form.



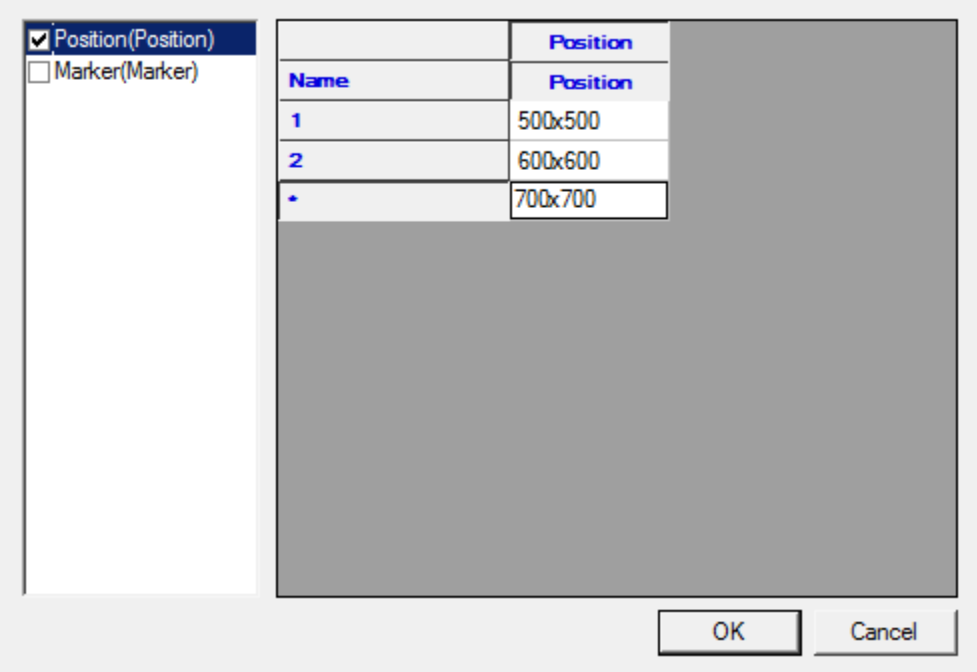
- *"Geometry"*. Used to change the appearance of the generic marker and define the insertion point.

The following commands are available in the drop-down menu:
Get new geometry. The button is used to change the appearance of the universal marker and define the insertion point.
Export marker source graphics. Exports marker geometry to a file.
Add Implemetnation. Adds additional performances, while the marker can change its appearance, by condition or by the user's choice.
When creating a new implementation, you are prompted to enter its name. When adding implementation, the *Tag - mcImplementation* variable is added to the properties table. This variable is responsible for the name of the current

marker implementation. It can be set to a calculated field or linked to a list on a form. Empty or *"Main"* is considered the main implementation.



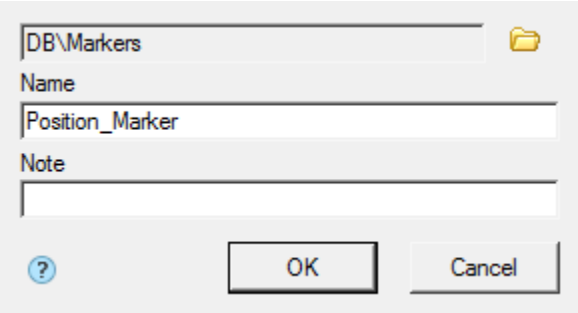
- *"Set parameters table".* Opens the marker parameter table editor. When inserting a marker and having a table of values, it is possible to assign a group of values to the marker attributes.



In the list, select the parameters whose values will be selected from the table. In the right part of the window, enter the required number of lines with values. Close the dialog with the *"OK"* button.

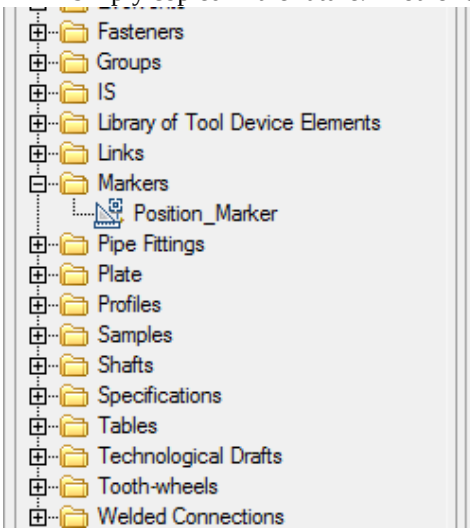
- *Save changes to odatabase.* The button is intended for writing a marker to library nanoCAD Mechanica 20.1.
- *Apply changes to all markers of this types.* Applies changes to all markers of this type in the drawing. Serves for changing the marker template for all markers made according to this template and inserted into the drawing.

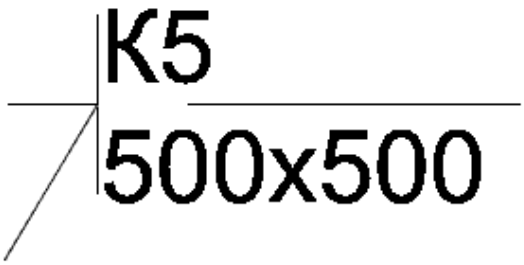
7. If necessary, add any number of hidden attributes, filling in the rows of the table in sequence. The attribute is added in the bottom line of the table, marked with an "*" symbol.
8. For future use of the marker, save it to the database as a named object.



In the *"Create element"* dialog box, specify the path to save, the name of the object and the note (for more information on working with the object base, see the nanoCAD Mechanica 20.1Object Manager). After you save the marker, a message appears prompting you to update the existing copies of the marker in the drawing.

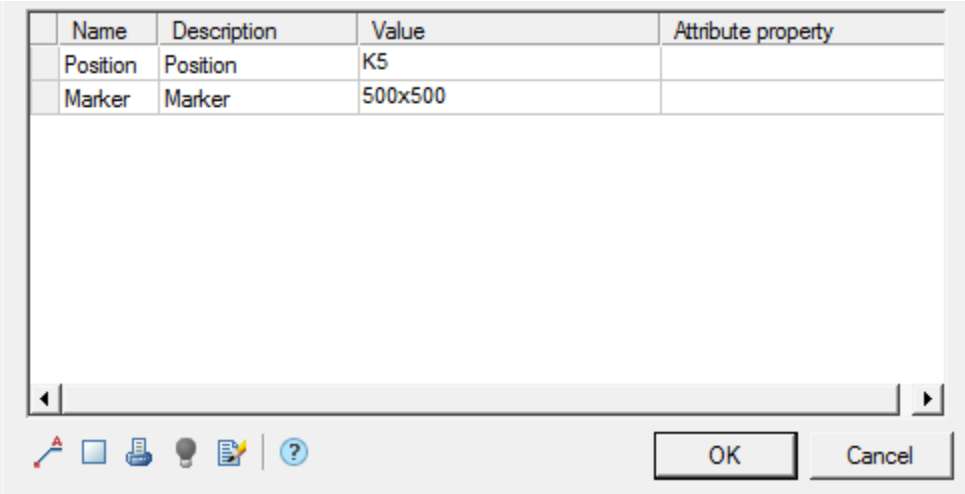
9. nanoCAD Mechanica 20.1 makes it possible to immediately apply the created marker. The command line prompts you to install the generated token. Pick an insertion point in the drawing. Identical markers can be simply copied in the future. In other cases, the marker is inserted directly from the base.





Edit marker

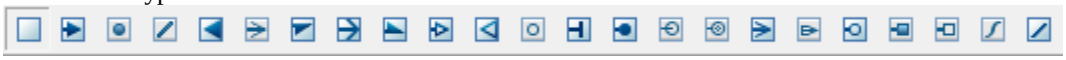
Editing of a marker is carried out in a dialog box, the title of which corresponds to the name of the marker. For example, when you call for editing the "Position_Marker" marker, the creation of which was described in the previous paragraph, the dialog box will look like this:



Additional commands are located at the bottom of the window:

- *"Marker draw mode"*. When you click on this button, a drop-down list of rendering modes appears:
  - Simple marker.
  - Draw position line.
  - Linear align.

When you select the display method with the addition of a leader line, an additional button *"Arrow Type"* appears to select the type of arrow:



Click this button and select the desired extension line arrow type.
Arrow display parameters are defined in the "Settings - Symbols - Extension - Additional extension lines" settings.

- *"Unploted"*. When this button is enabled, the marker is placed on a special layer that is not displayed when printing. When this mode is activated, the icon on the ( ) button changes and the marker is displayed in gray on the screen (in accordance with the color of the non-printed layer).
- *"Highlight info source object"*. Highlights in the drawing the object with which the marker is associated (The function is available if the "Add property from object" mode is set for one or several attributes and the corresponding object was specified when inserting the marker).
- *"Edit marker properties"*. Turns on the edit mode, in which you can change the appearance of the marker, add or remove attributes.

System variables in marker

mcImplementation - Execution management. Empty or "Main" means the main view.
mcFormName - Form management. Empty or "Main" means the main form.
mcArrowSize - Arrow size control.
mcDefLayer - Controls the marker location layer. The "Value" field contains the name of the layer.
mcShowFormOnInsert - Controls the display of the dialog when inserting. If the value is 0, the dialog is not shown when inserting. The value can be calculated.

Template Wizard

Main menu: ***Mechanical - Standard parts - Markers, Groups - Template wizard***.

Ribbon: ***Mechanical - Library - Template wizard***.

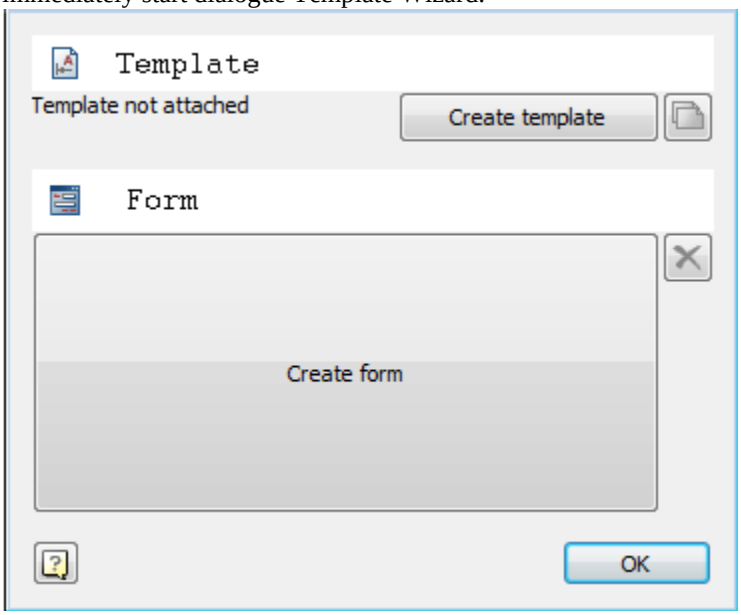
Toolbar: *Template wizard (* "Standard parts"*)*.

Command line: *MCTWIZ*.

Template Wizard is designed to create easy to use tools of commonly used design elements drawing.

Basic rules

When the command will be asked to select an object to create a template. If the object was preselected, then immediately start dialogue Template Wizard.



Command Options

Button *Create template* - creates a template in the database or applies the changes made in the editor.

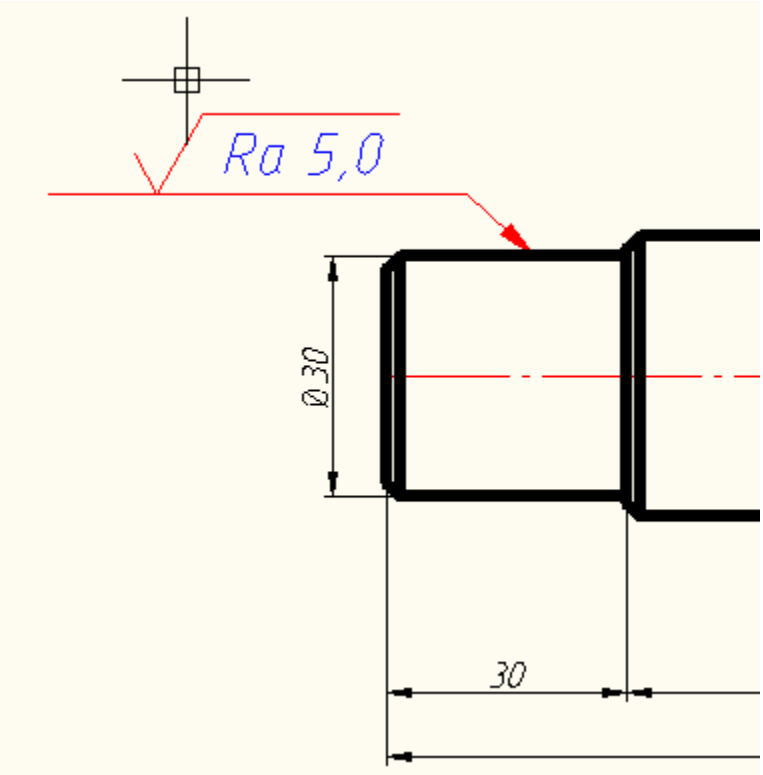
Button *From* - Starts the form template.

Button *Create a new template* - Creates a new template in the database based on the selected graphics and created form.

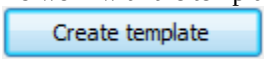
Button *Delete form* - deletes the created form template.

Example of creating a template

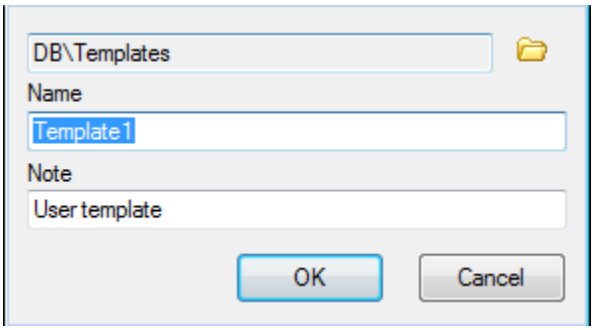
Consider the example of creating a template signs of roughness. To do this, the drawing shall bear one designation:



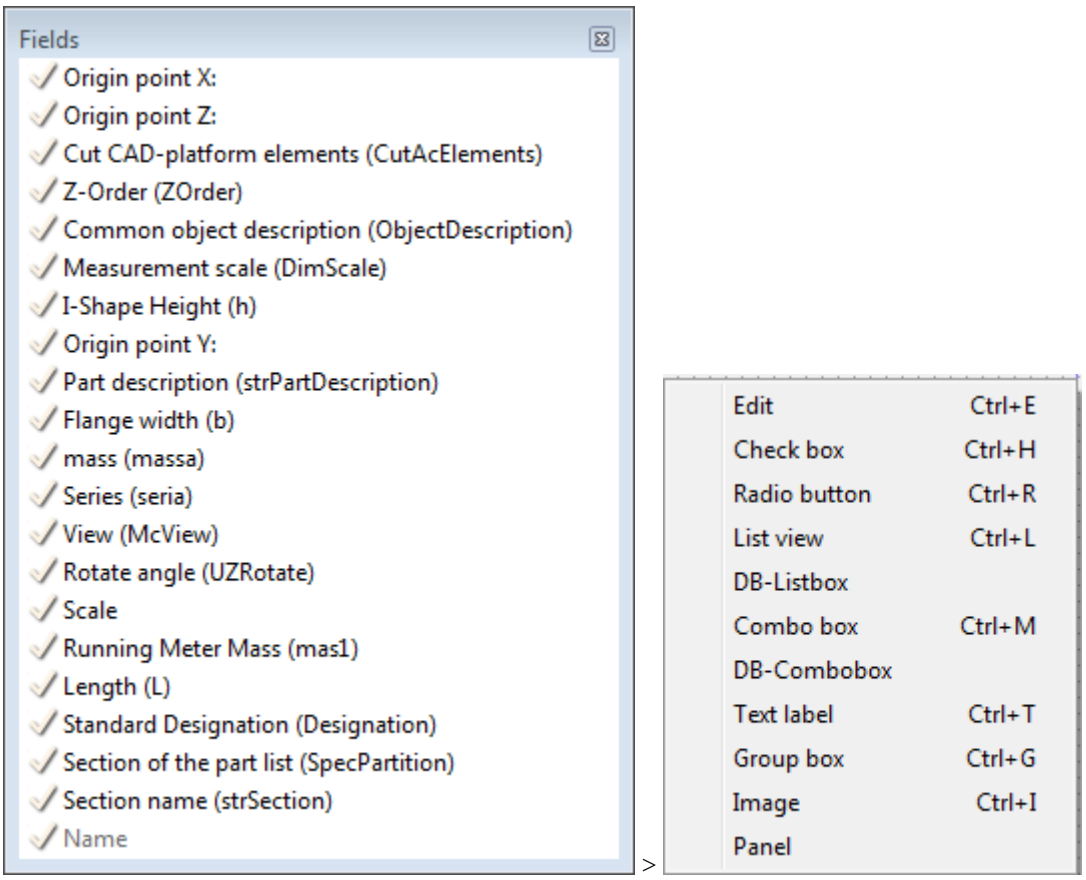
After you run the Wizard Templates chosen designation roughness.
To work with the template you must save it in the database that will be invited to do by pressing the button



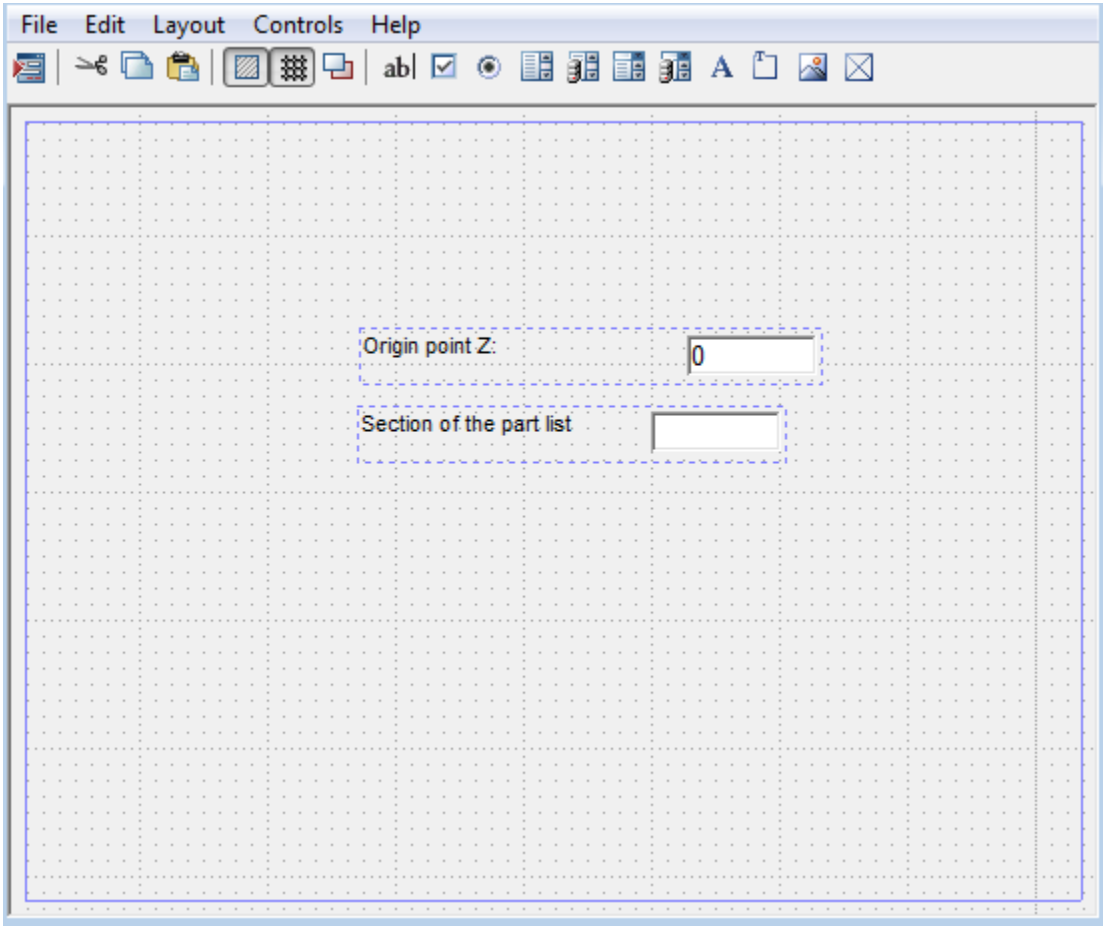
.



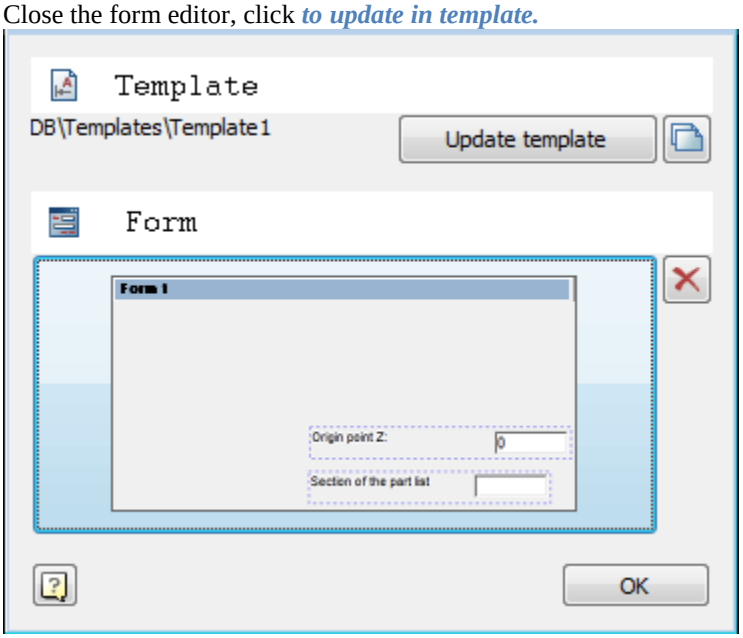
Then click on the ***Create a form*** and create a form using the notation tools online forms.
For example, we can take two parameters - the first line designations and direction of irregularities. Both parameters are added to the form as a combo box (drag while holding Ctrl).



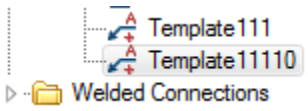
After editing, we obtain the form of the following form:



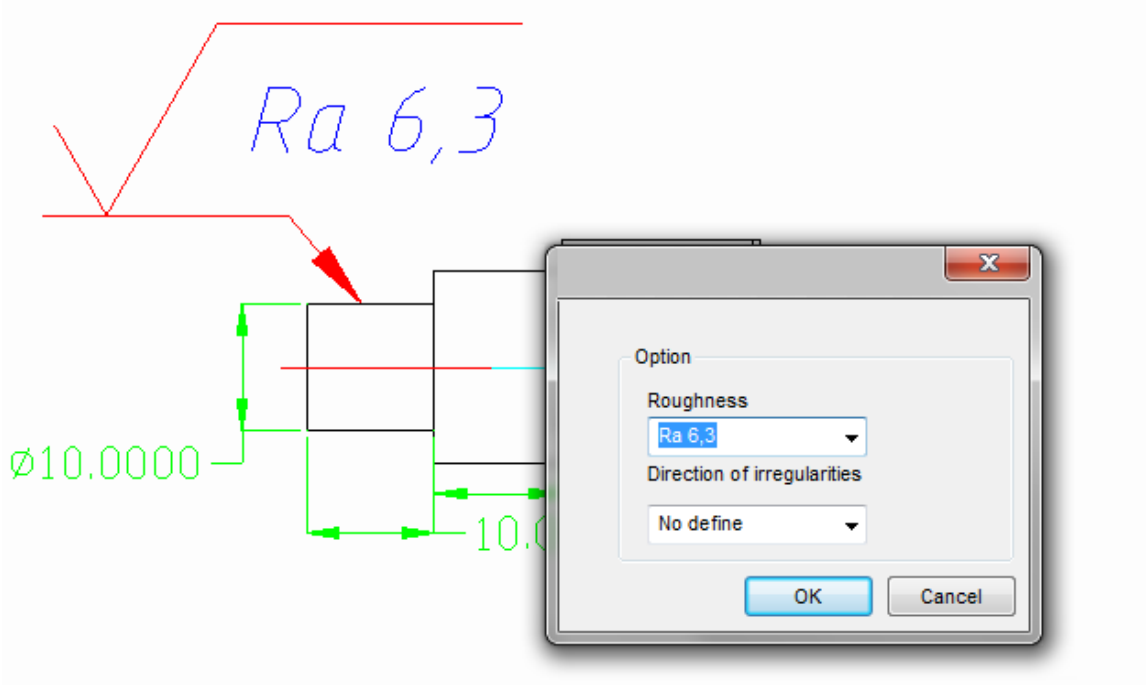
Close the form editor, click ***to update in template.***



Template will be saved in the database and will be displayed in the object manager.

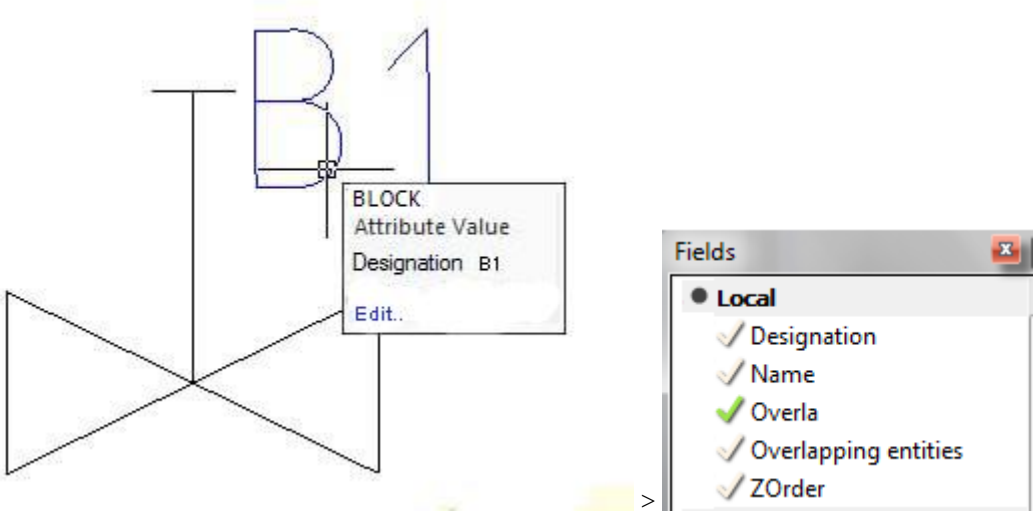


Now when you insert from the database will be able to also place the designation of roughness on the shelf line-outs, but when you edit your form will appear:

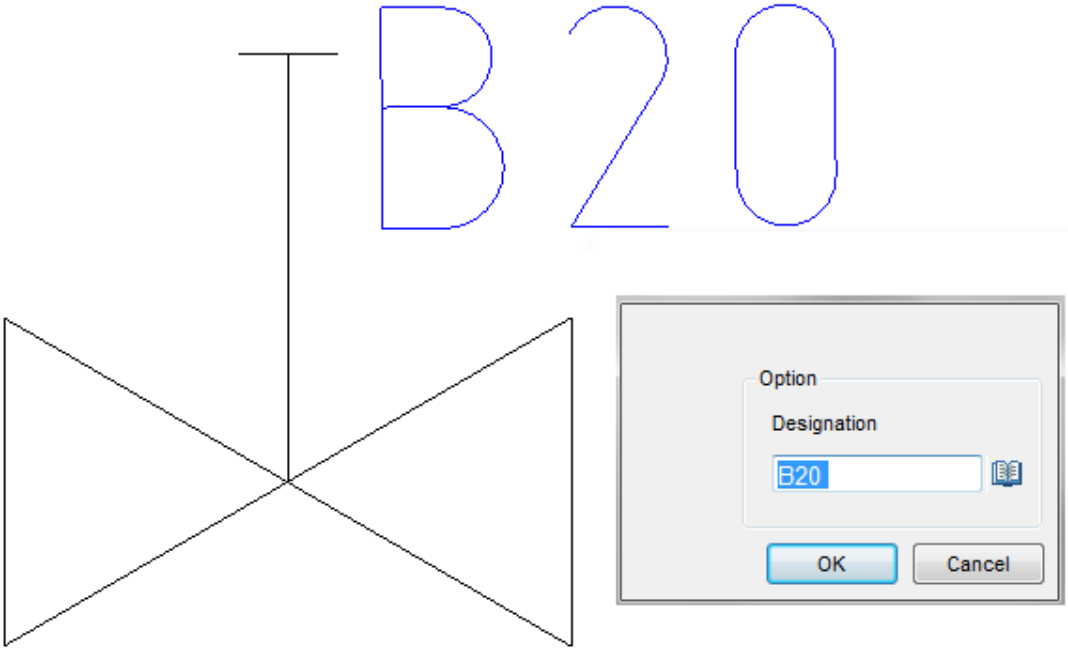


Editing blocks nanoCAD

If the original graphics selected block nanoCAD, its attributes are also displayed in the form editor as available variables.



When creating a form in the database and maintaining such a block is inserted into the drawing as a block of the same source, only the attributes can be set up via a custom form.



Control overlapping objects

Change hidden mode

Main menu: ***Mechanical - Standard parts- Cover -*** ***Change hidden mode***.

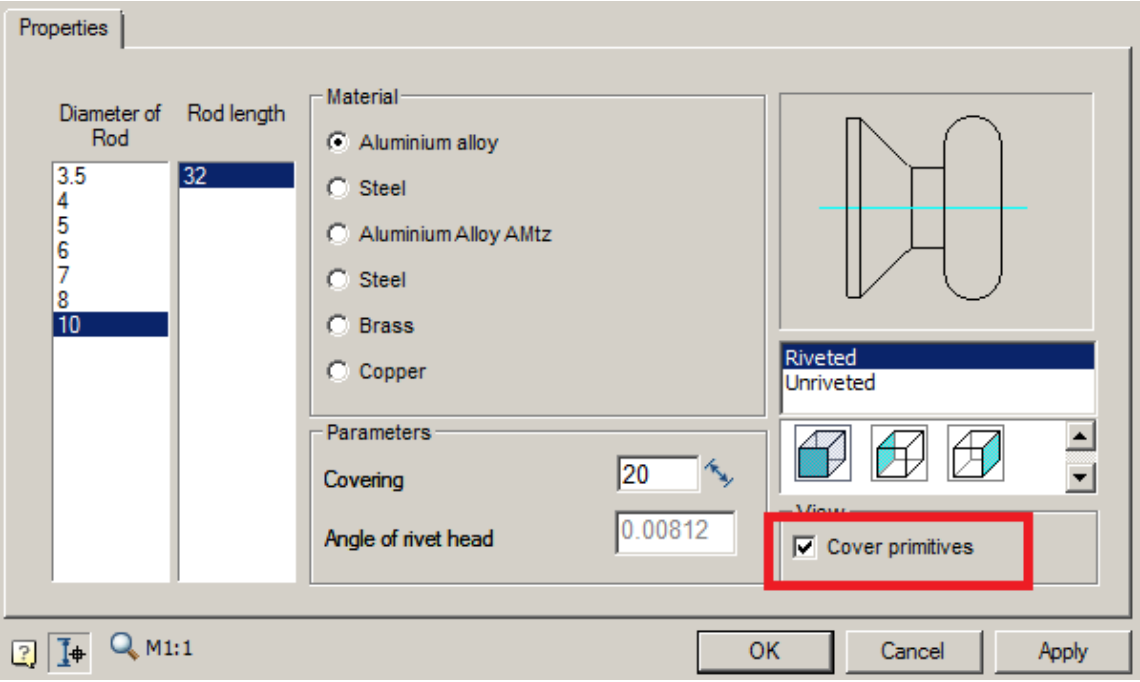
Toolbar: ***Change hidden mode (*** "Standard parts"***)***.

Command line: ***MCCHCOVER***.

The command switches the display mode of hidden lines.

nanoCAD Mechanica 20.1 allows you to hide individual graphics elements. The overlapping nature of the base object or user block is determined by a special parameter - the overlap level *ZOrder*. *ZOrder* - this is a numeric parameter whose value determines the overlap priority. An object with a large ZOrder value overrides an object with a lower value.

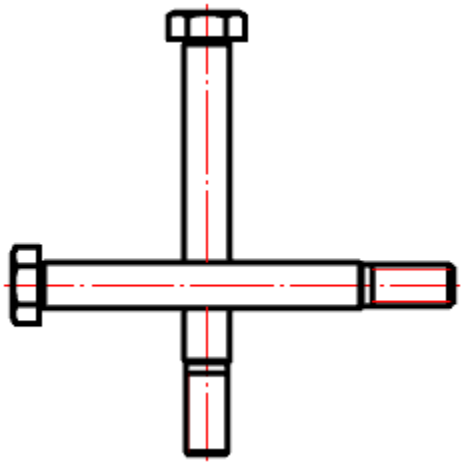
Primitives nanoCAD have the lowest overlapping level and therefore are always overlapped by objects from the database. That at an insertion of object nanoCAD Mechanica 20.1 objects nanoCAD did not overlap, it is necessary to disable the switch in the object insert dialog *"Cover primitives"*:



The area in the drawing that the base object covers nanoCAD Mechanica 20.1, is called *"Contour of suppression"*. An overlap exists within this contour.

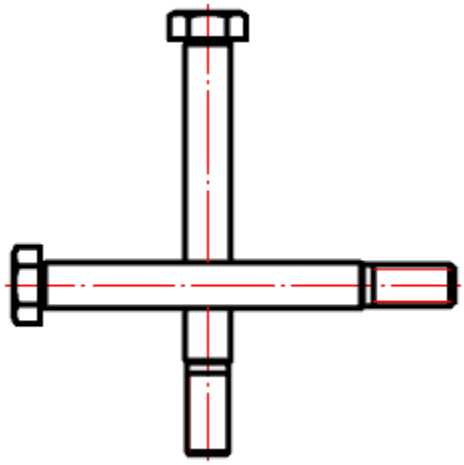
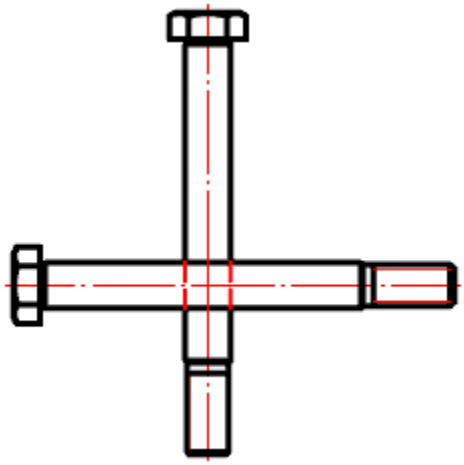
Procedure

1. Call command.



2. In the context menu, select the desired view of the overlapped objects *"Normal"*, *"Dotted"* or *"Switch"*.

Normal	Dotted	Switch

Normal	Dotted	Switch
		The command appends the value. Those. If a normal overlap has been established, then the dashed line becomes, and vice versa.

3. Select the secant frame for the objects for which you want to change the overlapping mode.
4. Repeat if necessary items 2 and 3, or complete the command on the key *"Enter"*.

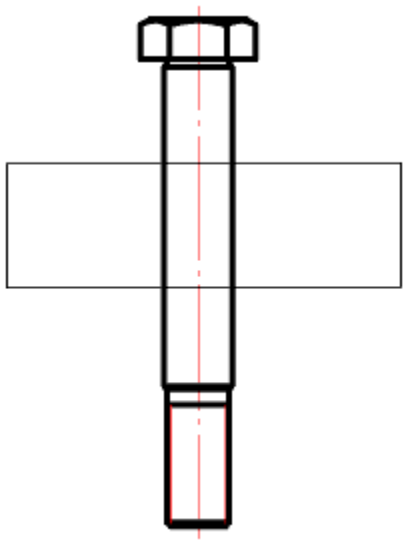
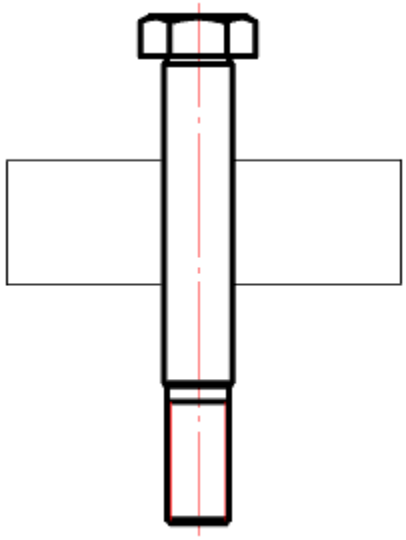
Regenerate suppression contours

Main menu: ***Mechanical - Standard parts - Cover -*** ***Regenerate suppression contours***.

Toolbar: ***Regenerate suppression contours (*** "Standard parts"***)***.

Command line: ***MCCOVER***.

Regenerate the drawing and redraw the contents of the current viewport.

Before	After
	

Move up

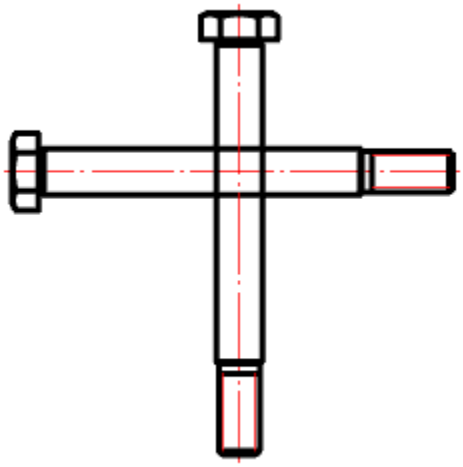
Main menu: ***Mechanical - Standard parts - Cover -*** ***Move up***.

Toolbar: ***Move up (*** "Standard parts"***)***.

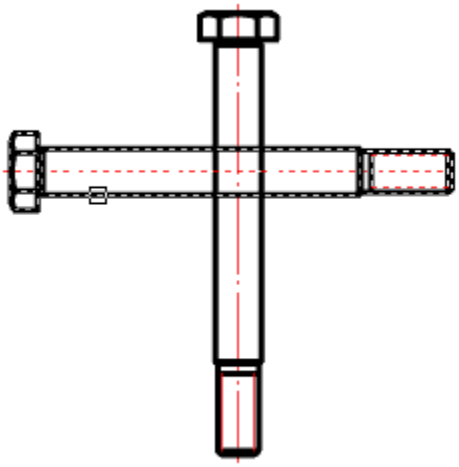
Command line: ***MCMOVEUP***.

Procedure

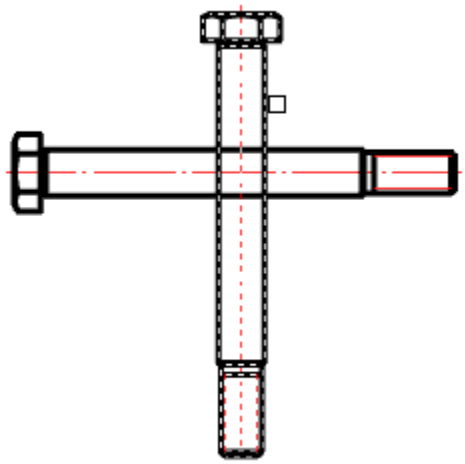
1. Call command;



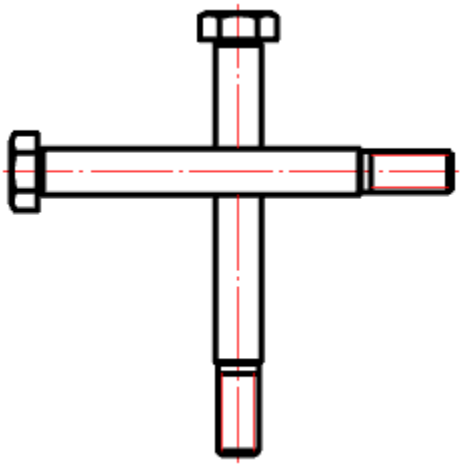
2. Select the objects you want to move up;



3. Select the objects to be compared;



4. After the selection is complete, the objects that need to be moved up will automatically increase the overlap level (ZOrder).



Move down

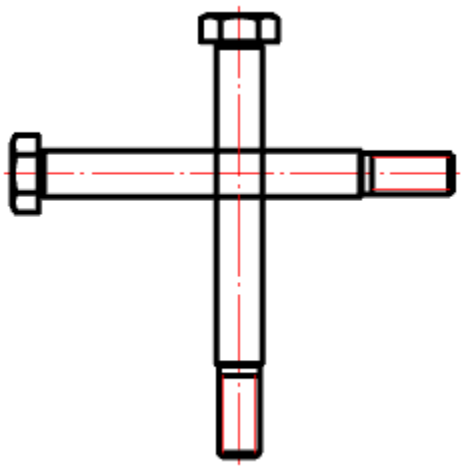
Main menu: ***Mechanical - Standard parts - Cover -*** ***Move down***.

Toolbar: ***Move down (*** "Standard parts"***)***.

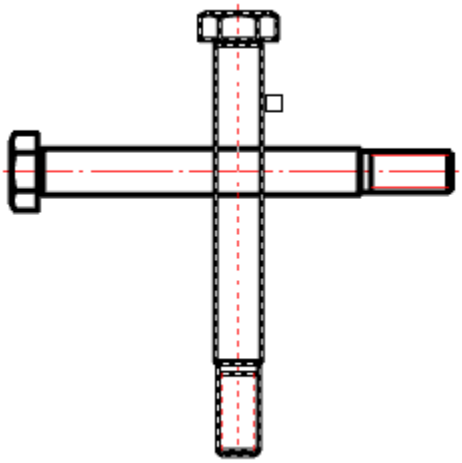
Command line: ***MCMOVEDOWN***.

Procedure

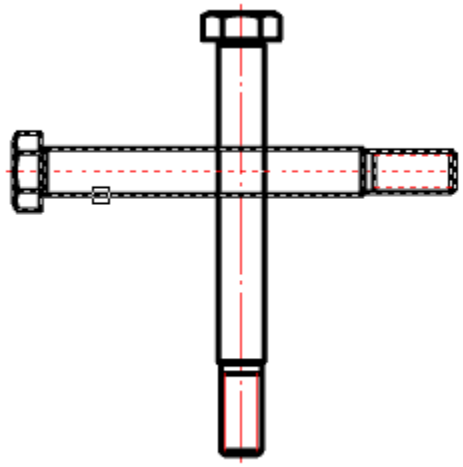
1. Call command;



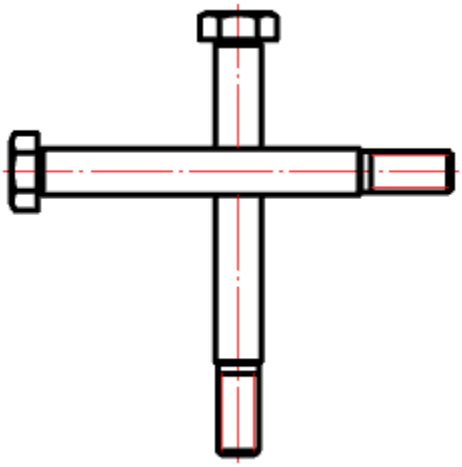
2. Select the objects you want to move down;



3. Select objects to be compared;



4. After the selection is complete, the objects that are to be moved down will automatically decrease the overlap level (ZOrder).



Part trim

Add part trim

Main menu: ***Mechanical - Standard objects - Part Contour Trim -*** ***Add part trim***.

Ribbon: ***Mechanical - Library -*** ***Add part trim***.

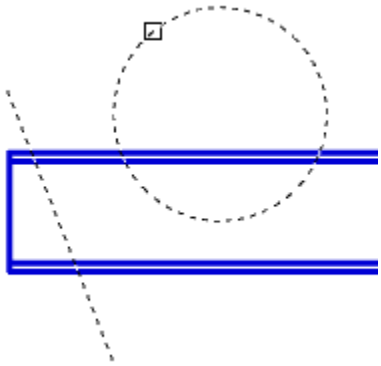
Toolbar: "Standard objects" - ***Add part trim***.

Command line: ***MCTRIM***.

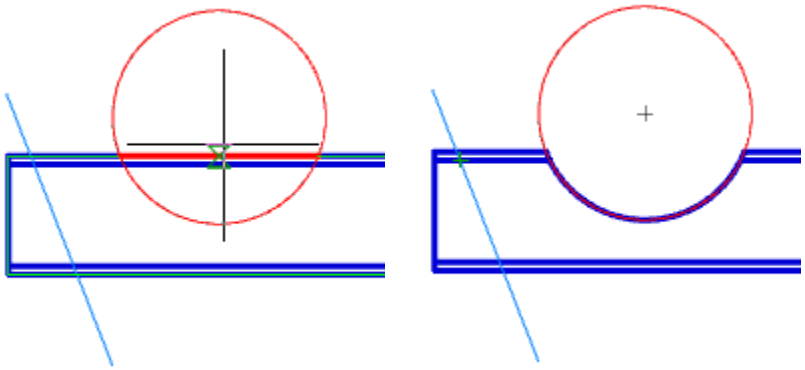
The command allows you to trim objects of metal structures.

Trimming with contour indication

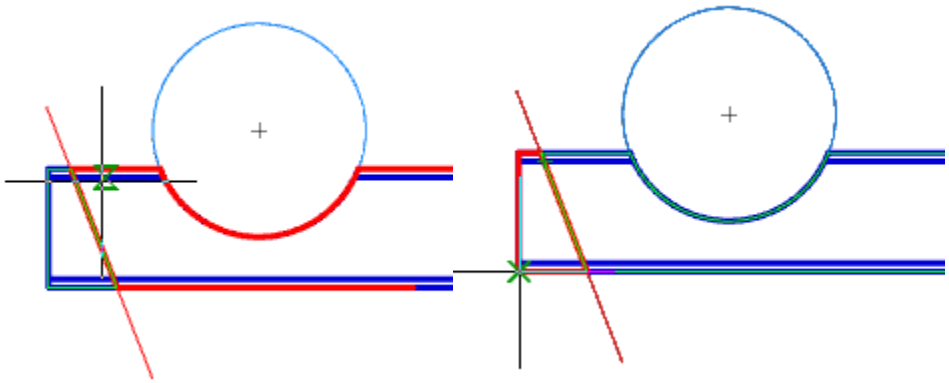
1. Call the command *"Add part trim"*.
2. Pick an outline. A contour can be a line segment, circle, arc, polyline (closed and open).



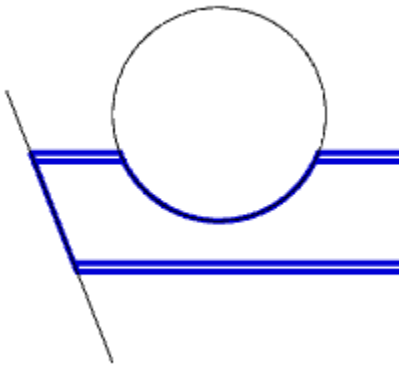
3. Specify trimming objects.
If a circle or closed polyline is used as a path, the trimming object is selected within the path.



If a linear path is used, the trimming object is selected from one side of the path.

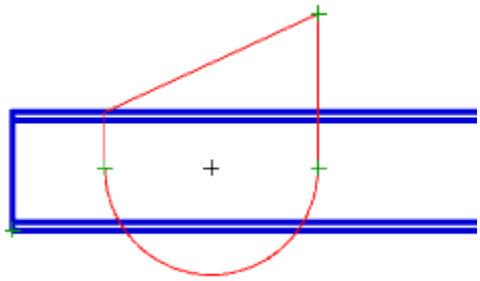


4. Press the "Esc" button to end trimming. The object will be trimmed.

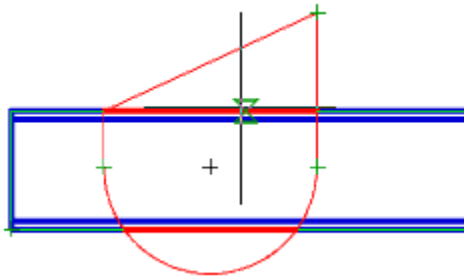


Trimming mode "COntour"

1. Call the command *"Add part trim"*.
2. Select trim mode *"COntour"*.
3. Draw a closed polyline path.



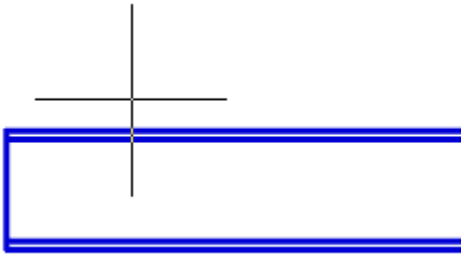
4. Pick a trimming object within the path.



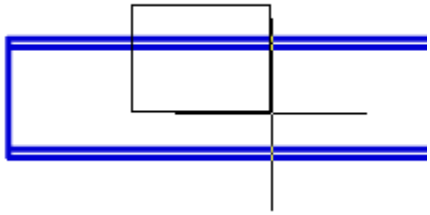
5. Trimming an object is made.

Trimming mode "rectangle by tWO points"

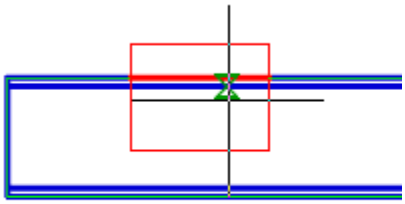
1. Call the command *"Add part trim"*.
2. Select trim mode *"rectangle by tWO points"*.
3. Pick the first corner.



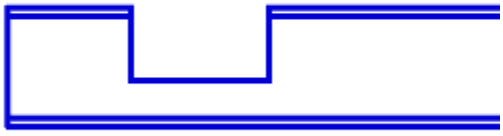
4. Pick the second corner.



5. Pick a trimming object inside the rectangle.



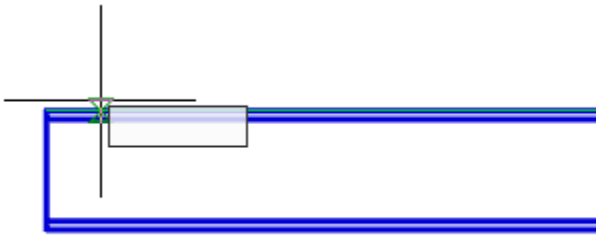
6. Trimming an object is made.



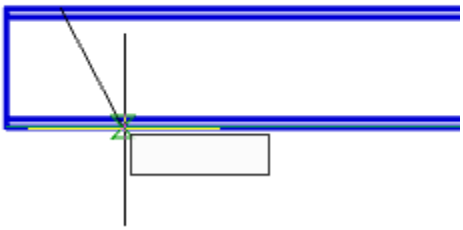
Trimming mode "rectangle by thrEE points"

Allows for trimming at the desired angle.

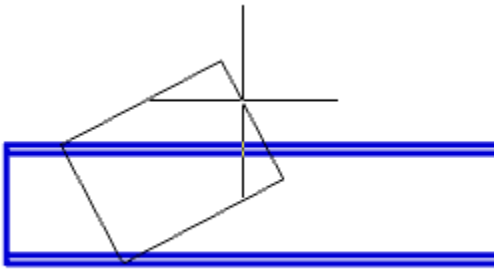
1. Call the command *"Add part trim"*.
2. Select trim mode *"rectangle by thrEE points"*.
3. Pick the first point.



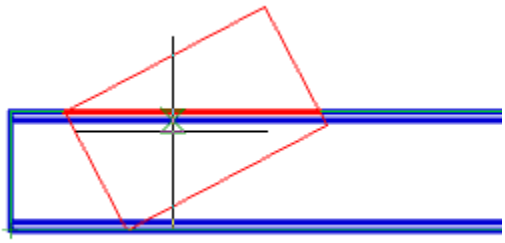
4. Pick the second point.



5. Pick the size of the outline.



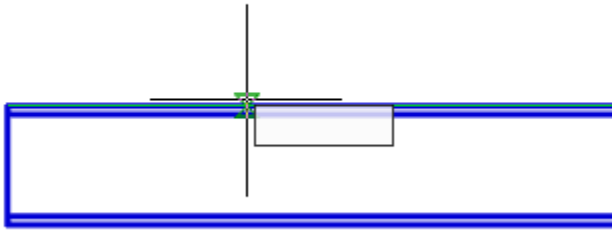
6. Pick a trimming object inside the rectangle.



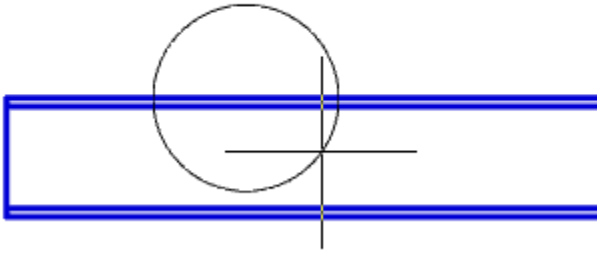
7. Trimming an object is made.

Trimming mode "HOle"

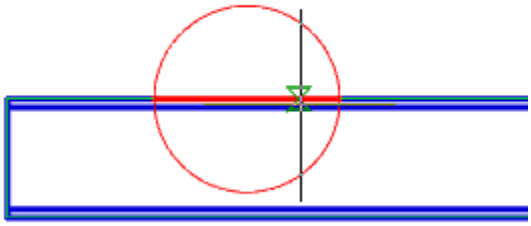
1. Call the command *"Add part trim"*.
2. Select trim mode *"HOle"*.
3. Pick the center of the hole.



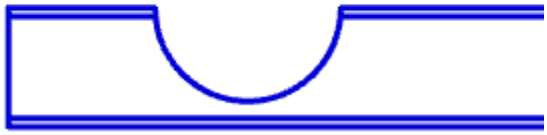
4. Pick a radius for the hole.



5. Pick a trimming object inside the circle.



6. Trimming an object is made.



Remove part trim

Main menu: ***Mechanical - Standard parts - Part Contour Trim -*** ***Remove part trim***.

Ribbon: ***Mechanical - Library -*** ***Remove part trim***.

Toolbar: "Standard parts" ***-*** ***Remove part trim***.

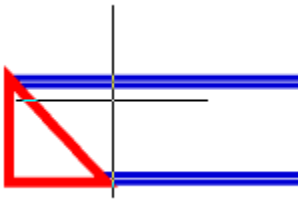
Command line: ***MCTRIMDELETE***.

The command removes a previously created trim.

Procedure

1. Call the command *"Remove part trim"*.

2. Specify the objects you want to reset and press *"Enter"* (or press *"Spacebar"*) to select all objects.



3. The specified objects will be restored to their original state.

Add View

Main menu: ***Mechanical - Standard parts -*** ***Add view***.

Ribbon: ***Mechanical - Standard parts -*** ***Add view***.

Toolbar: ***Add view (on toolbar*** "Standard parts"***)***.

Context menu: ***Add view (on select object)***.

Command line: ***MCADDVIEW***.

Designed to quickly add a new type of projection from the base of the inserted object to the drawing.

- Select the object to copy.
- Specify the insertion point of the object. Context menu commands ***Front / left / right / top / bottom*** standard projection (to insert a particular projection, it is necessary that it be recognized in the object).
- Context menu commands *Point, Angle* mode selection switch, respectively the insertion point or selection mode direction.
- Complete the cyclic command.

Adding a projection view to the object database can be done using the *Add View* from the context menu for the selected object.

Simplified / Full View

Main menu: ***Mechanical - Standard parts -*** ***Change representation***.

Ribbon: ***Mechanical - Standard parts -*** ***Change representation***.

Toolbar: ***Change representation (on toolbar*** "Standard parts"***)***.

Context menu: ***Change representation (on select object)***.

Command line: ***MCCHANGEREPRESENT***.

- Press the button and select the database object in the drawing.
- Context menu commands *Full*, *Simplified* select the object representation.

Switching presentation can be accomplished by using the ***Change view*** from the context menu for the selected object.

Mirror Part

Main menu: ***Mechanical - Standard parts -*** ***Mirrow part***.

Ribbon: ***Mechanical - Standard parts -*** ***Mirrow part***.

Toolbar: ***Mirrow part (on toolbar*** "Standard parts"***)***.

Command line: ***MCMIRROWSTD***.

- Press the button and select the database object in the drawing.
- Press ***Enter***.

Occur along the axis of revolution object ***X***.

Regenerate STD parts

Main menu: ***Mechanical - Standard parts -*** ***Regenerate Std Parts***.

Toolbar: *Regenerate Std Parts (on toolbar* "Standard parts"*)*.

Command line: *MCREGENSTD*.

The team needed to upgrade the graphics part of the object (s).

Operation

1. Call the command "Regenerate STD parts"
2. Specify the objects that you need to update or Enter, if you need to update all of the objects.
3. These objects will be updated

User defined object

Main menu: *Mechanical - Standard parts - User defined object*.

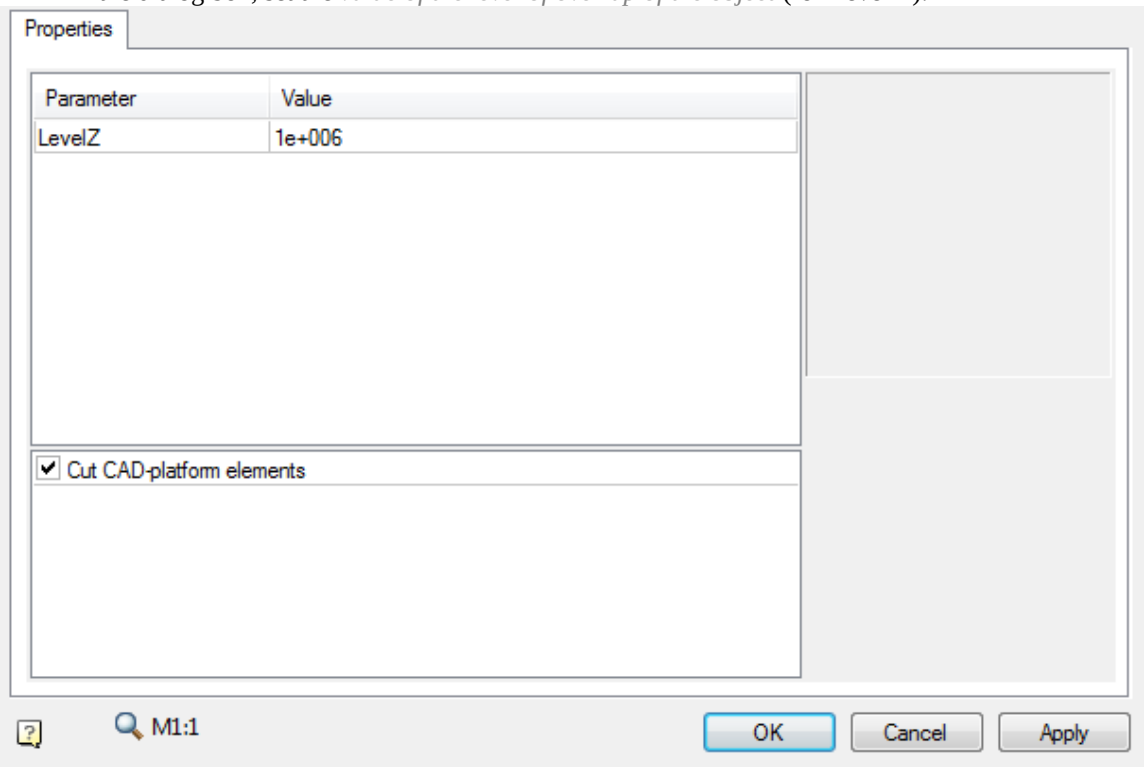
Ribbon: *Mechanical - Standard parts - User defined object*.

Toolbar: *User defined object (on toolbar* "Standard parts"*)*.

Command line: *MCCUSTOMOBJ*.

Tool *User object* makes it easy to group in objects nanoCAD and nanoCAD Mechanica20.1 into a single object. Custom objects support the ability to overlap other objects in the drawing.

1. Select the *objects to create a custom object.*
2. Specify *part origin point.*
3. Specify the drawing *insertion and direction.*
4. In the dialog box, set the *value of the level of overlap of the object* (for Level Z).



When you create a custom object contour suppression, external contour of the object is automatically detected contour suppression external contour of the object. Subsequently, the control circuit is used when objects overlap on the drawing..
Custom objects support editing with standard tools nanoCAD Mechanica20.1.

Managing Dependencies

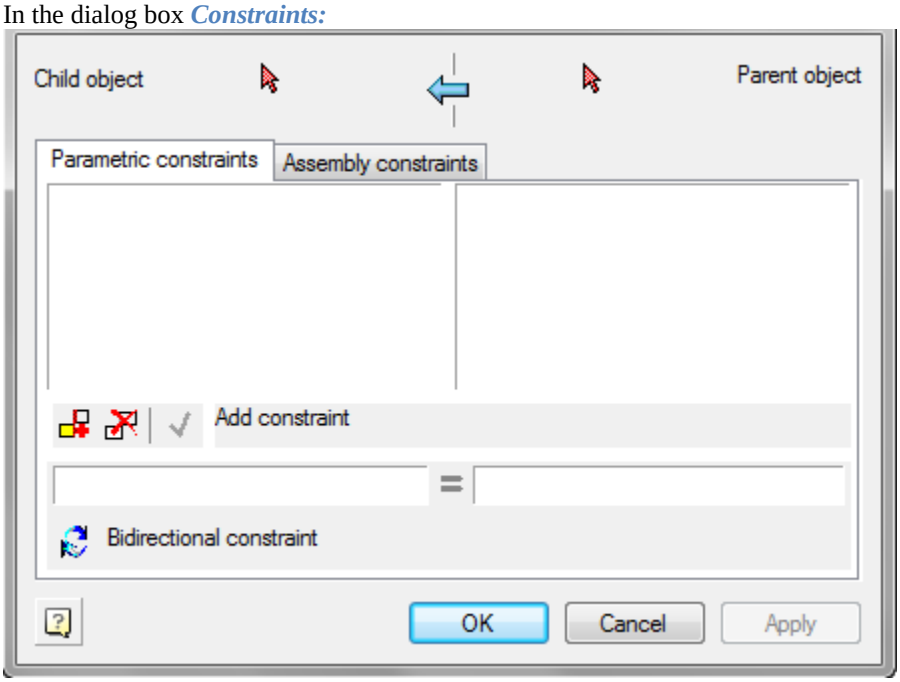
Main menu: ***Mechanical - Standard parts -*** ***Add constraint***.

Ribbon: ***Mechanical - Library -*** ***Add constraint***.

Toolbar: ***Add constraint (*** "Standard parts"***)***.

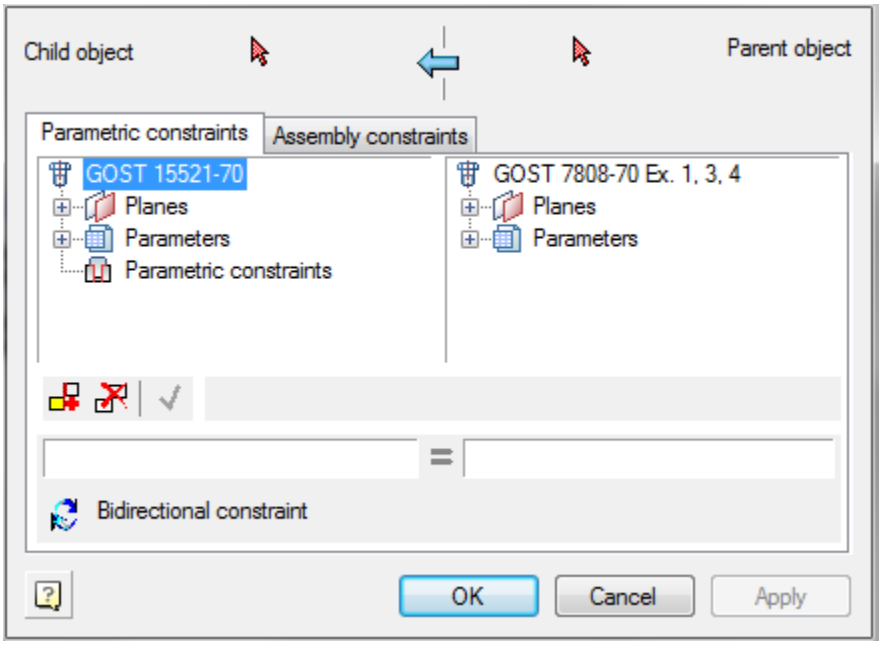
Command line: ***MCCONSTRAINT***.

In the dialog box ***Constraints:***



In the dialog box set parametric and assembly (geometric) dependencies between database objects.
The window is divided into two parts:
Left - *The child*
Right - *The parent object*
Use ***1-selected object*** and ***2 were selected to*** the choice made in the drawing, respectively subsidiary and parent objects. After selecting an object in the corresponding half of the window, a list of parameters of the object, which includes:

- Property name
- List surfaces (work surfaces)
- The list object's parameters
- List of installed parametric relationships
- List of installed assembly constraints



On two tabs ***depending Parametric*** and ***Assembly constraints*** established relationship between parent and child objects.

Parametric dependence connects the parameters of objects.

Assembly dependence binds workplanes objects.

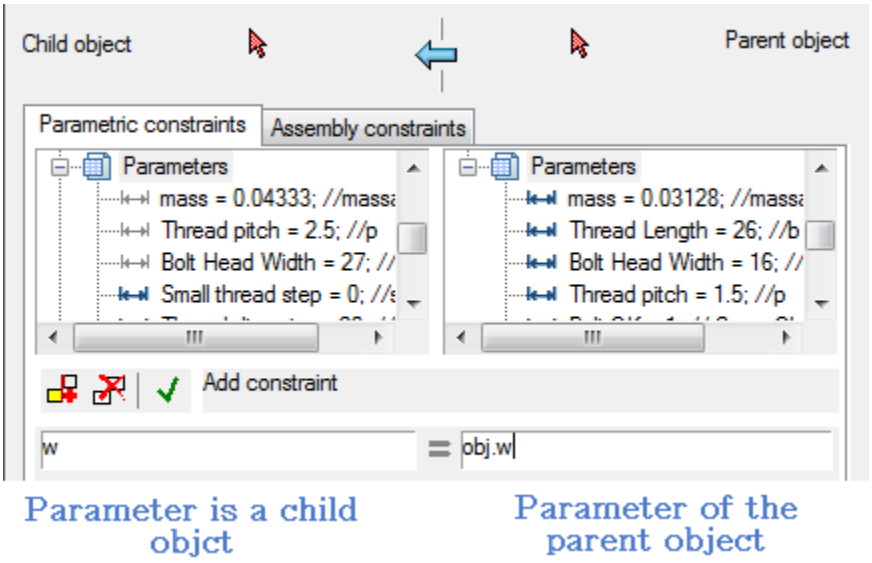
Button ***New dependency.*** Allows you to create a new dependency

Button ***Delete dependency.*** Deletes the selected list dependence

When adding or editing button is activated depending ***Save dependence***, which allows you to save your changes.

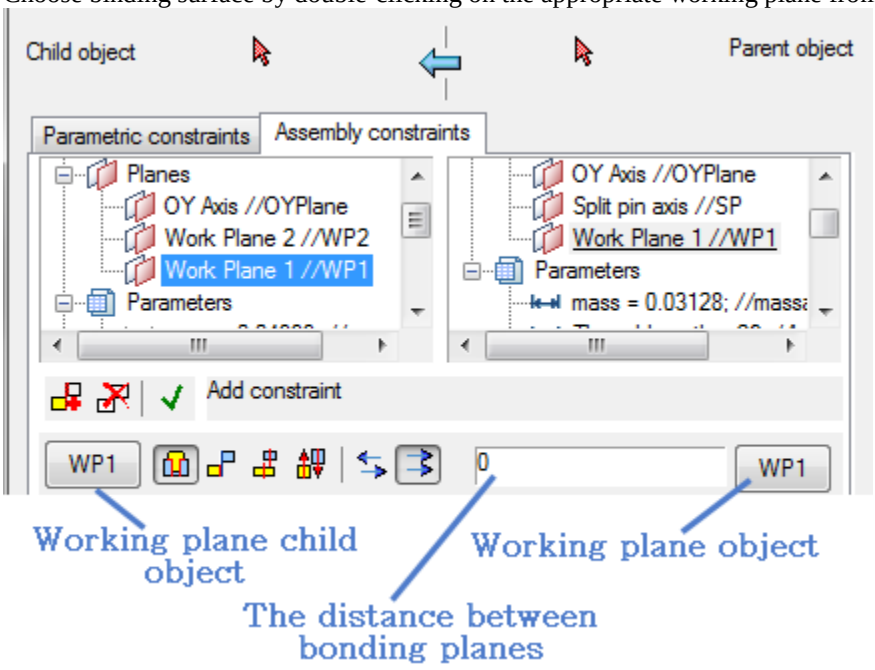
Installing the parametric dependence

When installing the parametric constraints is associated parameters are recorded in the corresponding input fields located below the list object's parameters. Selecting the link parameters by double-clicking on the corresponding parameters in the list.

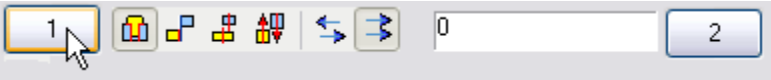


Installing assembly according

To install the switch assembly according to the tab ***Assembly constraints***.
Choose binding surface by double-clicking on the appropriate working plane from the list.



Selecting the plane can also be done visually in the drawing by clicking labelled plane and the linked marker indicating the working plane of the object.



The choice of method depends overlay using the buttons at the bottom:

Axial and surface alignment

Surface alignment

Combination axis

Direction

Choose mutual orientation related planes using the buttons:

Counter-directional

Codirectional

Button ***Bidirectional constraint***. Enables two-way relationship in which a related parameter or surface of the parent object changes with appropriate changes to the child object.

Compound objects

Main menu: ***Mechanical - Standard parts - Connect objects***.

Ribbon: ***Mechanical - Library - Connect objects***.

Toolbar: ***Connect objects (*** "Standard parts"***)***.

Command line: ***MCCONNECTTO***.

This command is used to overlay preset assembly and parametric dependencies on objects in the drawing.
After running the command must specify.

1. Child object - an object that will be joining.
2. Parent object - the object to which you will attach the child object.

Upon completion of the command will be imposed on the details of parametric and geometric constraints defined in their script..

Recognition graphics drawings

Main menu: ***Mechanical - Recognition - Symbol block recognition***.

Ribbon: ***Mechanical - Utilities - Symbol block recognition***.

Toolbar: ***Symbol block recognition (on toolbar*** "Recognition"***)***.

Command line: ***MCRECS***.

Using commands recognition of graphic objects will significantly speed up the process of registration as the projections of old drawings and new, but made by hand without the use of any application.
Character Recognition is offered to carry out automatic or semi-automatic mode.
nanoCAD Mechanica 20.1 recognizes the following symbols and signs on the drawing:

- ymbols, text callouts;
- formats;
- specification sheets;
- table.

Call Team

By default, the toolbar "Character Recognition" is not displayed.

To invoke it, place the cursor pointer on the main toolbar nanoCAD Mechanica 20.1and click the right mouse button. In the context menu, click on "Recognition".
You will see an additional toolbar:

	Character recognition
	Recognition formats
	Recognition specifications
	Table recognition

Automatic recognition

To perform automatic recognition of characters in the drawing context menu after recognizing any symbol.
Automatic character recognition requires checking the entire drawing, so it is not recommended for use in saturated drawings.
Recognize the main frame with the inscription enough to the source drawing contained within the Standard size, two contour lines and the title block.
Entries in the title block recognizable drawing are automatically entered in the dialog box title block, add additional fields and zones.

Recognition forms specifications

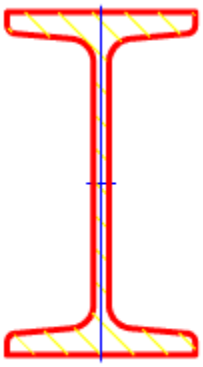
Information in the figure, placed on the forms specifications after recognizing transferred to Excel.
For correct recognition itself form the specification may be a block or drawn objects such as "Line".
The text in the cells of the form specification - an object of type "text".

Dialog "Overrides"

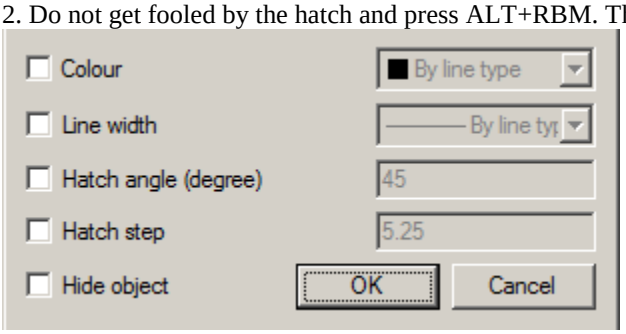
Shortcut: ***ALT+RBM (when you hover over a line or shading standard element)***.
It overrides the style of the individual lines in a standard cell.

Sample call

1. Insert a standard element, such as I-beams.In the properties, select the rendition of "left view" and the type of display "view with a cut".

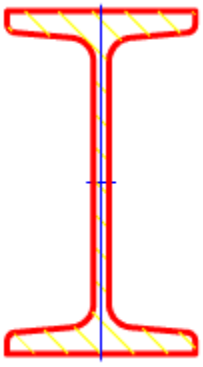


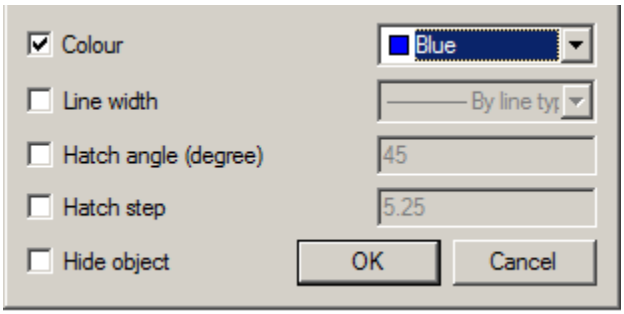
2. Do not get fooled by the hatch and press ALT+RBM. The "Overrides" dialog box.

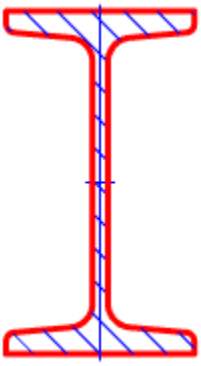


Options

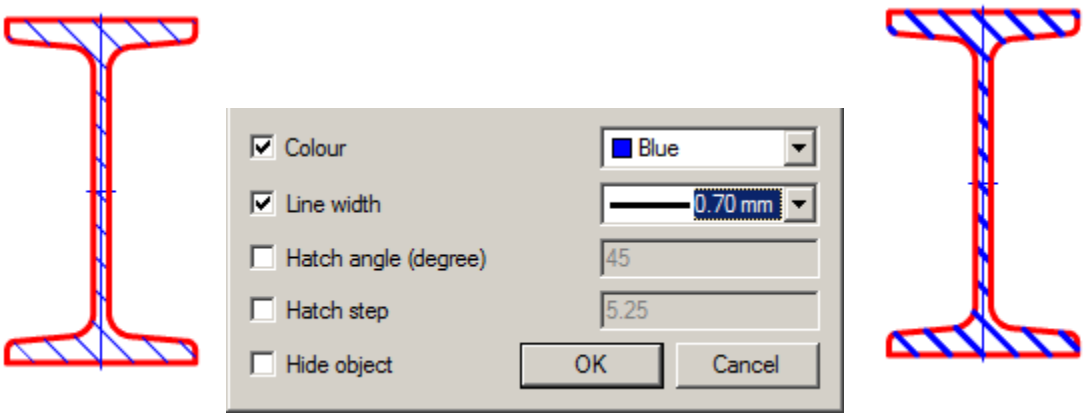
Colour -changes the color of the line.



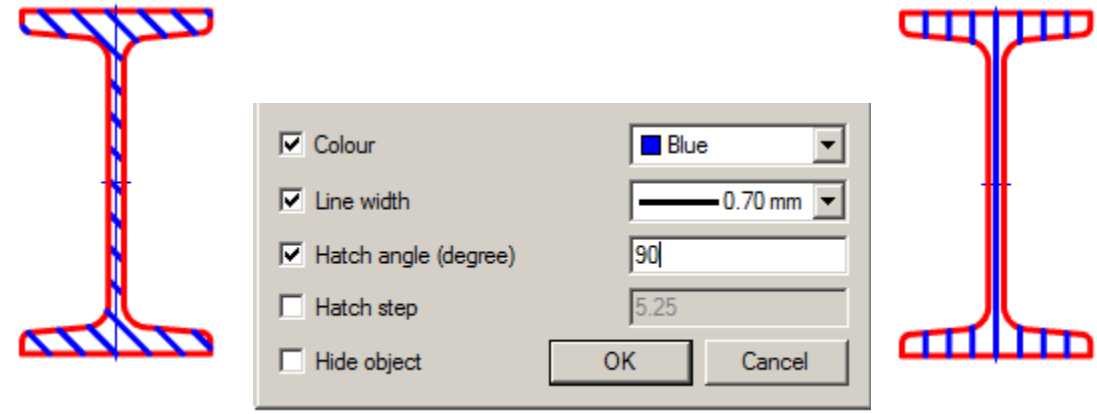




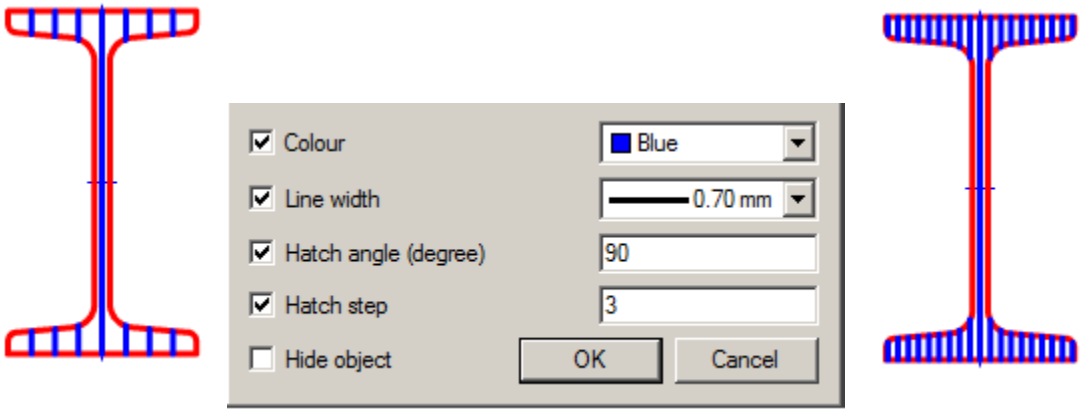
Line width - change the line width.



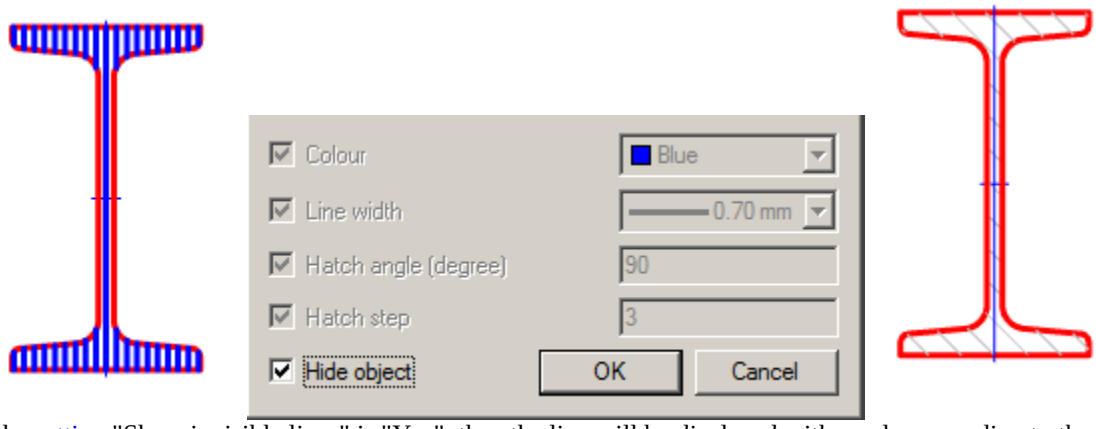
Hatch angle (degree) - controls the tilt hatch. The parameter is active only for the hatch.



Hatch step - changes hatching step. The parameter is active only for the hatch.



Hide object - it makes an invisible line. The other parameters are ignored.



If the setting "Show invisible lines" is "Yes", then the line will be displayed with a color according to the setting of "Color of invisible lines".

Designing

Design fasteners

Threaded fastening

Main menu: ***Mechanical - Design -*** ***Threaded fastening***.

Ribbon: ***Mechanical - Design -*** ***Threaded fastening***.

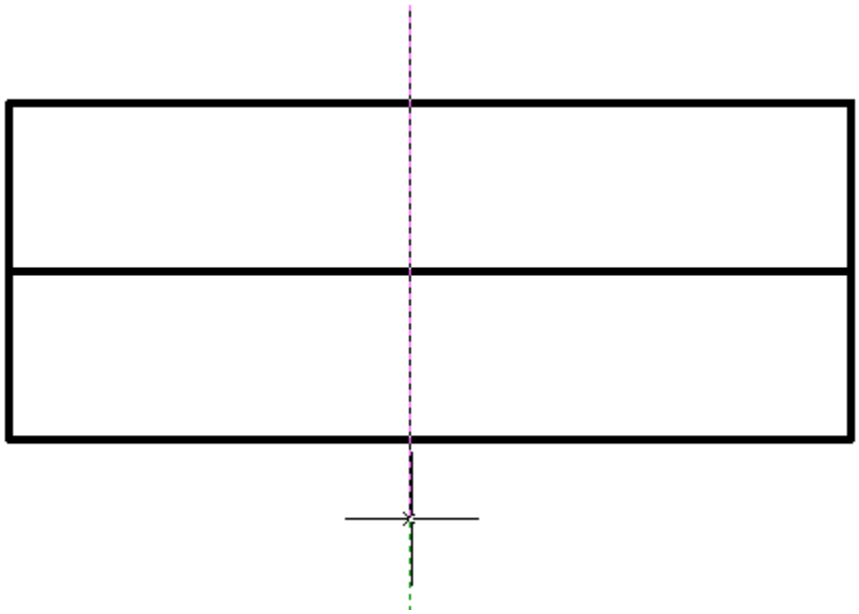
Toolbar: "Design" - ***Threaded fastening***.

Command line: ***MCJOINT***.

Tool for inserting a threaded fastening from library nanoCAD Mechanica 20.1.

Procedure

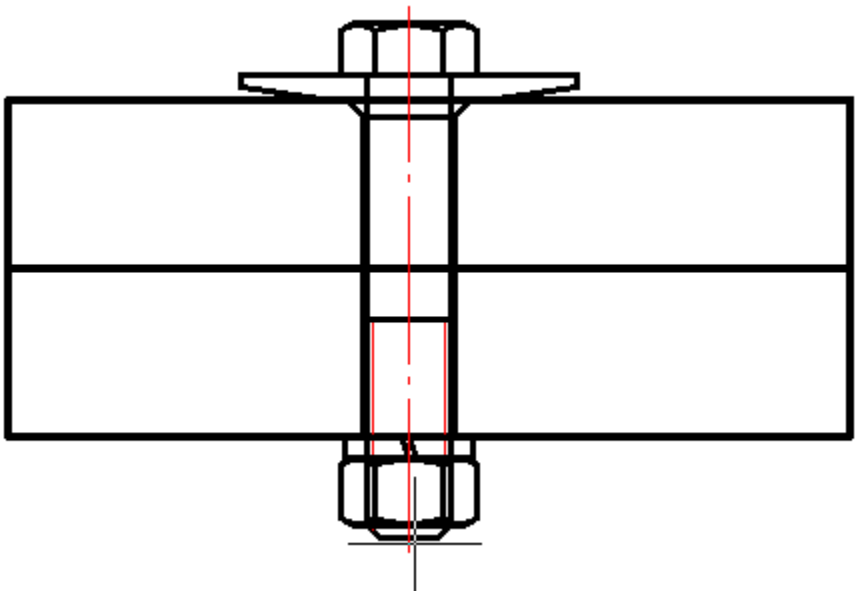
1. Specify the starting point of the threaded fastener in the drawing.
2. Specify the end point of the threaded fastener, the *"Fasteners"* dialog will open. When inserting, tools are available to select the direction of drawing. Specifying the start and end points defines the centerline of the fastener. The intersected lines make up the thickness of the part package being fastened.



3. In the *"Fasteners"* dialog box, customize the look of the threaded fastener.

Note:	The thread fastener insert dialog automatically saves the last selected template and thread diameter.

4. Confirm the settings with the *"Apply"* button, or with the *"OK"* button if you want to close the dialog at the same time. A threaded fastener will be built.

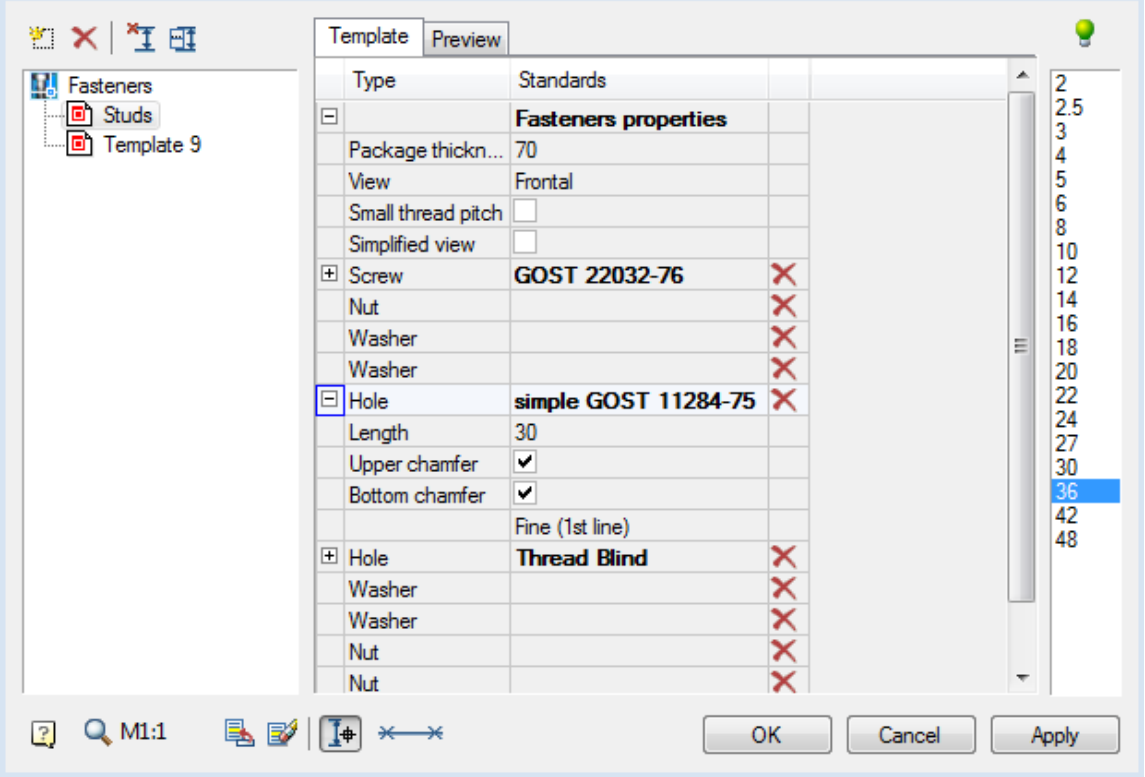


Editing a threaded fastener

You can edit a threaded fastener by editing the "Threaded fastener" object, or by editing individual parts included in the threaded fastener.
When editing, you can change the composition, diameter of the threaded fastener, as well as the parameters of the individual parts that are included in it.

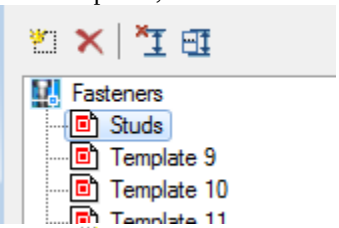
To open the threaded fastener setup dialog, apply any of the standard editing tools to the centerline of a group of parts.
If tooltips are enabled in nanoCAD Mechanica 20.1 settings, then when you hover the cursor over the centerline, the *"Threaded fastening"* tooltip appears.

Threaded fastener settings dialog box



Template list panel

It is possible to save a threaded fastener template for later reuse of assemblies with the same set of parts. To work with templates, use the buttons at the top of the dialog box, and below is a list of available templates:

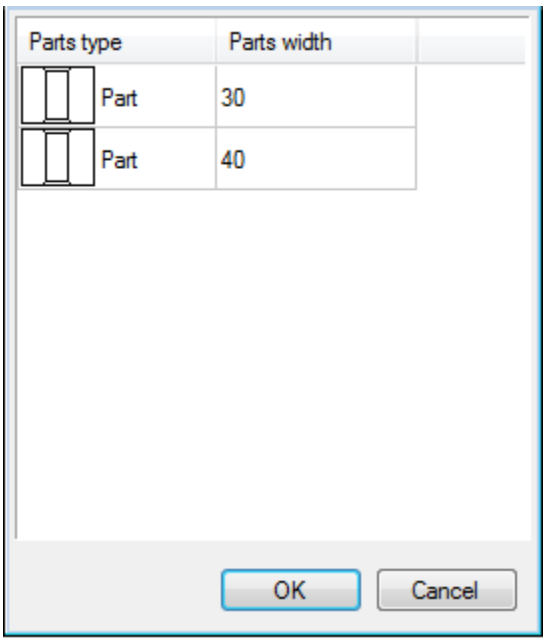


Button *"Create new template"* adds a new empty template. When creating a new template, the required number of holes is automatically added depending on the number of lines crossed in the drawing.

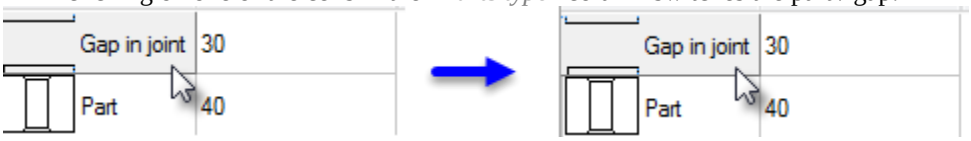
Button *"Delete template"* removes the selected template from the list. "Editing template" stores the current setting of the threaded fasteners. It cannot be removed.

Button *"Selecting insertion point"* is intended to reselect the insertion point and the length of the threaded fastener in the drawing.

Button *"Manually select holes length and placement"* opens a window for setting parameters of a package of connected parts.



LMB clicking on one of the cells in the *"Parts type"* column switches the part / gap:



In the *"Parts width"* column, the thickness of the corresponding part (or gap) is set. After setting the parameters of the connection package, close the window with the *"OK"* button.

The context menu of the list of templates contains the commands:

- Add folder
- Delete object
- Rename
- Import object
- Export object
- Send by email
- Copy
- Configuration tool

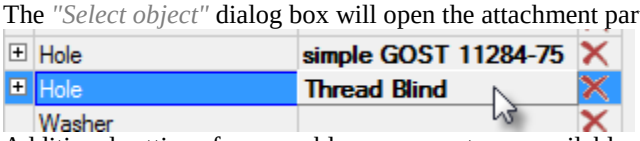
These commands are similar to those used in the object manager nanoCAD Mechanica 20.1.

Tab "Template"

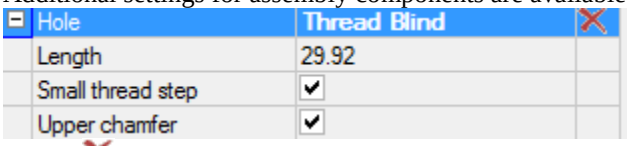
The following settings are available on the *"Template"* tab in the center of the dialog:

- Fasteners properties
  - *Package thickness* - thickness of the package of parts to be joined, in mm.
  - *View* - selection of standard image projection.
  - *Small thread pitch* - fine pitch switch.
  - *Simplified view* - simplified / full image switch.
- Details of the threaded assembly. The components included in the assembly are listed here in order.
  - Screw
  - Hole
  - Spot facing
  - Washer
  - Nut

To select a part of a threaded assembly from the nanoCAD Mechanica 20.1 base, click in the cell of the GOST column of the corresponding element.
The *"Select object"* dialog box will open the attachment parts folder.



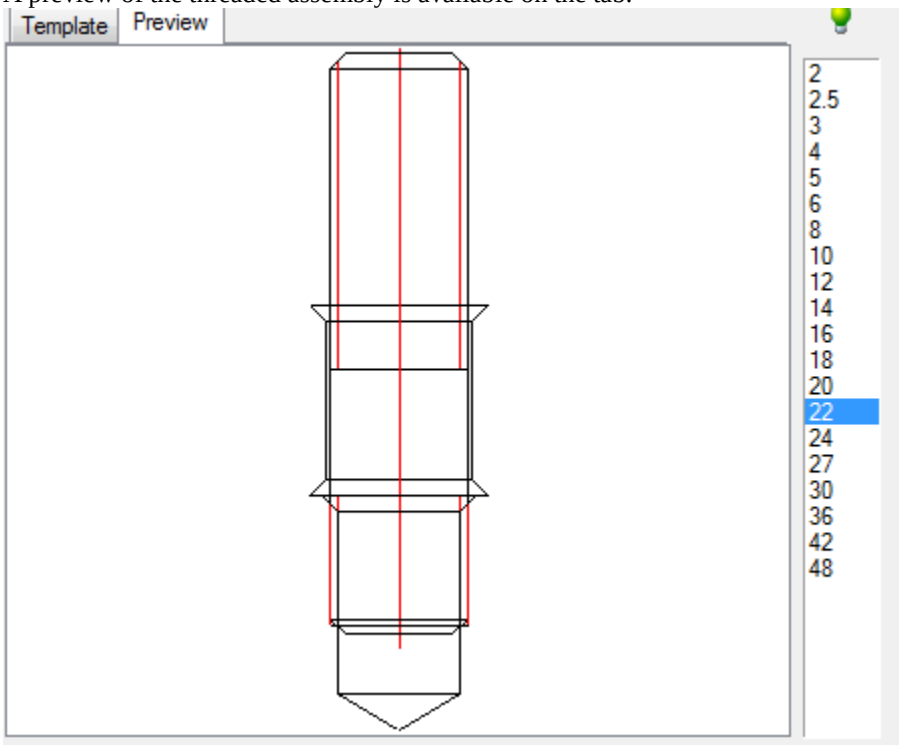
Additional settings for assembly components are available when expanding the list:



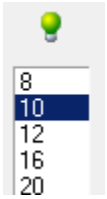
Button *"Clear"* in the right column of the list table clears the selected position.

Tab "Preview"

A preview of the threaded assembly is available on the tab.



List of thread diameters

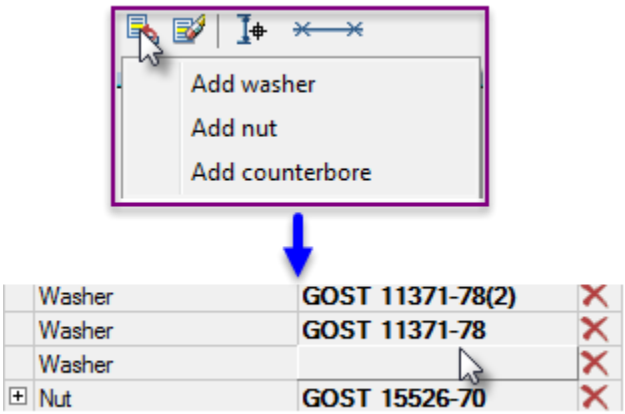


The *"lightbulb"* icon above the list indicates that the selected diameter value can be used for all objects in the assembly:

- The diameter value does not match one or more parts. Assembly is not possible.
- The diameter value applies to all parts in the assembly. Assembly is possible.

Button "Add part to template"

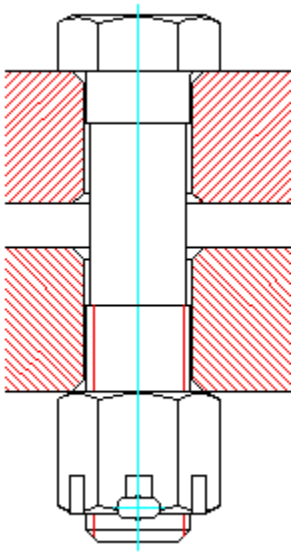
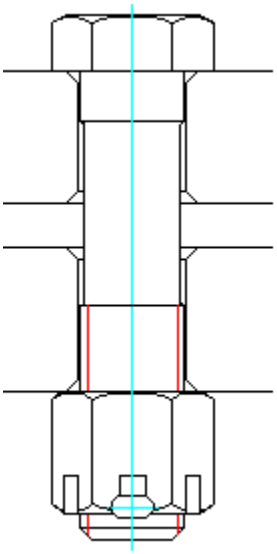
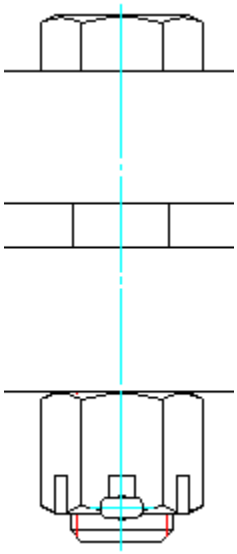
Adds a part to the assembly after being selected from the list. The list of details available for adding depends on the currently selected object:



Button ***"Remove part from template"***
Removes the selected object from the parts list.

Button ***"Dynamic selection"***
Turns on the mode of visual selection of the bolt length after closing the dialog.

Graphical display type selection list
When inserting a threaded fastener, 3 display methods are available, selected by the *"Select fasteners insert part"* button on the bottom toolbar of the dialog. The figure on the button displays the currently selected display method:

With local out	Normal insert type	Insert hidden fastener
		

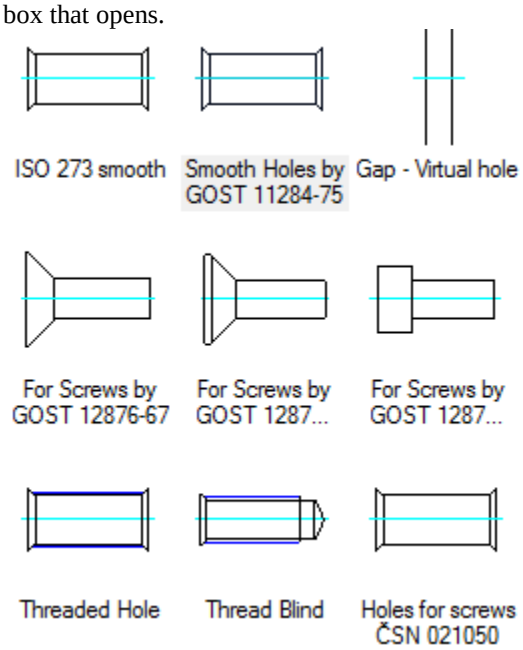
Displaying chamfers on holes

To select the options for displaying chamfers in the drawing, you must correctly place the marks in the hole properties.

⊟	Hole	simple GOST 11284-75
	Length	30
	Upper chamfer	✔
	Bottom chamfer	✔
		Fine (1st line)

Setting the hole type

To change the hole type, click on it in the connection part set and select a different type in the *"Select object"* dialog box that opens.



Rivet joint

Main menu: ***Mechanical - Design -*** ***Rivet joint***.

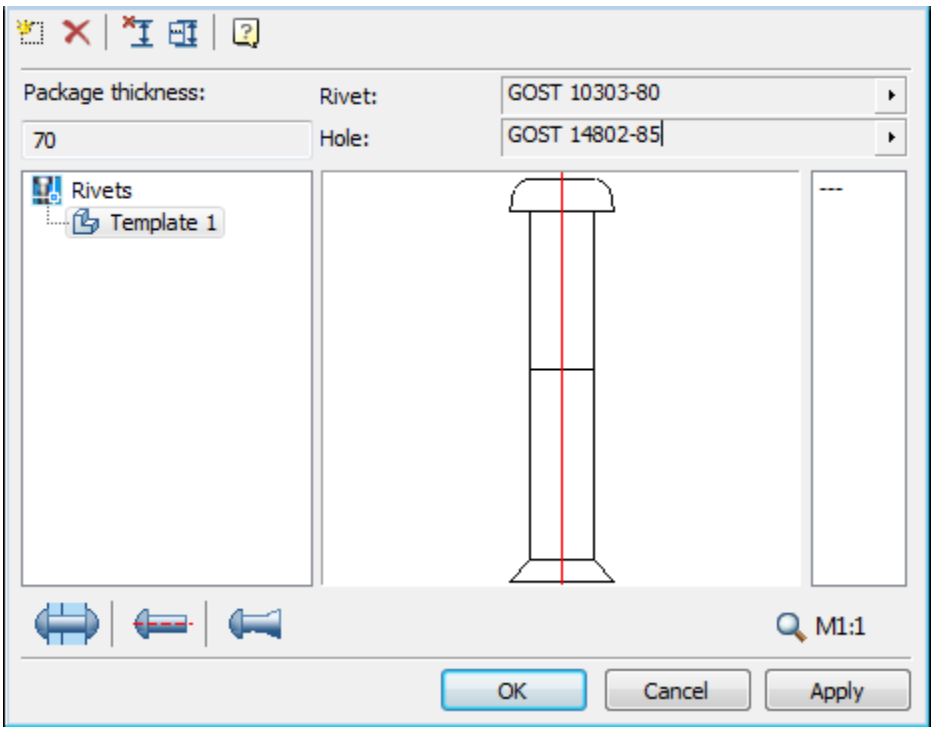
Ribbon: ***Mechanical - Design -*** ***Rivet joint***.

Toolbar: ***Rivet joint (*** "Design"***)***.

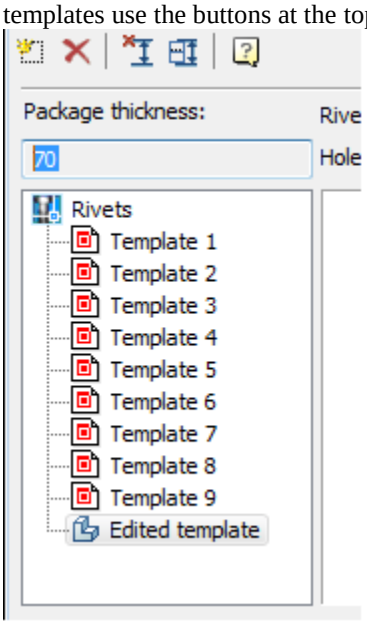
Command line: ***MCRIVET***.

Insertion tool riveted joint parts of the base standard elements nanoCAD Mechanica20.1.

1. Specify the starting point for drawing the rivet assembly.
2. Specify the endpoint of the rivet assembly. When you insert the available tools of choice destinations rendering .
3. In the dialog box ***mounting parts*** adjust the view of the rivet connections.



To select the type of rivet base nanoCAD Mechanica20.1 click in the *rivet*.
To select the type of base openings nanoCAD Mechanica20.1 click in the *hole for rivets*.
It is possible to save ***the template riveted joint*** for later reuse assemblies with the same set of items. To work with templates use the buttons at the top of the dialog box, and below is a list of available templates:

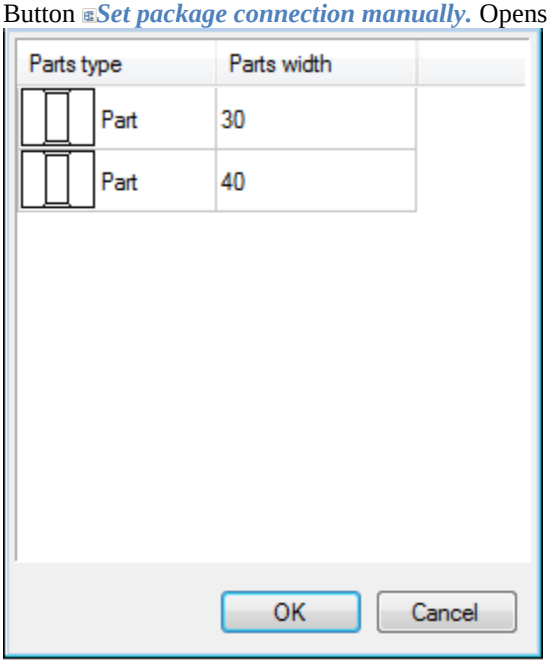


"Editable Template" stores the current settings bolting. It can not be deleted.
Button ***Create a new template.*** Adds a new blank template.
Button ***Delete template.*** Deletes the selected template from the list.

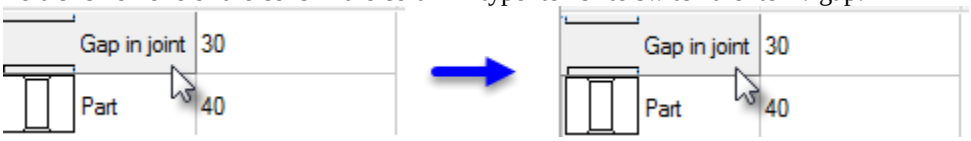
In the context menu templates available commands:

- Add Folder
- Delete object
- Rename
- Import from file
- Export to file
- Send by email
- Copy

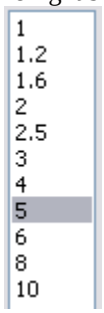
These commands are similar to those used in [in the Object Manager](#) nanoCAD Mechanica20.1.
Button ***Select the insertion point and the length of the riveted joint.*** Designed to re-select the insertion point and the length of the rivet assembly drawing.
Button ***Set package connection manually.*** Opens setup package of the parts.



Left-click on one of the cells in the column "type items" to switch the item / gap:



In the "Thickness" is set to the thickness of the relevant part (or gap) After configuring the package connected, close the window by pressing ***OK***.
The right side of the dialog box *mounting parts* is a list of values diameter rivets.



When you insert the rivet connections are available 3 ways to display selected button ***Insert mode riveted joint*** at the bottom of the dialog box toolbar. Drawing on the button displays the currently selected display method:

With a local cutaway

Normal Box

Insert hidden

2 types available (front and side) display riveted joint:

Front view

Left side view

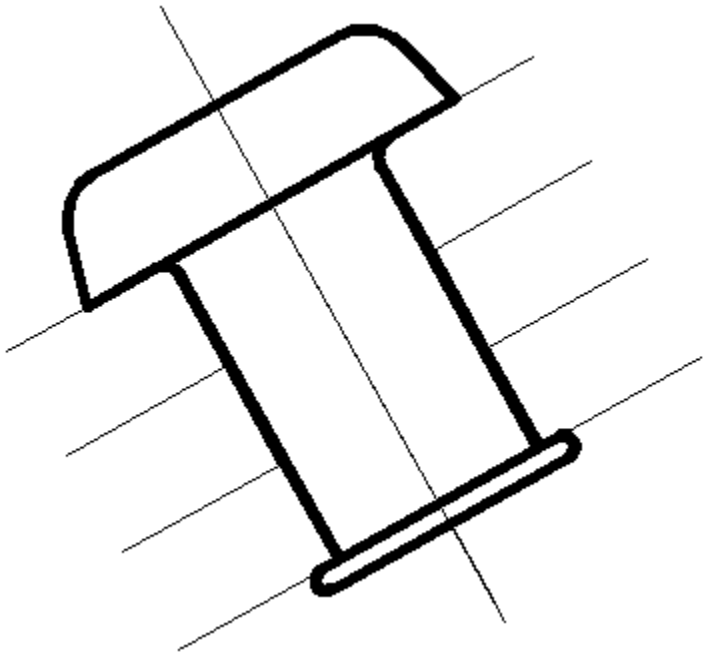
You can choose the *type rivet connections:*

	The number of types depending on the type of rivet.

Automatic detection of packet connection

When you insert the rivet connections made automatically recognize the boundaries of the parts. If the line crosses the insertion direction ***perpendicular to*** a few segments in the drawing, they are defined as the boundaries of parts and installs Connection:

Insertion point and the length of the rivet assembly are determined automatically based on the thickness of the package details.



Editing riveted joint

A dialog box settings riveted joint, apply any of the standard editing tools to the centerline of the group details. If the settings nanoCAD Mechanica20.1 included tooltips, then when you move the cursor to the center line will show "Rivet connection".

Remove fasteners

Main menu: ***Mechanical - Design - Delete parts***.

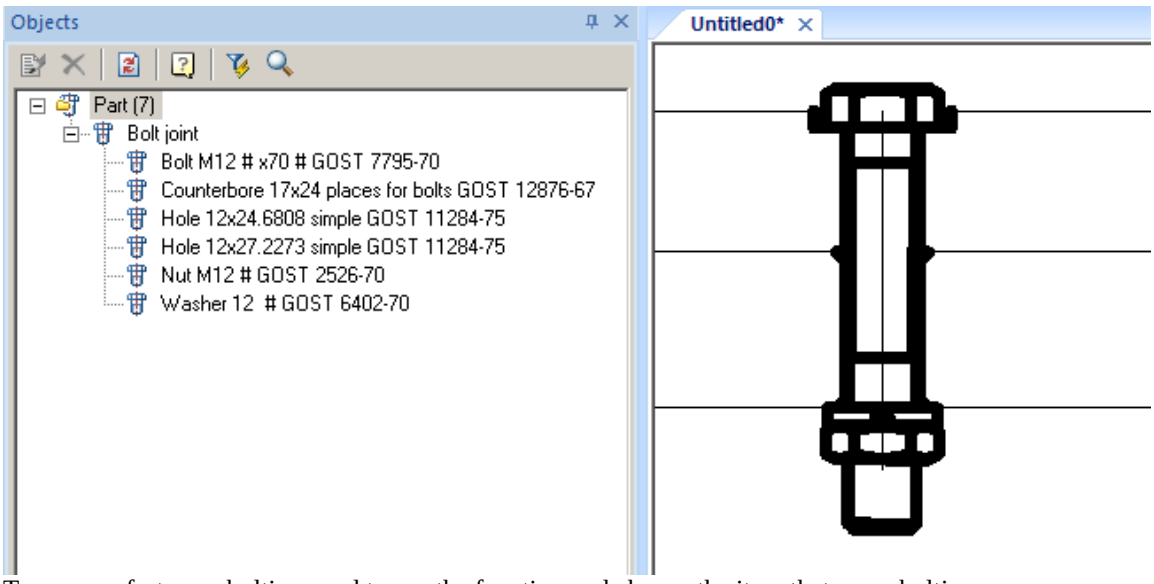
Ribbon: ***Mechanical - Design - Delete parts***.

Toolbar: ***Design - Delete parts***.

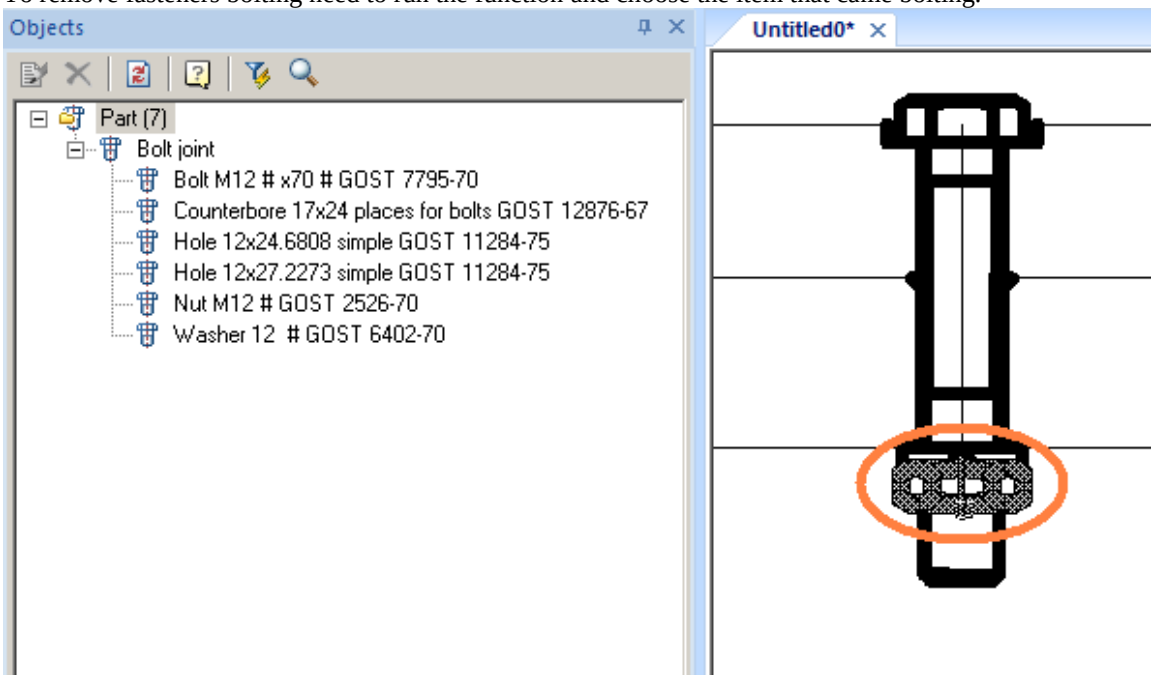
Command line: ***MCDELSPART***.

Operation

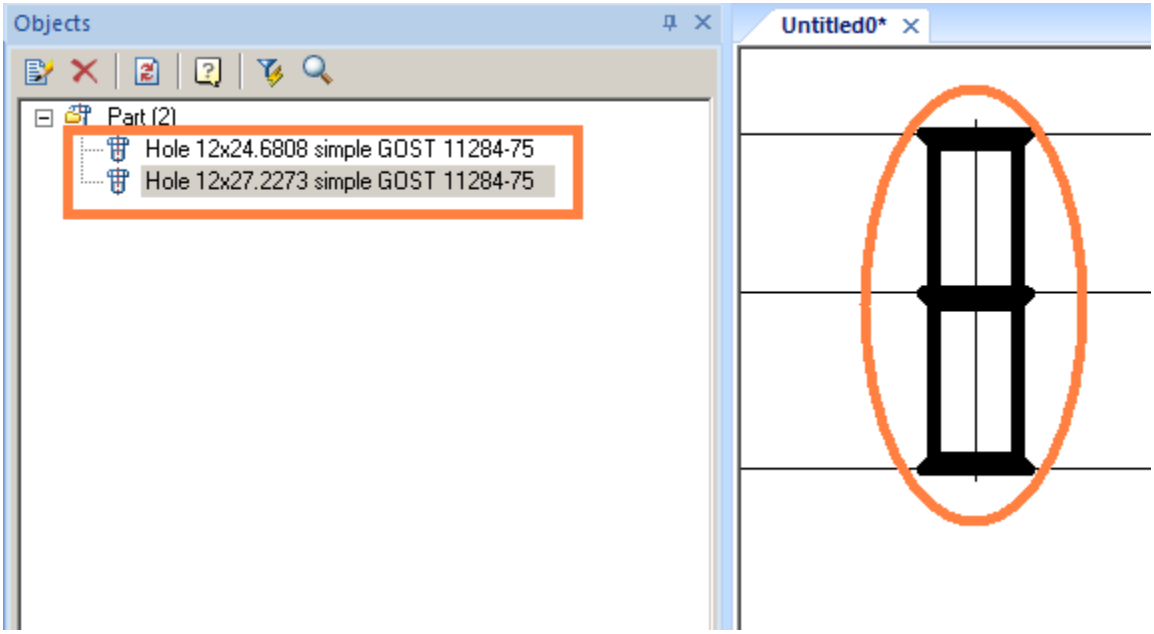
The function is intended to remove the fastener bolt or rivet connections. Openings by bolted or riveted to remain on the drawing and in the object manager.



To remove fasteners bolting need to run the function and choose the item that came bolting.



The remaining holes.



Details of rotation

Design details of rotation

Main menu: ***Mechanical - Design -*** ***Create shaft***.

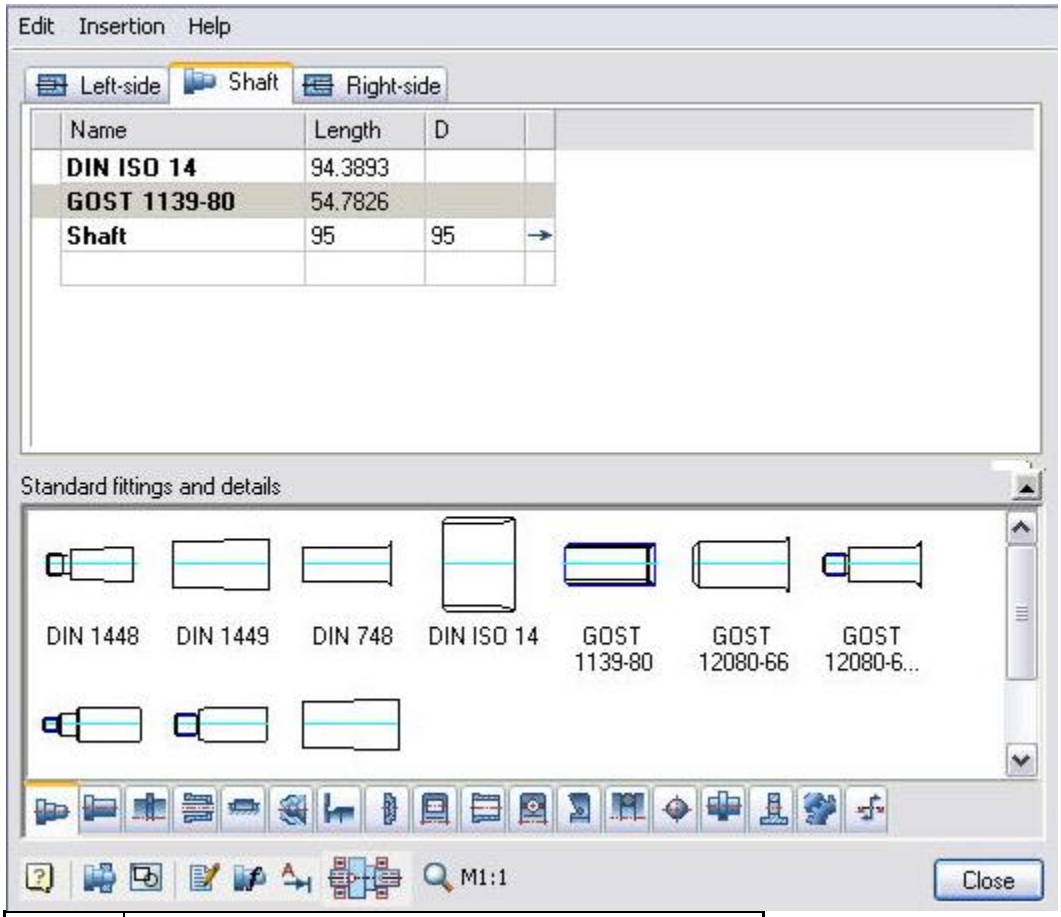
Ribbon: ***Mechanical - Design -*** ***Create shaft***.

Toolbar: ***Create shaft (*** "Design"***)***.

Command line: ***MCARBOR***.

Basic rules

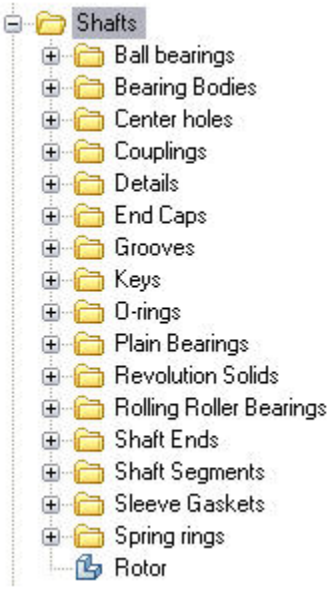
After invoking the command, pick an insertion point.
Set the shaft direction by moving the mouse.
Left clicking creates a shaft segment.The command remains active and allows additional segment to be drawn.
Construction site 20.1 auto-detects the insertion point of a graphic element which is to be inserted, and acquires its geometry from a standard database.The auto-dected elements are highlighted in color.After the insertion command has been invoked, elements are highlighted at the mouse movements.Left-click to confirm.
Dialog box ***Editg shaft***



	Show / Hide section ***Standard fittings and details sections***
	Add element
	Insert Group
	Edit Object
	Insert design
	Add view / section
M2:1	*Scale of details and shaft sections*

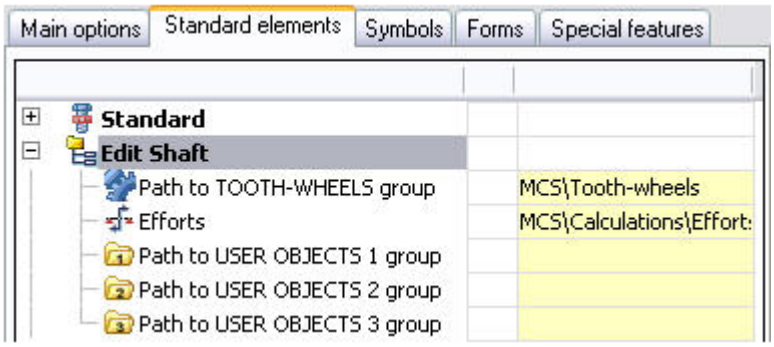
Shaft section displays methods

Composition tab "Standard fittings and details" is defined folder structure in the system folder "shafts" Database Construction Site 20.1.



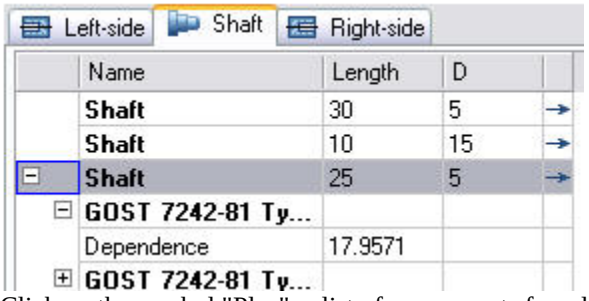
In addition, in the tabs of the Edit shaft dialog are included

- Tooth-wheels
- Efforts
- User folders



Path to these elements are specified in the option Construction Site 20.1 (***Standard Elements*** >***Edit shaft***).

Editing section of the shaft



Click on the symbol "Plus", a list of components for editing.
When you specify a string in the column names in the drawing selected component is highlighted.
To modify a standard component, double-click the left mouse button on the name of GOST. In introducing the new values for the editable portion of the shaft all the changes are immediately displayed in the drawing.

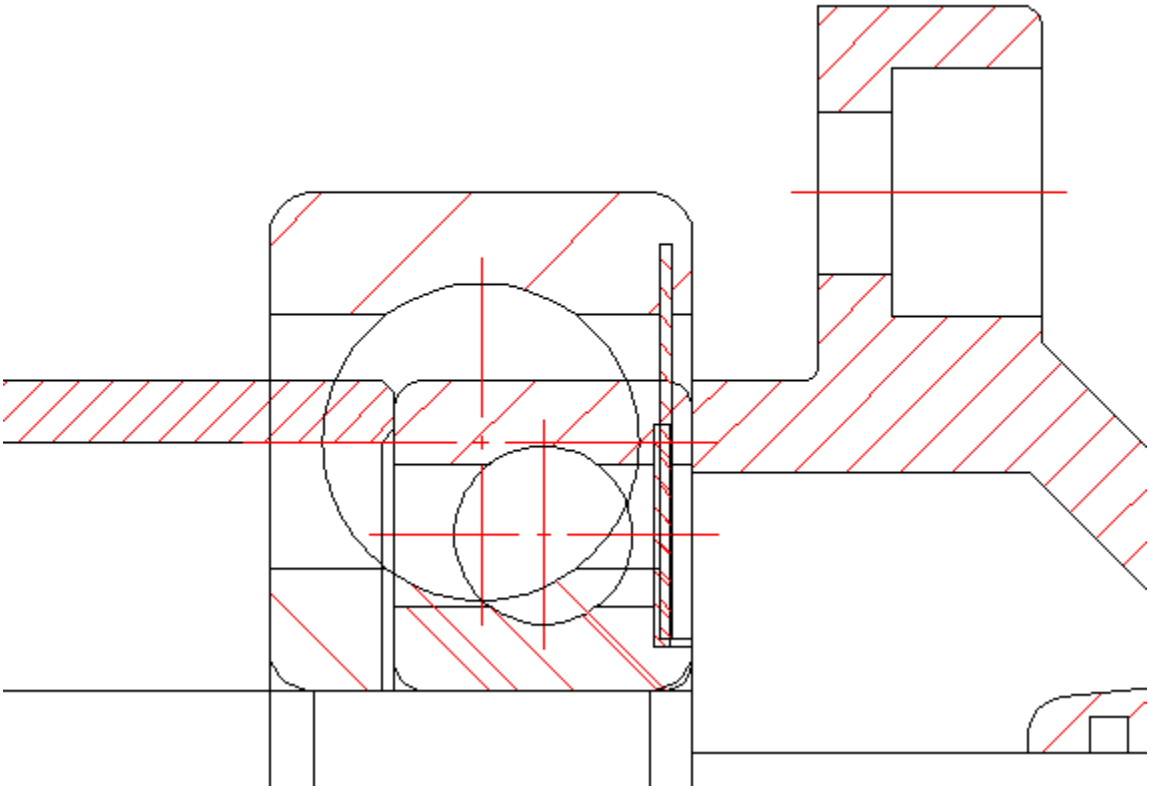
Purpose size details in the dialog box

To select the size of the base bearing the dialog window *Edit shaft.*
Specifying the column *name* to GOST bearing, check option; bearing the figure should highlight.

Double click the left mouse button on the designation of the bearing or click once on the icon to edit .
In the dialog box, this bearing sizes choose. Selection window will size bearing.
Complete the command by pressing OK and close the dialog box.
Similarly edit all items bearing assembly.

Dynamic display sizes details

The quickest way to selecting the size of parts - a dynamic view of all of its dimensions by moving the mouse pointer to the left-right and up-down.
Specifying the insertion point of the bearing on the shaft, select size bearing.



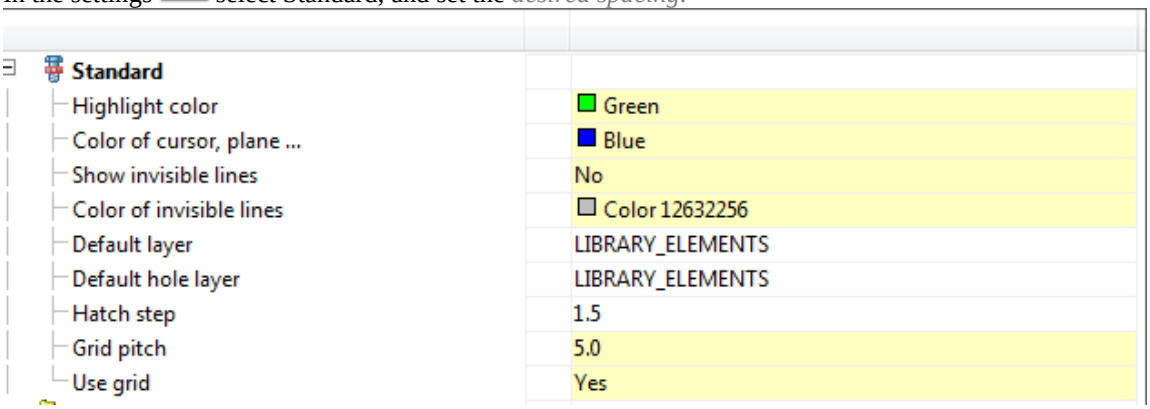
When you move the pointer of the available bearing sizes select those that meet the internal diameter of the shaft and the combination of the bearing width and outer diameter correspond to the current cursor position.

Export shaft to an external file

For detailing received shafts can be exported to an external file. Double clicking the left mouse button on the shaft axis edited activate dialog *Editing shaft.*
In the dialog box, display the *Edit* and select from the drop down list of commands string *Export to a file*.
Then you want to fill the frame and title block specify the file's location.

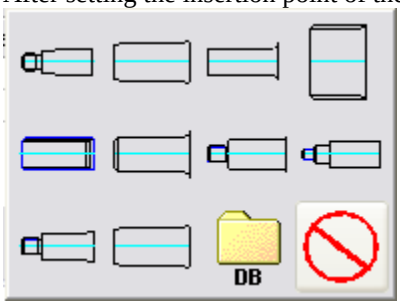
Setting step drawing

In the settings select Standard, and set the *desired spacing.*



Setting the stage shaft

After setting the insertion point of the shaft when rendering, you can use a dialog box to set options stage:



After clicking on the *shaft* , you can choose the types of shaft ends.

Determining the size of shaft section in the status bar

When rendering stage shaft in the status bar nanoCAD in the lower left corner of the window displays its properties.

Determining the size of shaft section in the dialog box

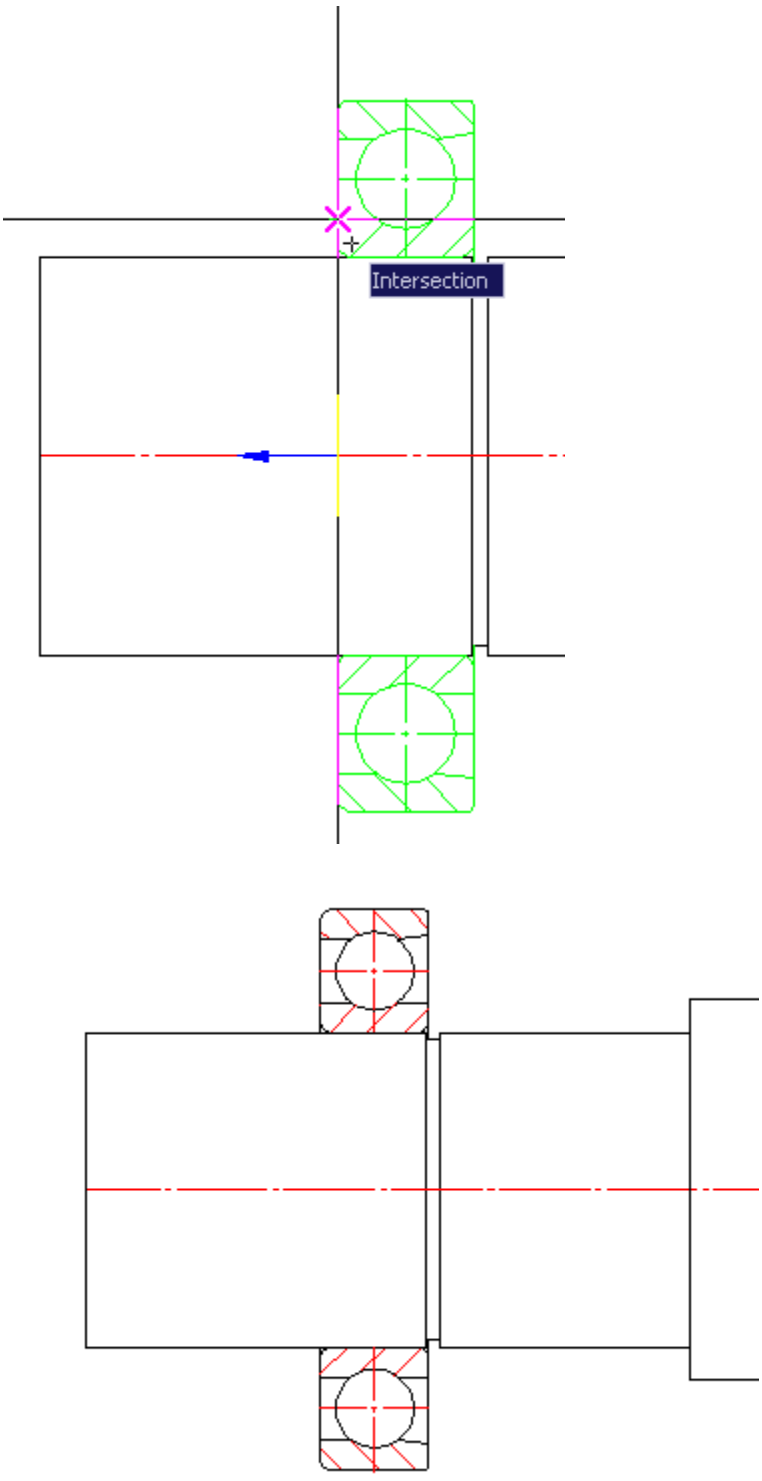
When you specify a dynamic image stage shaft, the right-click context menu and enter the D-line dialogue.
A dialog box shaft , in which you can enter the exact values of the diameter and length of the shaft portion.

How to edit areas of the shaft?

Edit shaft areas in several ways:
"Visual Editing": select an editable area and dynamically change its geometrical dimensions.
Specifying the exact plot sizes in the dialog box, double-click on the axis of the shaft left mouse button and change the section of the shaft in the dialog box *Editing shaft.*
Another way to call the same dialog box - point to the shaft axis and click the right mouse button.

Placing snap ring groove

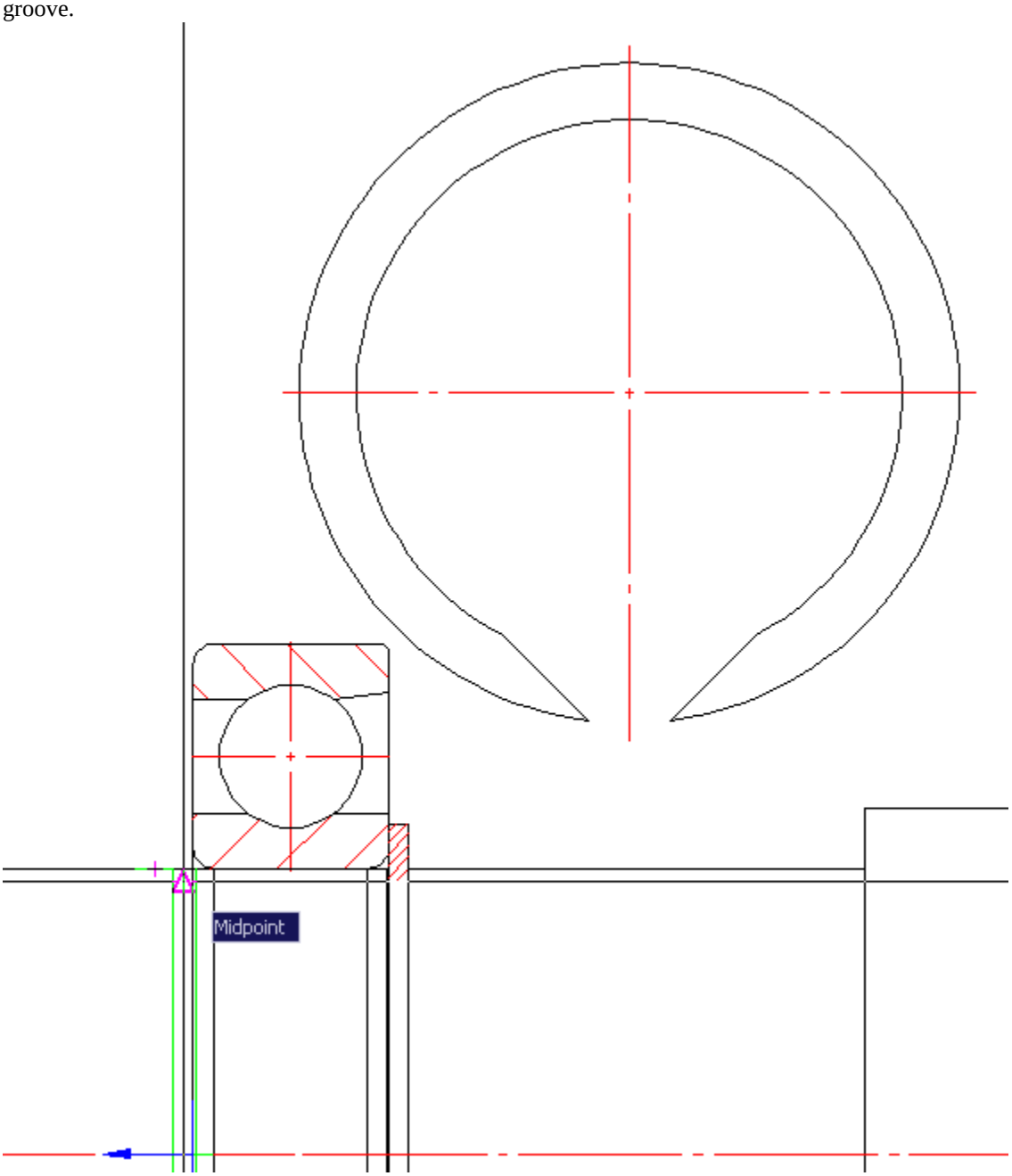
When placing grooves retaining rings for automatic recognition of the diameter of the shaft and the insertion point on the shaft must first be placed bearing.



To specify the position of the groove snap ring must be attached to the shaft or to the end of the bearing. Automatically set the dependence on the position and diameter of the retaining ring \ grooves.

Placement of the retaining ring

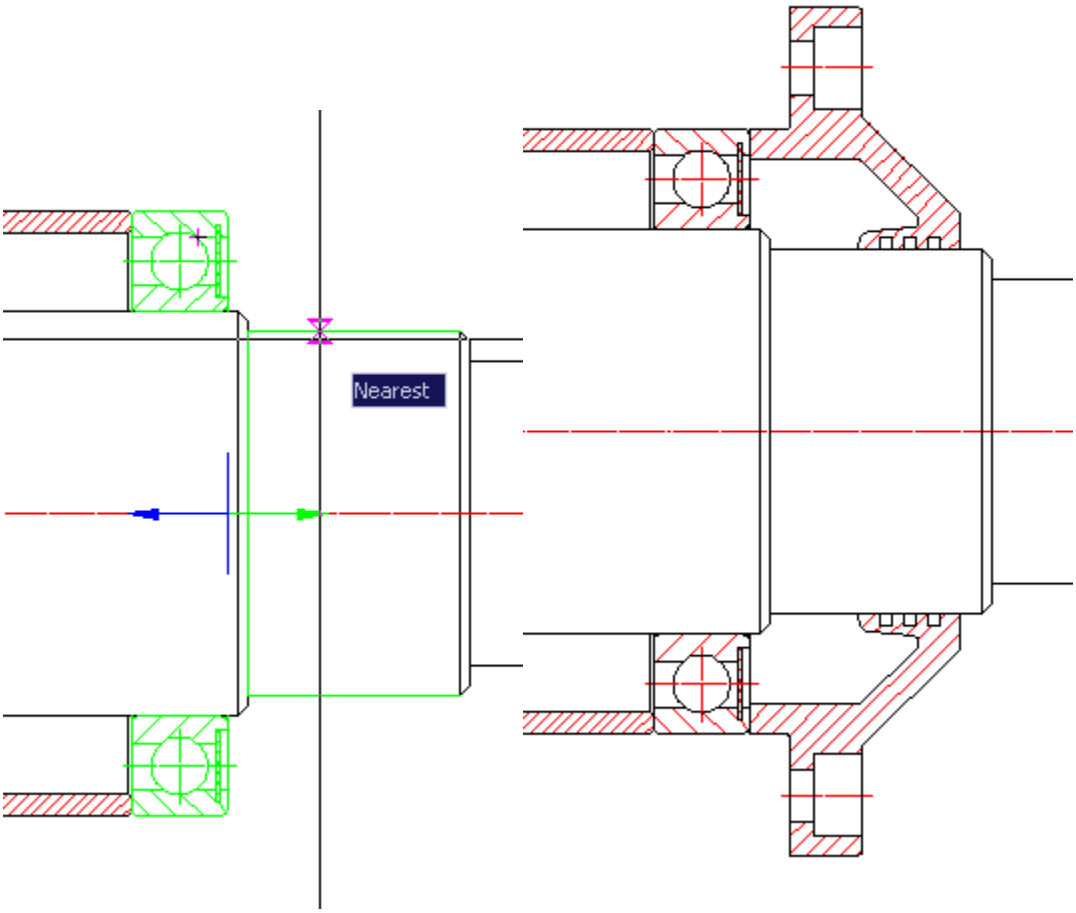
When placing the retaining rings for the automatic recognition of their size on the shaft must be pre-arranged groove.



To indicate the position of the ring is enough when you insert attached to the groove (it is highlighted in the appropriate color).

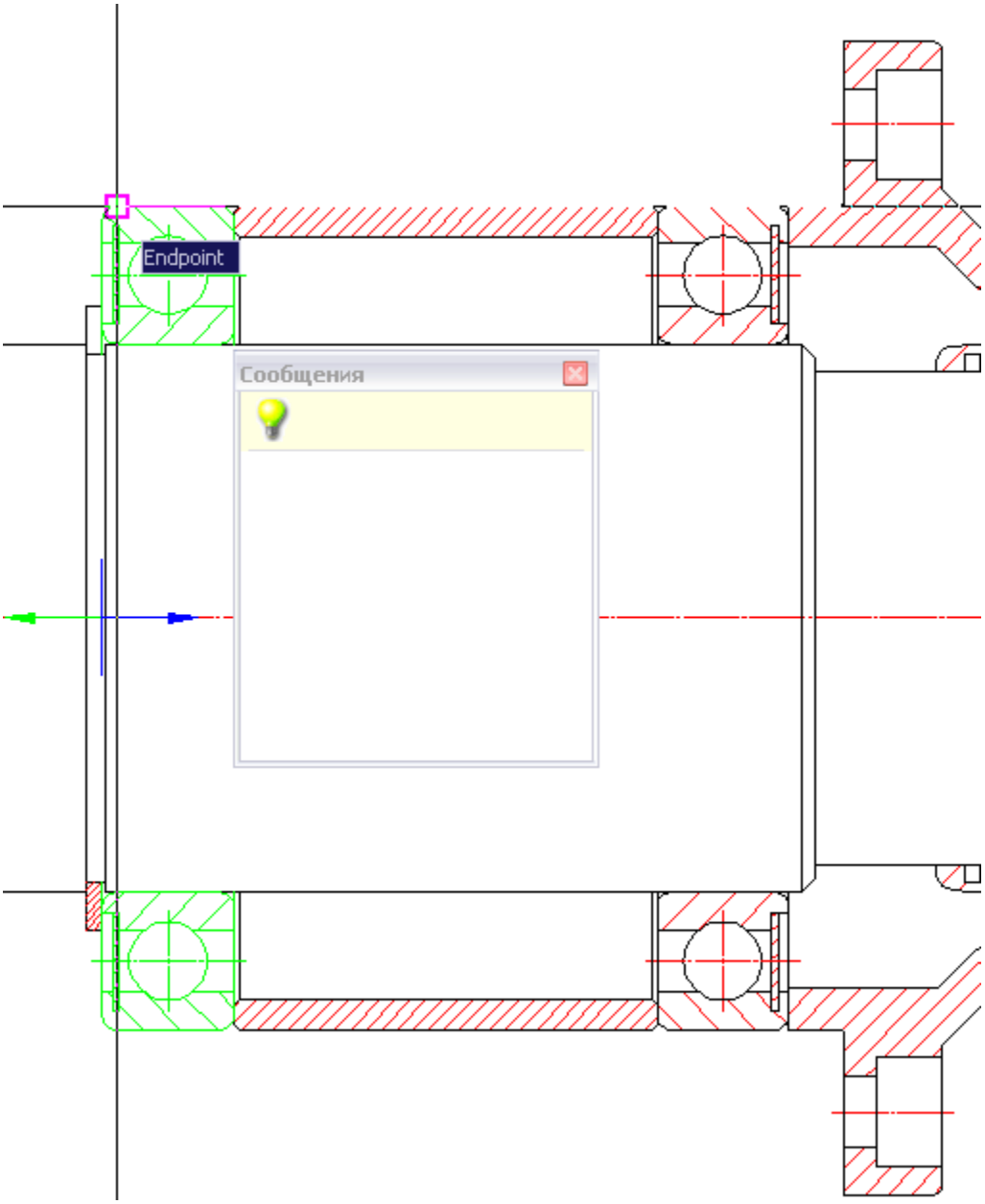
Accommodation details "cover"

For details of accommodation "cover" is necessary to consistently left-clicking on the bearing, and then on the shaft portion. Automatically creates a cap with the attached parameters of the situation, the diameter of the bearing and the shaft diameter.

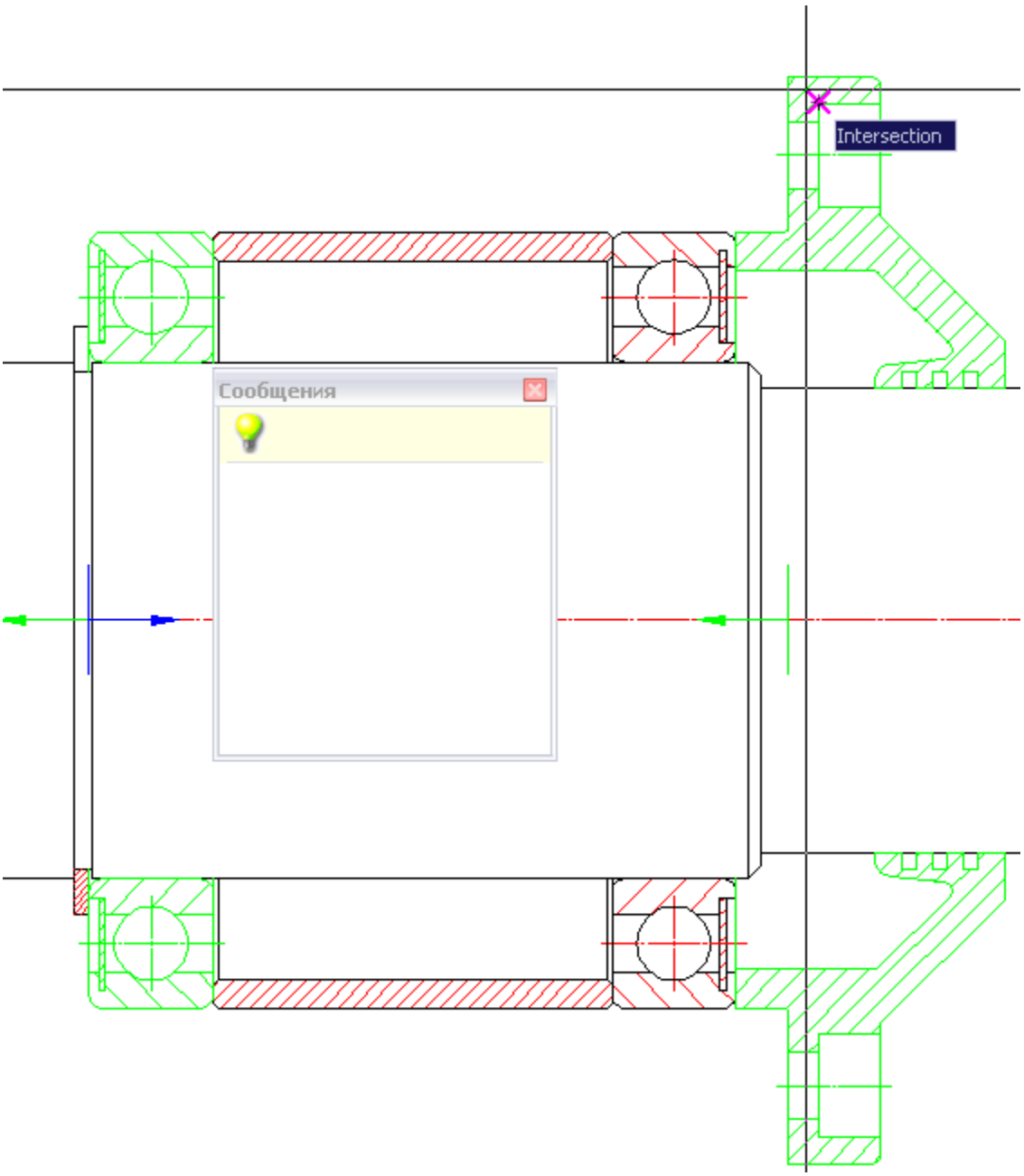


Accommodation details "Glass"

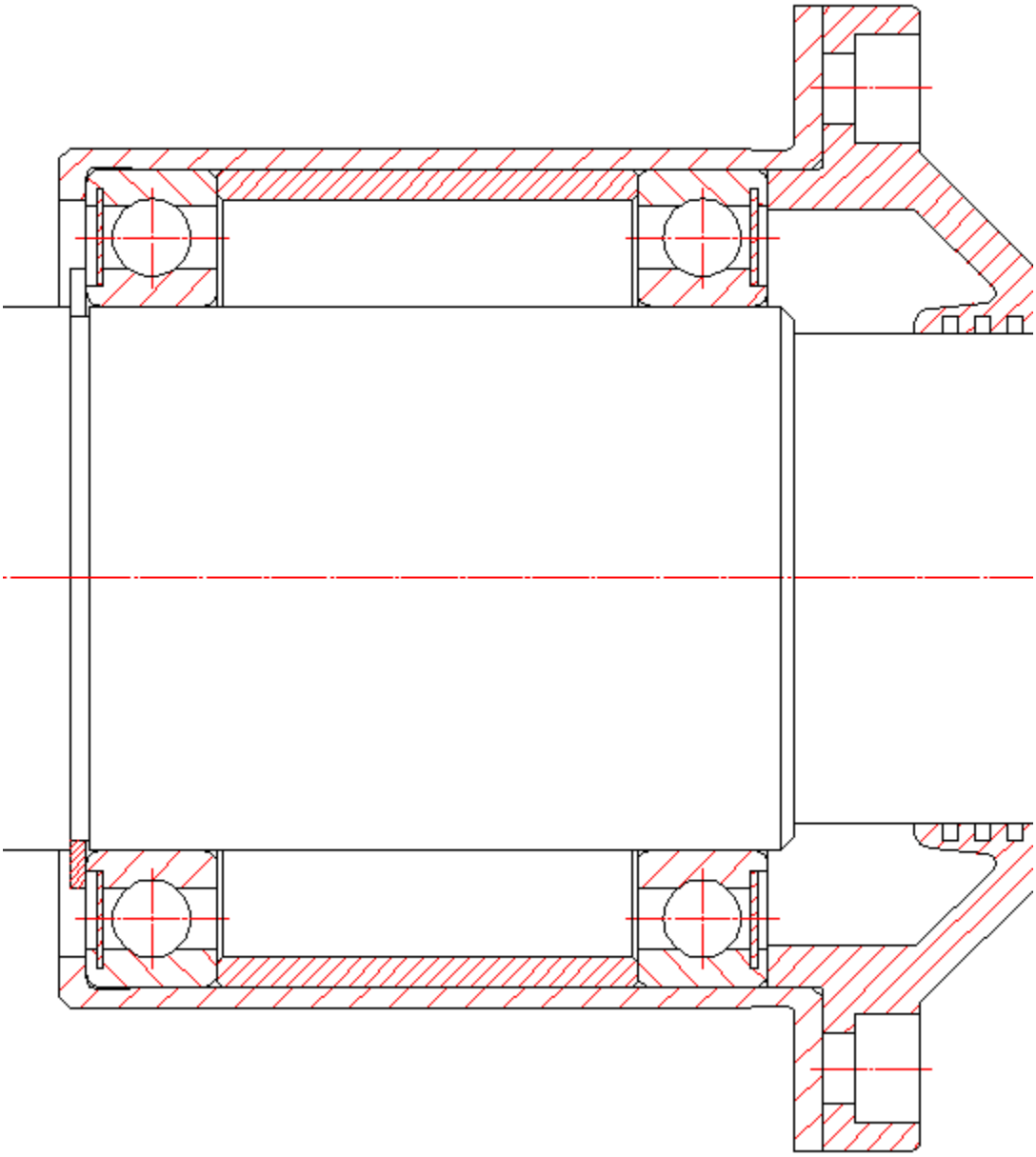
When placing parts "Glass" is necessary to specify the series bearing cap.
Choose from a glass base nanoCAD Mechanica 20.1 part, direct the cursor on the bearing - he highlighted. Click with the left mouse button.



It is further proposed to select the cover - direct the cursor on the cover, it is highlighted - click the left mouse button.



In the dialog that appears, click OK insertion cup.



Templates bearings

In the design of supports shafts are encouraged to use the mechanism of group insert multiple pieces of bearing select the desired template. If no suitable template support, it can be created. This technology is similar to technology insertion into a drawing fasteners from the selected template.
The pattern of support may include the following parts: bearings, wear rings, bushings, glass retaining ring grooves for them, locking cover, parts of gears and other parts of the base of standard parts.
Having a set of templates with a single command, you can immediately draw the option selected in the drawing of bearing.

Insert the template bearing support

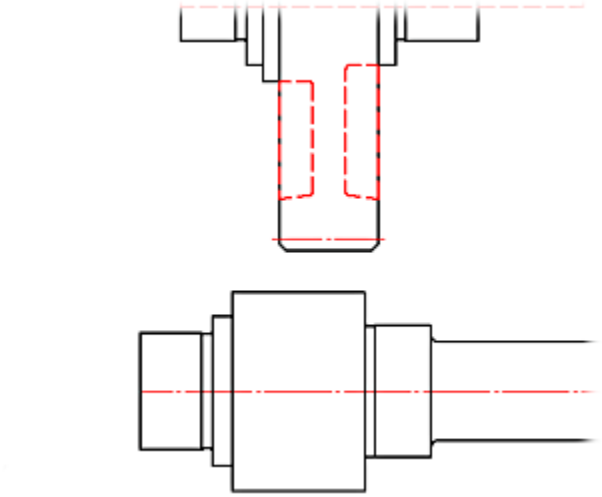
Double click the left mouse button on the axis of the shaft of the edited call the corresponding dialog box. At the bottom of the window, click on the icon *Insert group*.

Design pinion shafts

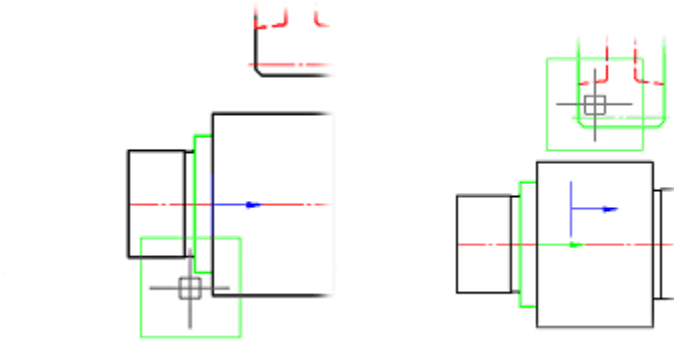
In the dialog box, select the folder Editing shaft gears and select a template cylindrical gear shaft.
After selecting the parts "cylindrical gear shaft" mouse pointer is displayed as a standard cross-hair, blue arrows and the image of a small green square (standard view). When you hover the pointer on the shaft, this area will be highlighted in green and contains the insertion point of the ring gear.
In the dialog box, spur gear option
Take the rate you need to click on the drop-down list of the calculations and indicate which item you want to transfer
render (or gear wheel).
The option is available if a calculation has been created. To insert an object into a chain of objects, click OK. After applying the gear, the cursor pointer takes the standard form. It is further proposed to continue drawing the next part of the engagement.

Drawing on the results of the calculation of gearing

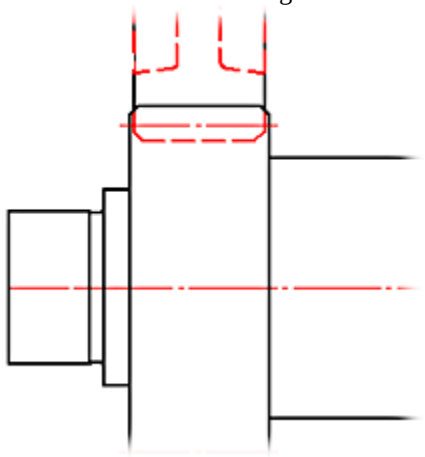
Original drawing:



Double click the left mouse button on the axis of the shaft, call the dialog box Editing shaft.
Tab gears.
Select a template cylindrical gear shaft.
By clicking the left mouse button, set the insertion point on the lower shaft gear.



Move the cursor pointer to the wheel on the upper shaft (ring gear is highlighted in green). Left-click to confirm your choice.
In the drop-down list of the calculations, specify the desired (in our example - calculation with the name of "Example of calculation") and select what should render the lower portion of the shaft (in this example - "Gear").
Click the OK button. At the request of the "Insert object in the chain of objects?" Answer "yes".
Press ESC to finish drawing commands engage and close the dialog box.



As a result, the drawing rendered pair of gearing with deposited assembly dependencies.

Shaft chamfer

Main menu: ***Mechanical - Design - Shafts -*** ***Shaft chamfer***.

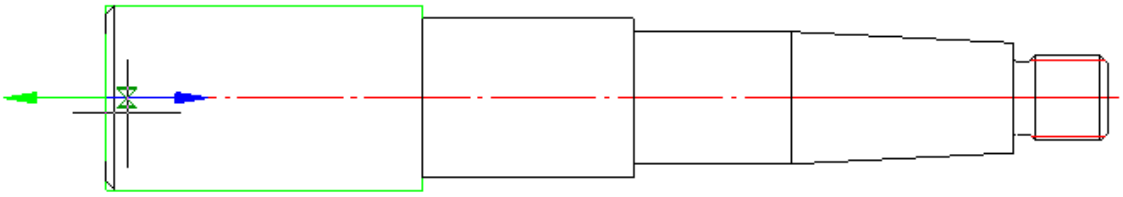
Ribbon: ***Mechanical - Design -*** ***Shaft chamfer***.

Toolbar: ***Shaft chamfer (*** "Design"***)***.

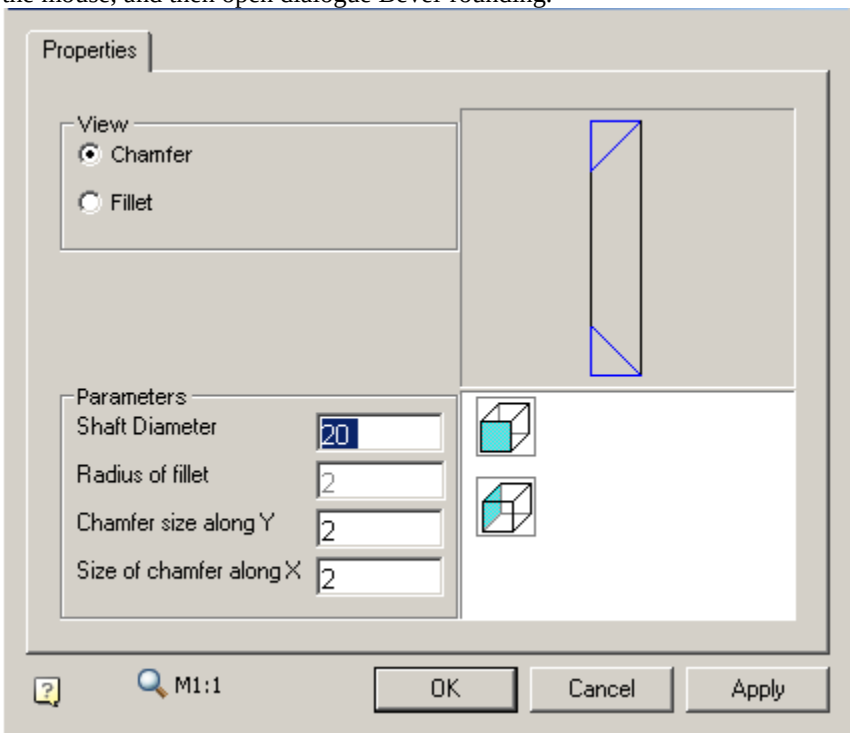
Command line: ***MCACHAMFER***.

Operating procedure

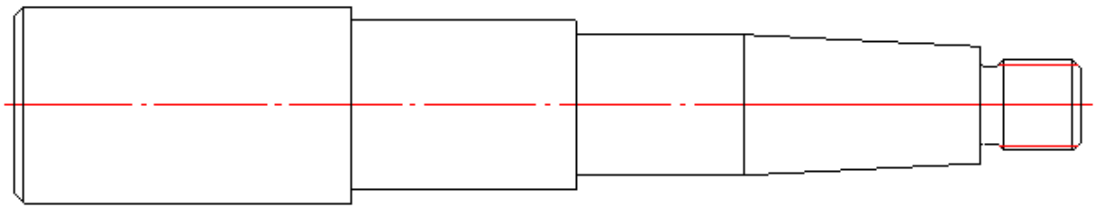
To insert a chamfer is necessary to run a command and select the portion of the shaft.



The chamfer is automatically adjusted to the diameter of the shaft portion. Fixing chamfer happens when you click the mouse, and then open dialogue Bevel-rounding.



After closing the dialog construct chamfer automatically.



Shaft fillet

Main menu: ***Mechanical - Design - Shafts -*** ***Shaft fillet***.

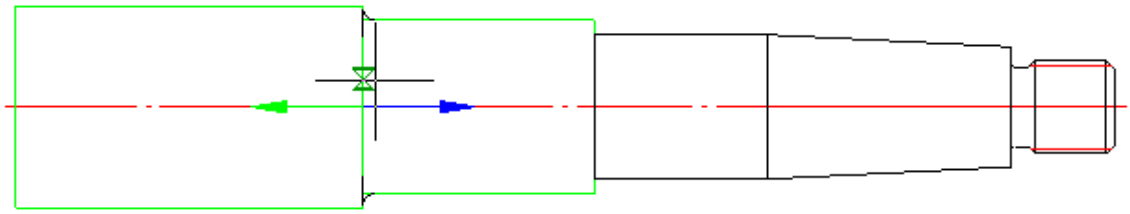
Ribbon: ***Mechanical - Design -*** ***Shaft fillet***.

Toolbar: *Shaft fillet (* "Design"*)*.

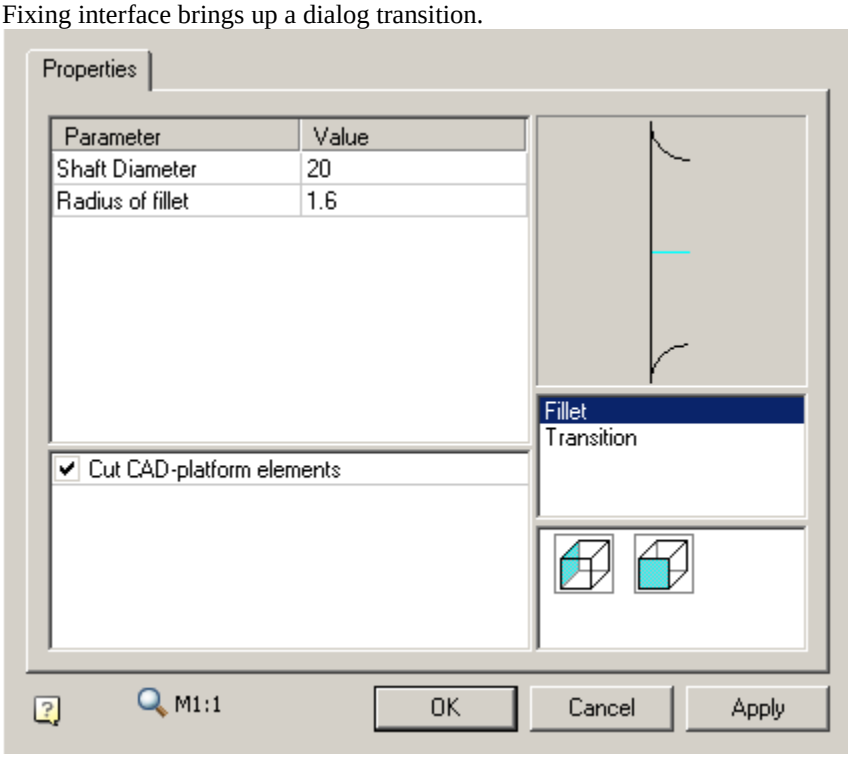
Command line: ***MCAFILET***.

Operating procedure

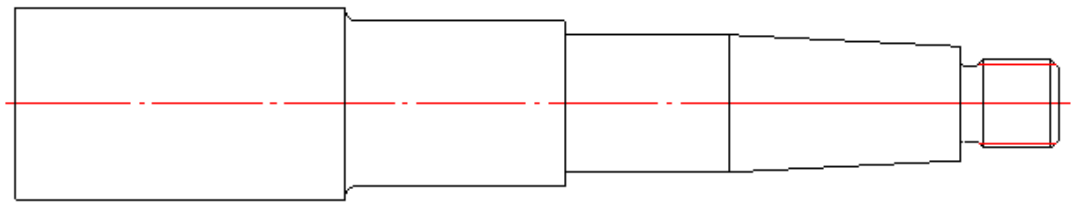
To insert, you must run the command interface and mating portions of the shaft.



Fixing interface brings up a dialog transition.



After closing the dialog interface to build automatically.



Pipelines

Main menu: ***Mechanical - Design -*** ***Pipelines***.

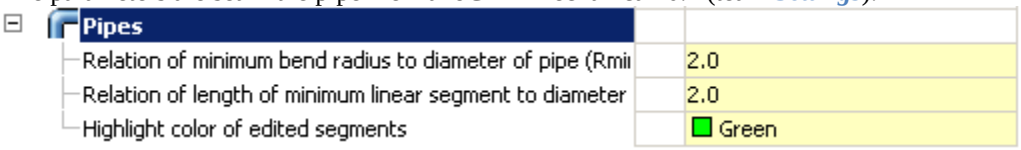
Ribbon: ***Mechanical - Design -*** ***Pipelines***.

Toolbar: ***Pipelines (*** **"Design"*)*.

Command line: ***MCPIPE***.

Two ways to build a pipe path

First - automatic connection of two pipe fittings elements.
When joining the program selects the shortest distance: pipe length is calculated from the minimum values of the linear section and minimum bending radius of the pipe diameter.
The parameters are set in the pipeline nanoCAD Mechanica 20.1 (team ***Settings***).



To automatically connect the two valve elements specify the connecting member (bushings, elbows, tees, crosses), hold down SHIFT. nanoCAD Mechanica 20.1 generate trajectory tubes and displays the dialog box with the geometric size of the segment of the pipeline. If the proposed path will not suit you, it can be edited in the same window.
Second - step by step construction of the pipeline with a consistent set the direction and length of the pipe segment.
Step task trajectory pipe begins with the first element of the valve. The insertion point and pipe diameter are determined automatically.
Next, follow the prompts on the command line, you can create the path of the pipeline, consistently introducing straight and arched portions.
For automatic connection of the pipe end to an element of reinforcement necessary to open the context menu.
Select option C bonds and provide an element of reinforcement with which to make the connection.
Call the following context menu again by pressing the right mouse button.
Select it as the pipe sections need render.
The compound can be performed in three or five sections.
By clicking the right mouse button to complete the command.

Sequence of piping design

Apply hole.

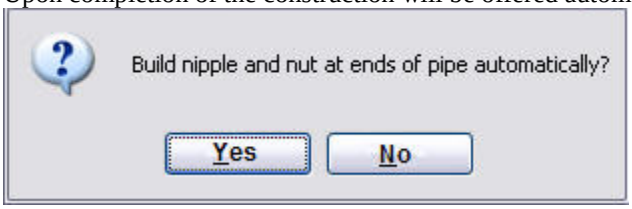
Execute command Inserting *pipeline fittings*.
Choose from a database element "Hole taper thread."
Specify the coordinates of the conical hole.
In the *opening taper threaded* set nominal thread. On the second tab, *Properties* , set the length and diameter of the hole drilling.
Complete the command by pressing ESC.
Place details of the reinforcement.
Assembly according to angles and taper thread throughs applied automatically.

Execute command *Inserting pipeline fittings*.
Choose from a database element "Prohodnik."
Hover your pointer to the hole and click the left mouse button

Complete the command by pressing ESC.

For pipelines on external cone

Apply pipe .
Upon completion of the construction will be offered automatically add nipples and nuts to the ends of the pipeline.



If nipples and nuts have been added automatically, you need to place them manually.

Select command *Inserting pipe fittings*.
Choose from a database element "Nipple".
Hover your pointer to the end portion of the pipe and left-click
Select size and click OK.
Complete the command by pressing ESC.

For pipelines on internal cone

When selecting the pipeline on internal cone selection dialog appears connects equipment automatically added to the compounds of the inner cone, depending on the type of environment and the type of nipple (spherical or hemispherical). For a spherical pin structure parts will be as follows:

- GOST 16043-70 nipple
- GOST 16048-70 lock ring
- GOST 16047-70 Bolting

For hemispherical pin composition details:

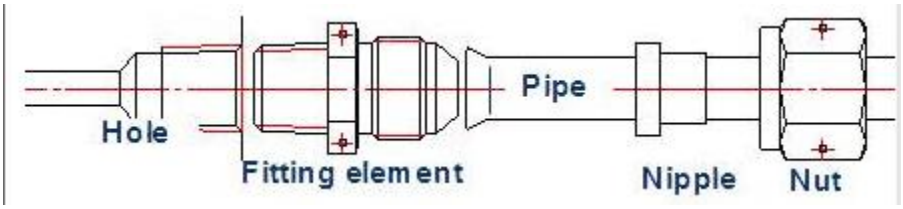
- GOST 16042-70 nipple
- GOST 16046-70 Bolting

Thread the other end of the pipeline construction include:

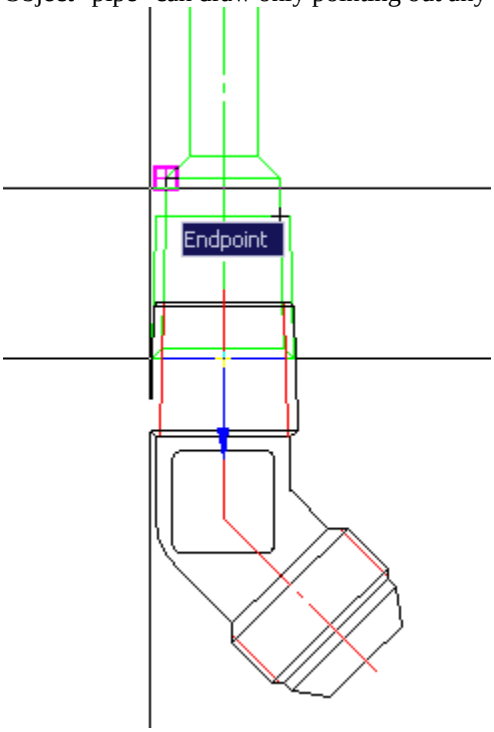
- Choose hardware - reappears dialogue "set of equipment."
- Building and - at the second end of the conduit parts to be added to set the same parameters and composition as the first one.
- Do not build - further details will not be built.
- Example of construction
- Collective inserting objects
- Set the method of overlapping pipelines
- How to make a connection in the context?
- Editing trajectory pipeline

Example of construction of the pipeline on external cone

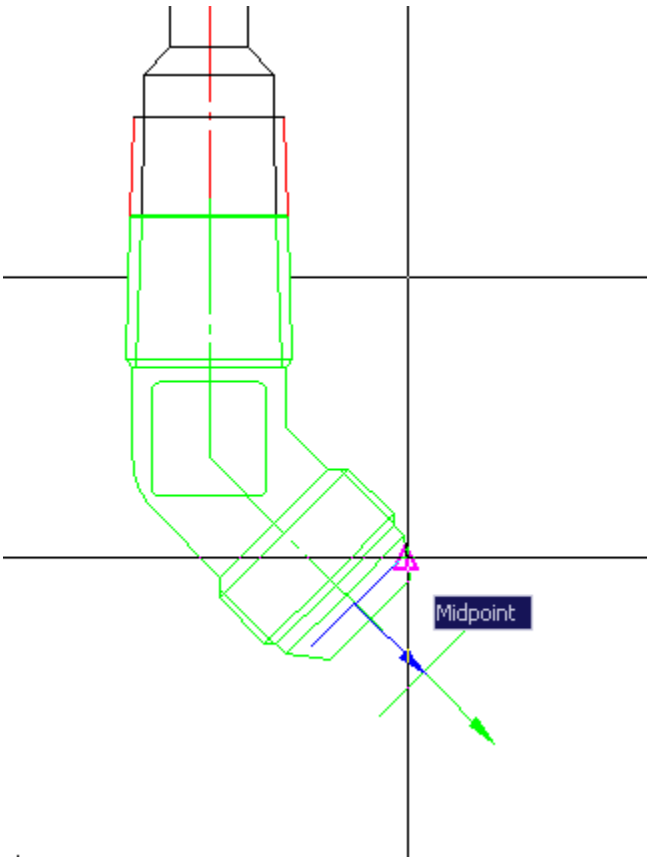
To the connection details automatically recognize your size and were object-dependent, the figure should be placed initially *element reinforcement* or *opening.*
In our example, we will adhere to the following order: Insert pipe nipple and nut.

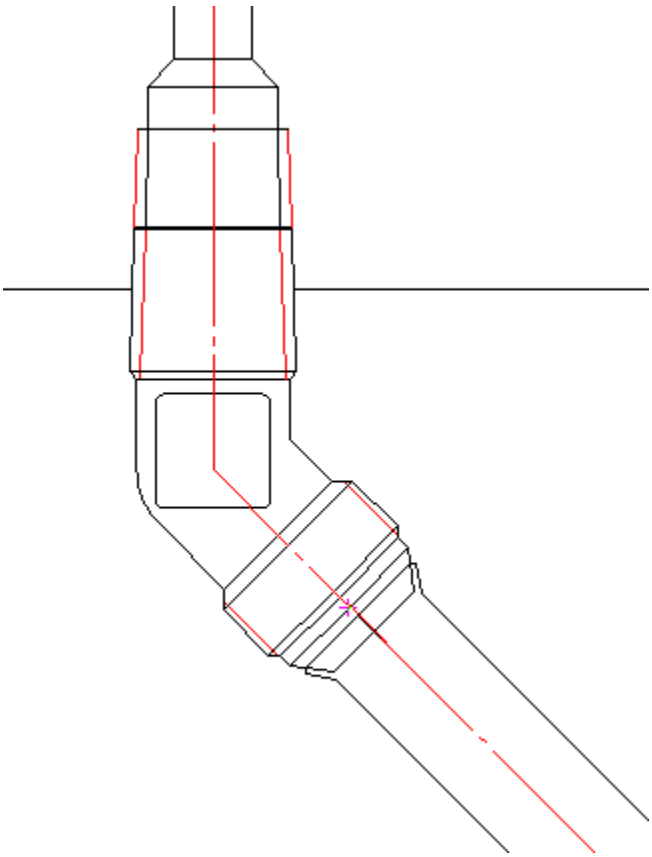


Object "pipe" can draw only pointing out any *element of reinforcement.*

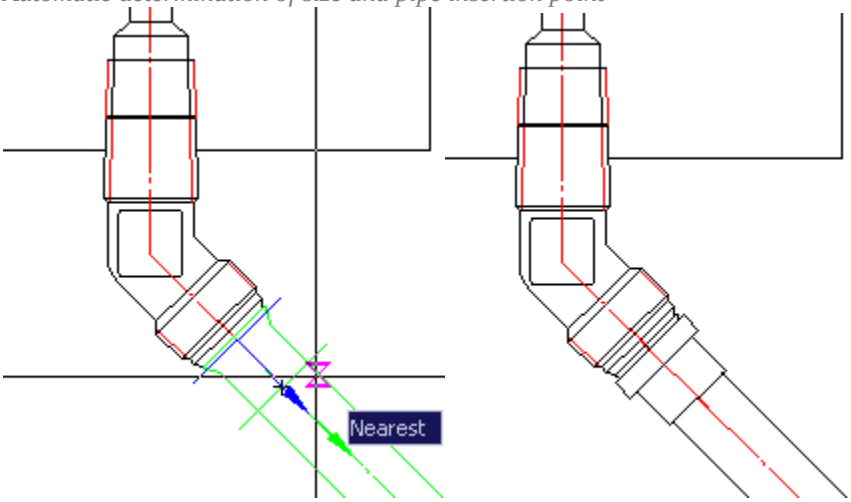


For applying object-dependent connection details on previously placed component is enough to point to the workpiece aperture cursor pointer. Base item is highlighted in color.

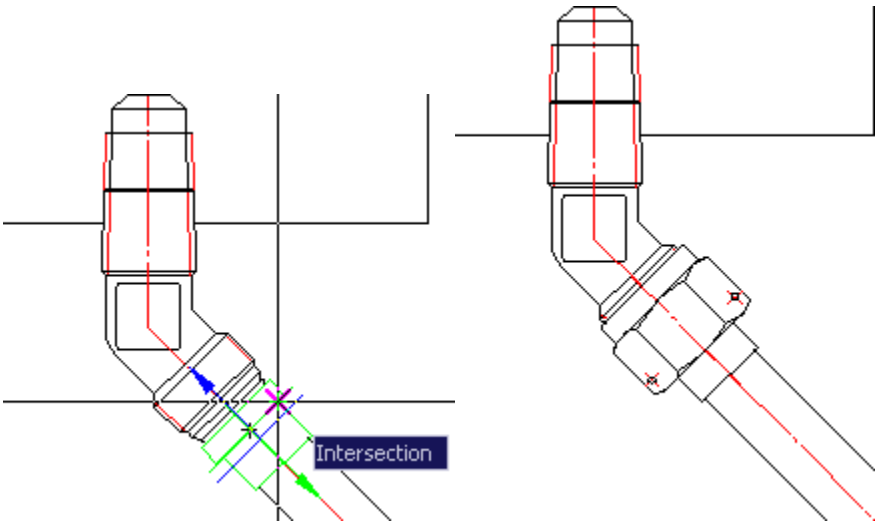

Midpoint



Automatic determination of size and pipe insertion point



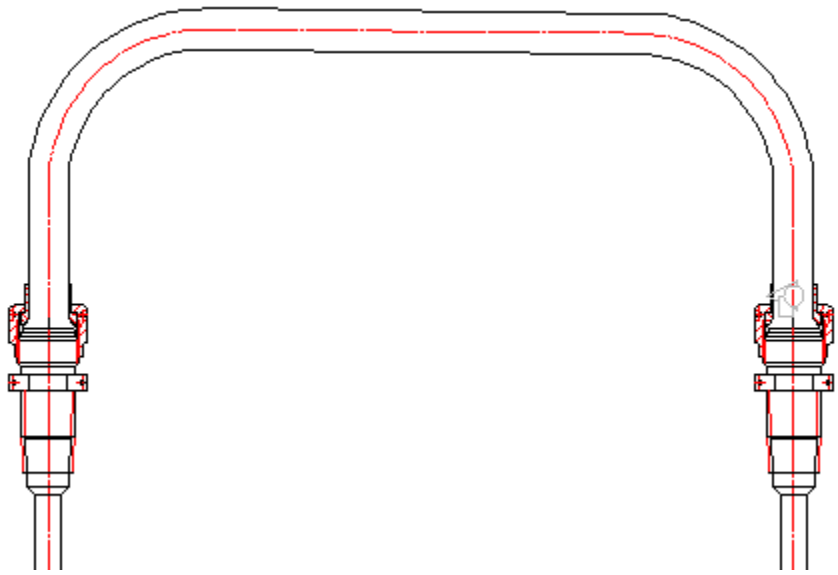
Automatic determination of size and the insertion point of the pin



Automatic determination of size and the insertion point nuts

Group inserting objects

For piping design, you can use a tool group insert objects.



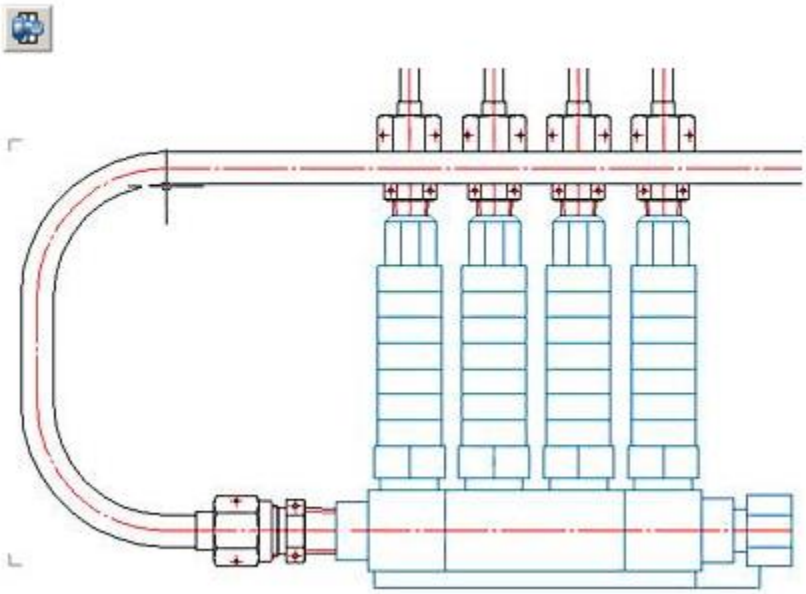
*Dialog box **Insert group** and organize the group result.*

To use her group must first create it. To do this, use the following steps:

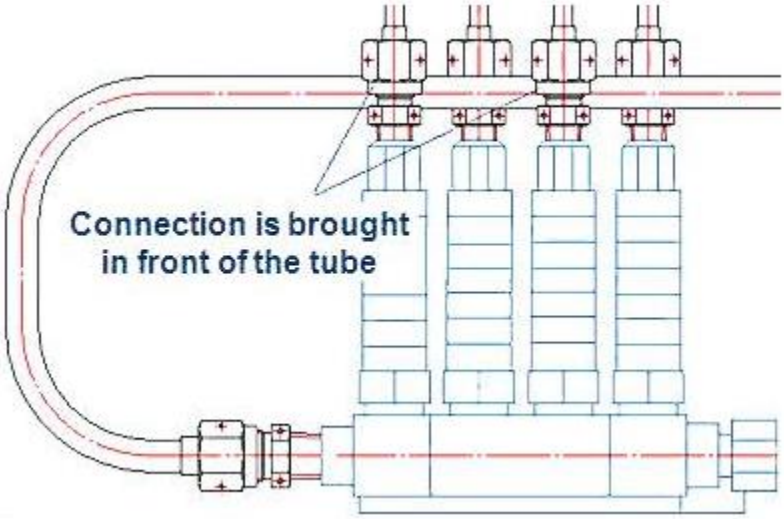
- Added to the drawing details of the object manager
- Through dialogue Dependency Management, or automatically when you insert constrain standard parts are dependencies between components.
- Group is created through dialogue to create groups.

The group must be incomplete, ie does not include fittings or pipe.
Instead, when you insert the missing fittings group will be requested item (items) in the drawing.

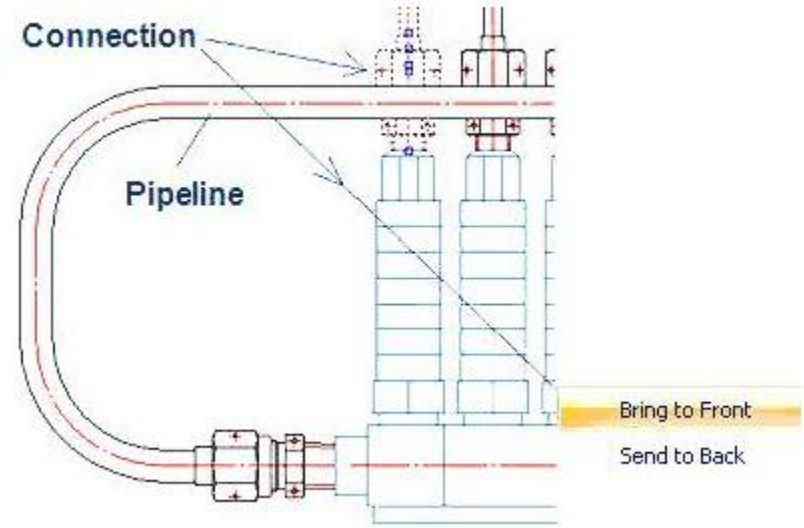
Set the method of overlapping pipelines



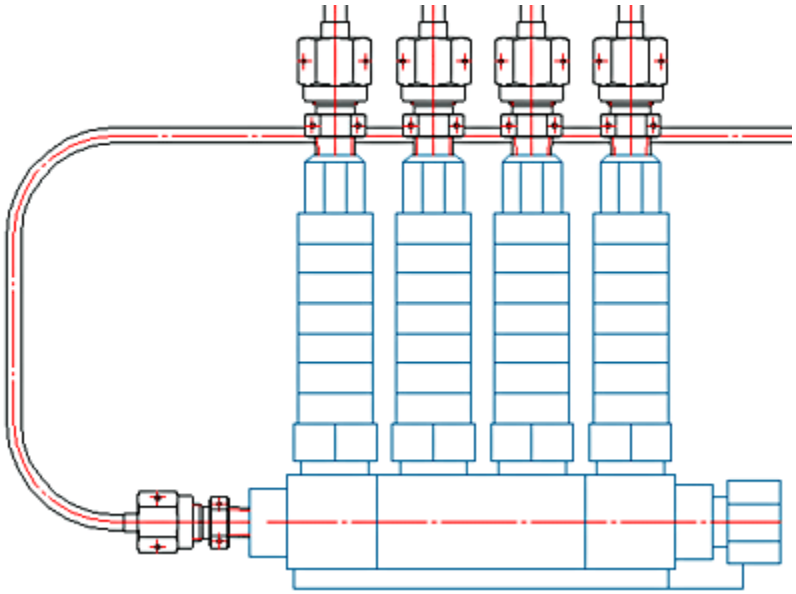
We indicate the horizontal pipe as an object to which the selected throughs must be higher.



After clicking on the icon Overlapping objects get a new version of the layout.



Distinguish compounds and by right-clicking, we show in the context menu line Move Up. Horizontal pipe is above all the elements.

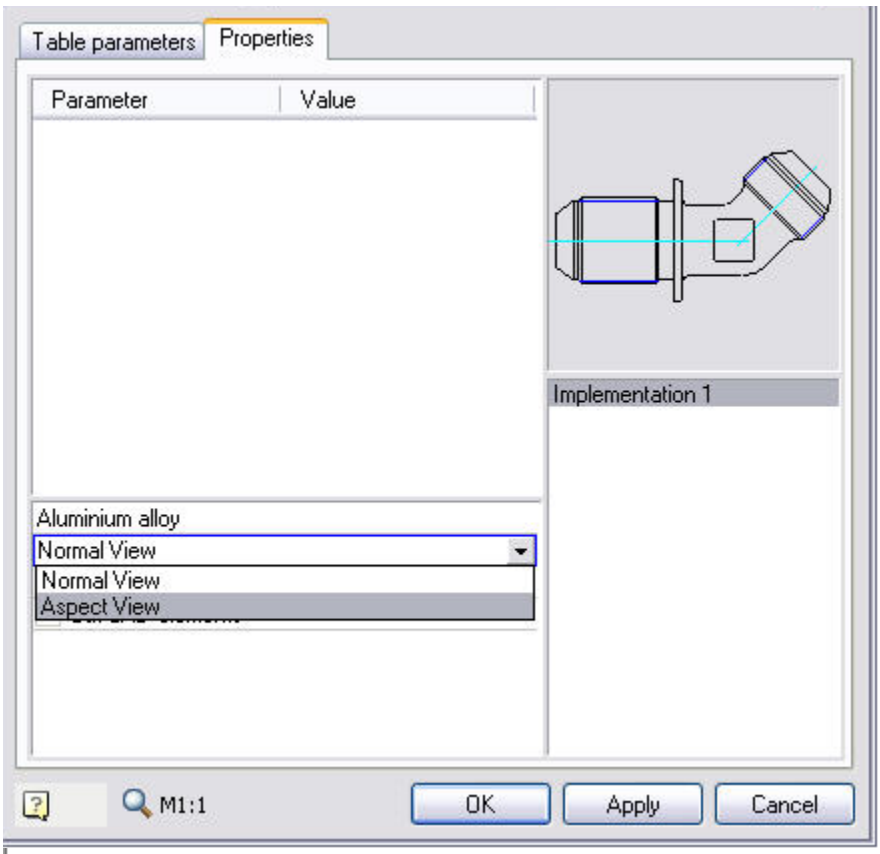


Creating a cross-section of a connection

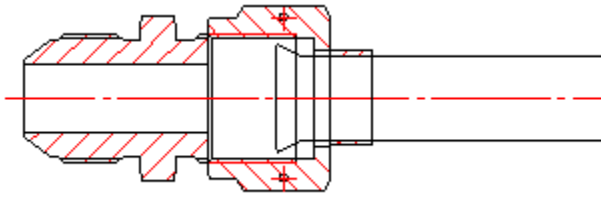
A cross-section of a fitting connection can be created at fitting insertion time, as well as by editing a previously created connection.

To change the view type of a fitting connection, each component must be edited individually.

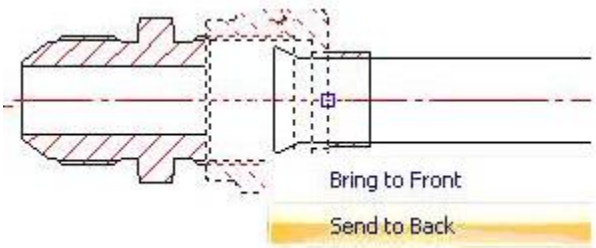
In the GOST number-titled dialog, switch to the *Properties* tab and activate the *Aspect View* option.



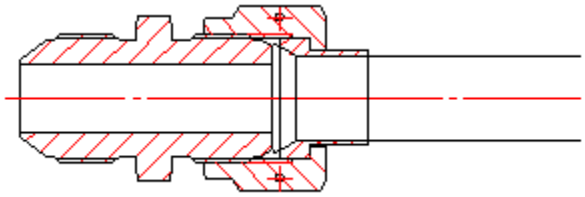
Once the representation type has been changed for all the connection components, their overlapping method should also be adjusted. To achieve this, pick a component, call up the context menu and place it above or under another component.



Select the nut and invoke the content menu , choose send to back , Then choose the object behind which the nut should be placed.

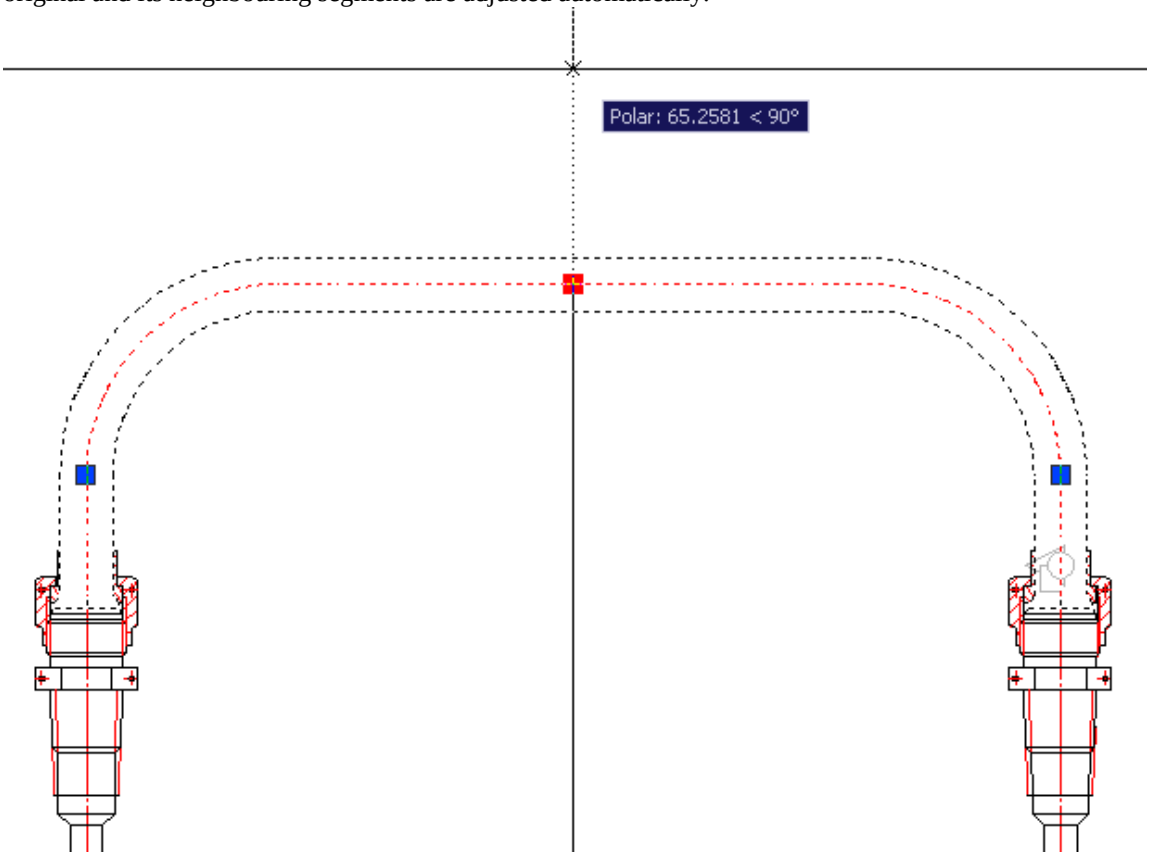


Giving the command overlapping objects, we corrected geometry.



Editing pipeline trajectory by Grip Editing

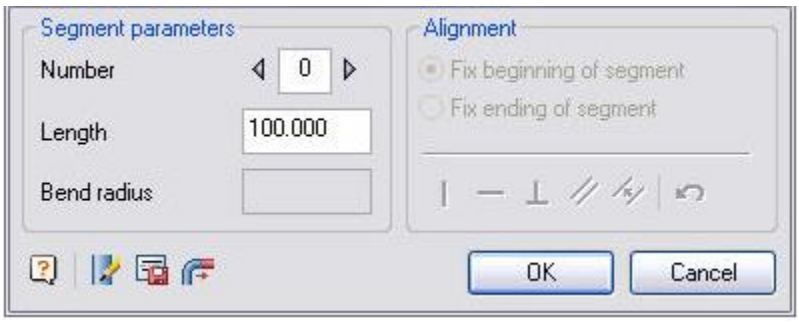
Select a pipeline and use the grips to move the nodes.If a midpoint node is edited , the relocated tube is parallel to its original and its neighbouring segments are adjusted automatically.

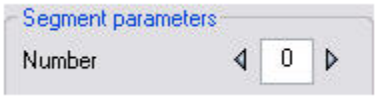


Nodes located on the endpoints of linear segments may only be relocate along the axis of the current segment.

Editing pipeline dialog

Double-clicking on a tube to open the edit pipe dialog:





To edit a trajectory using the step by step connector , first pick a tube segment. The segment will show in different color.
For linear section given length parameter is set.

For arc sections,the bent radious parameter is set.

The following sections of pipelines alignment options:

 Vertical alignment.

Horizontal alignment.

Object perpendicular alignment.

Object parallel alignment.

Object parallel alignment by distance.

Undo last alignment

In addition, the following editing options:

Base parameters change.

Editing properties of pipe

Editing tube properties , start the base parameters change command to set the material and geometry parameters of the tube in the dialog.
On the Properties You can change the display pipeline (filling, cut).

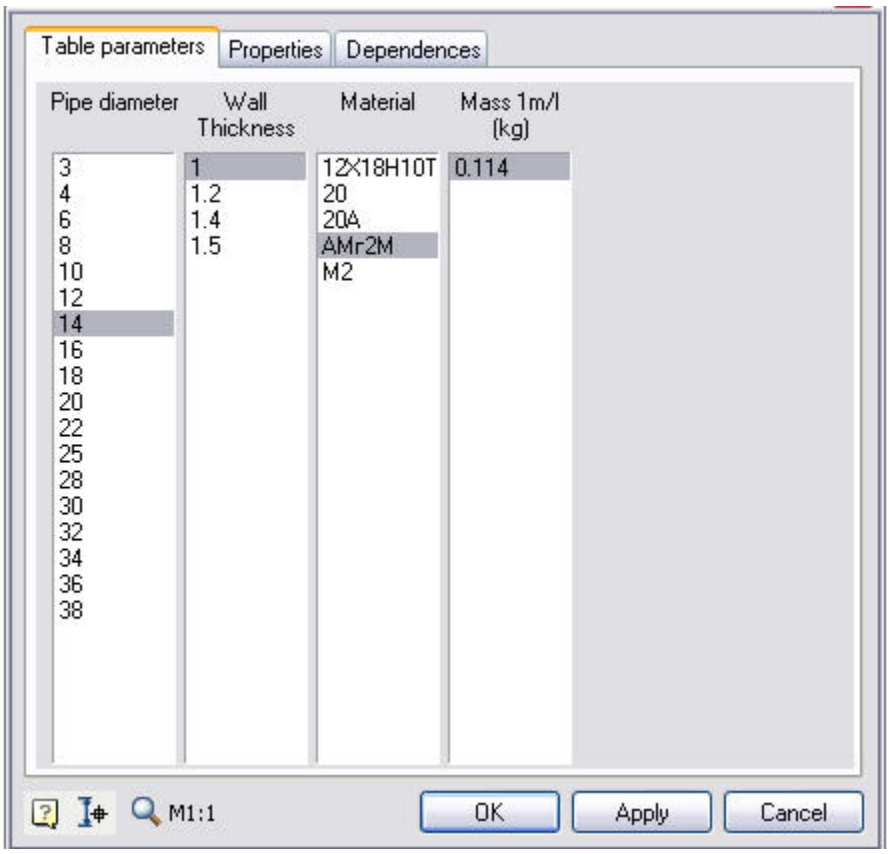
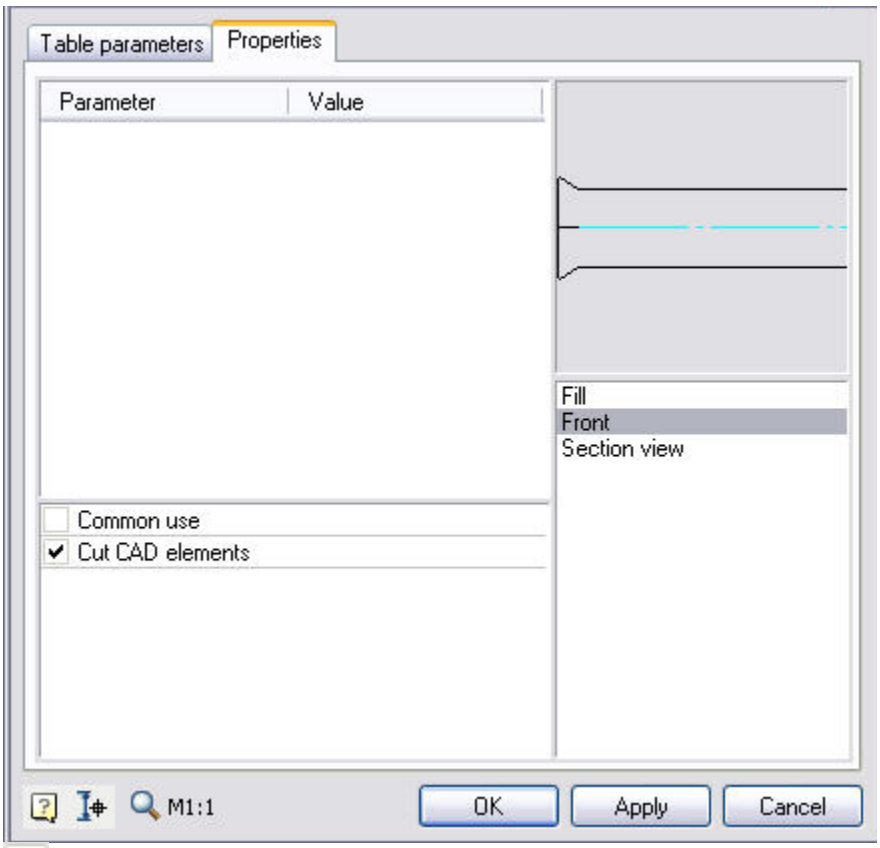


Table parameters
Properties
Dependences
Pipe diameter
Wall Thickness
Material
Mass 1m/l (kg)
3
4
6
8
10
12
14
16
18
20
22
25
28
30
32
34
36
38
1
1.2
1.4
1.5
12X18H10T
20
20A
AMr2M
M2
0.114
M1:1
OK
Apply
Cancel



Saving the pipe to an external file
After selecting this command, a dialog box *title block*. When information is entered in the title block, it is proposed to specify a filename and location.

Extend pipe
This function connects the pipe to a fitting or to a free end of another pipe.

Fitting incut for connections on external cone

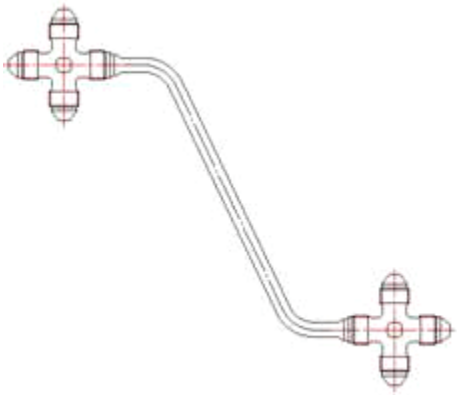
Main menu: ***Mechanical - Design - Insert fittings in pipes***.

Ribbon: ***Mechanical - Design - Insert fittings in pipes***.

Toolbar: ***Insert fittings in pipes (*** "Design"***)***.

Command line: ***MCINSHNODE***.

Fundamental rules

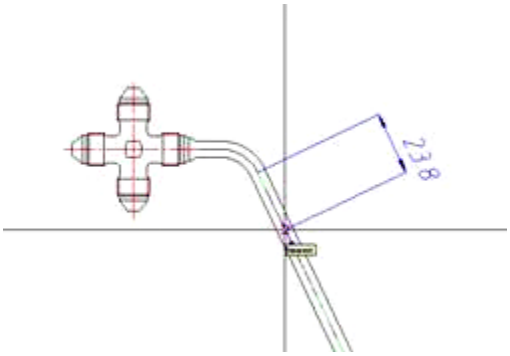


To edit the pipe connections and insert incuts, use the *Fitting incut* command. The pipe is automatically partitioned into two sections.

At the generation of the two new pipes, it will ask you to specify ports for connection of the pipeline ends. The trajectory of the pipes will be changed depending on the chosen ports.

You can separate the tube only into linear segments if the resulting segments will satisfy the condition of the minimum length of the linear segment.

The sequence of commands



1. Start the command and select the axis line of the linear segment of the pipe for insertion of the node. Position of an element is set dynamically.
2. After setting of the insertion point of the new element, the *Detail Selection* dialog will appear. Select the detail for insertion (two-way units, elbows, tees, crosses).
3. The detail will be positioned automatically at the set point. Further, it will offer to specify ports for the pipeline's connection to the inserted fitting element.
4. Select the port to place an insertion point , and then select the port for the start of the second pipe , the result will be reconstructed connection.

Import PCB format IDF

Main menu: ***Mechanical - Design -*** ***Import *.idf (*.brd) files***.

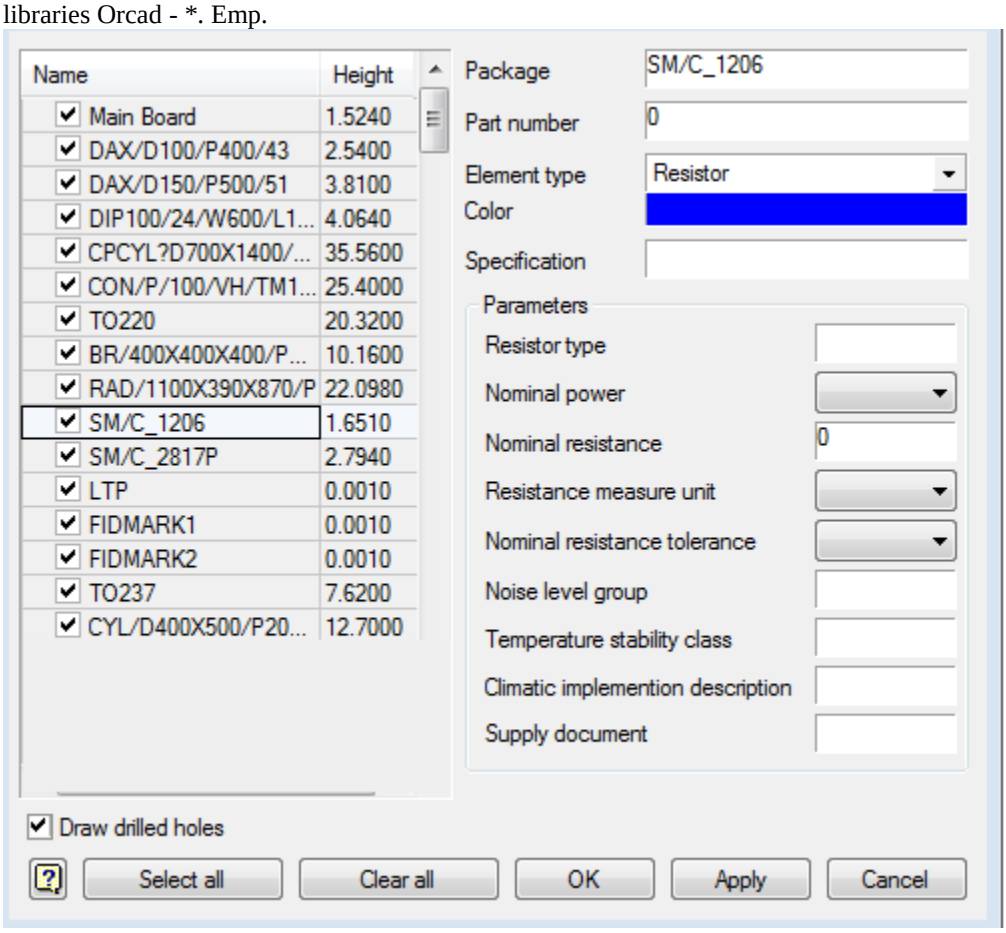
Ribbon: ***Mechanical - Design -*** ***Import *.idf (*.brd) files***.

Toolbar: ***Import *.idf (*.brd) files (*** "Design"***)***.

Command line: ***MCIDFCONVERT***.

After calling the command, the system asks for a file with the extension *.Emn (export file format from the IDF Orcad) or *.Brd (export file format from the IDF PCAD).

Caution! To import of PCB with components also required a library of components that must be maintained with the same name and in the same directory as the file of the PCB. PCAD library file has the extension *. Lib, file libraries Orcad - *. Emp.

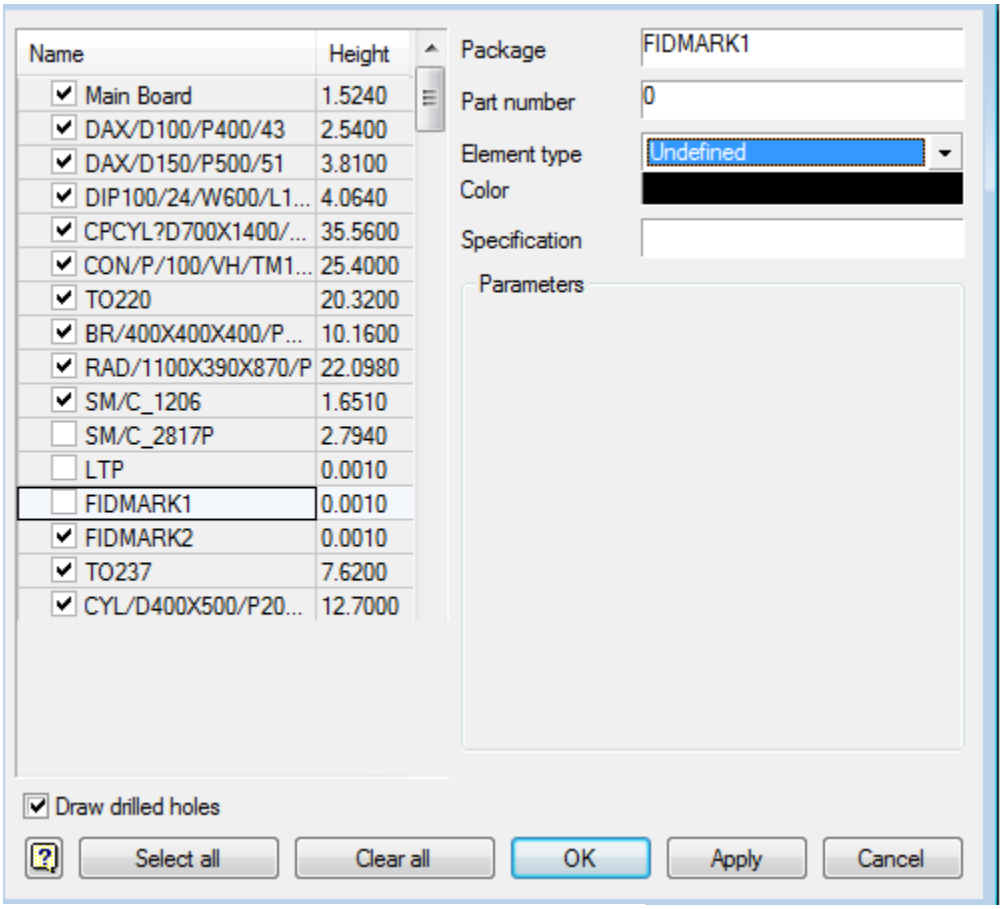


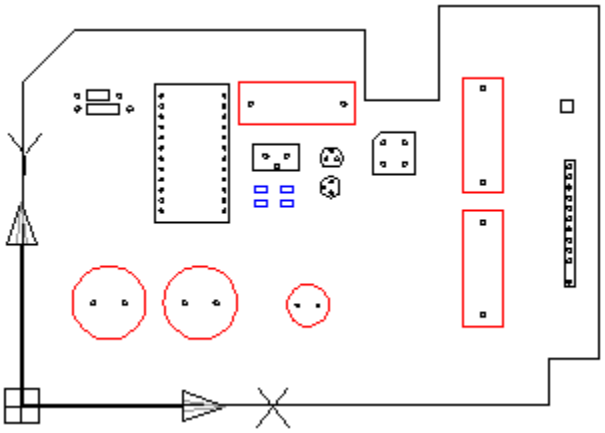
The system reads the information from the file. It is further assumed that the user can check and correct it with a dialog box IDF.

If by reference numerals may determine the type of element, the nanoCAD Mechanica 20.1 automatically assigns the appropriate circuit parameters and fill color (can be changed in settings).

Thickness (height) of the element is taken from the import file, and if it is not defined in it, the item is automatically assigned to one micron in height.

The user has the possibility to limit the detail card by disabling generation of the individual (or all) of the elements and hiding holes in the board.





After clicking the Apply or OK button - automatically generated drawing board. It is placed at the origin, and can thereafter be moved by the handles.
The Apply button allows you to continue editing the appearance and composition of the board with a graphical preview of the drawing.
PCB editing by double-clicking on it and in the same dialog box, change its structure or appearance.

Engineering analysis

Calculation wizard

Main menu: ***Mechanical - Design - Calculations -*** ***Calculations wizard***.

Ribbon: ***Mechanical - Design -*** ***Calculations wizard***.

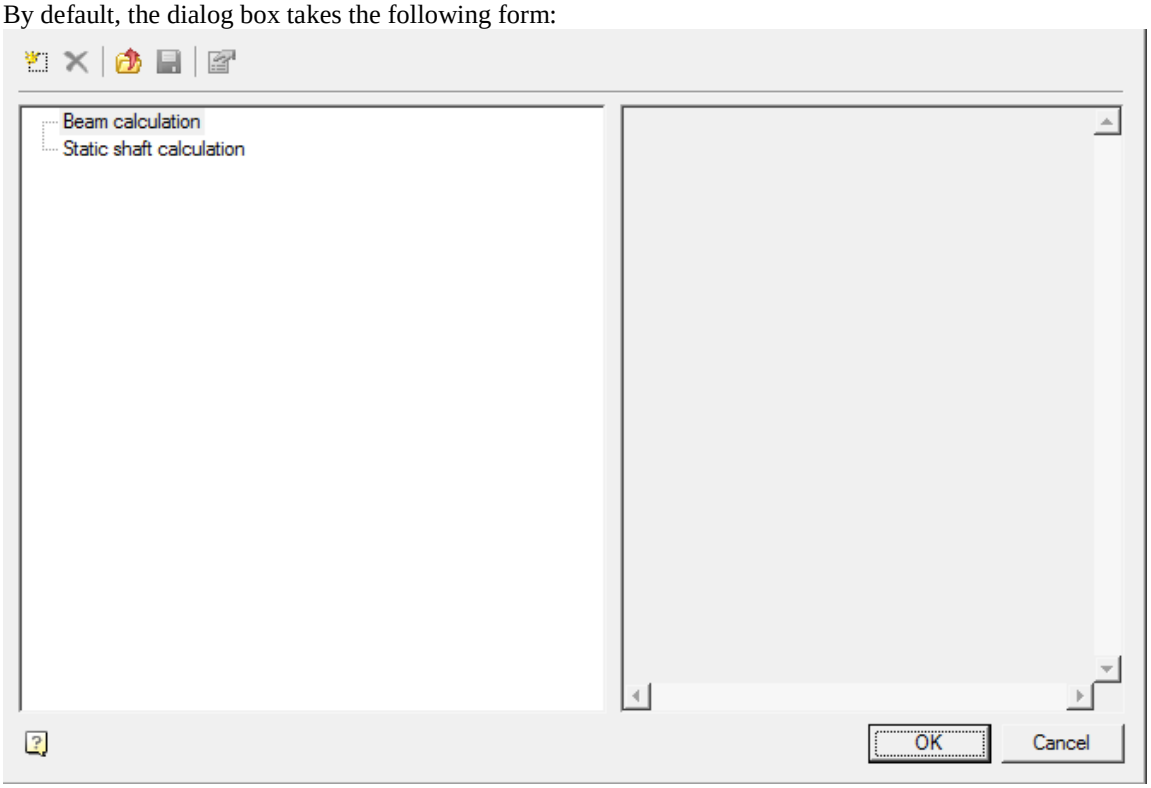
Toolbar: ***Calculations wizard (*** "Design"***)***.

Command line: ***MCCALC***.

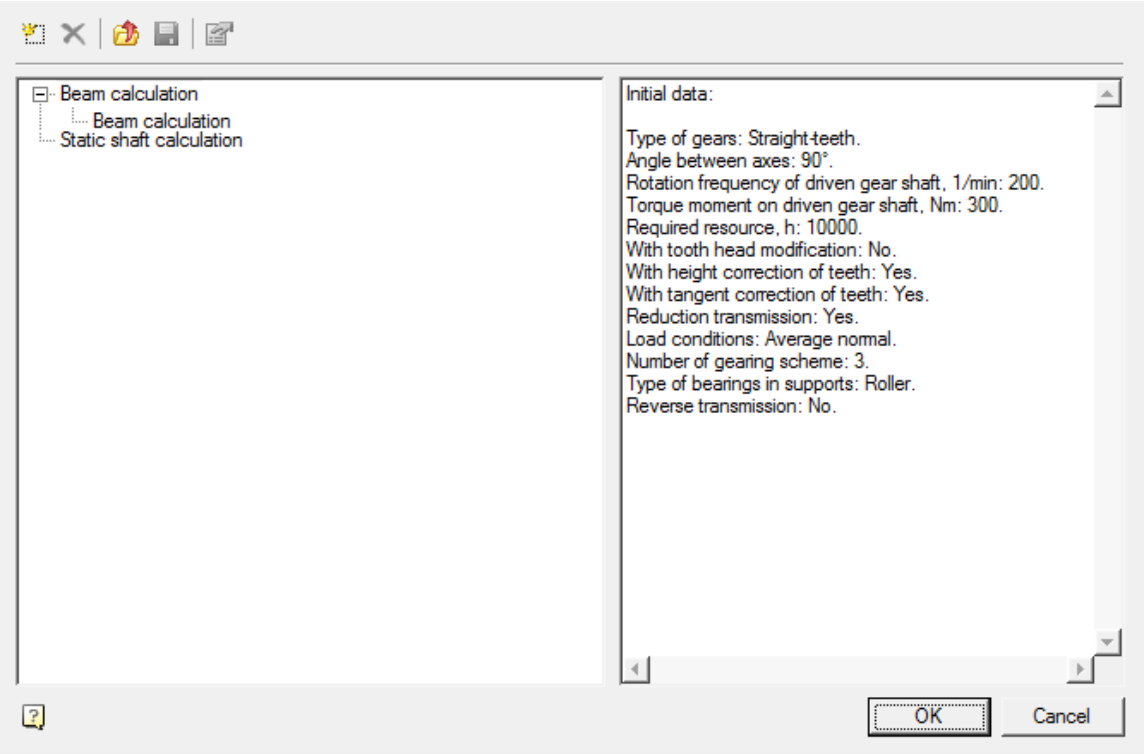
Fundamental rules

1. Calculations wizard allows you to edit all the calculations are available in the file - regardless of the graphics platform. To create a new calculation required, specifying the desired type, click *"New calculation"*.
2. The calculation results can be displayed directly in the calculation manager.
3. The type and number of parameters displayed in the dialog can be edited by the user at his discretion. To do this, double-click in the right part of the dialog box cause the viewer to edit.

By default, the dialog box takes the following form:



When installing the mouse cursor on the name of the calculation are available additional options.



Toolbar

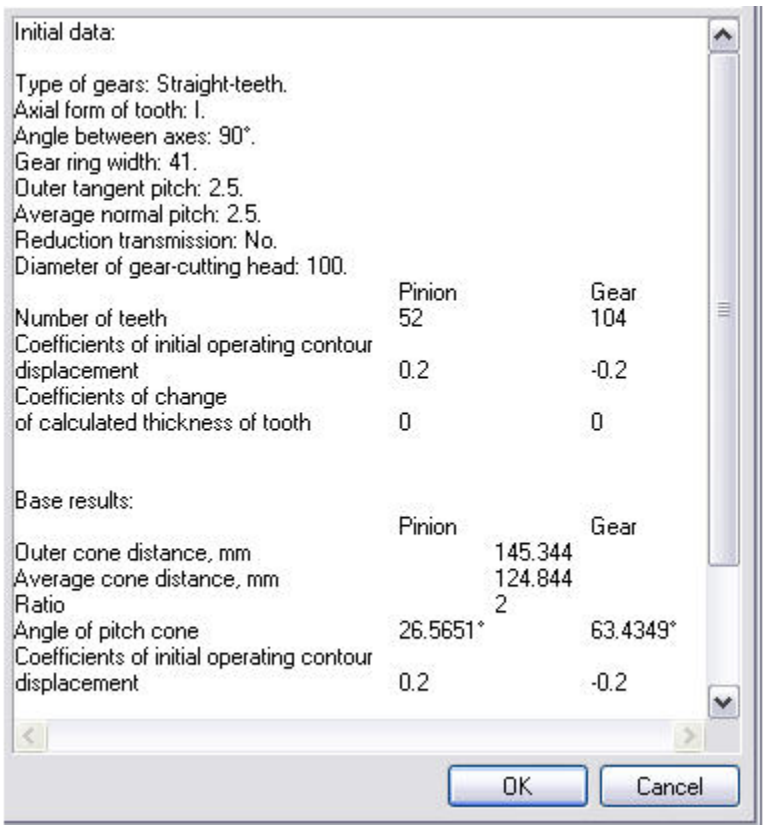
Create - Activates the calculation type depending on the selected window.

Delete - Removes the created calculation

Open - Opens previously saved on the disk calculation

Save - Saves the selected calculation on disk in a file * .mcc

Edit - This command opens the window for editing the calculation results



Upon activation, the report window, the user can enter their own value and remove unnecessary.
Default report templates for calculating gearing placed in a folder
C:\Program Files\Nanosoft\nanoCAD Mechanica 20.1\Reports Templates

Calculation of gears

The calculation is made on the basis of GOST 21354-87 (hereinafter - the Standard) "Cylindrical gears involute external engagement for strength calculation.". The standard establishes the basic design according to the definition of the contact strength of the active surfaces of the teeth and tooth strength at bending metal cylindrical involute external gear transmission for general industrial applications with source circuit in accordance with GOST 13755-81, modulus m / 1 mm, working with lubrication oil at peripheral speeds v<=25 m/s.

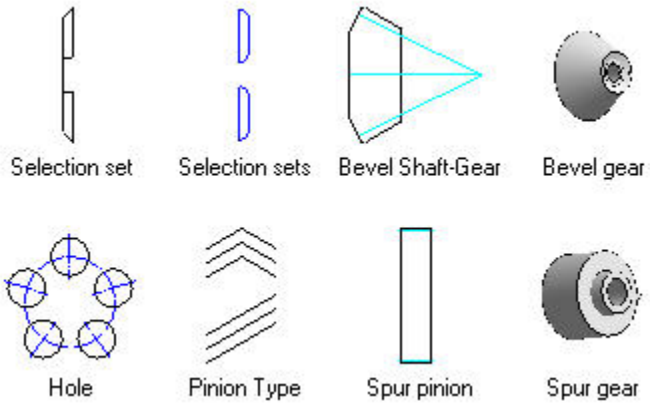
Methods for rendering the gear

Designing gear can begin in any sequence. What to do in the first place - the calculation or drawing gear - it does not matter.
The wheels that were drawn without calculation, it is always possible to adjust the results of the calculation. All geometry is adjusted automatically.
Approaches to the design of the gear in nanoCAD Mechanica 20.1:

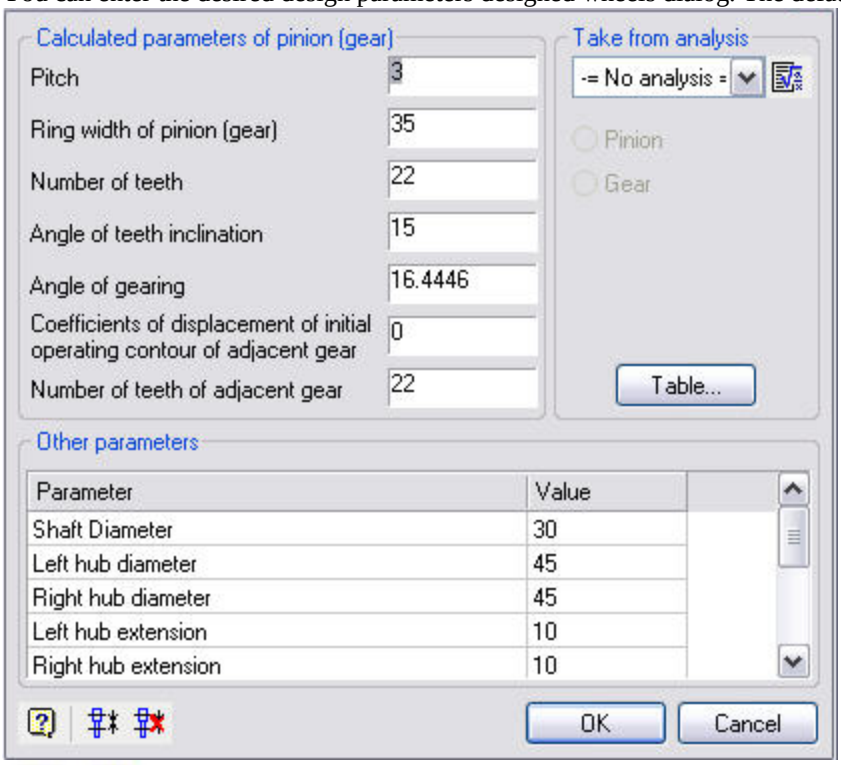
1. Draw gear without calculation by entering the wheel settings dialog.
2. To calculate the gear and the results of calculation render mesh.
3. To calculate the transmission stage and assign the result to the transfer were drawn without prior calculation.
4. Use the template file with the initial data on the calculated transmission gear stages and drawn by the circuit of the gearbox. By changing the raw data calculation, to update the geometry file template steps gears.

Drawing wheels (gears) without calculating

We choose to base parts of the gear pattern.



We define the insertion point and the direction of its rendering.
You can enter the desired design parameters designed wheels dialog. The default render wheel without calculation.



Personal application assembly dependencies on a pair of gearing on the calculated spacing.

Drawing engagement of calculations

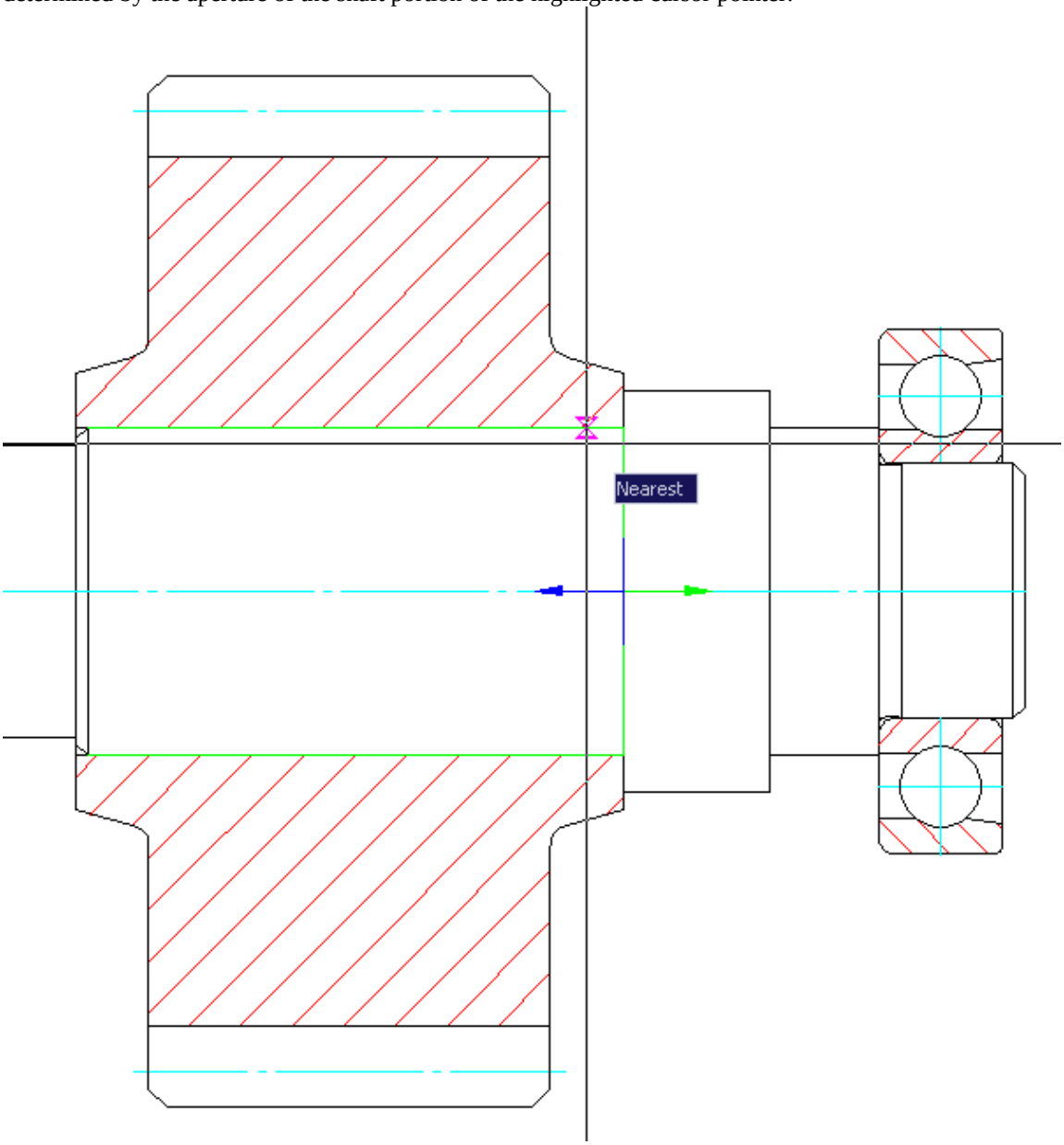
1. We choose to base parts of the gear pattern.
2. We define the insertion point and the direction of its rendering.
3. Since the calculation is done, select the desired calculation from the dropdown list.
4. Specify which element engagement render on the results of the calculation (wheel or gear) and click OK.

5. After drawing one element command remains active. To complete it, click with the ESC key.

To assemble the wheels and gears on the calculated spacing options use gearing and Remove gearing.

How to set the point of insertion of the gear?

After selecting a template nanoCAD Mechanica 20.1 prompts you to specify the insertion point and the direction of drawing of the selected type wheel.
If the drawing already has a shaft portion, which is supposed to place the wheel, the insertion point is automatically determined by the aperture of the shaft portion of the highlighted cursor pointer.



The approach to the calculation of gear

Gear is regarded as an independent object, for which the specified power and kinematic characteristics of the input or output.
Gear is an element of the designed gear. Pre-selection is carried out of the engine is determined by the overall gear ratio is carried out by a breakdown of the gear ratio gear stage determined power and kinematic characteristics of the gears of the individual stages.
In the proposed algorithm, the requirements of GOST completely soaked.
According to the calculation results can be obtained:

- geometrical characteristics of the gear train designed;
- data to monitor the accuracy of the gears;
- data to estimate the gear strength (allowable and actual voltage as in the contact and bending endurance calculation teeth, and the teeth in the calculation of the maximum load on the action);
- degree of loading gear on these kinds of stresses.

Estimator (a program for calculating the gear) allows you to perform two types of calculations:

- designing new gear for a given load, durability, desired gear ratio and other parameters;
- check the specified gear strength.

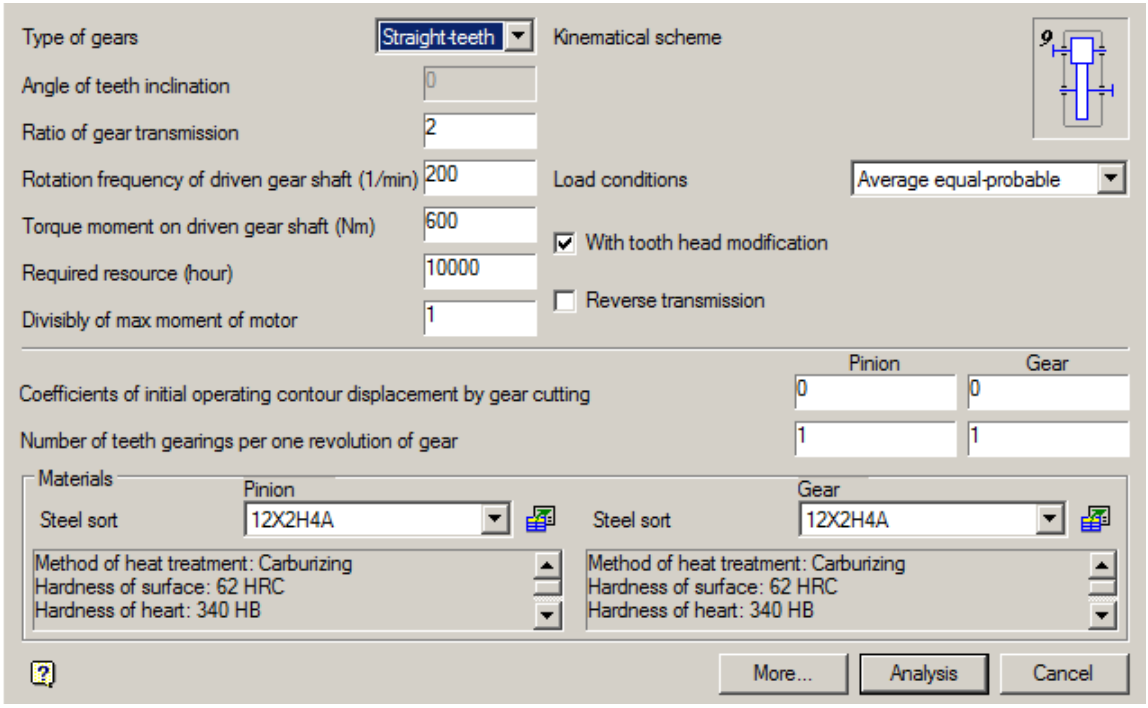
In the first case, the estimator first transfer designs (in our version - on the basis of conditions to ensure contact strength), that is, defines all of its geometric characteristics, including module and spacing, and then to check on the strength of all kinds of stress.
In the estimator of the second embodiment is not used for design, but only for testing the strength of the transmission, the geometric characteristics of which are already known or pre-defined.
For the two types of calculations in the dialog box Calculations toothed cylindrical gear are two options provided for data input:
Each type has its own form of payment report with its source data and its own characteristics of presentation of calculation results as well as its report templates.

Cylindrical Gearing Strength Analysis

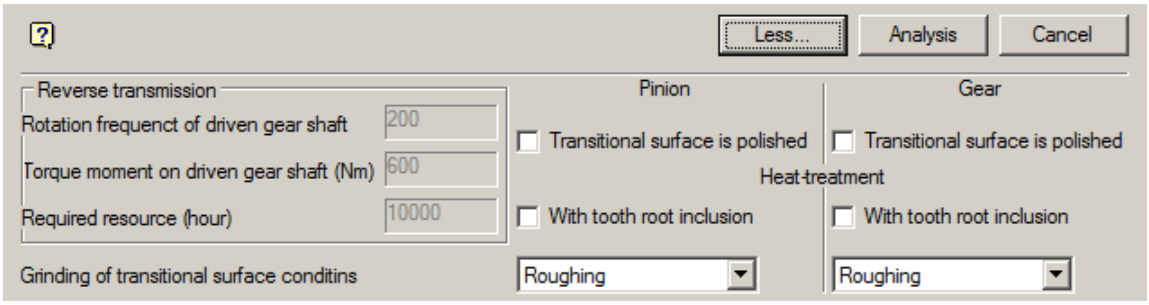
Library: Calculations - Spur gears - Cylindrical Gearing Strength Analysis
In what follows the parameters and characteristics related to the gear, given the index 1; relating to the wheel - the index 2.
The initial data are encouraged to use:

- typical transmission loading mode (GOST, features 32, p.106.) or load sequence diagram (GOST, features 14, p. 63);
- the torque on the shaft of the wheel gear - T2, N * m (if the load is variable, then how current torque accepted most of long-acting);
- Wheel speed - n2, min. - 1;
- the gear ratio gear set - u;
- the type of gears (spur, helical, herringbone);
- tooth angle - β° ;
- offset coefficients source generating circuit - x1 and x2;
- transmission type: reversible or irreversible. By reversing or non-reversing gear transfer meant a two-way or one-way load application;
- term transmission service - Lh, h;
- material and a method of chemical-thermal hardening of gear wheels and transmission calculated;
- the location of the calculated gear in the drive according to the scheme in accordance with GOST, p. 58, fig. 13a.

Enter the initial data for the design of the transmission by means of the dialog box, shown in figure:



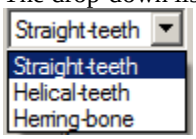
By clicking on the button "More ..." open additional field data input for the reverse gear, the method of machining the tooth flank and its heat treatment.



Initial data

Type of gears

The drop-down list of the gear types allows you to select one of three options: spur, helical, herringbone or wheel:



Angle of teeth inclination

When choosing a helical or herringbone wheels in the "angle of the teeth" should enter a value in degrees of the desired angle of inclination of the tooth line in the recommended range (typically 8-22 ° for helical and 25-40 ° for chevron wheels).

If this field is left blank, the program installs by default a certain angle, which lies within the specified limits. Most often, this angle imposed by the previous calculation. It can be changed either directly or by subsequent adjustment calculation.

Ratio of gear transmission

The field is filled in accordance with the terms of reference and the expected characteristics of the engine.

Rotation frequency of driven gear shaft (1/min)

The field is filled in accordance with the terms of reference and the expected characteristics of the engine.

Torque moment of driven gear shaft (Nm)

The field is filled in accordance with the terms of reference and the expected characteristics of the engine.

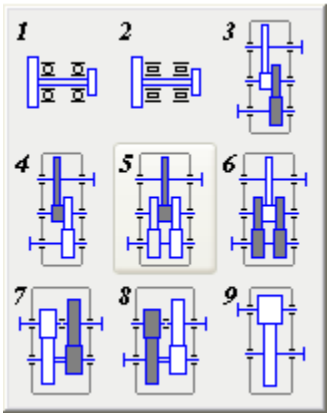
Required resource (hour)

The field is filled in accordance with the terms of reference and the expected characteristics of the engine.

Divisibly of max moment of motor

The field is filled in accordance with the terms of reference and the expected characteristics of the engine.

Kinematical scheme



Click on the field with the image of the kinematic scheme reveals nine schemes of gear reducers, from which you can choose the layout of the gear pair corresponding to the estimated case. Estimate the case highlighted in white.

Load conditions

Load conditions is selected from the dropdown list.

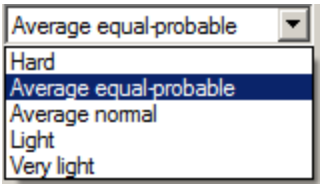


Table of values of equivalence ratios for each loading mode:

Load conditions	μ_h	μ_F	
		Normalization, improvement q=6	Hardening, cementation q=9
0 - Simple	1.0	1.0	1.0
1 - Hard	0.5	0.3	0.2
2 - Average equal-probable	0.25	0.14	0.1
3 - Average normal	0.18	0.06	0.04
4 - Light	0.125	0.038	0.016
5 - Very light	0.063	0.013	0.004

Check "With tooth head modification"

Check "Reverce transmission"

It makes active group of source data "" Reverse transmission ".

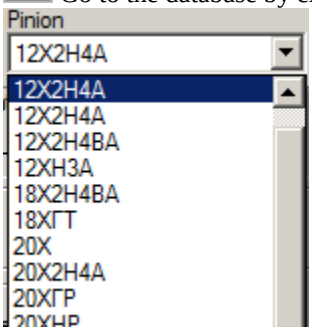
Koefficients of initial operating contour displacement by gear cutting

Number of teeth gearings per one revolution of gear

Group "Materials"

Materials of gears and their characteristics are selected from a drop down list or from the database.

Go to the database by clicking on the appropriate button to the right of the selection grade of steel.



Type of material selection group gears and wheels:

For convenience, you can select to see comments to the selected material.

Group "Reverce transmission"

The group contains the following fields:

- Rotation frequenct of driven gear shaft

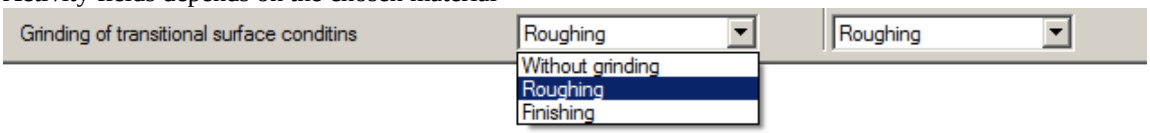
- Torque moment of driven gear shaft (Nm)
- Required resource (hour)

Check "Transitional surface is polished"

Check "With tooth root inclusion"

Grinding of transitional surface conditins

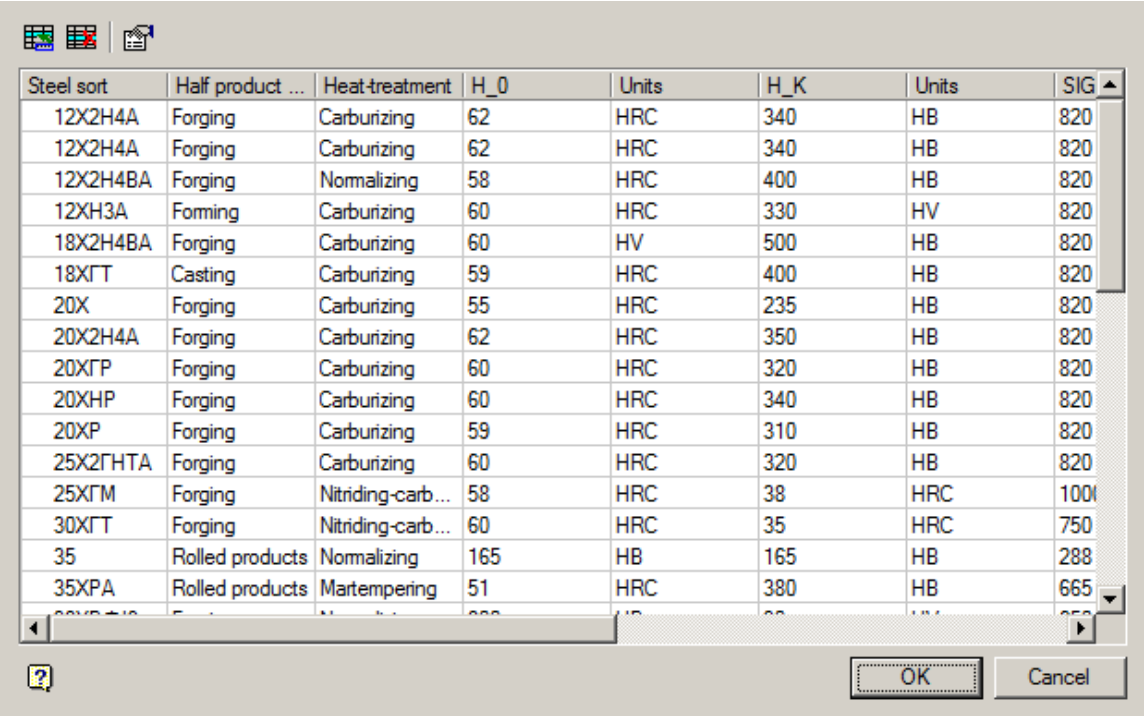
Activity fields depends on the chosen material



If the transition is made grinding surface of the teeth, the required grinding mode can be selected from the corresponding drop-down list.

Editing Material Database

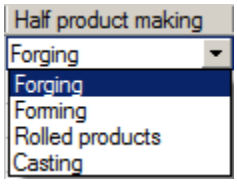
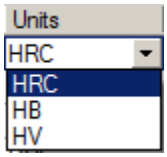
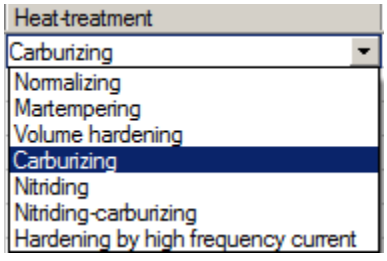
The user can edit any of the selected characteristics of the material, but it should be borne in mind that some of them change must be accompanied by a change in the other. In particular, the limit of endurance in bending teeth, corresponding to the basic number of stress cycles $\sigma^0_{Flim\,b}$ for normalized, improved nitrided and wheels, as well as the base value limit stress teeth flexural maximum load σ^0_{FSt} for normalized and improved gear depends on the hardness of the teeth.
Therefore, changing the hardness of the wheel teeth with a chemical-thermal treatment, need to change their performance and durability. If you are not sure of the correctness of the total introduced to them changes, to edit the properties of the material is better to use filling material properties dialog box (see. below).



Steel sort	Half product ...	Heat-treatment	H_0	Units	H_K	Units	SIG
12X2H4A	Forging	Carburizing	62	HRC	340	HB	820
12X2H4A	Forging	Carburizing	62	HRC	340	HB	820
12X2H4BA	Forging	Normalizing	58	HRC	400	HB	820
12XH3A	Forming	Carburizing	60	HRC	330	HV	820
18X2H4BA	Forging	Carburizing	60	HV	500	HB	820
18XГT	Casting	Carburizing	59	HRC	400	HB	820
20X	Forging	Carburizing	55	HRC	235	HB	820
20X2H4A	Forging	Carburizing	62	HRC	350	HB	820
20XГP	Forging	Carburizing	60	HRC	320	HB	820
20XHP	Forging	Carburizing	60	HRC	340	HB	820
20XP	Forging	Carburizing	59	HRC	310	HB	820
25X2ГHTA	Forging	Carburizing	60	HRC	320	HB	820
25XГM	Forging	Nitriding-carb...	58	HRC	38	HRC	100
30XГT	Forging	Nitriding-carb...	60	HRC	35	HRC	750
35	Rolled products	Normalizing	165	HB	165	HB	288
35XPA	Rolled products	Martempering	51	HRC	380	HB	665

OK Cancel

The options available in the database, with the following designations are:

	Half product making
	Units of measurement of hardness.
	The choice of heat treatment of the material.
SIGMA_Flimb	The limit of endurance in bending teeth, corresponding to the basic number of stress cycles MPa.
Y_g	Factor which takes into account the impact of the transition surface grinding teeth (GOST, table. 13, p. 10.4).
Y_d	Factor which takes into account the effect of strain hardening or

	electrochemical treatment of the transitional surface of the tooth (GOST, table. 13, p. 10.5).
S_F	Calculated safety factor based on the bending resistance (GOST, table. 13, p. 11).
SIGMA_Fst	The safety factor in the calculation of the teeth on the maximum flexural load (GOST, table. 18, n. 2).
SIGMA_T	Yield gear material MPa.
H_O	Surface hardness of the teeth.
H_K	The hardness of the teeth of the core.

Updating and changing the base made by the three buttons located in the upper left Materials:

New material

Delete material

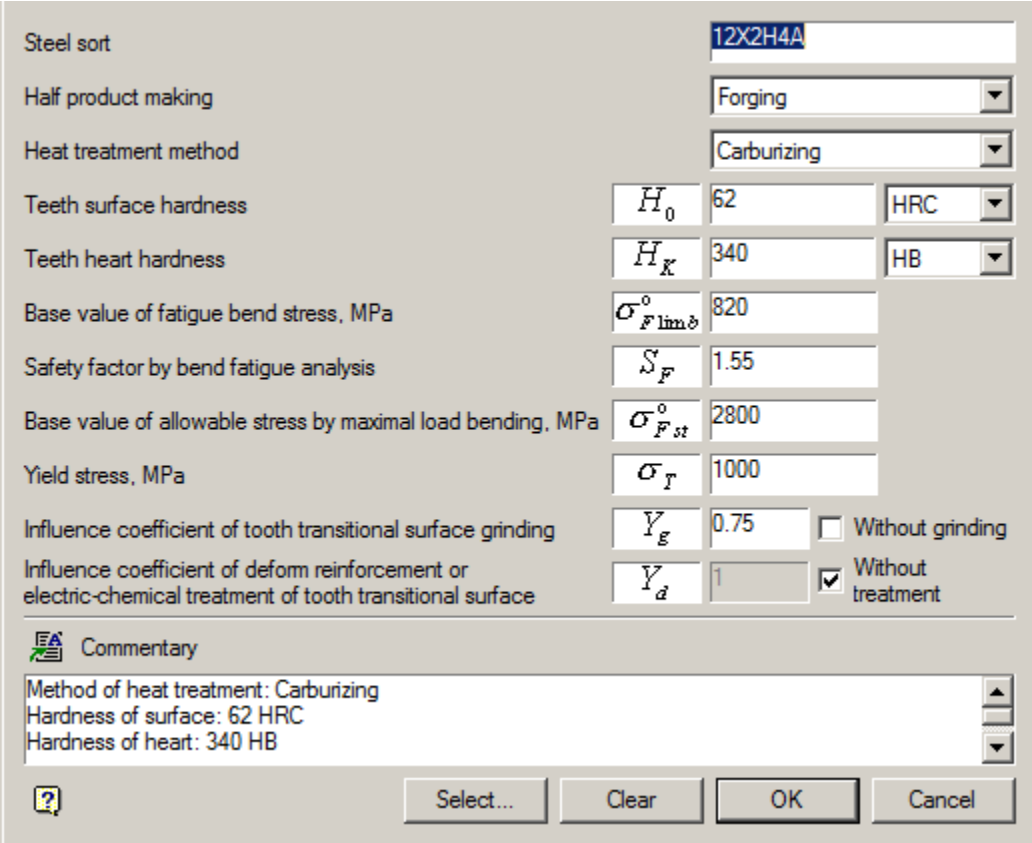
Edit

For example, click on the New button opens a dialog box material properties of the material. For the correct insertion of a new material in the base must be filled in all the fields of the window.

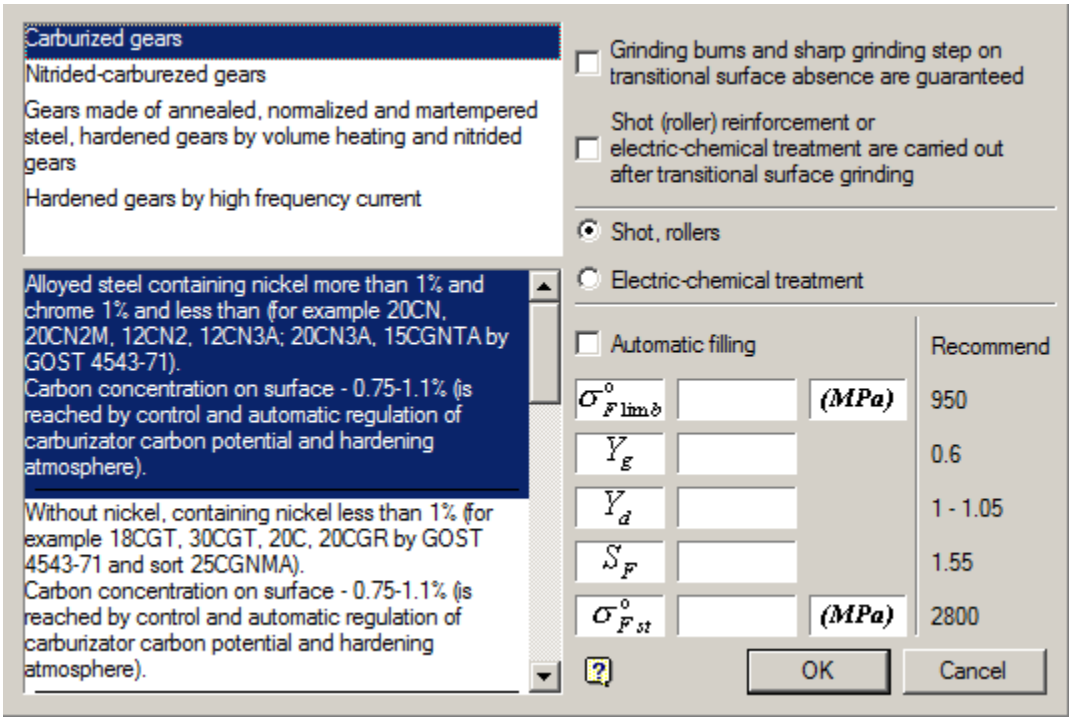
Entering a new material

In the field of material input parameters are copied to the properties of the previously selected metal. Thus, the user is given the opportunity to create its database in relation to specific production and technological capabilities of the enterprise.

All fields of the material properties of a window can be filled by the user immediately. However, given that in accordance with GOST limits contact endurance and stamina for bending at various thermochemical treatment of steels defined by different dependencies, at this stage it is advisable to fill the database go to the next dialog box, activating the button Select ...



The corresponding filling material properties dialog box (the button Select ...) allows you to select the characteristics of aggregated groups of materials and thermochemical treatment (upper left field of the window).

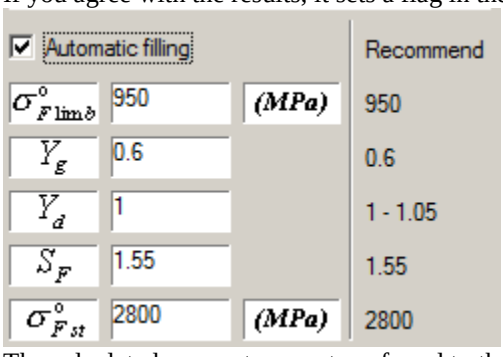


These can then be clarified in relation to specific groups of steels (bottom left margin of the window) and see the calculated values of the respective limits of endurance (Recommended field).
We have already said that strength characteristics of the gears of the normalized and improved, as well as nitrided steels depend on the hardness of the teeth. Therefore, in determining the strength characteristics of gears exposed to these types of chemical and thermal hardening, in advance, before calling the dialog box Filling material properties in the Properties dialog box material required hardness of the teeth must be installed.

In this case, the parameters $\sigma^0_{F\lim b}$ and σ^0_{FSt} in the "Filling the properties of the material" will match hardness adopted.

Autofill material properties

If you agree with the results, it sets a flag in the autocomplete field.



The calculated parameters are transferred to the corresponding open field characteristics of the particular material. These fields can be filled manually, if desired.

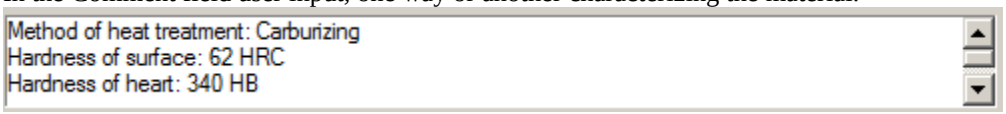
Removing material from the base

Selecting material removal cancel command does not work. To return to the base material should be re-entered.

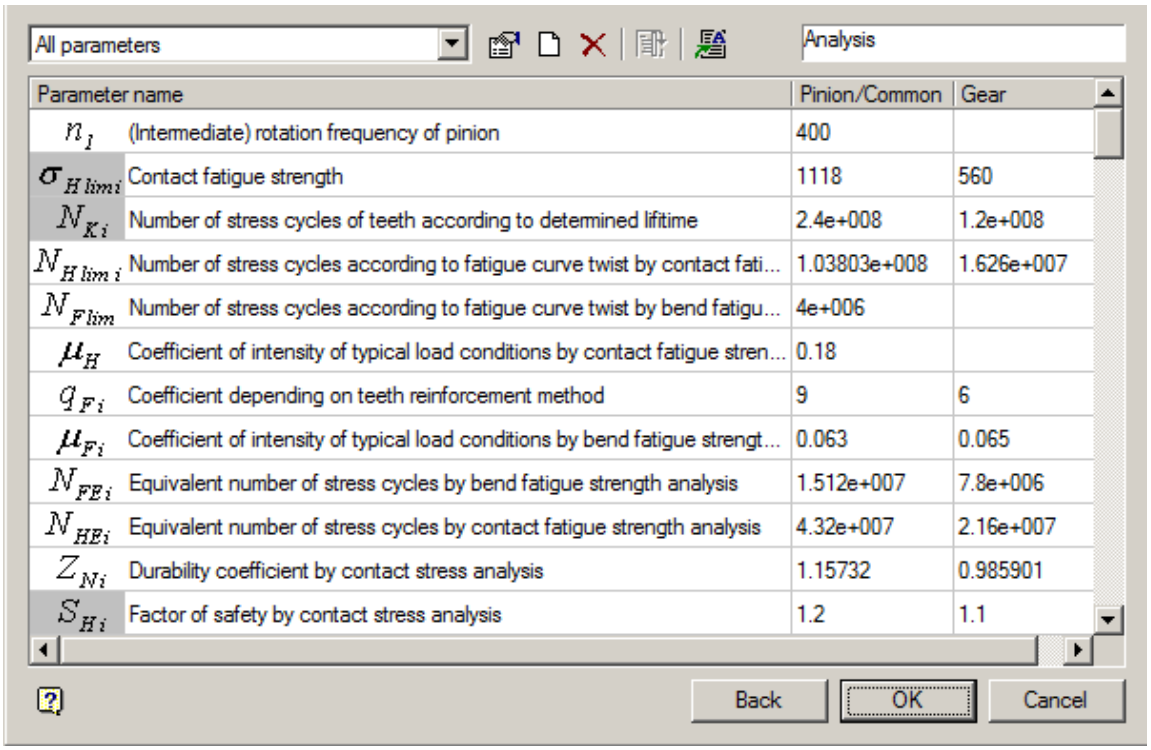
Edit the properties of the material

When you select the Edit window Materials Database of materials displayed earlier submitted Material Properties dialog box.

In the Comment field user input, one way or another characterizing the material.



The calculation results



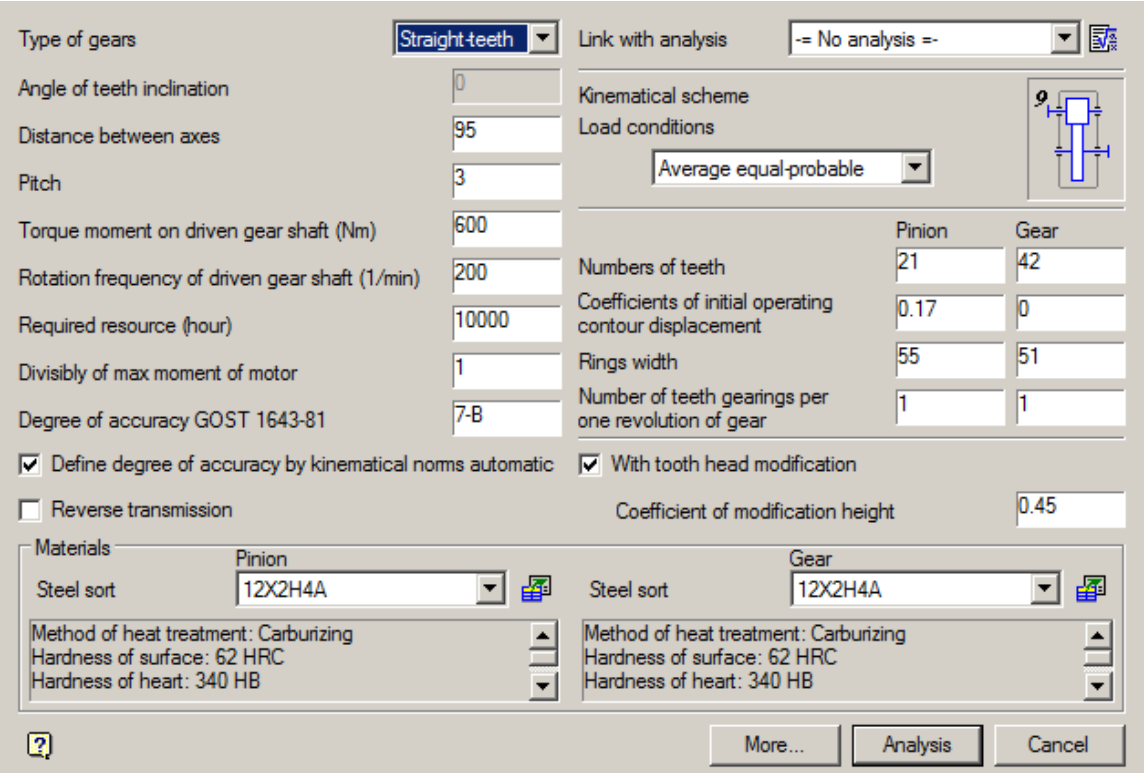
This dialog shows all the parameters of how the raw data and calculate results. By clicking the Back button, you can return to the dialog box to make changes to the initial calculation data.

Strength calculation of spur gearing

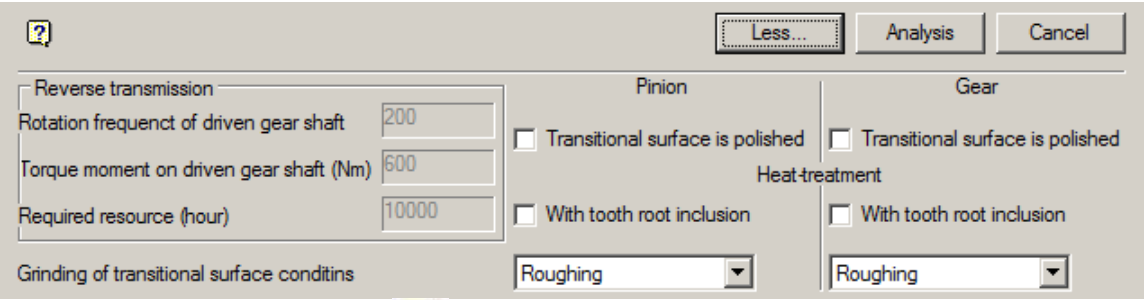
Library: Calculations - Spur gears- Strength calculation of spur gearing

Operation of the second embodiment does not use calculators for design, but only for testing the strength of the transmission to the known or predetermined geometrical parameters.

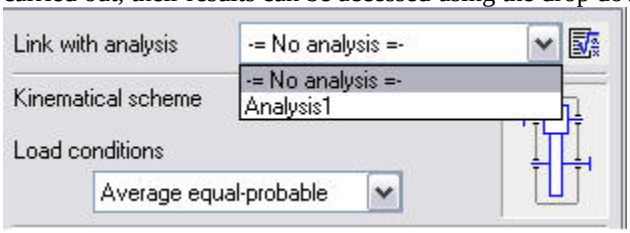
Checking calculations has its own form of the report and data input. In all other respects similar to the method described above.



By clicking on the button "More ..." open additional field data input for the reverse gear, the method of machining the tooth flank and its heat treatment.

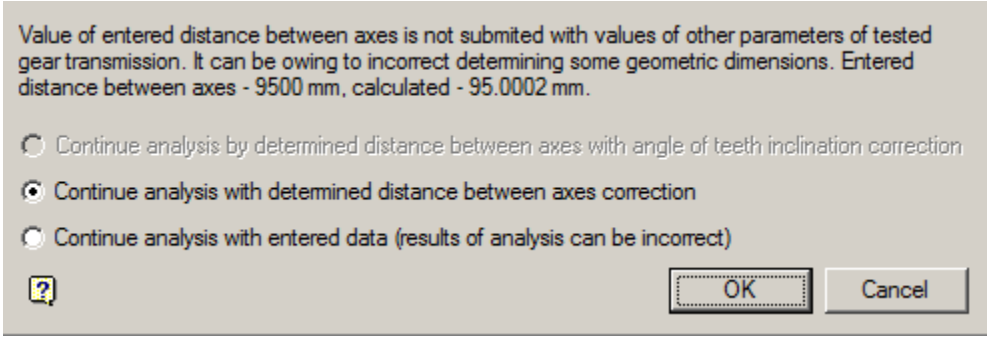


From this window, click on the icon You can go to the Calculation wizard. If any payments were previously carried out, their results can be accessed using the drop down menu.



If the entered spacing does not match the entered calculation parameters, then when it changes you will receive a message which will be offered options:

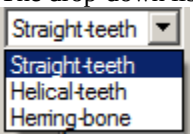
- Continue with the correction of the calculation specified center distance
- Continue with the entered payment data (calculation results may be incorrect)



Initial data

Type of gears

The drop-down list of the gear types allows you to select one of three options: spur, helical, herringbone or wheel:



Angle of teeth inclination

When choosing a helical or herringbone wheels in the "angle of the teeth" should enter a value in degrees of the desired angle of inclination of the tooth line in the recommended range (typically 8-22 ° for helical and 25-40 ° for chevron wheels).
If this field is left blank, the program installs by default a certain angle, which lies within the specified limits. Most often, this angle imposed by the previous calculation. It can be changed either directly or by subsequent adjustment calculation.

Distance between axes

Pitch

Rotation frequency of driven gear shaft (1/min)

The field is filled in accordance with the terms of reference and the expected characteristics of the engine.

Torque moment of driven gear shaft (Nm)

The field is filled in accordance with the terms of reference and the expected characteristics of the engine.

Required resource (hour)

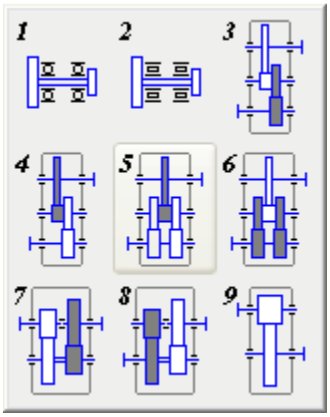
The field is filled in accordance with the terms of reference and the expected characteristics of the engine.

Divisibly of max moment of motor

The field is filled in accordance with the terms of reference and the expected characteristics of the engine.

Degree of accuracy GOST 1643-81

Kinematical scheme



The two-stage and three-stage gear units can be run on deployed or coaxial. In simple circuits deployed, each stage consists of a pair of gears. The disadvantage of such schemes - an uneven load distribution between the bearings, so that the resulting deformation of shafts occurs the load concentration along the length of the tooth, causing the need for a shaft with high rigidity.
This disadvantage is eliminated by using bifurcated stages. Preferably in the two-stage gear as forked use a high-speed, and in the three-stage - the intermediate stage.
The large width of a low-speed stage in these cases does not cause a significant load concentration along the length of the tooth due to the symmetrical arrangement of gears with respect to the supports.

Load conditions

Load conditions is selected from the dropdown list.

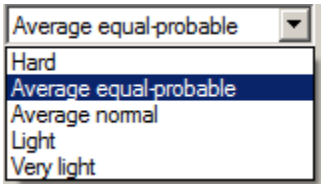


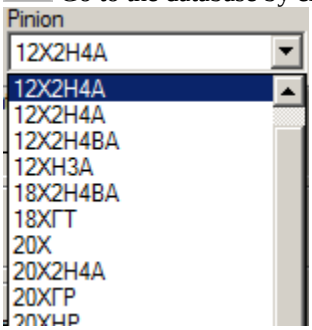
Table of values of equivalence ratios for each loading mode:

Load conditions	μ_h	μ_F	
		Normalization, improvement q=6	Hardening, cementation q=9
0 - Simple	1.0	1.0	1.0
1 - Hard	0.5	0.3	0.2
2 - Average equal-probable	0.25	0.14	0.1
3 - Average normal	0.18	0.06	0.04
4 - Light	0.125	0.038	0.016
5 - Very light	0.063	0.013	0.004

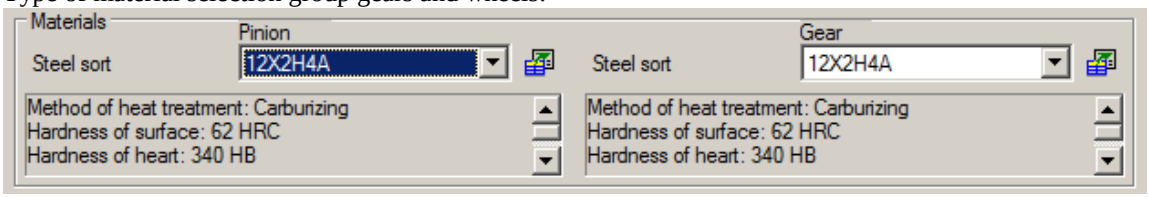
Group "Materials"

Materials of gears and their characteristics are selected from a drop down list or from the database.

Go to the database by clicking on the appropriate button to the right of the selection grade of steel.



Type of material selection group gears and wheels:



For convenience, you can select to see comments to the selected material.

Group "Reverce transmission"

The group contains the following fields:

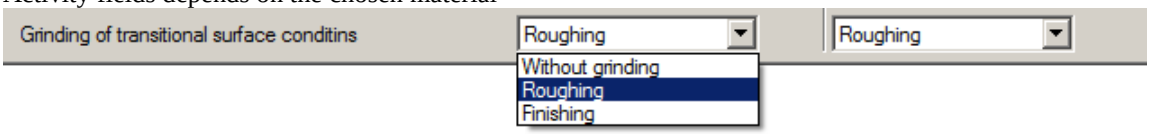
- Rotation frequenct of driven gear shaft
- Torque moment of driven gear shaft (Nm)
- Required resource (hour)

Check "Transitional surface is polished"

Check "With tooth root inclusion"

Grinding of transitional surface conditins

Activity fields depends on the chosen material

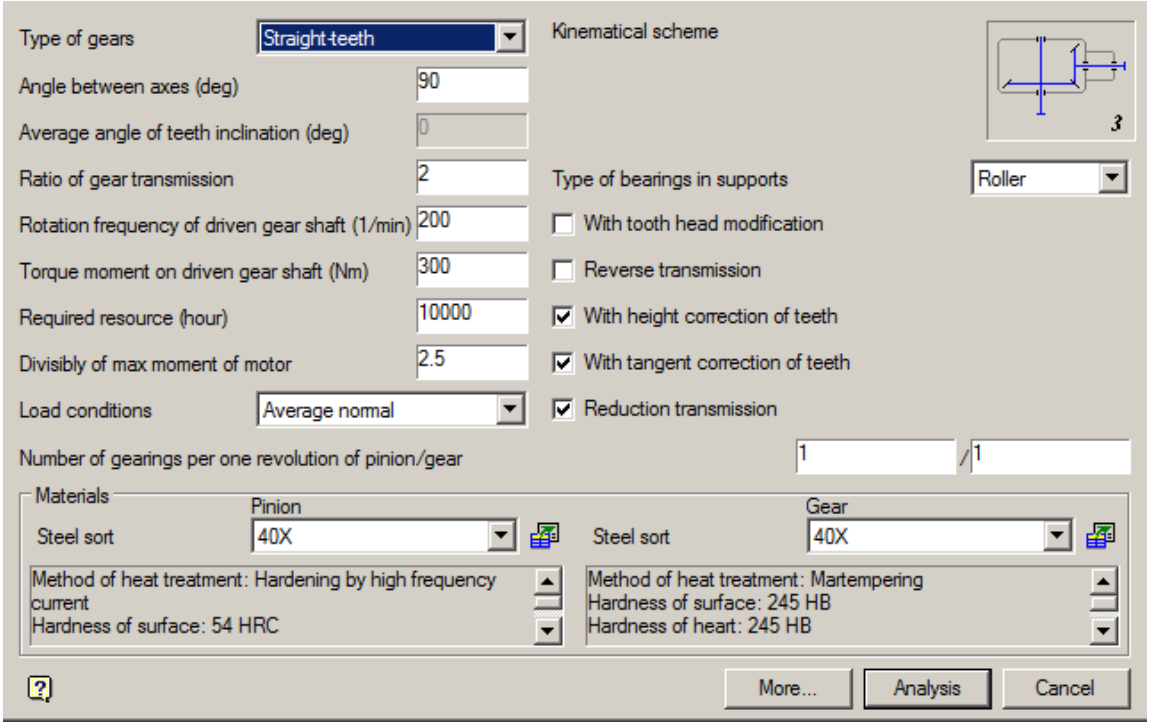


If the transition is made grinding surface of the teeth, the required grinding mode can be selected from the corresponding drop-down list.

Bevel Gearing Strength Analysis

Calculation wizard: Bevel Gearing Strength Analysis.

Library: Calculations - Bevel Gears - Bevel Gearing Strength Analysis

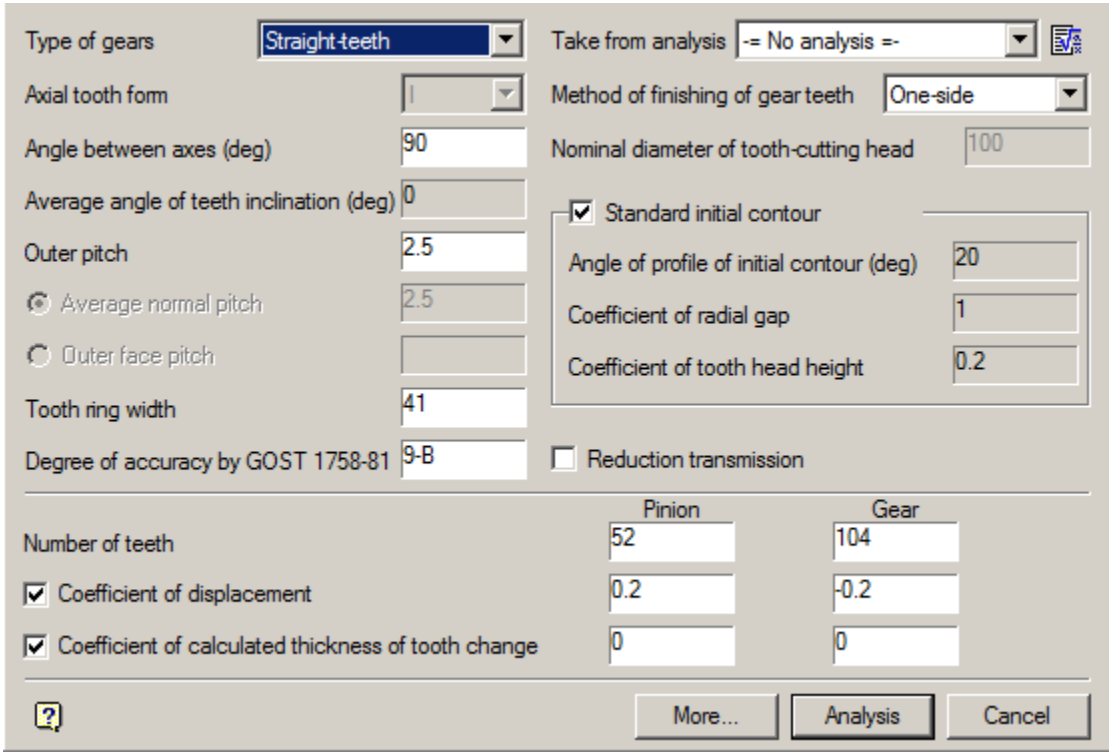


Geometric Calculation of Bevel Gearing

***Calculation wizard**:* Geometric Calculation of Bevel Gearing.

Library: Calculations - Bevel gears - Geometric Calculation of Bevel Gearing

Calculation of the geometric parameters and the conical gearing elements can be made based on user input (checking the strength of finished transmission) or on the basis of the strength calculation.



Static beam calculation

Main menu: ***Mechanical - Design - Calulations - Static beam calculation***.

Toolbar: ***Calculation beams (toolbar*** "MCS Calculations"***)***.

Command line: ***MCBEAM***.

Library: ***Calculations - Loads - Beam***.

Library: ***Calculations - Loads - Calculations - Beam calculation (applied to the finished beam affixed to the supports, loads and moments, see. Calculation)***.

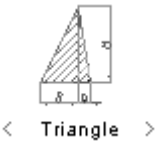
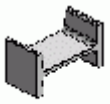
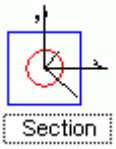
Calculation wizard: ***Baem calculation (applied to the finished beam affixed to the supports, loads and moments, see. Calculation)***.

This command is used to calculate the strength of the direct beam of constant cross-section.
After selecting the command a dialog box opens ***beam calculation***.

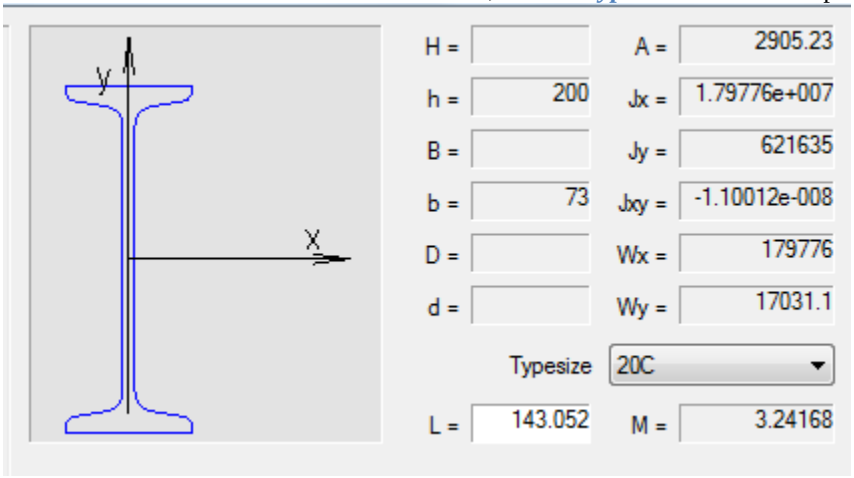
1. On the left side of the dialog box, select the ***cross-section of the beam***.
2. Right set ***sectional dimensions*** and ***length L beams***. For visual selection of the beam length, click ***Dynamic input*** at the bottom of the dialog box.
3. Enter the ***physical characteristics of the material*** and the ***cut rotation***.
4. Specify moment and the loads acting on the beam.

Setting section

There are three ways to specify the section:

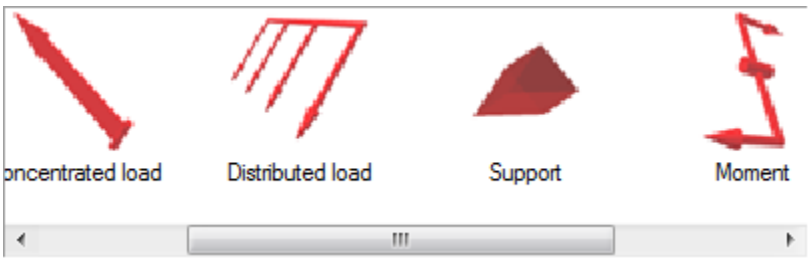
	Standard geometric cross section (e.g., triangle).
	Rolled metal base elements of nanoCAD Mechanica 20.1 (e.g, I-beam GOST 8239-89). To use the base when calculating the beams need to set a Class 725.
	Arbitrary section defined by the user. Sizing is carried out in several ways: • Click inside the closed area. Automatically determined by the outer contour of the closed area and the region is added to the set. • Clicking on a closed polyline or circle. Added to the area bounded by a circle or polyline • Reselection area removes it from the set for the calculation of the cross section. After selection of the drawing section in the dialog box ***Beam calculation*** appears custom cross-section:  Change the name of the custom section by selecting it and clicking the left mouse button on the title. Delete a custom section of the list is made by pressing ***delete***.

For the standard section of the database items, choose ***Type size*** from the dropdown list.

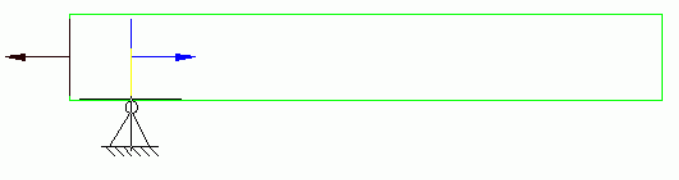


Beam Load and moments

1. In the dialog box, double-click the desired ***type of load*** or ***moment***.



2. Select ***insertion point*** or the load bearing on the beam. At the time of selecting the insertion point move the cursor to the beam to install dependencies.



3. Set the ***parameters*** or load bearing in the dialog box.

- ***Support***

Point - the distance from the starting point to the point of insertion beam bearing in millimeters. Use keys to select the type of support.

- ***Concentrated load***

Position - the distance from the starting point to the point of insertion beam load, mm.
Value - the value component of the load in the selected direction, H.
Point - the distance from the insertion point to the point of load application of the load in the selected direction, mm.
Diagram on the left side of the dialog box explains the meaning of the input parameter.
Tabs on the top of the window are used to select the coordinate system in which the values are set:
Rectangular
Spherical
Cylindrical

- ***Distributed load***

Position - the distance from the starting point to the point of insertion beam load, mm.
Length - length of the section, which is attached to a distributed force, KN/mm.
In input fields, set the value component of the load in the selected direction, H.
Diagram on the left side of the dialog box explains the meaning of the input parameter.
Tabs on the top of the window are used to select the coordinate system in which the values are set:
Rectangular
Spherical
Cylindrical

- ***Moments***

Diagram on the left side of the dialog box explains the meaning of the parameters.
X, Y, Z - the components of the bending moment , KN*m.
C - distance from the starting point to the point of insertion beams torque , mm.

Calculation

Click ***Calculate beam*** at the bottom of the dialog box to perform the calculation.

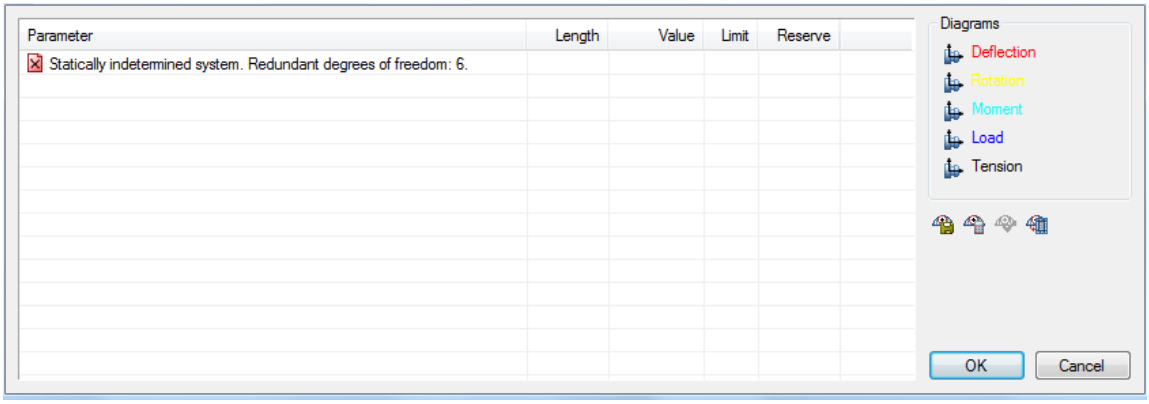
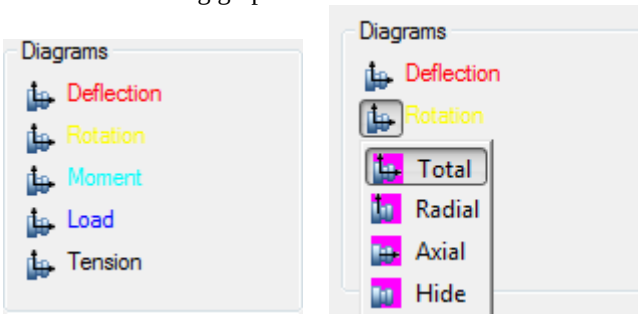
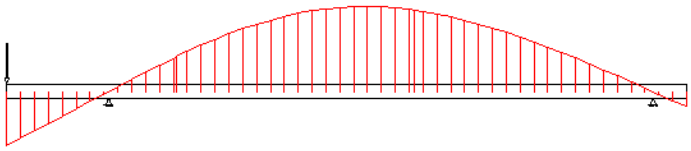


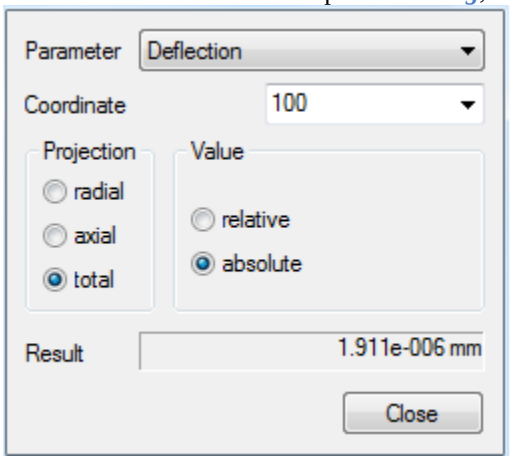
Table of results of calculating comprises calculating the maximum deflection and stress values as well as the distance from the starting point of the beam to a point at which the maximum of each parameter.
To insert a drawing graphs strain and effort to select the desired type of graph for each feature:



Button ***Apply*** adds graphics drawing.



Button ***Save Report***. Stores the results of the calculation of the beam to an external file.
Button ***Calculate value***. Opens ***Meaning***, in which parameters are calculated at an arbitrary point of the beam.



Button *Export diagrams*. Sell calculation results in a table in Construction Site 20.1. After selecting the command, specify the insertion point on the drawing table.

Calculate section characteristics

Main menu: *Mechanical - Design - Calculations - Calculate section characteristics*.

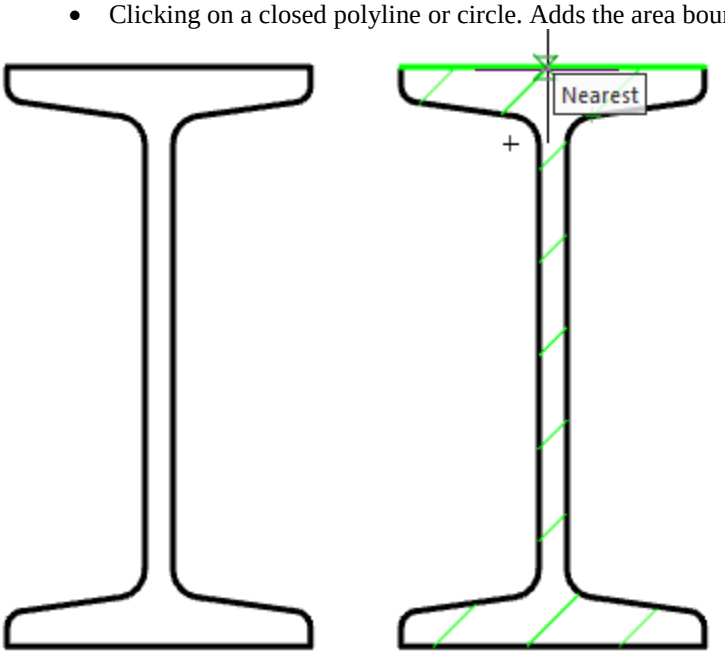
Toolbar: *Calculate section characteristics (toolbar* "Calculations"*)*.

Command line: *MCGCS*.

This command is used to calculate the geometric characteristics of the complex cross-sections with respect to arbitrary axes.

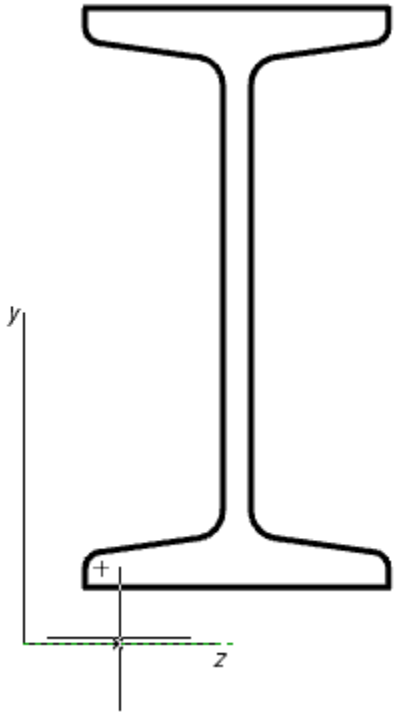
Calculation procedure

1. Call command.
2. In the drawing, then click the area closed by one of the ways:
   - Clicking inside a closed area. Automatically determined by the outer loop of the closed area, and the area is added to the set.
   - Clicking on a closed polyline or circle. Adds the area bounded by polyline or circle.

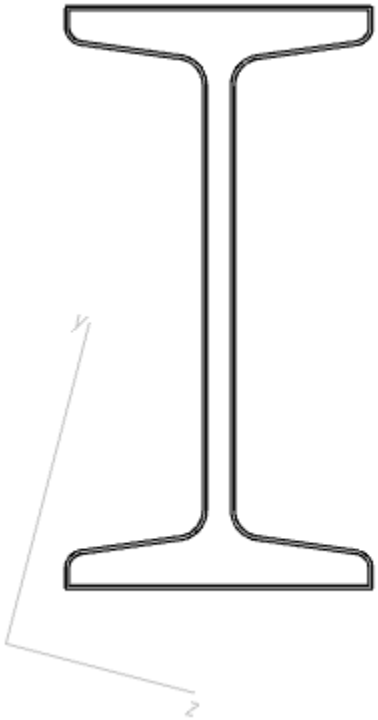


Note: Repeated selection of the field deletes it from the set to calculate the cross-section.

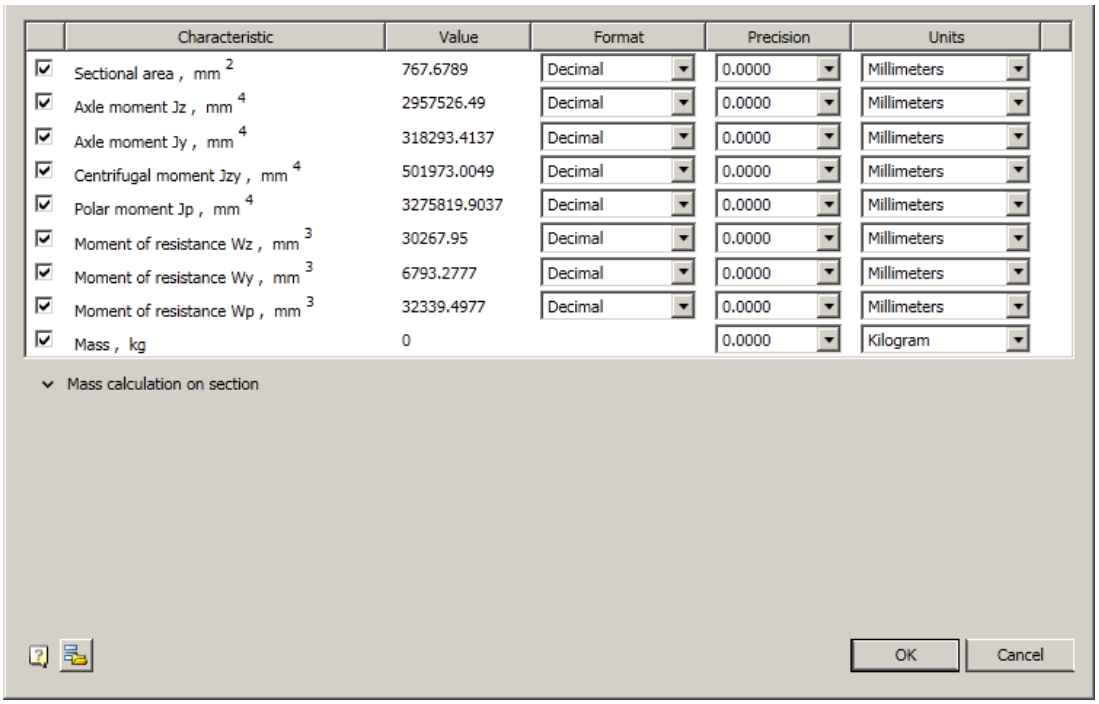
3. Complete the selection by clicking the right mouse button or the Enter key.
4. Specify the origin point or [central axis]. If you select the "central axis", the point of origin is automatically set in the center of mass of the cross section with an angle of rotation equal to 0 (step 5 is skipped).



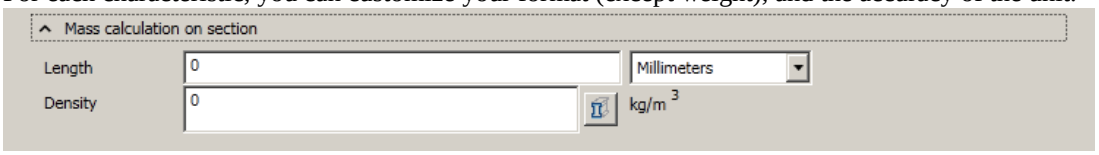
5. Specify the angle of rotation of the coordinate system.



6. In the dialog box "Geometric section properties" Adjust the characteristics included in the summary table of the results.

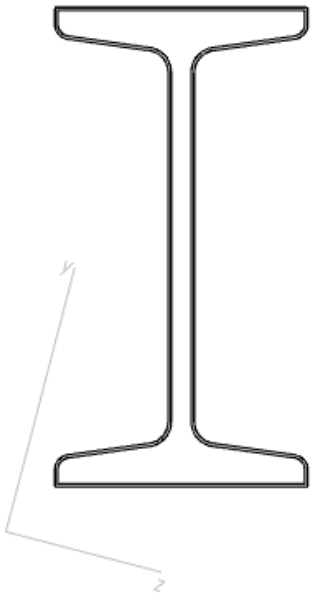


For each characteristic, you can customize your format (except weight), and the accuracy of the unit.



To calculate the weight you need to open an additional menu "Mass calculation on section", which indicate the product density and length. When you specify the length of the need to expose the units. When you specify the density, you can use the base materials, by clicking on the button "Material selection".
Customized settings display characteristics (size, precision, units) stored for later calculations sectional characteristics. To return to the default settings, press the button "Load default settings".

7. Click "OK" in the dialog box and enter the drawing an insertion point of the table with the calculated parameters.



Parameter	Value	Units
Sectional area	767.6789	mm^2
Axle moment Jz	2957526.49	mm^4
Axle moment Jy	318293.4137	mm^4
Centrifugal moment Jzy	501973.0049	mm^4
Polar moment Jp	3275819.9037	mm^4
Moment of resistance Wz	30267.95	mm^3
Moment of resistance Wy	6793.2777	mm^3
Moment of resistance Wp	32339.4977	mm^3
Mass	0	kg

After the calculation is added to the drawing object nanoCAD Mechanica 20.1, comprising a cross-section schematic outline and the main central axis. The lengths of the axes are proportional to the values of the corresponding axial moments of inertia.

Fastening calculation

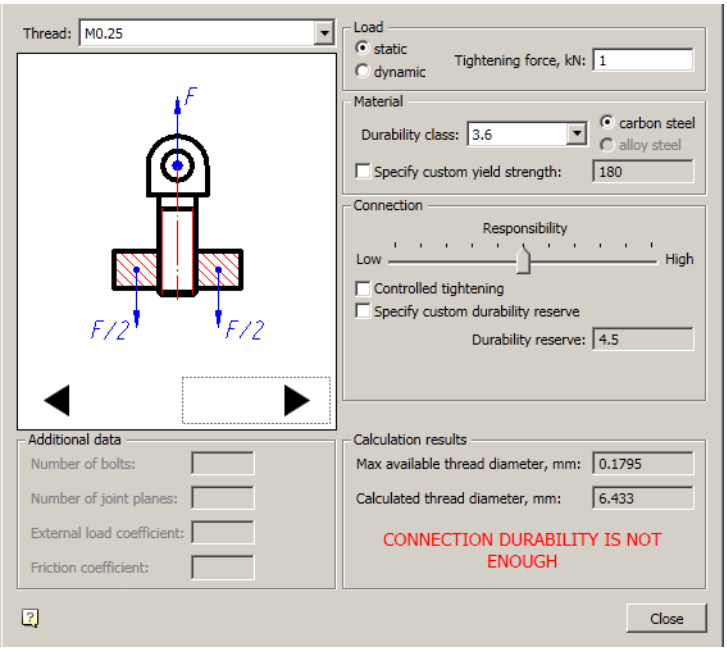
Main menu: ***Mechanical - Design - Calculations - Fastening calculation***.

Ribbon: ***Mechanical - Design - Fastening calculation***.

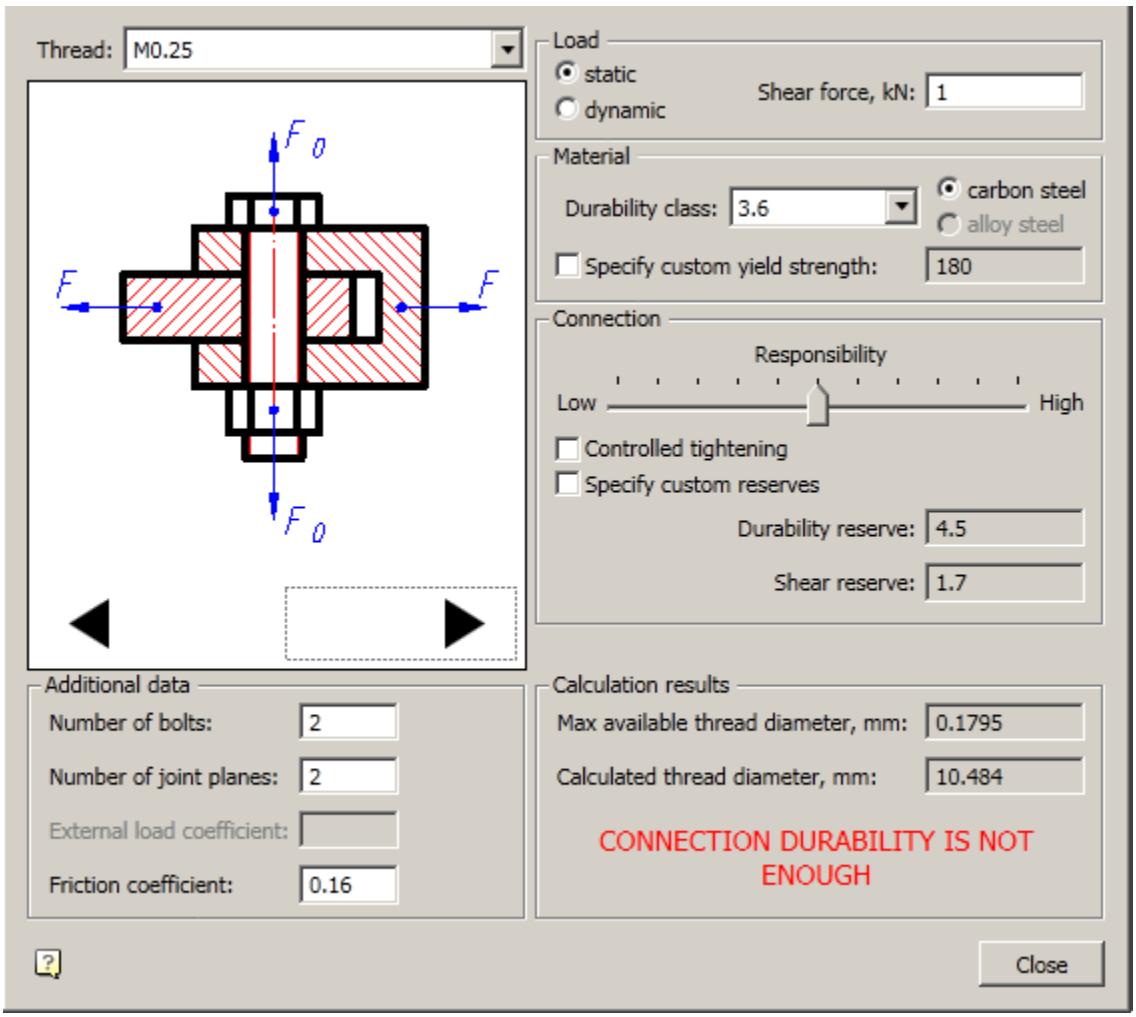
Toolbar: ***Calculations - Fastening calculation***.

Command line: ***MCBOLTDESIGN***.

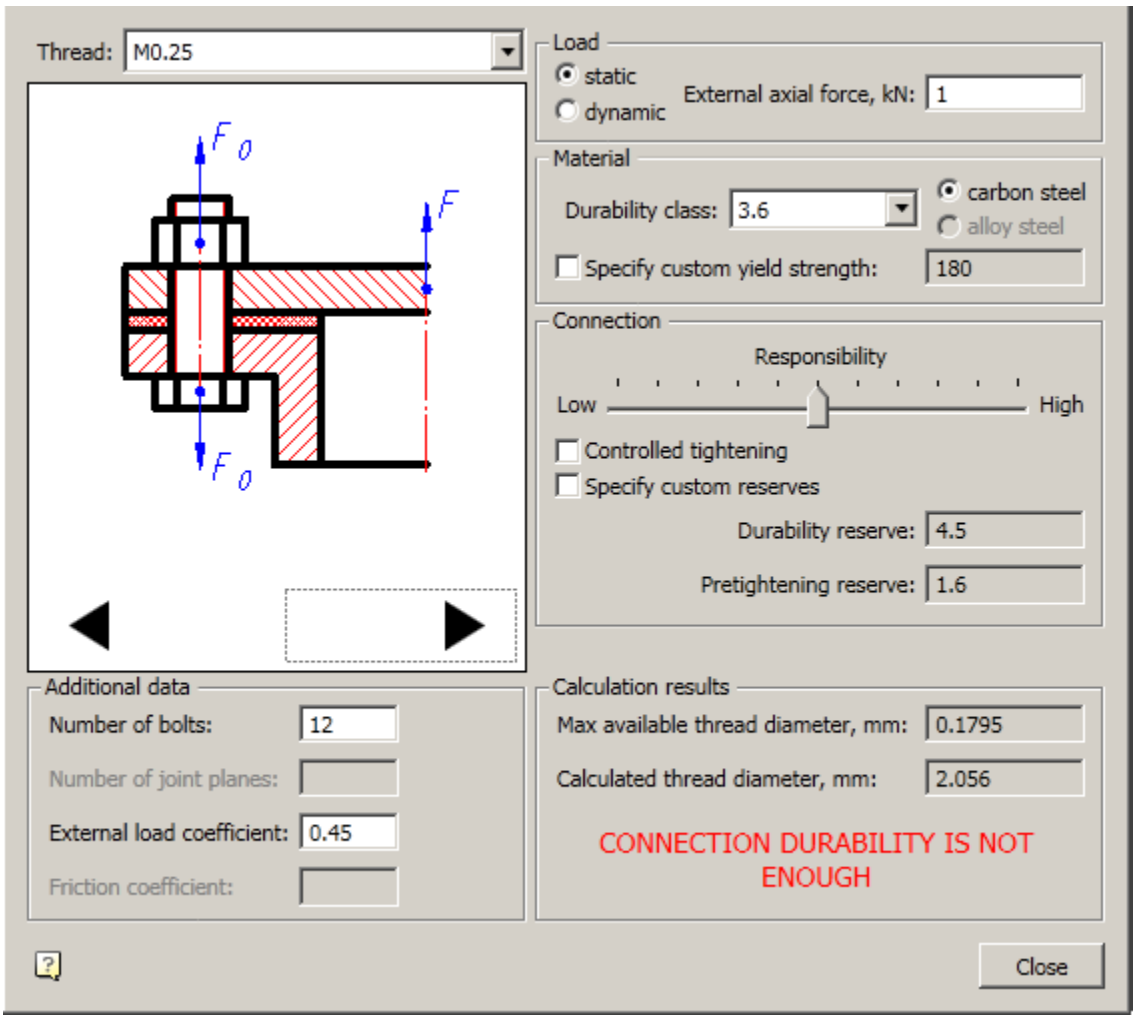
Verification calculation to determine the admissibility of the selected thread for the specified load conditions.
The basis, when checking calculations bolts adopted methodology contained in the textbook "Kuklin NG, Kuklin GS machines Details - M .: Vyssh.shk., 1987".
In calculating the user is able to take into account the recommendations of "GOST 1759.4 - 87 Bolts, screws and studs Mechanical properties and test methods.".
When you call the command, a dialog box "Calculation check bolt". By controlling the arrows in the graphic menu, select the version of the calculation.
1 variant - The connection is tightened during assembly, there is no external load.



2 variant - Compound loaded transverse force.



3 variant - Connections are tight and loaded external axial tensile force.



Depending on the options available to calculate the various parameters.

Note:	The result of the calculation is the message in the "Results of calculation" on the sufficiency of the bond strength. Furthermore, in the "calculation results are" options are displayed: Max. valid thread diameter (mm) - inner diameter of the thread selected bolt; Current thread diameter (mm) - the calculated internal diameter. If the first parameter is greater than the second, the strength is sufficient.

Dimension chains

Teams of calculating dimensional chains are located in the Object Manager nanoCAD Mechanica. To access it you need to switch to the "Dimension chains" panel object manager nanoCAD Mechanica.

- Fundamental rules
- Calculation methods
- Icons tree calculation
- Toolbar
- The sequence of calculation

Fundamental rules

It proposed two methods of calculation: minimum-maximum method and the probabilistic method.
Measures involved in the calculation may be specified as one or several separate drawings.
The project is monitored dynamically link with the size of the drawings. If the size is different from the drawing or missing from the drawing, it will be highlighted in bold.
Calculated closing unit contains the calculated values of the nominal value, the upper ES and lower EI deviations.
All dimensions are displayed in the 1: 1 scale.
In edit mode, you can create new master links, remove the entire branch graph, edit size, surface combine different branches (the movement of the nodal points "grips").
At the end of the table shows the calculation of the calculated deviations closing units.
It supports only linear dimensional chains.

Calculation methods

Maximum / minimum

Nominal closing link:

$$N_{\Delta} = \sum_{i=1}^{n} N_i$$

n - number of units, except the locking.
The upper limit deviation of the closing unit:

$$ES_{\Delta} = \sum_{i=1}^{n} \begin{cases} ES_i & \text{positive direction of the circuit} \\ -EI_i & \text{the negative direction of the circuit} \end{cases}$$

Lower limit deviation of the closing unit:

$$EI_{\Delta} = \sum_{i=1}^{n} \begin{cases} EI_i & \text{positive direction of the circuit} \\ -ES_i & \text{the negative direction of the circuit} \end{cases}$$

Probabilistic

Nominal closing link:

$$N_{\Delta} = \sum_{i=1}^{n} \left(N_i + \frac{ES_i - EI_i}{2} \right)$$

ES - upper deviation;
EI - lower deviation;
n - number of units, except the locking.
The upper and lower deviation of the closing level will be symmetrical:

$$ES_{\Delta} = -ES_{\Delta} = \frac{t}{2} \sqrt{\lambda^2 \sum_{i=1}^{n} (ES_i - EI_i)^2}$$

t - risk ratio.

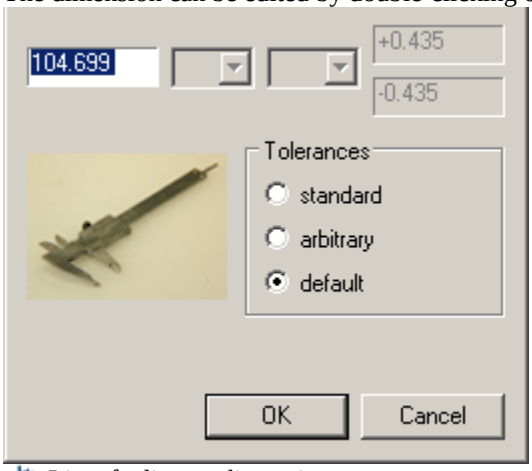
$$t = 5 - \frac{2.45}{P^{-0.14}}$$

P - risk percentage;
λ2 - coefficient taking into account the law of distribution of real denominations.
Adopted 1/9 for the Gaussian distribution, for the distribution of 1/6 and 1/3 Simpson for uniform distribution.

Note:	When calculating the maximum / minimum denomination of an unknown can be set on a single link, the upper deviation of the unknown - on the other, the lower - in the third. The probabilistic method such chain is not counted.

Icons tree calculation

Project
Group size may contain several cycles of size.
View thumbnails while editing the graph.
Drawing
Group sizes, taken from a drawing. Path Drawing dropped, compared to only a name, that is the size of the C: \ Work \ Drawing1.dwg and C: \ Acad \ Drawing1.dwg will be placed in the same group "Drawing1".
Dimension
Dimension added to the drawing.
Next to the icon indicates the value of the size tolerances. Instead, you can enter any name (for example, the width of the bearing).
The value of the dimension can be viewed in a tooltip. If the dimension of the drawing was linked text below the dimension line, its name will be assigned automatically. To display the nominal, enter an empty name.
The dimension can be edited by double-clicking or from the context menu.



List of adjacent dimensions
The icon reflects the relative position of a given size (blue rectangle) and the adjacent (green rectangle).
Unknown closing link
You can add a calculation for editing or drawing a graph by adding the size in which the value and / or deviations are "?". It contains a list of increases or decreases dimensions (presumably).
Switching link
It contains the calculated values of the deviations:

- face value ;
- upper deviation ES;
- lower deviation EI.

Note:	When you edit or delete any dimension calculated counting all units nulled.In project

	dynamically tracked due to the size on drawings. If dimension different from the drawing or missing from the drawing, it will be highlighted in bold fonts. Again take the value from the drawing context menu (How drawing).

Toolbar

Create calculation
If the button is not available, were selected to be the calculation start or finish editing the graph.
The project can be any number of calculations. They are translated independently. By default, a calculation is named "New".

Open project
Projects are stored in a special format SPJ. If an existing project is not saved, it will be lost. Warnings are not provided.
Project files can be transferred to other folders on other machines. The relative path from the project file to the file associated drawings should not be modified so as not lost contact with the size of the project in the drawings.

Save project

Save project as…

Add dimensions
From the selected objects based on the current filter will be selected either all or only the horizontal or only the vertical dimension. In the column all sizes will be rotated horizontally.

Change filter
Setup select the type of dimensions (horizontal, vertical).

Start/finish editing
The beginning and the end of the calculation of dimensional chains. This command displays the current drawing on the graph for the current calculation.
Pressing will clear the graph with the drawing and will in the calculation of the changes.
In edit mode, you can create new master links, remove the entire branch graph, edit size, surface combine different branches (the movement of the "grips").

Set to drawing
Makes changes made in the current account, the original drawings. Warnings are not provided!

Get from drawing
Fixes dimension values to those given in the drawings.

Calculate chain
Calculates all master links the current calculation.

Create report
It puts the current drawing table the report and the distribution schedule tolerances.

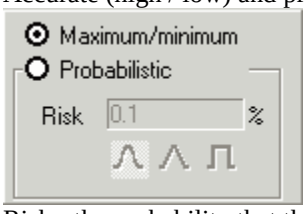
Closing link
Adds closing link. The command is only active during editing of the graph. After pressing need to sequentially select two surfaces of one or more branches of the graph (in the drawing).

Non-specified limiting tolerancies

Sets limiting tolerancies for unspecified tolerances. Limiting tolerancies is selected from the drop down menu.

Calculation method

Accurate (high / low) and probabilistic.



Risk - the probability that the real value of size does not provide a closing link in getting the calculated border. The possible values are from 32% to 0.001%. Recommended - 0.01%.

The law of distribution of real values of size. From left to right: a normal distribution (Gaussian), triangular (Simpson) and linear (uniform).

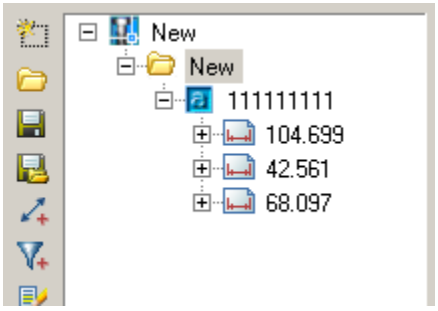
If the law is unknown, it is recommended to take a linear distribution.

Help...

The sequence of calculation

1. By clicking on the icon "Create calculation" in the browser creates a folder sizes. By default, it is named "New".

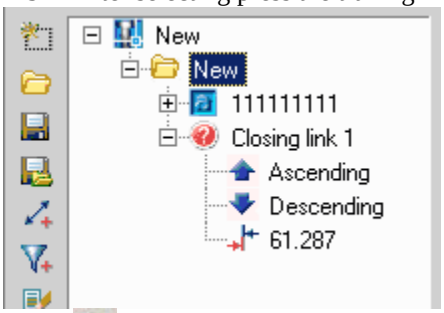
2. After clicking on the icon "Add dimensions" the drawing must specify the dimensions involved in the calculation.



Information on the selected size appears in the browser.

3. Press "Start/finish editing".
4. Becomes available team "Closing link". Enter the drawing the locking unit (multiple master links).

5. After selecting press the trailing links "Start/finish editing". The information in the browser changes.



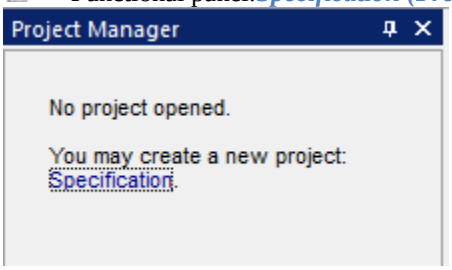
6. Calculate chain. The browser displays the results of the calculation.

The calculation results are displayed in a table in the drawing by clicking on the icon "Create report".

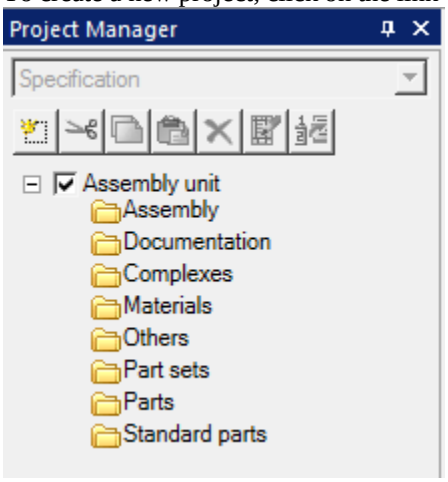
Drawing specifications

Project Manager

Functional panel:***Specification (Project Manager)***.



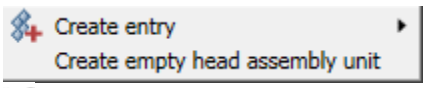
To create a new project, click on the link *"Specification"*.



When creating a project, a head empty assembly unit is created. An assembly unit includes default specification sections.

Toolbar

Create - the command allows you to create an entry or empty head assembly unit.



Cut - the command cuts an entry for later insertion into another assembly unit.

Copy - the command copies the entry for subsequent insertion into another assembly unit.

Paste - command inserts a previously cut or copied entry. The insertion is made in the same section of the specification from where the entry was cut or copied.

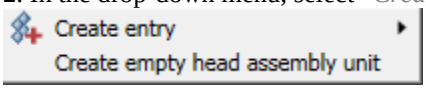
Delete - command deletes entry.

Delete specification - the command opens the specification editing dialog.

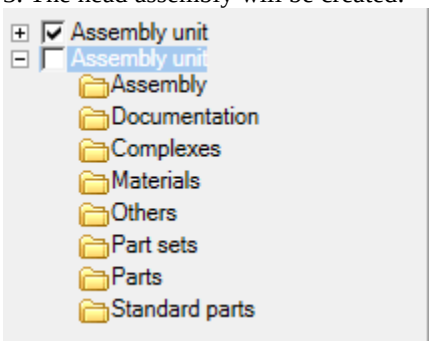
Create position - the team creates a position on the occurrences: Assembly units, Details, Standard products, Other products and Materials.

Creating an empty head assembly unit

1. On the toolbar, click *"Create"*.
2. In the drop-down menu, select *"Create empty head assembly unit"*.



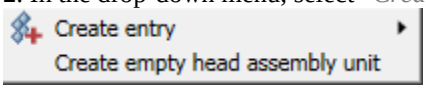
3. The head assembly will be created.



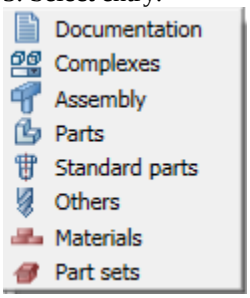
The active build is marked with an enabled switch.

Creating entries

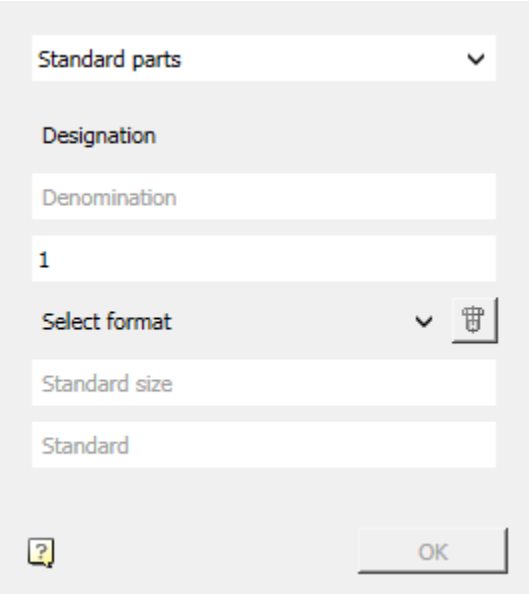
1. On the toolbar, click *"Create"*.
2. In the drop-down menu, select *"Create entry"*.



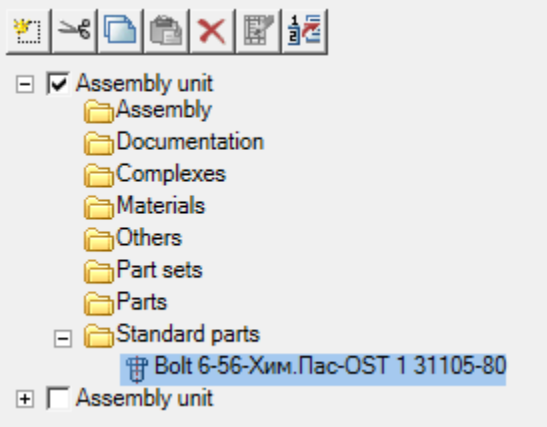
3. Select entry.



4. In dialogue *"Creating entry"* fill in the required parameters. Depending on the type selected, different fields will be available.

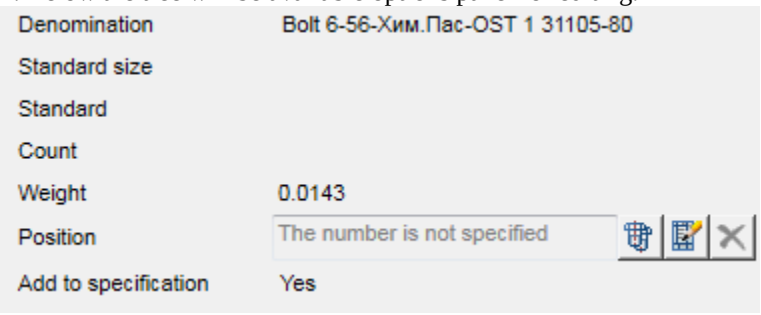


5. After entering all the necessary parameters, the "OK" button will be available. Confirm your entry.
6. The entry is added to the project manager tree.



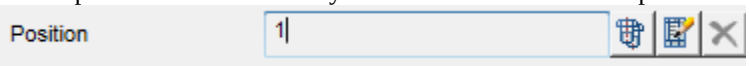
Entry Editing

1. Select an entry in the tree.
2. Below the tree will be available options panel for editing.



Create position

1. Select entry: Assembly units, Parts, Standard products, Other products or Materials.
2. Click the button on the toolbar *"Create position"*.
3. Insert a position in the drawing and fill in the data in *"Position leader edit"*.
4. The position field at the entry will be associated with the position in the drawing.

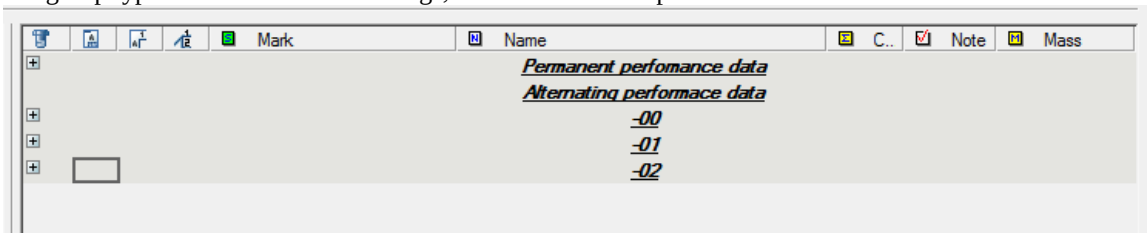


After creating a position, the following commands will be available:

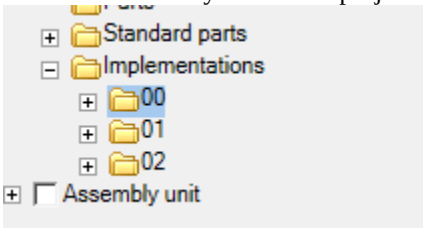
- *Show position* - shows the position on the drawing.
- *Edit position* - open *"Position leader edit"*.
- *Reset position number* - deletes position number.

Editing Performances

If a group type is set in the BOM settings, the sections of the performances will be available in the list of sections.



When placing the entry in the execution, the section "Executions" and the entry of the execution will be added in the tree of the assembly unit of the project manager.



The occurrence of the performance is called for editing by double-clicking the LMB. The dialogue allows you to create in the performance of new entries or copy them from the permanent data.
The dialog box consists of the sections: "Variable data" and "Permanent data".
In the section "Permanent data" are the occurrences of the specification. When you click on the "+" button, the entries are copied into the execution (in the section "Variable data").
In the section "Variable data" are occurrences of performance. Clicking on the "+" button will create an empty entry.
An empty entry can be edited by changing its Type, Designation, Name and Quantity.
To remove an entry from a performance, press the "-" button opposite the entry.

Specification

Main menu: ***Mechanical - Part list - Part list***.

Ribbon: ***Mechanical - Part list - Part list***.

Toolbar: ***Part list (*** "Part list"***)***.

Command line: ***MCSPECIFICATION***.

Refresh - refreshes numerals affixed, if they were not put down or changed the rules prostanovki item numbers.

Numbering - replaces characters "x" item numbers.

Export the drawing - creates the drawing table nanoCAD Mechanica 20.1 to output specifications for printing means nanoCAD.

Editing operation positions Specifications:

Cutting position of editor.

Copying position in the buffer.

Paste position.

Add record.

Deletes the current record.

Move up and move down - changes the position of the selected row specification.

Sort - sort position alphabetically within sections.

1

Add balloon - balloon to create the BOM item, if has not been set previously.

Properties -Setting Specifications.

Find the drawing - moves the current view and highlights the position of the drawing.

In the upper left corner of the specification contains the fields relating to the weight of the product.

Text field contains the calculated value of the product weight.
When you turn jackdaws *Update* estimated value product weight is automatically saved to the "weight" of the title block of the drawing.

Gray shaded box in the column specification section shows that this section does not contain any information.
Left mouse click on the symbol "Plus" can open (close) the contents of this specification section.

Adding assortment (material)

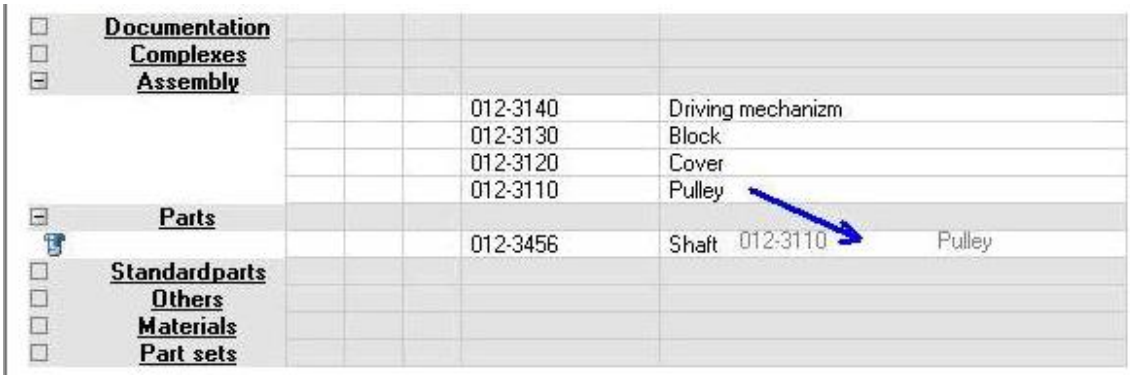
To create the material in the BOM editor uses the same mechanism as when filling the field *Material* titleblock drawing.

When you press the *Add Material* dialog box appears *material*.
Clicking on the icon *from the database* a dialog box material database or database TechnologiCS. Selected assortment appears in the editor window specifications as a simple string delimited prefix, the numerator and denominator:
Fractional representation of entries is restored when you export the drawing specifications.

Pressing the *search by keyword* run a full-text search database materials.

Relocating the entry between partitions

To transfer the recording details in another section hold down the left mouse on the portable recording and release it above the title of the desired topic.

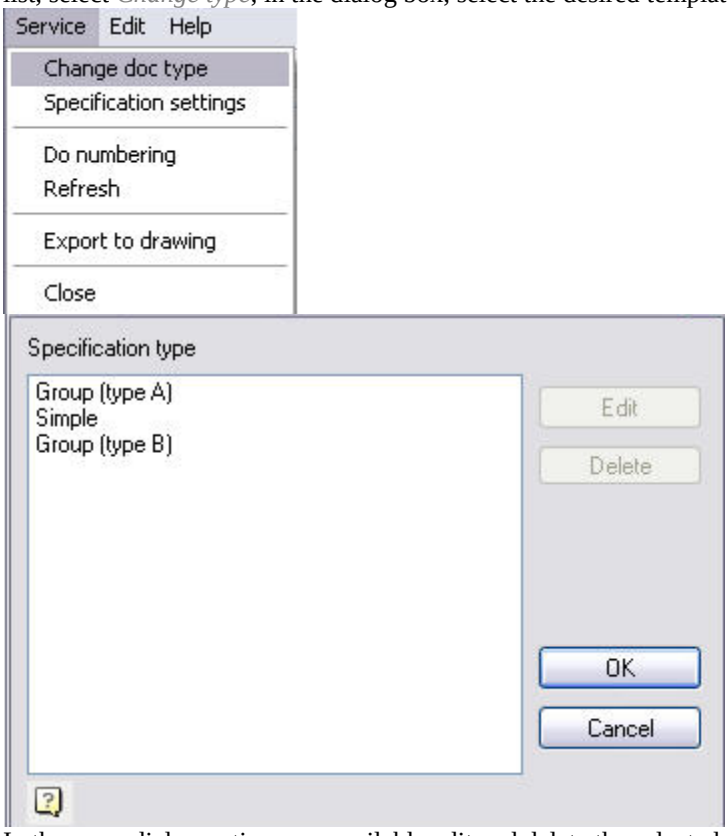


*Transferring records to section **Assembly Units***

Setting Specifications

Style setting specifications

To edit a template specification in the drop-down menu window *Editor specifications* tap *Tools*. In the drop-down list, select *Change type*, in the dialog box, select the desired template specification.



In the same dialog options are available edit and delete the selected template - through the configuration dialog specification.

Specifications Configuration Dialog

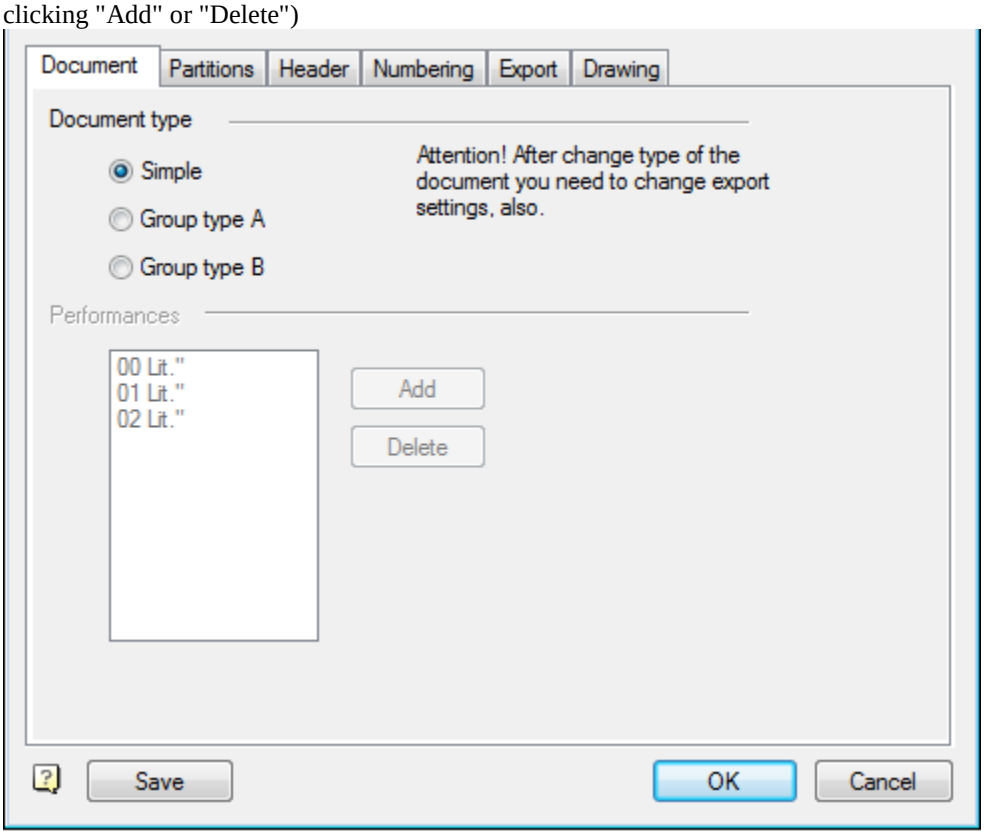
Settings dialog specification includes the following tabs:

Document tab

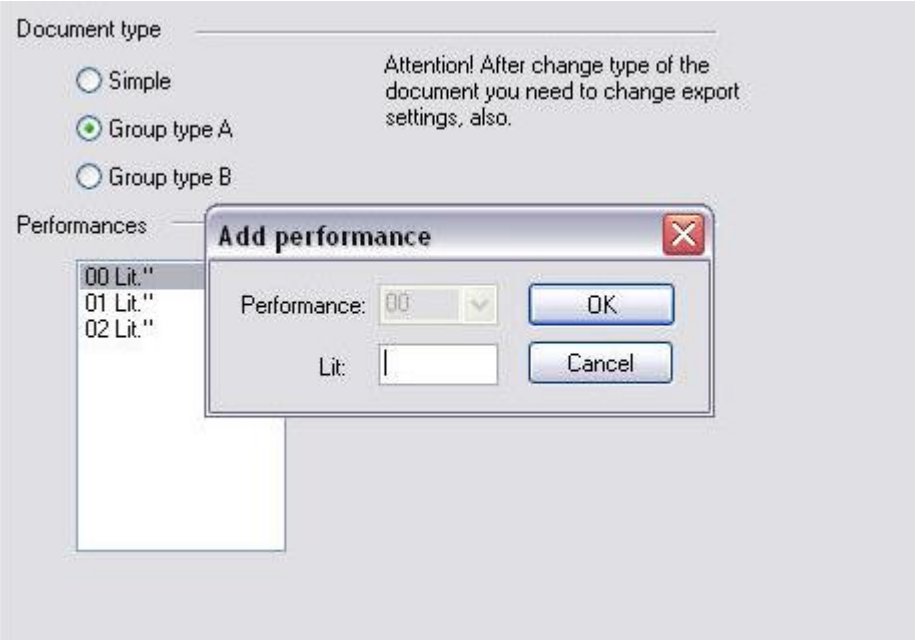
Tab document defines the type of document and marking performances.
The following types of specifications:

- Simple
- Group Type A
- Group B Type

When choosing a Type A or Type B list becomes active performances. List of performances can be edited (by clicking "Add" or "Delete")

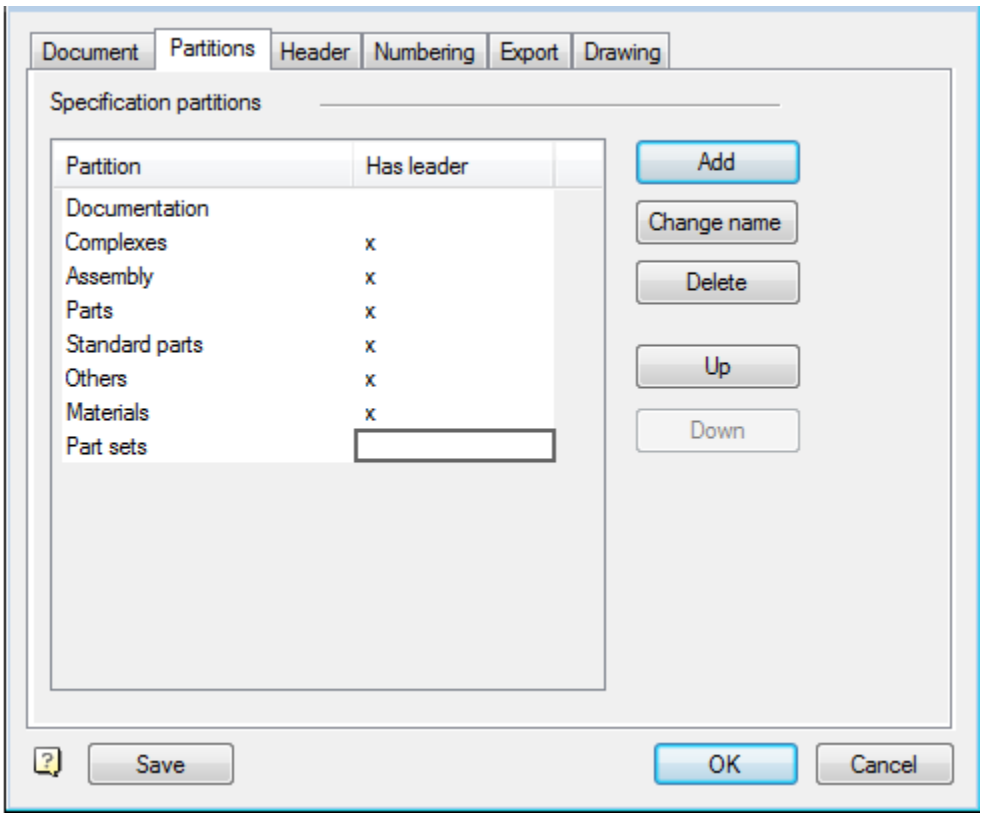


Double-clicking on performance causes him to edit. In the editing dialog can associate the letter execution.



Tab Sections

Categories tab allows you to customize the specifications section.
The partition table determines which sections and in what sequence will be present in the specification, and for which one has a notation of the specification callouts. Double-click on the field "Has the callout" enables or disables the callout for the corresponding section.
Controlling the placement of sections in the specification is performed using the "Up" and "Down."

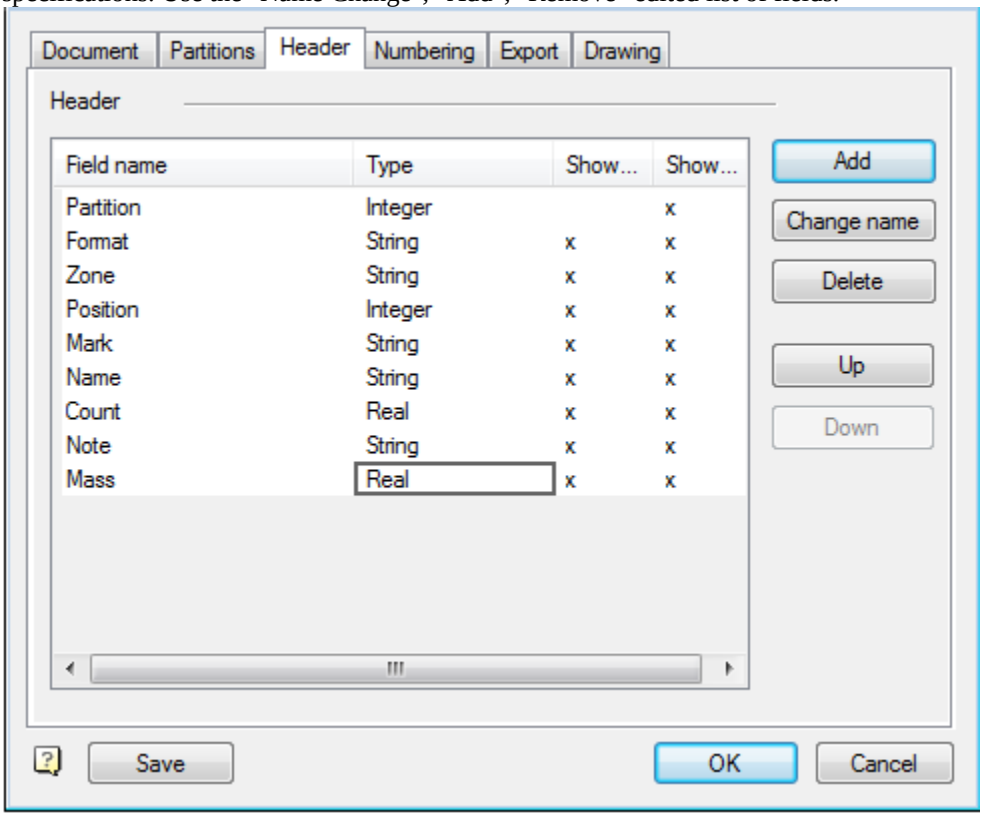


The "Name Change" Allows you to change the name of the selected partition.
"Add" button brings up a dialog to add a topic specification. Sections are added one by one. You can add a section with the standard name (if not already present in the specification) or with an arbitrary name. Galka "Has callout" duplicates the corresponding value in the partition table.

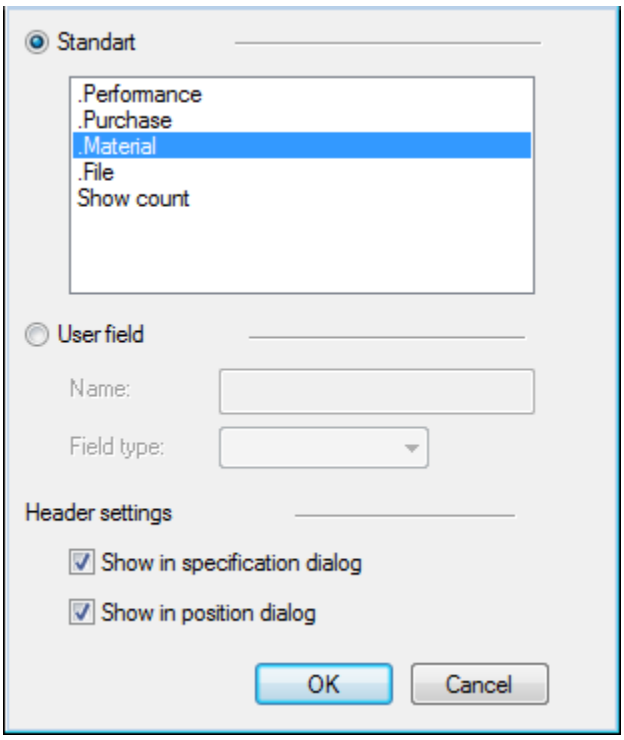
Tab Title

Title tab controls the composition and order of the fields used in the construction specifications.

Table fields specification includes the field name, type, and switches the display position in the editor and the editor specifications. Use the "Name Change", "Add", "Remove" edited list of fields.



Button "Add field" brings up a dialog where you can add fields from a predefined list, or an arbitrary field, specifying the type and display in the editor position and specification.

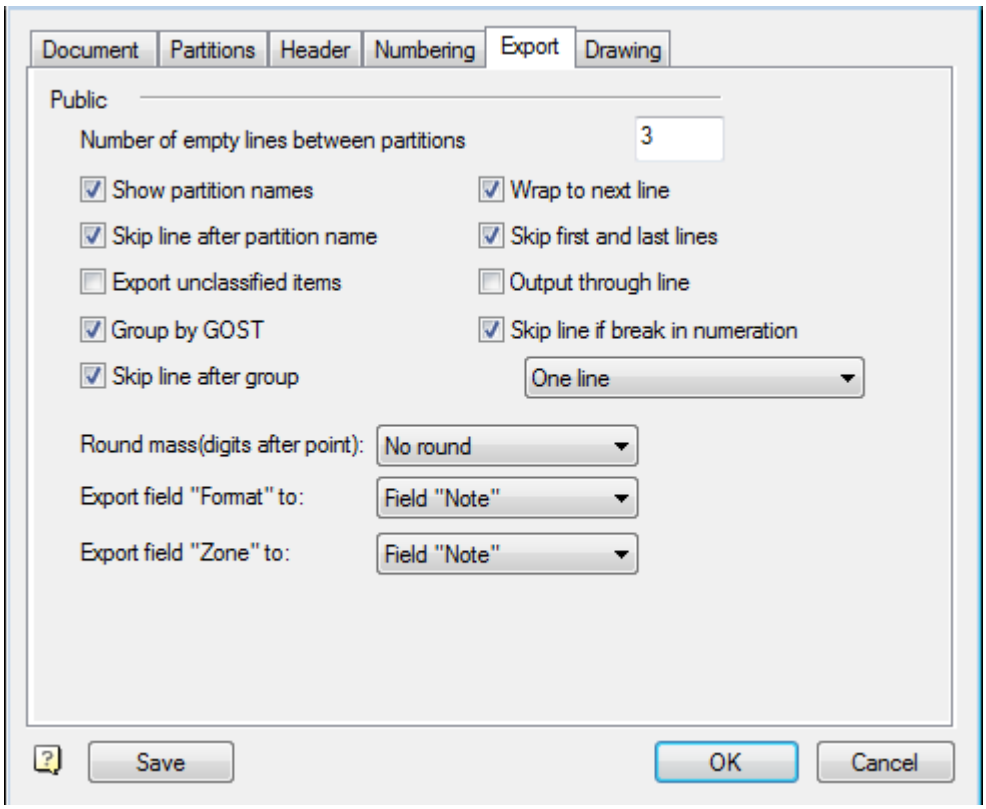


Numbering tab

On this tab, adjusted the numbering of positions in the specification.

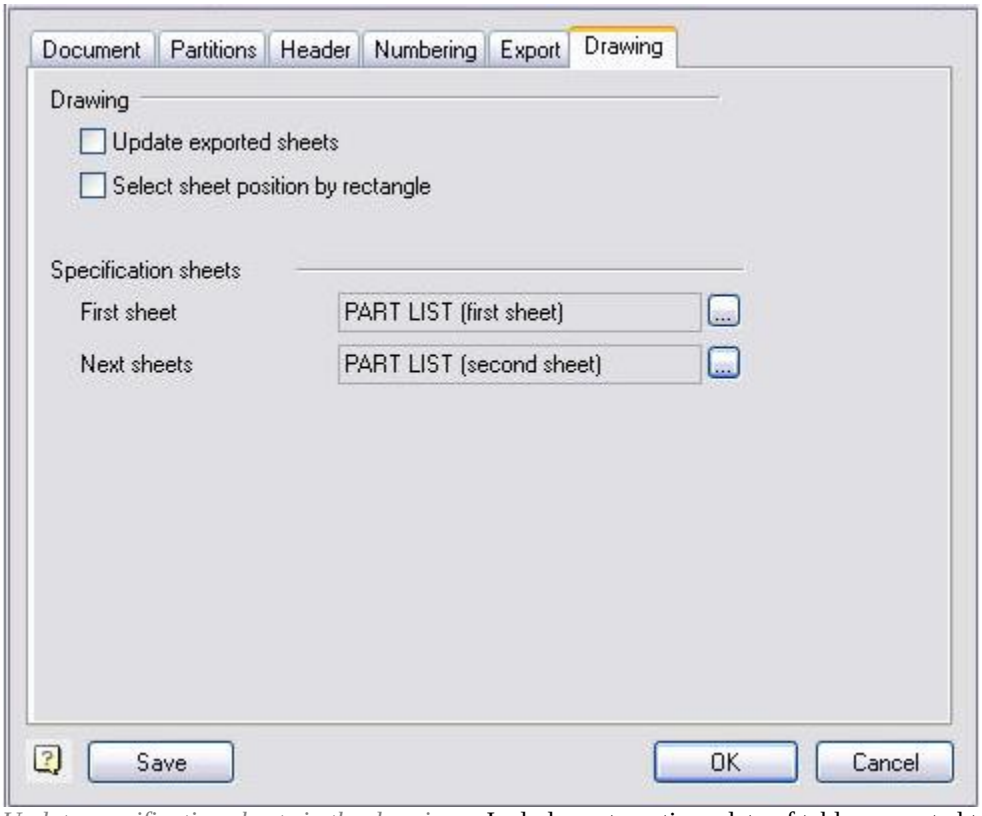
Export tab

On the Export grouped items export settings in the drawing and specifications in Excel.



General settings:
See sections - controls the display section headings in the exported specification.
Skip the line after section - includes extra blank line in the specification table between sections.
Must positions without partitions - if enabled, the position is not related to any section will be included in the export specification.
Grouped by GOST - one standard parts will be grouped general heading (for example, "nut OST 26-2041-96").
Skip the line after group - add a blank line after grouped according to GOST positions.
The number of skipped lines between sections - number of blank lines between sections of the specification.
Migrate to the next line - includes transfer to the next line with a long name products. If this option is disabled, the item name will be entered into a table cell.
Skip first and last lines - pass includes the first and last row of the table in the template specification.
Output at the line - each position specifications will be displayed with a space in one line.
Skip the line at break numbering - if the positions do not follow the order, after nonconsecutive lines added a line under a missing item. If the drop-down list shows a miss "single line" - that regardless of the number of missing items will be added only one empty line. If the drop-down list select a miss "depending on the gap," the number of rows to skip will depend on the size of the gap. (For example, if after 6 position followed by the position 9, then 2 lines will be skipped).
Followed by the output settings of the field "Format" field and "Zone" - show them in the "Comments", "Format" and "Zone" respectively, or do not show at all.
Option *Rounding weight* controls rounding up masses of parts made to specification. If nobhodimosti rounding on weight, you should select the required word length.

Drawing tab



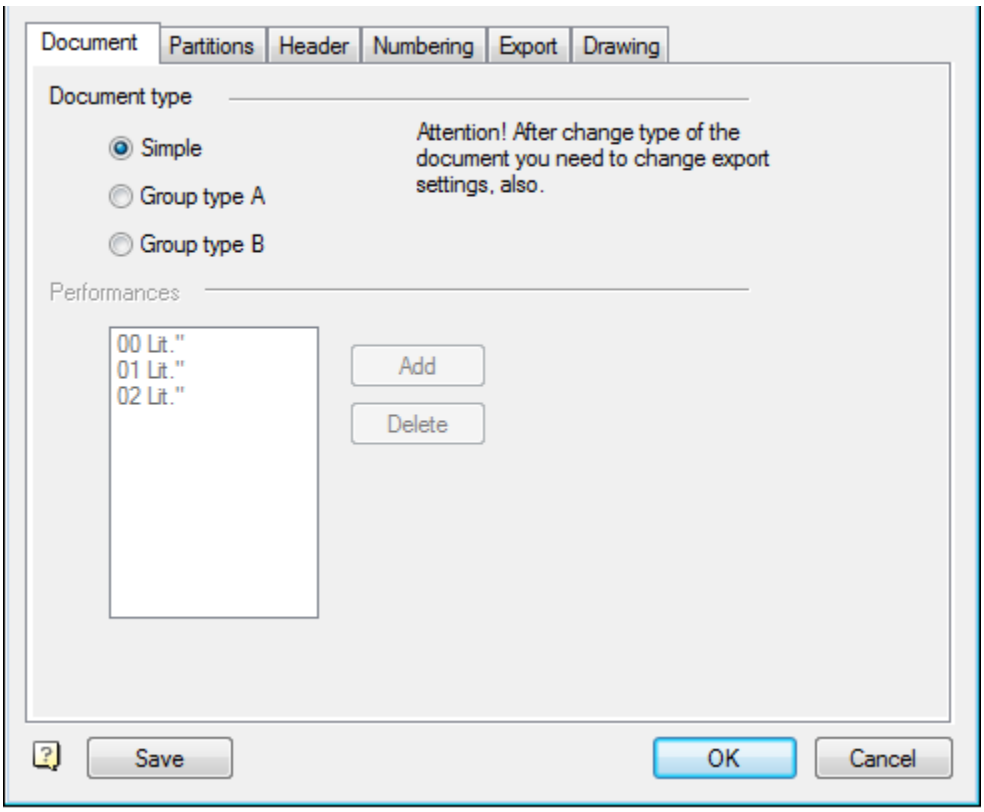
Update specification sheets in the drawing - Includes automatic update of tables exported to the drawing specifications. The update is performed on the export specification drawing. Accommodation in this specification is not requested again.
Specify the insertion point selection rectangle - when this option is enabled the position of each sheet exported to the drawing specifications not given an indication of the insertion point, but an indication of the rectangular boundary of the placing on the drawing sheet.

Export Settings on the drawing table designation include serving template specification for the first and subsequent sheets.

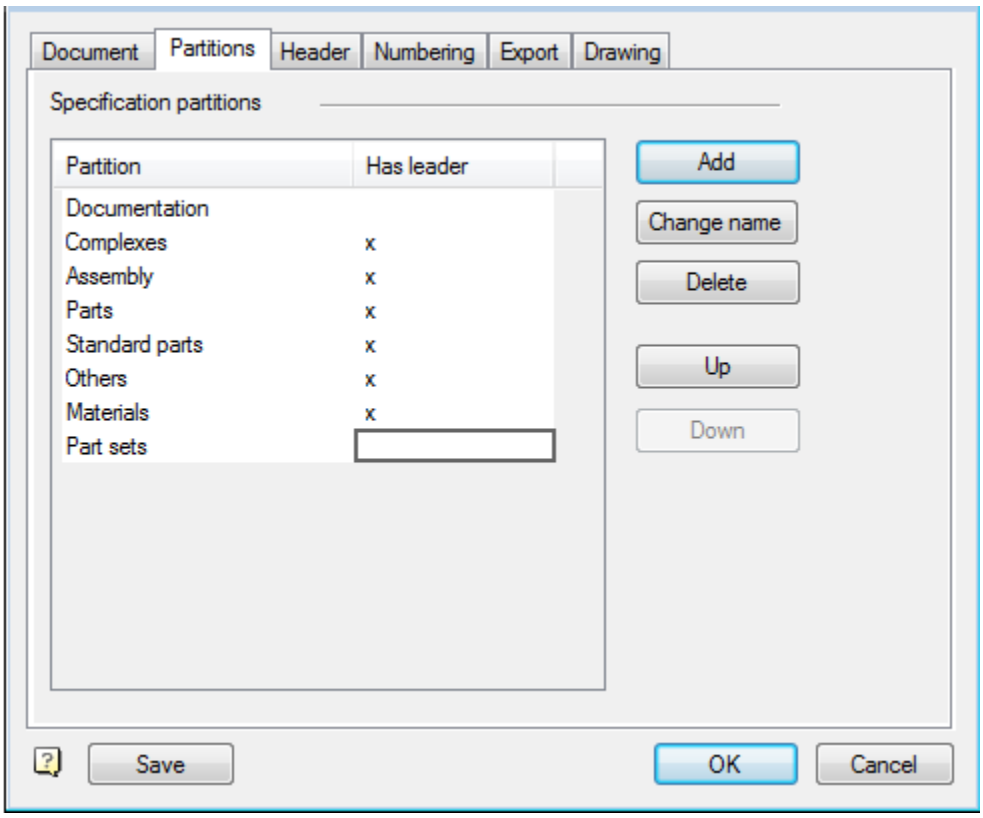
When you click on the table selection dialog opens.

Setting up a custom type specification

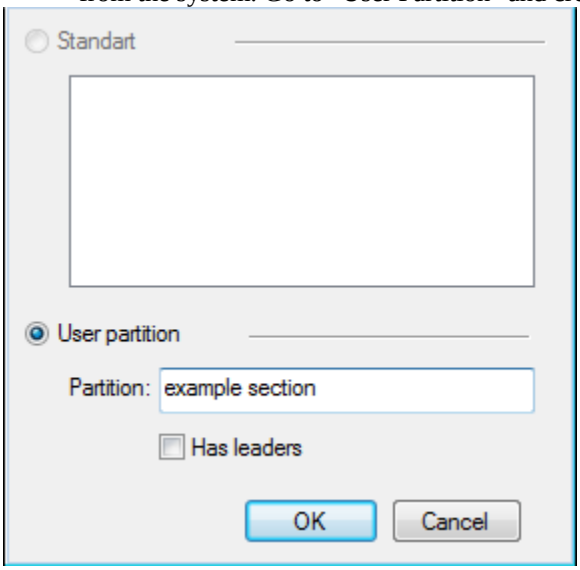
- Go to the drop-down menu *Specification type simple*.



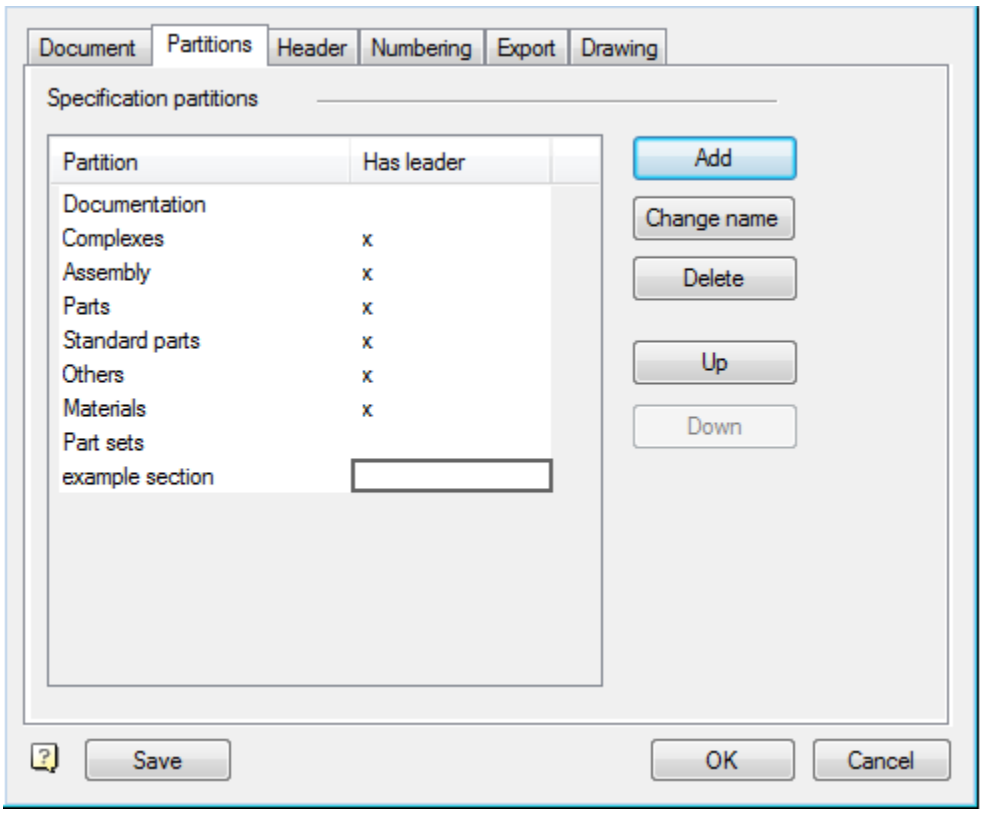
- Go to the tab "Partitions".



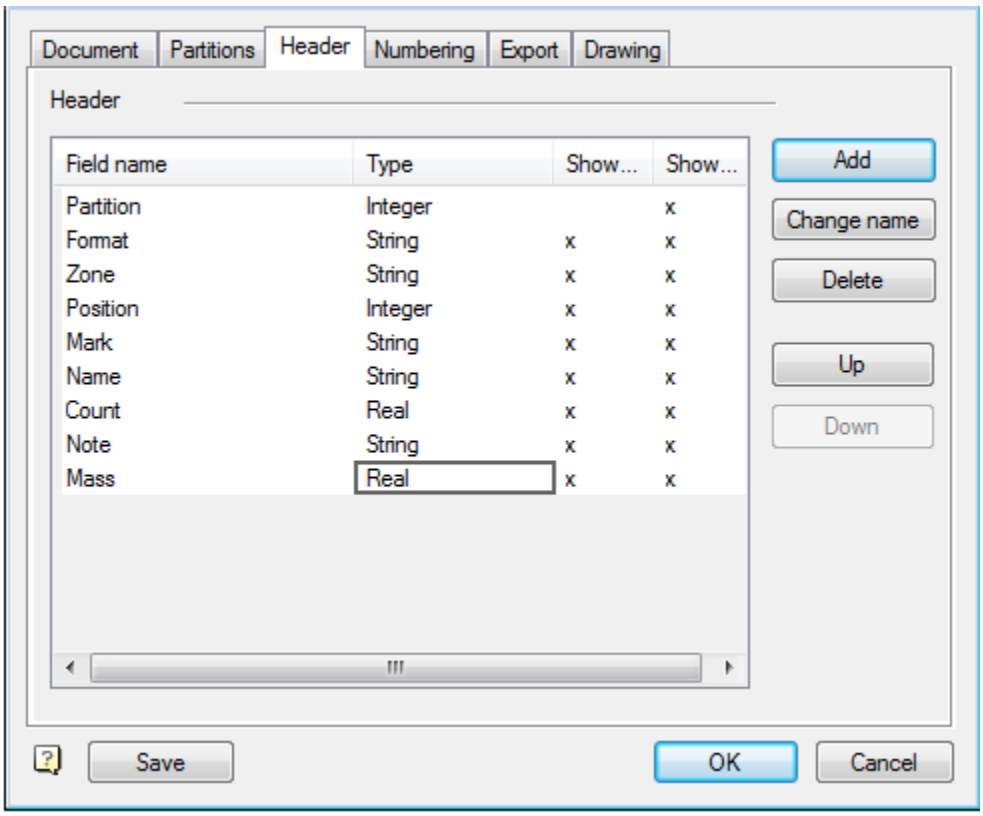
- Click on the *Delete* button and select the sections that you do not need or you do not use. In the sections of specifications provided by the State standard, The current setting, can be removed but removing them from the system is impossible. At any time, you can edit the style specification and return these sections.
- Click on *the Add* button. At the top of the dialog box, in the "Standard", posted sections that are removed from the system. Go to "User Partition" and create your profile.



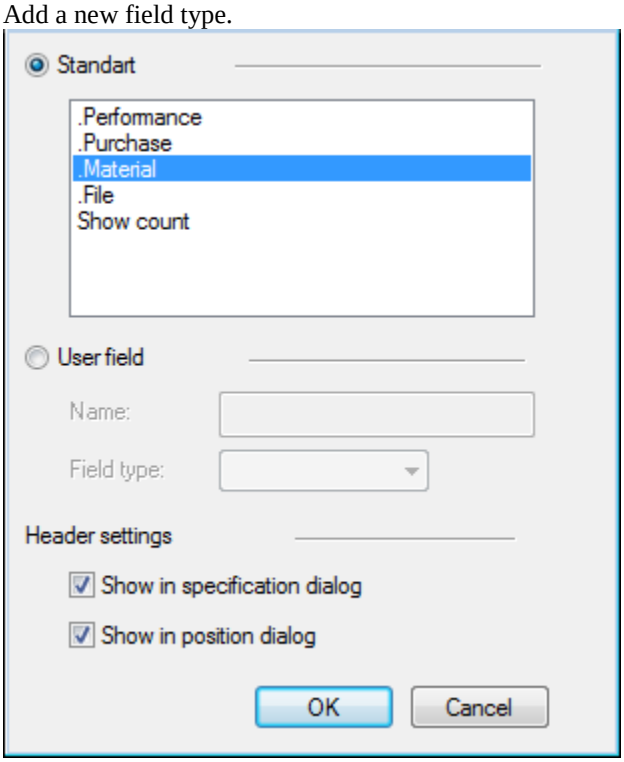
The result can be seen in the figure below.



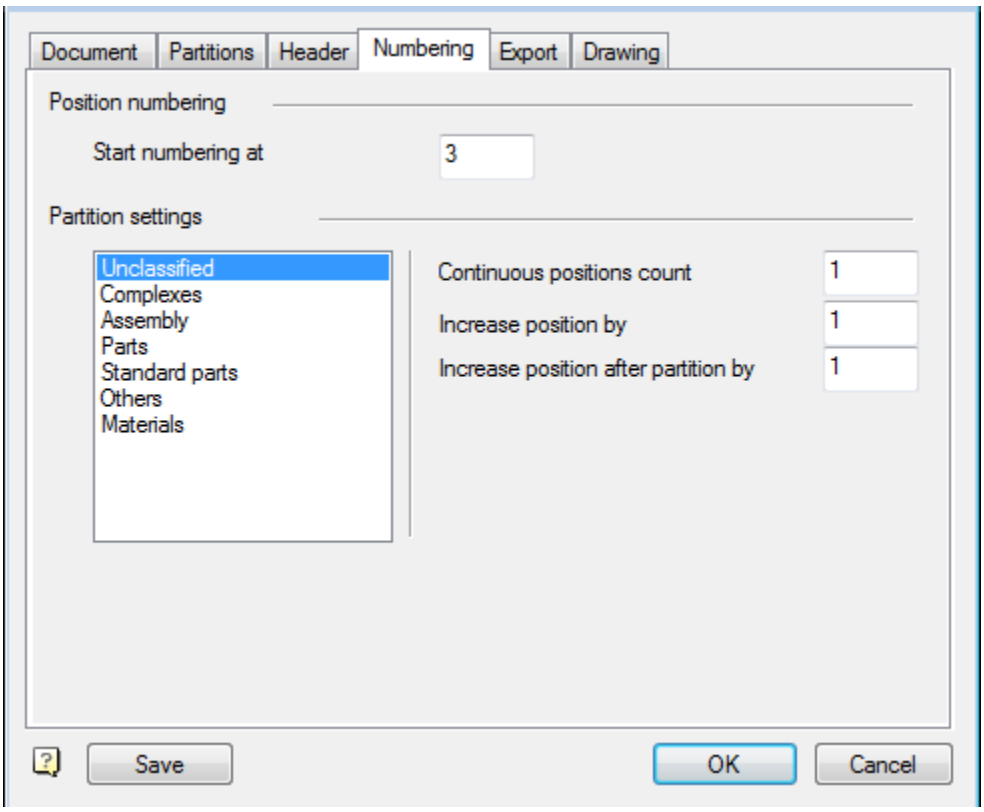
- Go to the tab "Header".



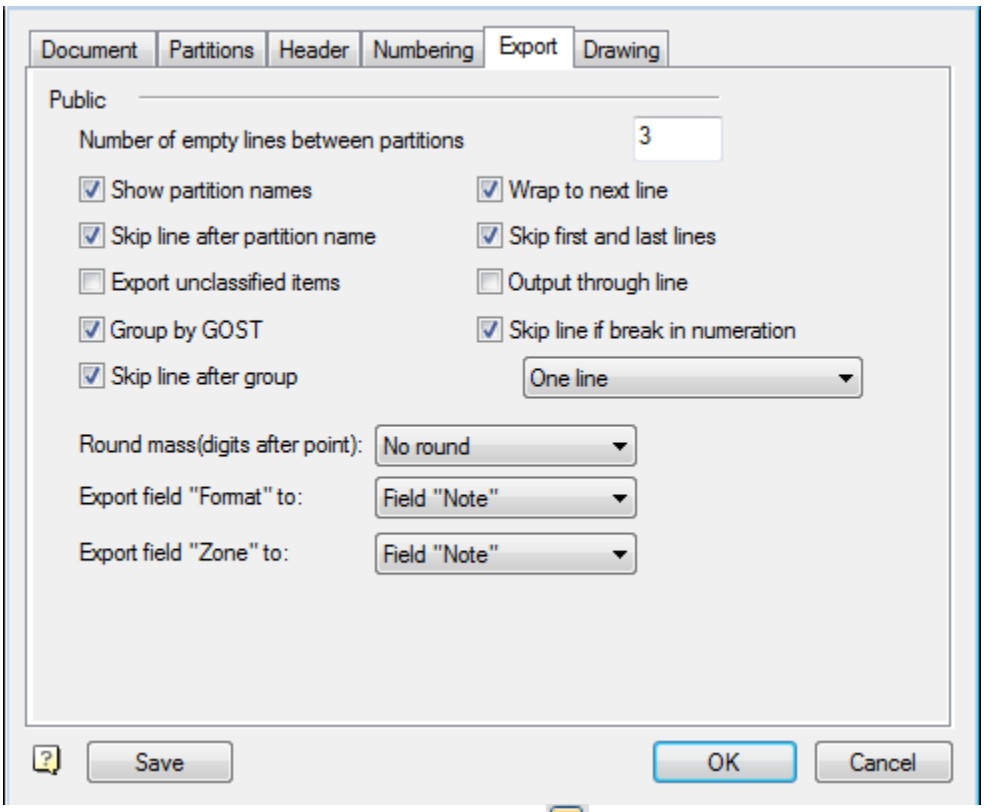
Add a new field type.



- On the "Numbering" it is possible to set a rule for the numbering and skip to the desired position.



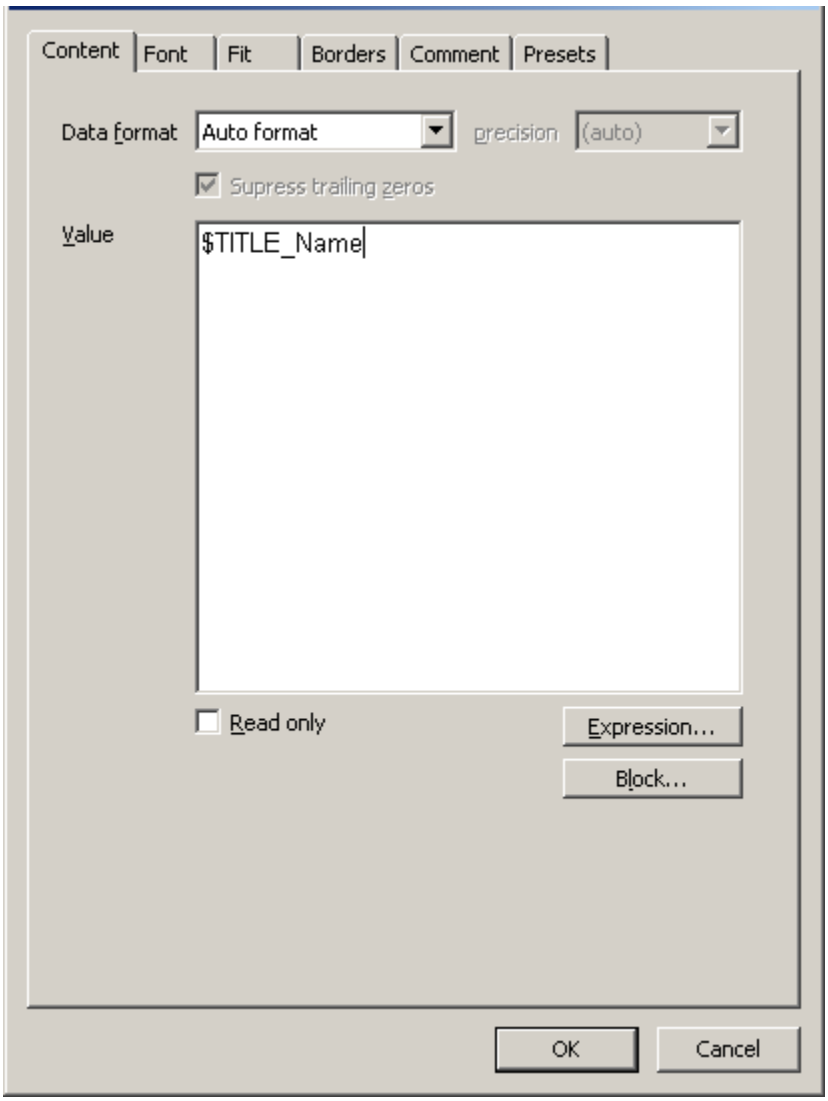
- Tab "Export" link provides specifications and user table that allows you to customize the specification of any complexity.



- Tab "drawing." On this tab, click the button . In the dialog box, choose from the database tables for the first and subsequent sheets.

Create a table for a custom specification

Open Table Editor and create a new table.
To create a custom specifications in the table must specify the header field specification and the field "title block".
Export records to be made in the fields below fields defined as "Title".
To set the BOM header must select a table cell and, from the context menu *"Properties"*. In the *"Value"* enter the header field specifications with the prefix ***$TITLE_***.
For example ***$TITLE_Name***.



To limit the filling column specification adjustment must be at the bottom line allowed to enter in the *"Value"* line ***$END***. Otherwise the filling will be made before the end of the table or subtable.
To transfer data from the main drawing caption (for example, the fields of the title block specification) must be in the *"Value"* enter the value with the prefix ***$SHTAMP_***. The cell named ***$SHTAMP_Name*** be entered value of the *"name"* of the title block of the drawing.
After registration table accordingly to save it in a database under the appropriate name for later use.

Balloons

Main menu: ***Mechanical - Part list - Balloons***.

Ribbon: ***Mechanical - Part list - Balloons***.

Toolbar: ***Part list - Balloons***.

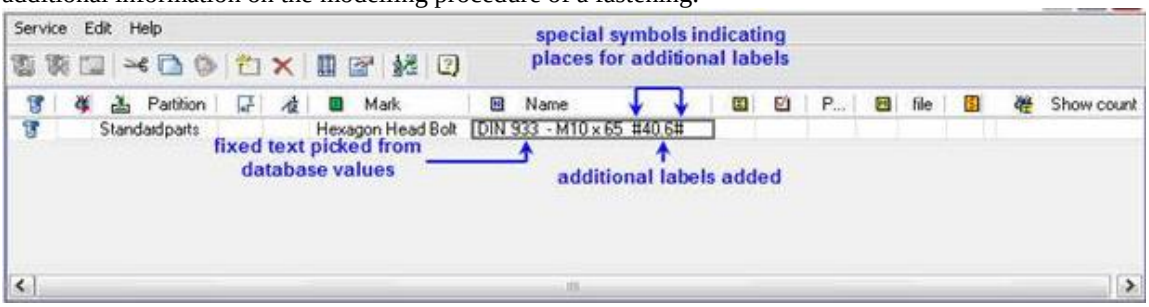
Command line: ***MCPOSITION***.

Fundamental rules

When you point to elements of attachment parts for nanoCAD Mechanica 20.1 recognizes the size of parts mounting connections without additional indication of its components.
When making a specification of the assembly drawing mechanism can be set dynamically related to the title block of assembly component external DWG-file.
All changes in the linked drawing die subassembly (rooms, mass node) will automatically be reflected in the general specification.

Notation of the position of the standard item

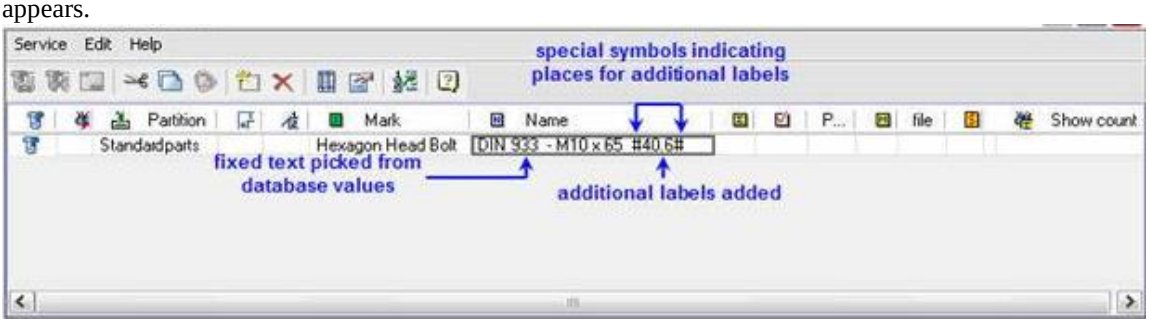
When selecting compounds for a fastening nanoCAD Mechanica 20.1 Auto detects the typical preset sizes of fastening elements with no additional input. The construction indicators-fit , degree of hardness and the cover for standard parts should be set individually, depending on the specific conditions of the assembly operation.
Additional lables(Precision , modelling procedure, thread tolerance , degree ogf hardness etc , must be entered among the truss symbols. In the ballon editor (or in the part list editor) , double click the entry field to enter additional information on the modelling procedure of a fastening.



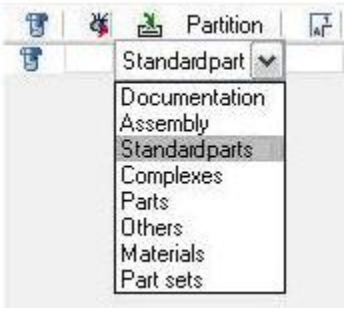
The information input on performance of standard details.

Creating a Balloon

Pick the start point and the second point , defining the position of the underscore , the position leader edit dialog appears.

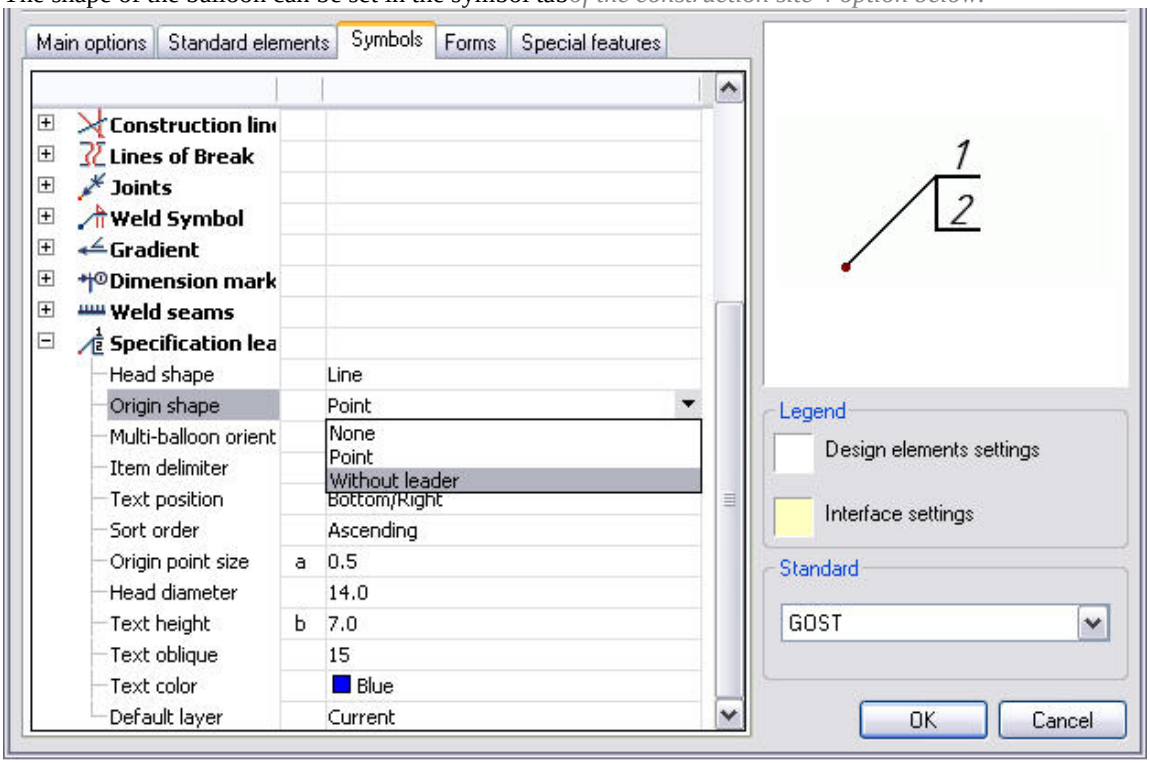


To group balloons according to the type, left click on the partition field.From the drop-down menu choose the desired option to which the part is referring.

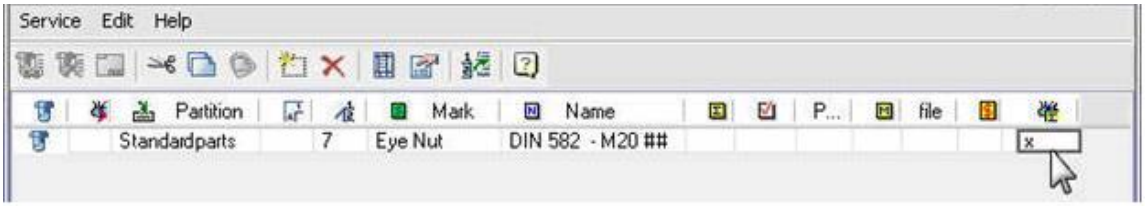


The mark, Name, note and mass fields can be set by the user.
If the item name is too long, you can enable the forced transfer it to the next line. To do this, add a control character \n.

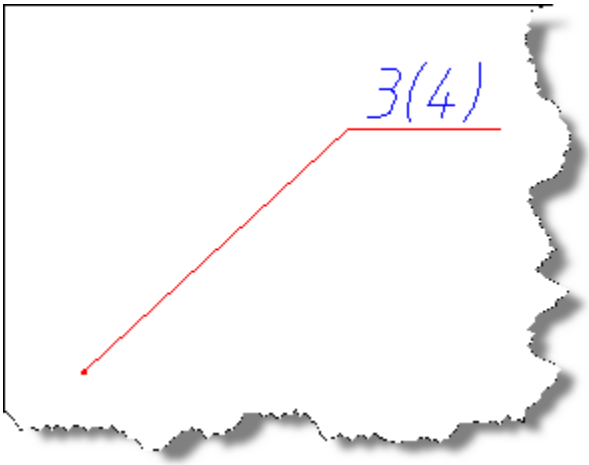
At the specification export, the posiotion name will be broken onto two lines.If the format is on the drawing then for the specification table the drawing zone is added automatically.
If a format is present on the drawing for positions on the specifications, the zone of the drawing is automatically added.
The shape of the balloon can be set in the symbol tab*of the construction site 4 option below.*



To display the quantity of positions connected with given leader, it is necessary to select the mark.



The total of the details of the given position will be displayed on the drawing next to the position number within brackets.

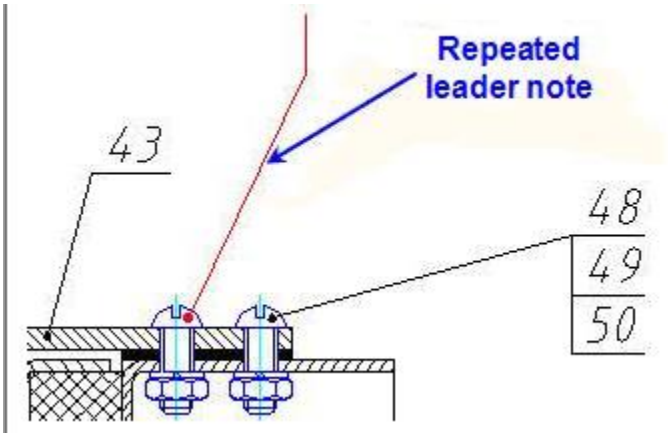


Going into the editor specifications

Checkboxes for the number of items, click on the icon Specification. This icon is located in the dialog ***editor specifications*** and in the main toolbar nanoCAD Mechanica 20.1.

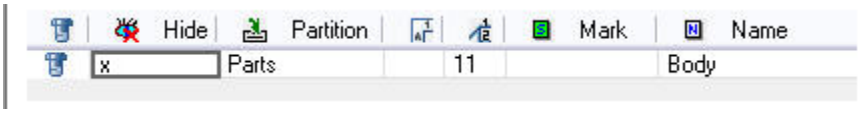
Blocked Ballon Insertion on the drawing

If the ballon leader is applied to the item that has already been entered in the part list , it is shown in the figure with a vertical extension line.



Repeated leader

If such repeated ballons are meant to be any reason , double click the red leader.
In the dialog window, double-click on each of x-symbols in the column used to block ballon insertions.



Close the window by clicking x symbol in the upper right corner the part numbers will appear in the drawing.

Creating performance and item positions , before inserting into the drawing

To create an additional position for an item which has already been identified on the drawing, use the *Add position/performance* option.

Click on this icon and the *Select position record* dialog will open. In it, the earlier insertion positions are displayed.

Note! In the *Select position record* dialog, only those positions that have been identified by number are displayed.

If the specification editor settings mean that a group view of the document is shown, the ***Select position record*** dialog contains two parts.

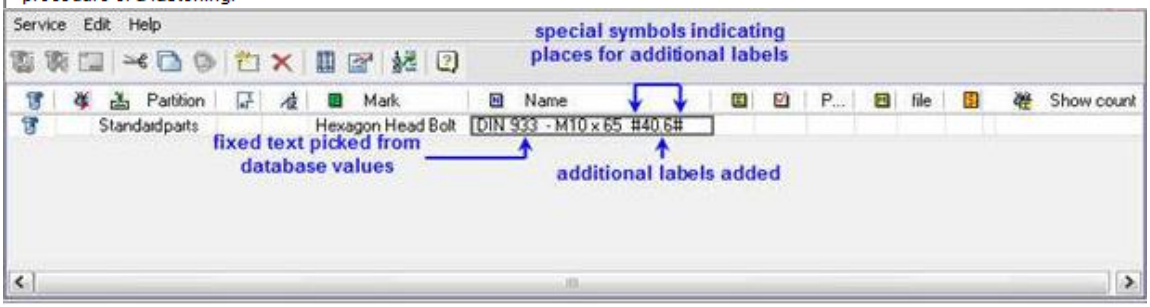
The top part of the window displays the list before the inserted positions, and the bottom part of the window - the list of position performances.

If a line is chosen in the top part of a window, the new performance of the selected line will be added in the position editor. If a line is chosen in the bottom part of a window, the selected performance will be added.

Adding Construction Indicators to Standard Parts

When selecting components for a fastening, WiseMechanical auto-detects the typical preset sizes of fastening elements with no additional input. The construction indicators - fit, degree of hardness and the cover for standard parts should be set individually, depending on the specific conditions of the assembly operation.

Additional labels (precision, modelling procedure, thread tolerance, degree of hardness etc.) must be entered among the truss symbols. In the balloon editor (or in the part list editor), double-click the entry field to enter additional information on the modelling procedure of a fastening.



The information input on performance of standard details

Creating performance and item positions , before inserting into the drawing

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Note! In the *Select position record* dialog, only those positions that have been identified by number are displayed.

If the specification editor settings mean that a group view of the document is shown, the ***Select position record*** dialog contains two parts.

The top part of the window displays the list before the inserted positions, and the bottom part of the window - the list of position performances.

If a line is chosen in the top part of a window, the new performance of the selected line will be added in the position editor. If a line is chosen in the bottom part of a window, the selected performance will be added.

Position Align

Main menu: ***Mechanical - Part list - Position align***.

Ribbon: ***Mechanical - Part list - Position align***.

Toolbar: ***Position align (*** "Part list"***)***.

Command line: ***MCPOSALIGN***.
Command allows you to align the numbers callouts specification. To do this, you must first select which items will be subject to equalization, then - in the context menu to specify alignment options.

Following options are available:

- Vertically.
- Horizontally.
- Line - an indication of the line along which the rooms are arranged
- Rectangle - specifies a rectangular area around which aligned numerals.

Recognition specifications

Command line: ***MCRECSPEC***.
Master recognition specification enables automatic search and replace specifications created from individual primitives nanoCAD, transforming them into an object "Specification" nanoCAD Mechanica20.1. Initial primitives are in the drawing.

Drawing design

Drawing borders

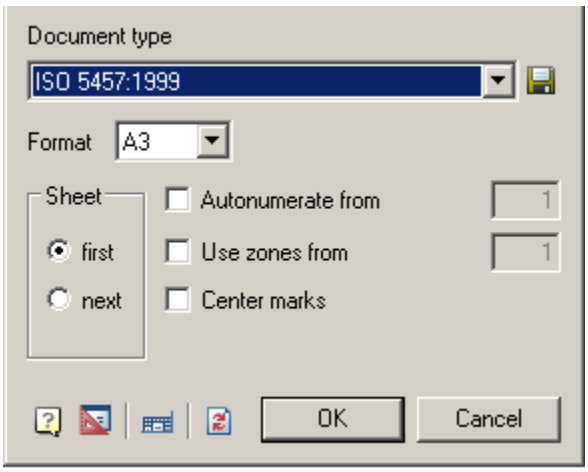
Working with standard formats

Main menu: Mechanica - Drawing title/borders - Drawing title/borders.
Toolbar: Drawing title/borders ("MCS Drawing title/borders").
Command line: MCFORMAT, FORMAT.

- Filling in the fields of the title block
- Drawing title/borders recognition

Fundamental rules

Each format type is an element of the database nanoCAD Mechanica20.1, which is located in the folder "Drawing border templates" library nanoCAD Mechanica20.1.
emplates major labels and additional fields are located in the subfolder formats "Title block templates". Renaming or deleting existing items in the folder "Drawing border templates" would make it impossible to insert existing formats.
Insert the format by clicking on the icon Drawing title/borders. This dialog box appears *Format*:



This window is defined by:

- Format.
- Sheet.
- Autonumerate from.
- Use zones from.
- Center marks.

After selecting the type of document, as well as the size and orientation of the sheet is inserted into the selected format.

- To edit the frame size, double-click the left mouse button on the contour of the frame.
- To edit the text in the title block, double-click on the text of the left mouse button.
- To edit entries in the title block, double-click the left mouse button on any line of the title block.

If you remove the first sheet of the scope of its information from the title block is automatically transferred to the second sheet.

To save the settings template attached stamps and their parameters in the database, click on the button , then enter the path stored in the database and the template name.

General Principles

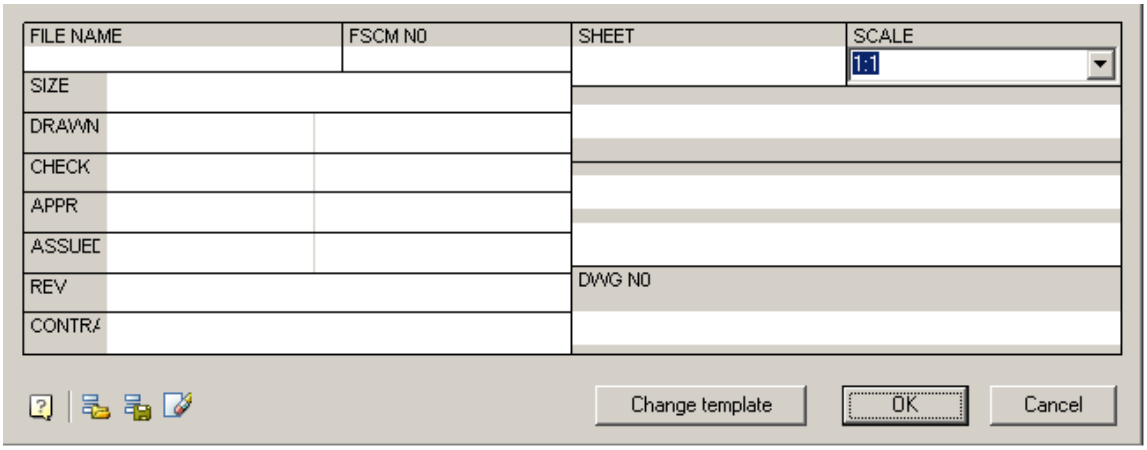
To change the line frames the template files, edit the frames and other fields.
"Document Type" - Select the document type from the list format template in section database. Depending on the document type is changed appearance and availability of individual dialog items "Format".
"Format" - the choice of the size of the sheet on the basis of standard size and multiplicity (not available for the types of documents based on the standard ISO, because the multiplicity inherent in the choice of format).
"Leaf" - the choice of template title block for the first or subsequent sheets.
"Orientation" - select page orientation (vertical or horizontal).
"Autonumerate from" - Automatic Numbering sheets. It lets you specify the number of the first sheet.
"Use zones from" - avtoprostanovka zones. It lets you specify the number of the first zone.

Edit main stamp

After you select the document type and size under the proposed set the insertion point of the frame. If you press Enter or right mouse button, the lower left corner of the frame is aligned with the coordinate 0,0.

Set default stamps - restores the template format those stamps which were overwritten while maintaining format template database.

Filling in the fields of the title block



To fill the title block fields and additional graph, double-click on any line of the title block.
The main inscription is filled directly into the dialog box, "Stamp"
While filling in the title block previously entered information stored in the section "History" menu. To open the context menu, place the cursor in an editable field and click the right mouse button.
Recent values can be saved in the section "Recent" menu.

Automatically fill fields of the title block

Save as default
The command saves the current filling of the stamp in the memory. In the following fields will be filled automatically stamp

Load defaults
The command loads the stored filling.

Clear
The team clears stamp field

Change template *Change template*
The command opens a drop-down menu to change the pattern of the stamp.

Drawing title/borders recognition

Main menu: Mechanica - Recognition - Drawing title/borders recognition.
Toolbar: Drawing title/borders recognition ("MCSRecognition").
Command line: MCRECF.
Master recognition format enables automatic search and replace formats created from individual primitives nanoCAD, transforming them into an object of the "Format" nanoCAD Mechanica20.1. Initial primitives are in the drawing.
Recognition takes place in two phases: the first searches for the formats in their dimensions, the second selected title block. Selection is made by comparing the serial patterns of stamps from the library with the existing geometric objects in the drawing. The remaining seals are installed by default. After the successful recognition format graphs stamps are automatically filled with values from the drawing, and the format is placed on the current layer.

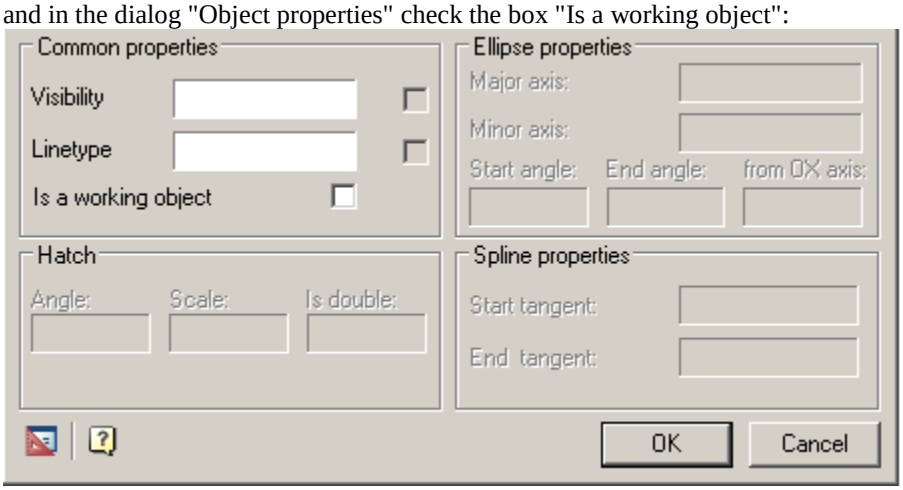
Creating custom formats and stamps

Before you create a custom size or stamp is necessary to draw a pattern nanoCAD means in accordance with the required desired view object being created. When drawing templates should adhere to the following rules:

- The template can contain lines, arcs, and circles, text (single-line and multi-line), the objects of the library of standard parts nanoCAD Mechanica20.1, blocks nanoCAD;
- the thickness of the lines that are assigned to the template thickness is not equal to zero, and not "layer" to be adopted in accordance with the thickness of the closest standard value;
- the color lines and text included in the template will be saved according to the selected values;
- multiline text (mtext) is inscribed in the boundaries of the rectangular area of text is selected when you create it.

You can use overhead lines and graphic objects, which are not drawn when inserting format or stamp in the drawing.

To install a drawing object as a service, use the button *"Set parameter"* toolbar MechWizard, select the object, and in the dialog "Object properties" check the box "Is a working object":



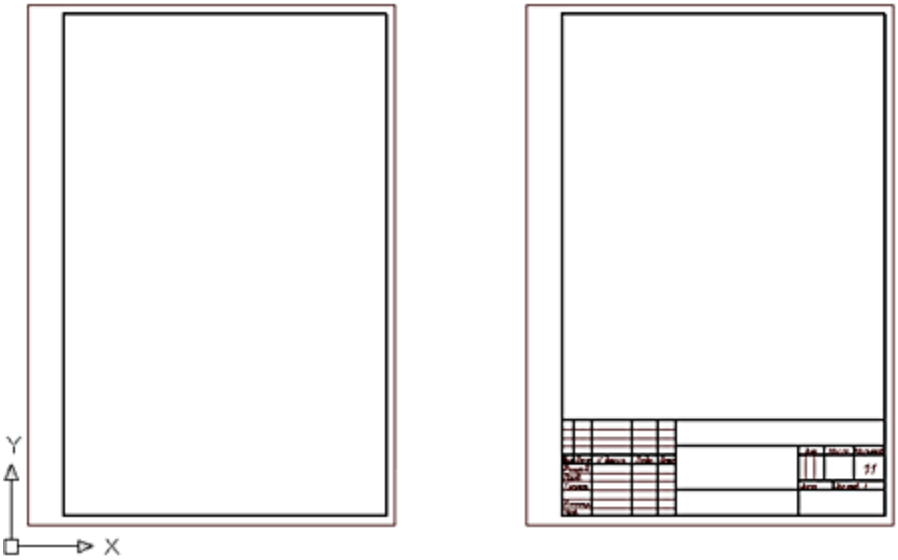
- Use the links in the templates
- New border wizard
- New title wizard

Use the links in the templates

Stamps in the format template

Custom format may contain links to templates stamps available in the library. To add a stamp in the format used by the relevant rules.

Stamp Position and orientation when you insert a drawing format will be determined by the insertion point and orientation of text links:



the figure shows the format template added referring to stamp basic inscription. When you insert a drawing format drawn stamp recorded in the library under the specified name.
the format can be added to any number of references to the stamps from the library.

Using text fields

In the pattern of the stamp may be used special signage text fields.
Text added to the template and starting with the single character "$", is recognized as a reference pointer on the field (the Count) input. In the input field of the dialog box, you can edit the stamp to record the text (for example, the name of the developer or check). Name field can contain characters of the Latin alphabet and the national or spaces.
Dimensions of the input fields are defined primitives surrounding the insertion point, the reference index. If you found another symbol "$", the text after it considered the description field:
$Field_name [$field definitions].
the contents of the text box will be displayed in the list of object properties.
To limit the size of the input field or eliminate overlapping fields in the dialog, you can use the service line.
To create a stamp based on an existing base in the sample, the sample to be inserted into the drawing by setting in the dialog box "Stamp" mode "Display field names" . After insertion into the drawing die can be used as the layout of the new stamp.
Standard fields stamps have a description in Russian (GOST), English (ISO) or German (DIN) language, the values of the fields available in the list of object properties.

Name	Description	Contact with the properties of the document Inventor	Note:
$Designation	Description	+	
$DrawingName	Name	+	
$Drawing type	Drawing View		
$Material	Material	+	
$Litera 1	Letter 1		
$Litera 2	Letter 2		

Name	Description	Contact with the properties of the document Inventor	Note:
$Litera 3	Letter 3		
$Mass	Weight		
$DimScale	Scale	+	Sets the size of generated within the format
$Sheet	Sheet	+	Changes in the automatic numbering
$SheetCount	Sheets	+	Changes in the automatic numbering
$Add.text	Position the additional inspection		
$Author	Developed	+	
$Control	checked	+	
$TechControl	Technical control	+	
$Add.subject	Last additionally checked	+	
$NormControl	the Regulatory Control	+	
$Approve	Approved	+	
$Author (sign)	Developer Signature		
$Control (sign)	the signature checking		
$TechControl (sign)	Signature of the technical control		
$Add.subject (sign)	Signature additional inspection		
$NormControl (sign)	Text regulatory control		
$Approve (sign)	the signature approving		
$Author (date)	the date of	+	
$Control (date)	Date of Inspection	+	
$TechControl (date)	Date engineering controls	+	
$Add.subject (date)	Date further verification	+	
$NormControl (date)	Date of regulatory control	+	
$Approve (date)	Approval Date	+	

Name	Description	Contact with the properties of the document Inventor	Note:
$Original #	Part Number script		
$Sign & date 1	Signed and dated 1		
$Instead #	Instead, inventory number		
$Duplicate #	Inventory number duplicate		
$Sign $date 2	Signed and dated 2		
$Reference #	the reference number	+	
$First use	the primary use	+	
$Format	Format	+	
$Enterprise	Enterprise	+	

In the pattern of the stamp can be used table nanoCAD Mechanica20.1. Included in the stamp table when you insert a drawing can be edited as regular tables nanoCAD Mechanica20.1 (except that the die table can not be modified in the editing mode screen).

Additional features

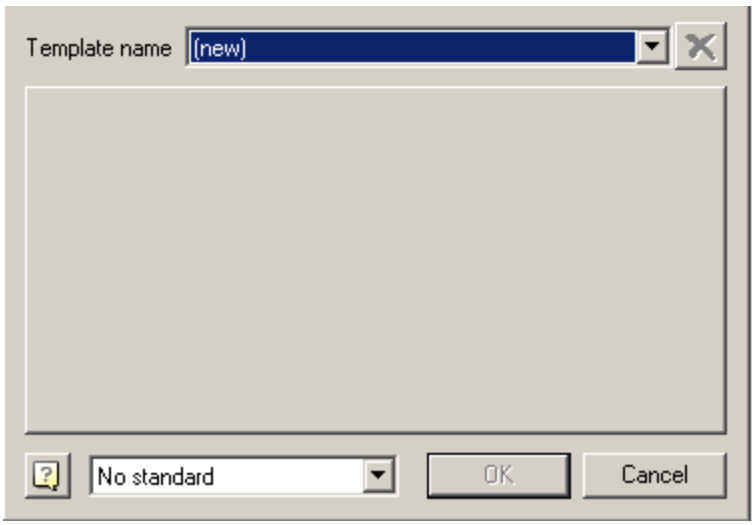
Converting from previous versions - In file %ProgramData%\Nanosoft\nanoCAD Mechanica20.1\DataRO\ru-RU\settings.xml defined fields for name mapping formats of old and new versions. If an entity applies the custom stamps of previous versions, the names of the fields, you can add them into the appropriate section of this file.

Integration with other objects nanoCAD Mechanica20.1

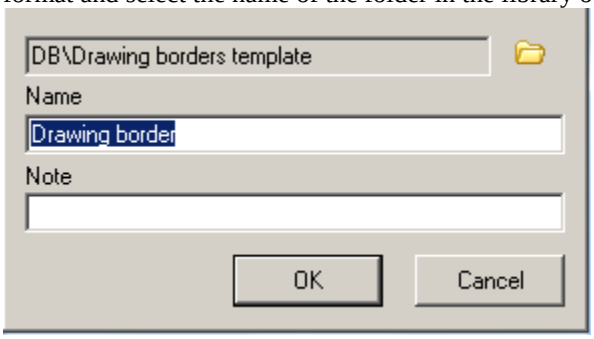
Tables - Format allows you to use tables as stamps. This preserves virtually all the functionality built-in table.
Universal marker - The values of any fields format can be used in a universal marker.
Dimensions - When you create within the format size, the size takes out set in the title block of the format.
Legend roughness - The format may be associated with the designation of a roughness surfaces unspecified. Icon roughness is automatically transferred to the upper right corner of the format.
Property document - The value of the standard field format is synchronized with the attributes of the file in which it was created.
Technical conditions - Format calculates the coordinates of the columns to accommodate the technical requirements.
Attribute documents TechnologiCS - Exchange values between the fields and the format of the document attributes. You can set the matching fields automatically (by the same name), and then specify manually. Establishing appropriate stored for later use.
Zone format - The format can be calculated, is there any point in it. If so, you can receive the designation area containing the point.

New border wizard

Main menu: Mechanica - Drawing Title/Borders - New border wizard.
Toolbar: New border wizard ("MCSDrawing Title/Borders").
Command line: MCFORMATWIZ.



To change the existing format, select it from the drop-down list of templates. The field, located in the center of the panel will be a general view of the selected format.
To add a new format, select the drop down list "Template name" paragraph (new).
Click the left mouse button on the center field, select the drawing created by pre-layout format template, and confirm the selection click OK.
In the dialog box "Format creation" will be recognized image format.
If you create a new format after closing the dialog box by pressing OK, a dialog "Create element", which follows the format and select the name of the folder in the library of standard parts, which will save the created format:



By default the path "DB \ Drawing borders template". If you want to keep the format in another folder, press and select the folder that appears in your browser.
Creating a template format is similar to creating templates stamps. Embedded tables are not supported. The texts, starting with the single "#" character, determine the type and position of the stamp format. When you insert the size of the database will be loaded stamped with the name specified in the text (the symbol '#' is ignored and the text "main message" will be replaced with the box on a stamp with the name of "Title Block"), if it is in base. If the name starts with "$" and contains only hexadecimal digits, it is interpreted as UID template stamp in the database. Coordinates, direction and alignment of the text will be applied to the stamp.

Note:	See stamp UID by inserting the drawing title block of the database manager nanoCAD Mechanica20.1.At the command prompt displays the hexadecimal object UID

Predefined templates are stored in the formats "DB \ Drawing borders template". It is recommended that the new template is stored in the same folder, but when loading the template can be seen the whole base.
Frame custom format has a fixed size - the one set in the template. There is a special recognition mode that allows the use of standard scalable framework with their stamps. To enable this mode, select from the list in the dialog master the standard by which the frame will be built. In this mode, all of the graphics in the template are ignored,

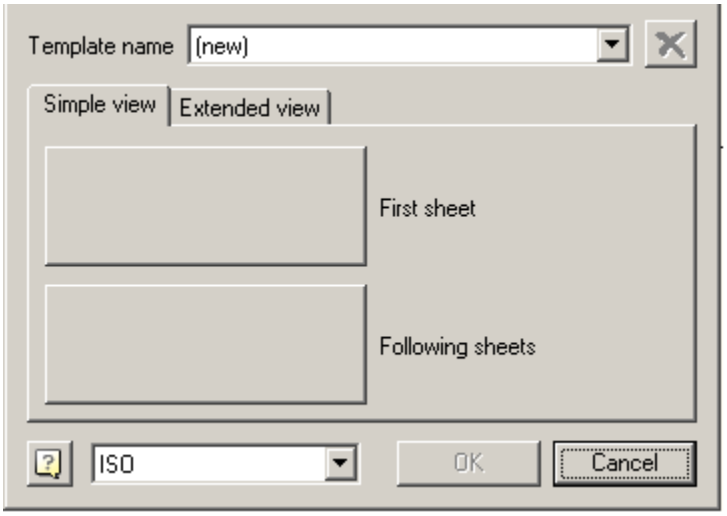
except for text and bounding box. The frame can be of any size - in the final format it replaced with standard frame. All texts are considered names stamps. There are two variants of binding stamps:

- If the text begins with the "#" symbol, it is stored offset from the insertion point to the nearest corner of the bounding box.
- If the symbol "#" no stored offset from the text insertion point to the bottom left corner of the bounding box.

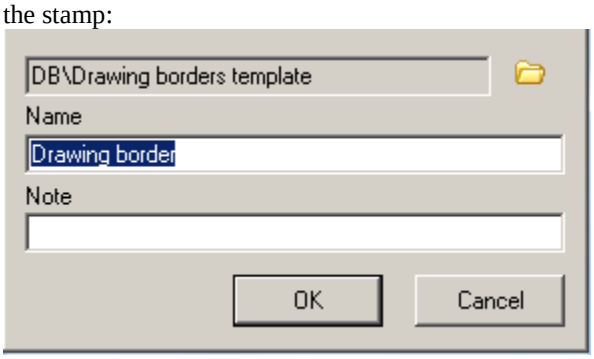
Note:	Standard sizes have patterns in the database - their patterns are embedded in the program code.

New title wizard

Main menu: Mechanica - Drawing Title/Borders- New title wizard.
Toolbar: New title wizard ("MCSDrawing Title/Borders").
Command line: MCSTAMPWIZ.



To change an existing stamp, select it from the drop-down list of templates.
To add a new format, select the drop down list "Template name (new)".
A template stamp can have up to four different graphical representations: simple and full (advanced) type, including the form of the first sheet and the view for subsequent sheets.
Select one of the tabs: "Simple view" or "Extended view". Click the left mouse button on one of the fields marked with a stamp type, and select the drawing template you created earlier mock punch. Confirm the selection click OK.
In the dialog box "Create stamp" will be recognized image of the stamp.
If you create a new stamp, after closing the dialog box by pressing OK, a dialog "Create element", where you must enter the name of the stamp, and choose a library of standard parts folder in which to save the template created by the stamp:



New templates stamps recommend that you save to the folder "DB \ Drawing borders template \ Title block templates".
If the layout for the die table to use, it will retain its editing capabilities even being introduced to the format. For these tables are only supported editing in place.
The pattern die may be up to four types:

- the main type of the first sheet,
- he main type of follow-up sheets,
- an expanded view of the first sheet,
- an expanded view of the next sheet.

The texts, starting with the single character "$" are recognized as the names of the fields. The editing dialog stamp they will be displayed in the form input fields. Name field can contain spaces and extended characters. Dimensions of the field defined primitives surrounding the point of its insertion. If you found another symbol "$", the text after it considered the description field. This text will be displayed in the list of properties:
$<name_field>[$<description_field>]
To limit the size of the field or eliminate overlapping fields in the dialog, you can use the service line. This is a primitive with zero thickness or line marked as the utility from mcWizParam. Set the parameter.
If you set daw "Work object", then stamped on tobrazhatsya these primitives will not.
To create a stamp based on an existing base in the sample, the sample to be inserted into the drawing, and in his dialogue press "Show Field Names". View selected stamp button "View".
After that, the object can be divided (explode), edited and used as the layout.
The dialogue is a choice of the standard, which is taken from a stamp. This standard is used for the selection of the frame formats in recognition of the primitives.

Tables

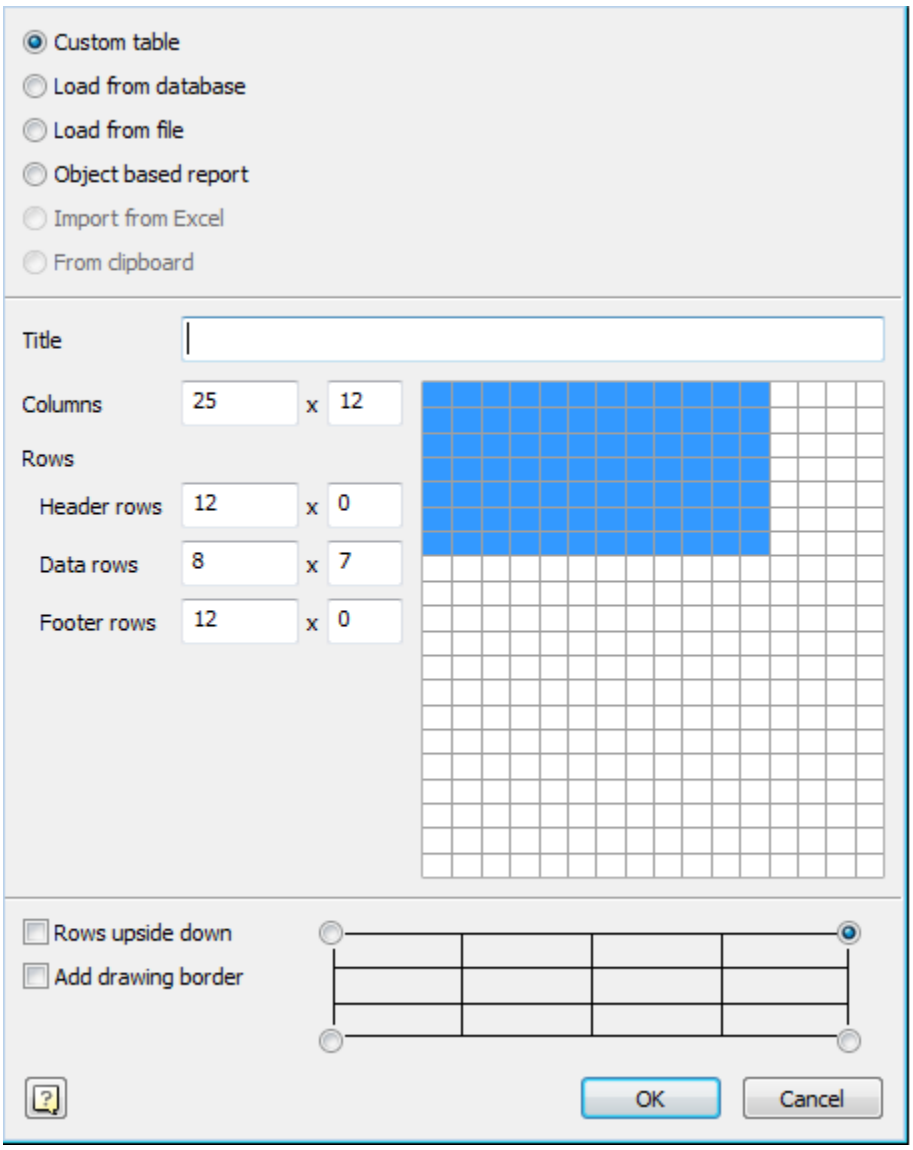
Tables

Main menu: ***Mechanical - Tables -*** ***Tables***.

Ribbon: ***Mechanical - Symbols -*** ***Tables***.

Toolbar: ***Tables (on toolbar*** "Tables"***)***.

Command line: ***MCTABLE***.
Causes dialog box to insert a new table.



- To create a custom table, select *Custom*. In input fields specify the dimensions of the created table, or on the layout, drag the mouse to the right as many rows and cells as required. When subsequent editing the number of rows, columns, cell sizes can be changed.
- To insert a standard table, select *load from database*. You can insert a standard table of the base elements Building site . In the dialog box, select the table you want. Database elements Building site is available for all major types of standard tables.
- To insert a table from a file, select *Load from file*.

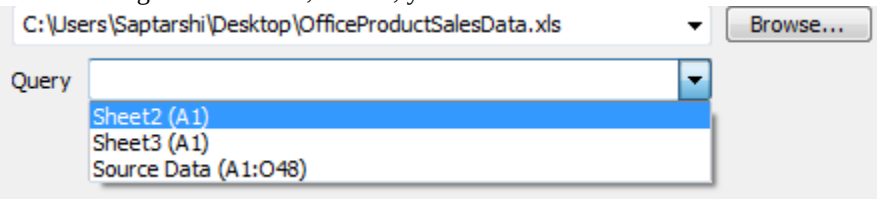
Click *Browse* and select the file table.

Supported formats:

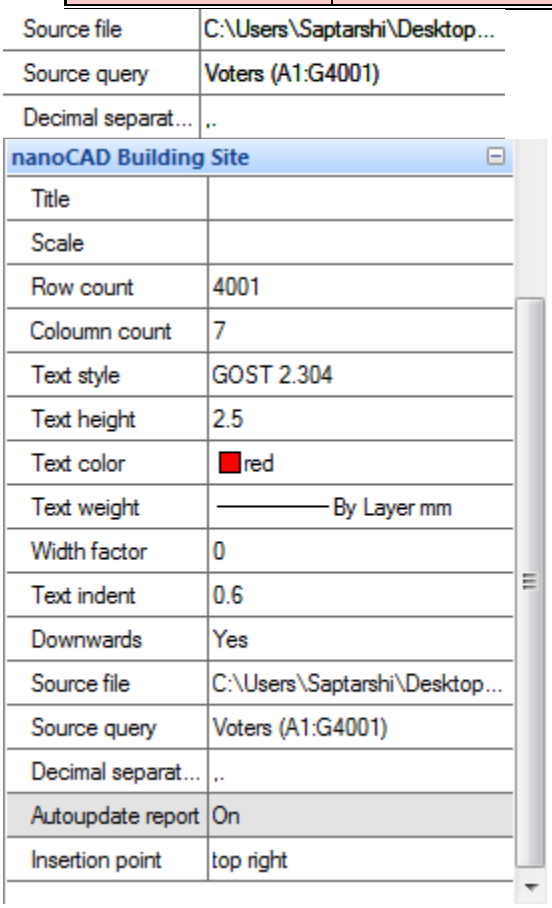
- tbl - tables files
- dat - data file or a text file
- mdb - database Microsoft Access
- xls - Spreadsheet Microsoft Office Excel
- xlsx - spreadsheet Microsoft Office Excel 2007

- csv - table cell separated by commas
- txt - standard text file
- xml - XML document
- odf - OpenDocument Format.

When loading a table of xlsx, xls file, you must select the Excel document sheet.



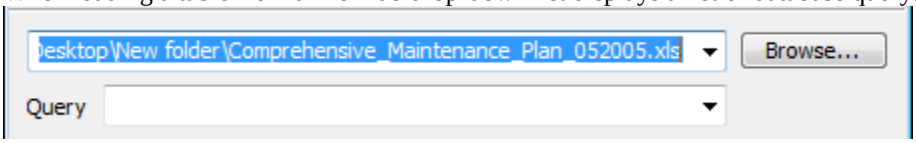
Important!	Select the sheet can be made after a table is loaded from the file.



In the properties of a table row *source file* displays the path to the source file of the table.

Select line *Request* to enter the desired page of the document. Then in the Table Editor, click *Update Table from an external source*.

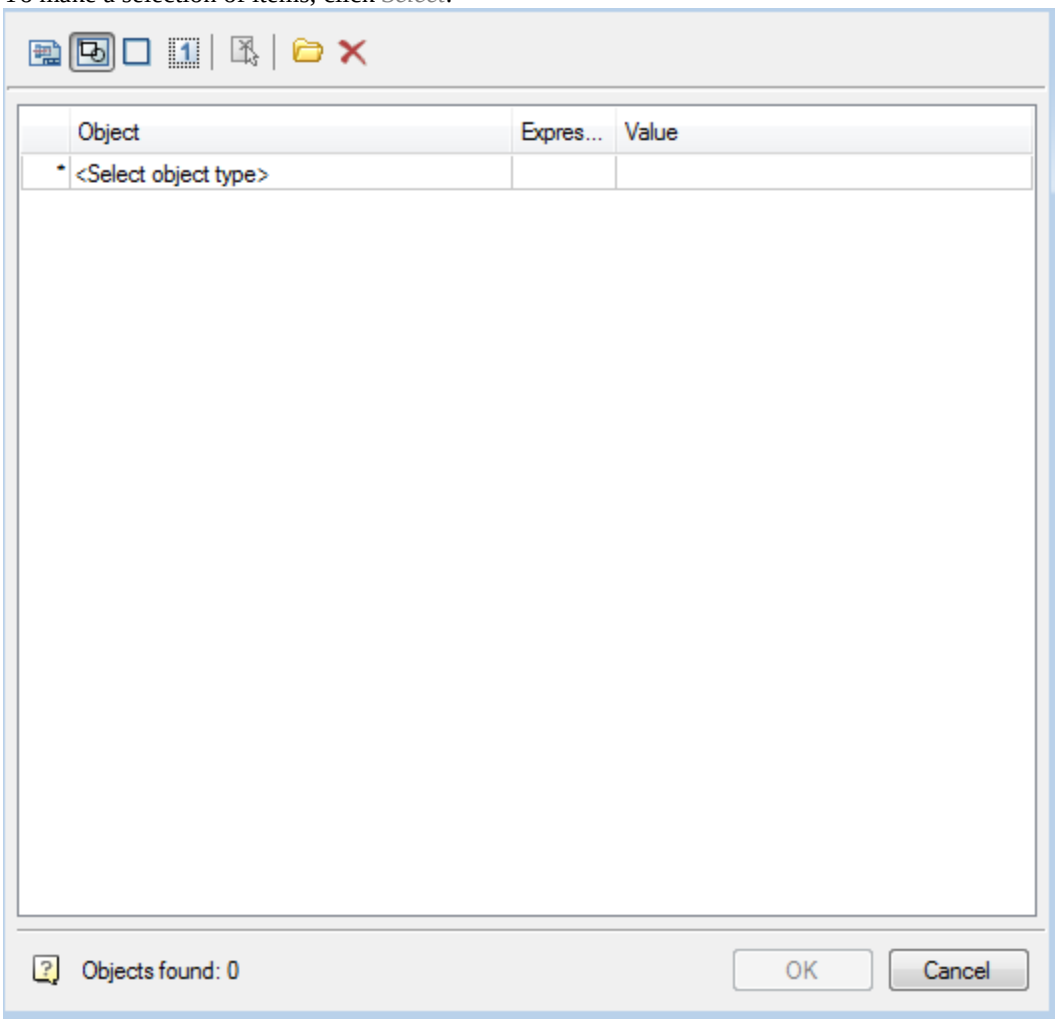
When loading a table from a file mdb drop-down list displays a list of database query.



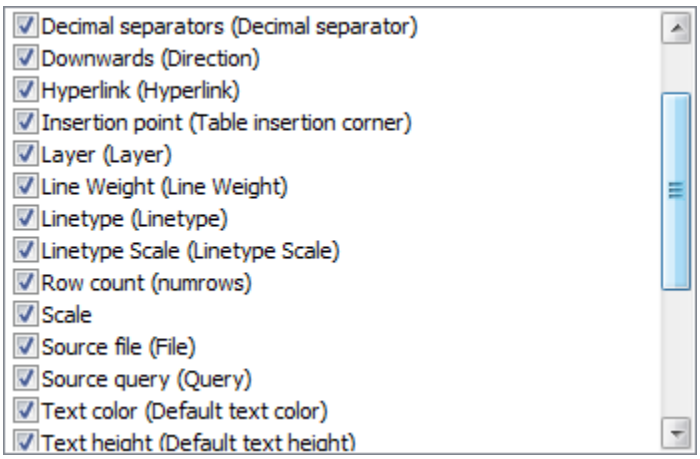
Note:	To import tables from files, you can drag a file from Explorer to the dialog table . When you drag a file from Windows Explorer in the imported table is added to the existing table. When you drag a file from Windows Explorer dialog box replaces the existing table to be imported.

- To generate a report based on a sample of objects, select *the objects in the sample report*.
-

To make a selection of items, click *Select*.



The search conditions are set in the *Quick selection* dialog.
Further dialogue is given based on what the attributes are and creates a report on the selected objects.



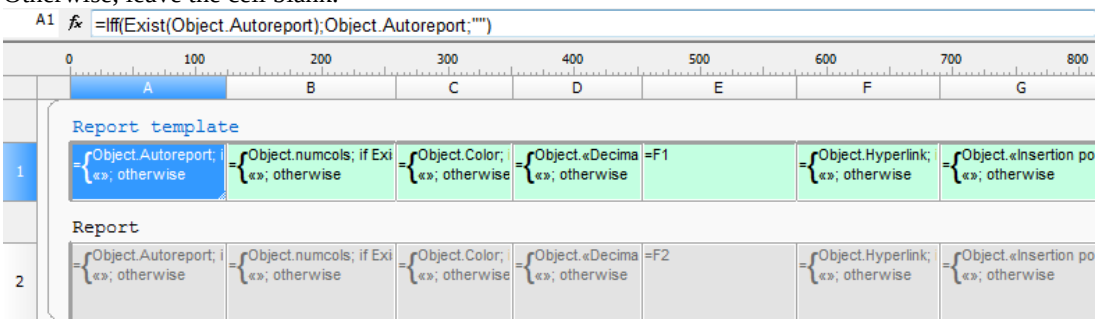
Includ attributes for the report in the list by selecting flags

- ***All***- selects all attributes
- ***None*** - the choice is removed from all the attributes
- ***Invert*** - inverted selection attributes

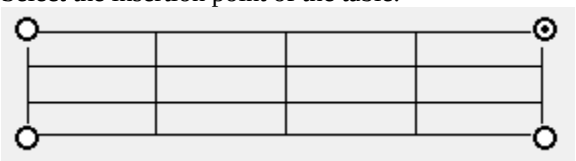
The table is created with a template report form

=Iff(Exist(Object."Autoreport");Object."Autoreport";"")

This formula checks whether there is a line for the attribute named *Autoreport,* and displays its value in a cell. Otherwise, leave the cell blank.



The number of columns in the table determined by the number of selected attributes.

- To import a table from Excel, select *Import from Excel*. To activate this item, the document should be opened.
- Select the insertion point of the table.
- 
- Select the numbering of rows in the table.
-

- If the switch *rows upside down* is enabled, the line numbered in reverse order.

- When the checkbox

after you insert the table will start a dialogue format.
- Click OK and the insertion point on the drawing table.

Note:	If the drawing pre-select a group of objects, and then run the ***tables***, you will be prompted to create a report based on a sample of objects.

Import, export from Excel

Import

Main menu: ***Mechanical - Tables - Import table from Excel***.

Ribbon: ***Mechanical - Symbols - Import table from Excel***.

Toolbar: ***Import table from Excel (on toolbar*** "Tables"***)***.

Command line: ***MCTABLEEXCELIMPORT***.

Transmits data from an open book MS Excel.

- In an open Excel worksheet, select the cells whose data you want to import into a table nanoCAD Mechanica 20.1.
- Go to the nanoCAD, editor of the table, and then click *Import tables from Excel*.
- Data will be transmitted to the table.

Export

Main menu: ***Mechanical - Tables - Export table to Excel***.

Ribbon: ***Mechanical - Symbols - Export table to Excel***.

Toolbar: ***Export table to Excel (on toolbar*** "Tables"***)***.

Command line: ***MCTABLEEXCELEXPORT***.

Designed to transmit tabular data in MS Excel. After clicking on the button opens a new Excel workbook and it migrates all the tabular data with formatting cells established in the table.

Important!	The tables nanoCAD Mechanica20.1 not to Excel the following data: • *Blocks nanoCAD* • *Formatted text* • *Formulas that do not match the format of Excel*

Table recognition

Main menu: ***Mechanical - Recognition - Table recognition***.

Ribbon: ***Mechanical - Utilities - Table recognition***.

Toolbar: ***Table recognition (on toolbar*** "Recognition"***)***.

Command line: ***MCRECTABLE***.

Table recognition enables automatic search and replace tables created from individual primitives nanoCAD object "Table" nanoCAD Mechanica20.1. Sources are primitives for drawing.

1. Use the command *table recognition.*
2. Specify the drawing point at the table.

The table is created in the current layer. Dimensions of the recognized table are rounded to integer values.
Text belongs to the workpiece table is recognized as the values of the respective cells of the table.
Real dimensions of the original graphics are multiplied by the scale design, so if it turns out the table with zero height lines, you need to change the scale clearance in accordance with the size of the original graphics.

Editing a table in the drawing

Main menu: ***Mechanical - Tables -*** ***Edit table in-place***.

Ribbon: ***Mechanical - Symbols -*** ***Edit table in-place***.

Toolbar: ***Tables -*** ***Edit table in-place***.

Command line: ***MCIPEDITTBLIPEDITTBL***.

Shortcut: ***CTRL+RBM on table frame***.

When editing a table on-screen, the *"Edit Table"* toolbar appears.

The active cell is highlighted with a green frame. The text entered into the cell is automatically pre-pressed.

Menu

Group cells.
Serves to combine adjacent cells into one.
Procedure:

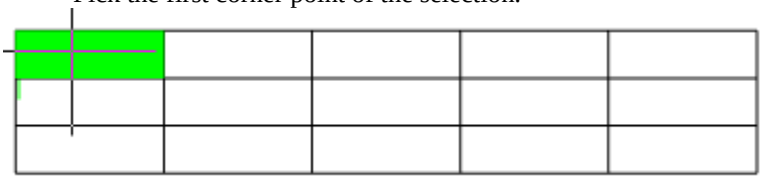
- Call command.
- Pick the first corner point of the selection.
- Pick second corner point of selection.
- Selected cells will merge.

Ungroup cells.
Undoes the grouping of a previously merged cell from multiple cells.
Procedure:

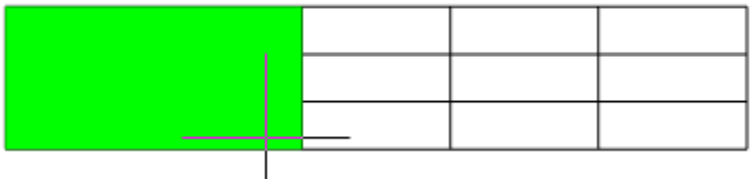
- Call command.
- Pick the first corner point of the selection.
- Pick second corner point of selection.
- Selected grouped cells that fall within the selected area will ungroup.

Split cells.
Serves to divide cells into several smaller cells.
Procedure:

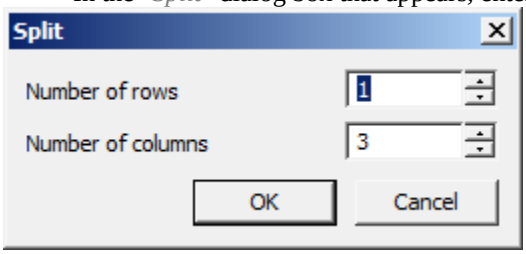
- Call command.
- Pick the first corner point of the selection.



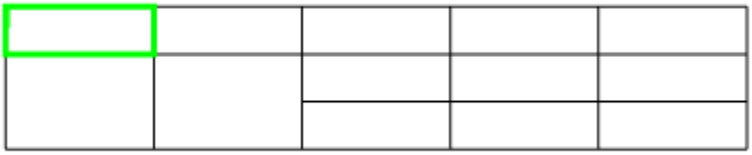
- Pick second corner point of selection.



- In the *"Split"* dialog box that appears, enter the required numerical values for repartitioning.



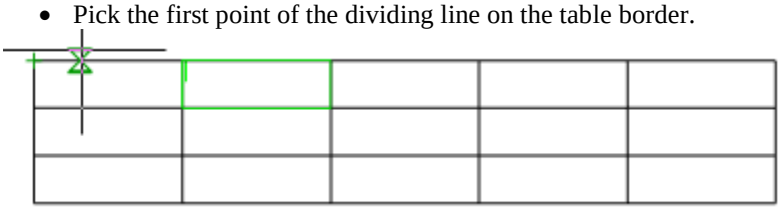
- This will result in a new breakdown.



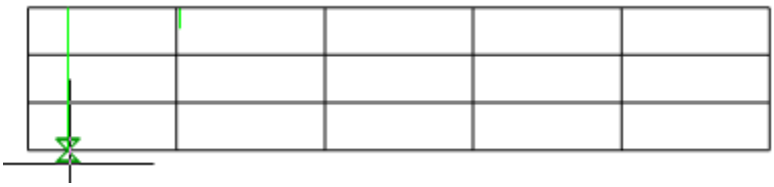
Note: When using the *"Split"* command, the height and width of adjacent cells may be lost.

Pencil.
Serves for splitting cells into several by creating additional borders.
Procedure:

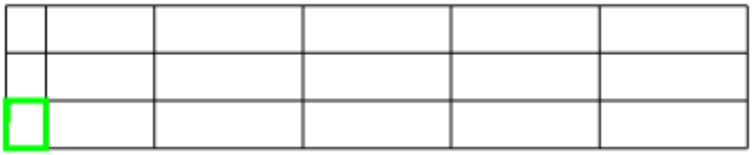
- Call command.
- Pick the first point of the dividing line on the table border.



- Pick the second point of the dividing line on the table border.



- The cells that the dividing line will pass through will be split into new cells. Each cell is independent.



Add row. Adds one row at the bottom of the table.

Add column. Adds one column to the end of the table.

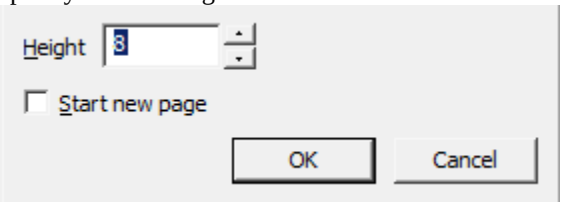
Insert row. Adds one row to the current position of the table cursor.

Insert column. Adds one column to the current position of the table cursor.

Delete row. Deletes the row containing the selected cell.

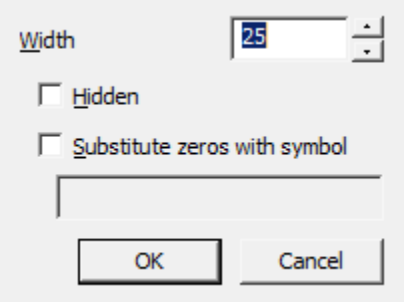
Delete column. Deletes the column that contains the selected cell.

Row properties. Changes the height of the row containing the selected cell. A dialog box is called up in which to specify the line height in millimeters.



The *"Start new page"* parameter splits the table in an arbitrary place, i.e. where the cell is selected. Other pagination methods are available in the full editor interface.

Column properties. Changes the width of the column that contains the selected cell. Enter the column width in millimeters in the dialog box.



The *"Hidden"* option hides the selected column. You can return visibility to a column from the full editor interface.
The *"Substitute zeros with symbol"* parameter replaces the null values of the column cells with the row data. In this case, the format of the cells must be numeric.

Cell properties. Calls up the "Cell Properties" dialog box, which configures the properties of the selected table cell.

Cells properties. Setting properties of a group of table cells. Select the required cells and in the appearing "Cell Properties" dialog box set the parameters of the selected table cells.

Sum

Calculates the sum of the selected area.
Procedure:

- Select the result cell.
- Call the command.
- Pick the first point of the marquee.
- Pick the second point of the marquee.
- The formula will be entered into the result cell.

Selective sum

Calculates the sum of the selected cells.
Procedure:

- Select the result cell.
- Call the command.

- Specify cells. Press "Enter" to finish.
- The formula will be entered into the result cell.

Notebook

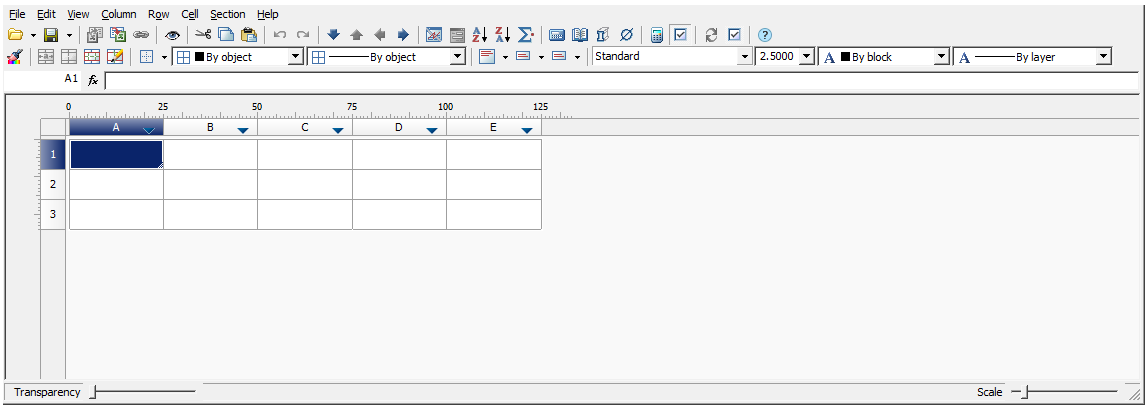
Calculator

Special symbols

Quit. Exits the table screen editor. You can also exit the on-screen table editor by right-clicking on an empty space in the drawing.

Table editor

The interface is opened by double-clicking on the table, or by selecting the *"Edit"* command from the context menu of the selected table.



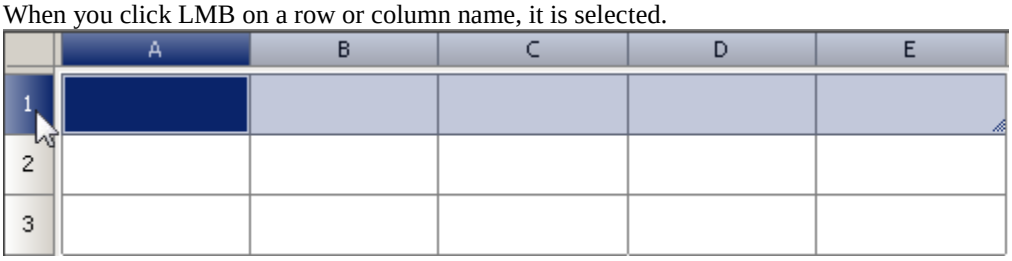
The dialog box contains:

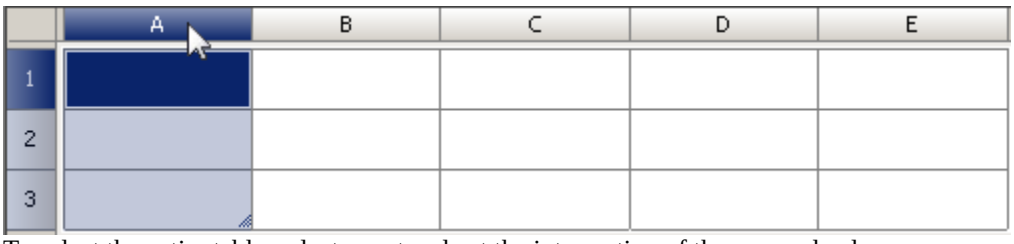
- main menu;
- toolbar;
- parted cells with names (addresses) of rows and columns;
- status bar with transparency and scale caret;

General Provisions

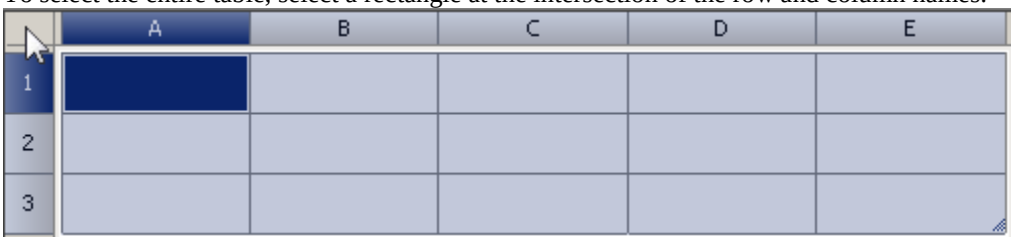
The names (addresses) of rows and columns are adopted similarly to the MS Excel spreadsheet editor. Vertical columns from left to right are named in alphabetical order in English letters (A, B, C, D,…, Z, AA, AB, etc.), horizontal rows are named in numbers in order.

When you click LMB on a row or column name, it is selected.

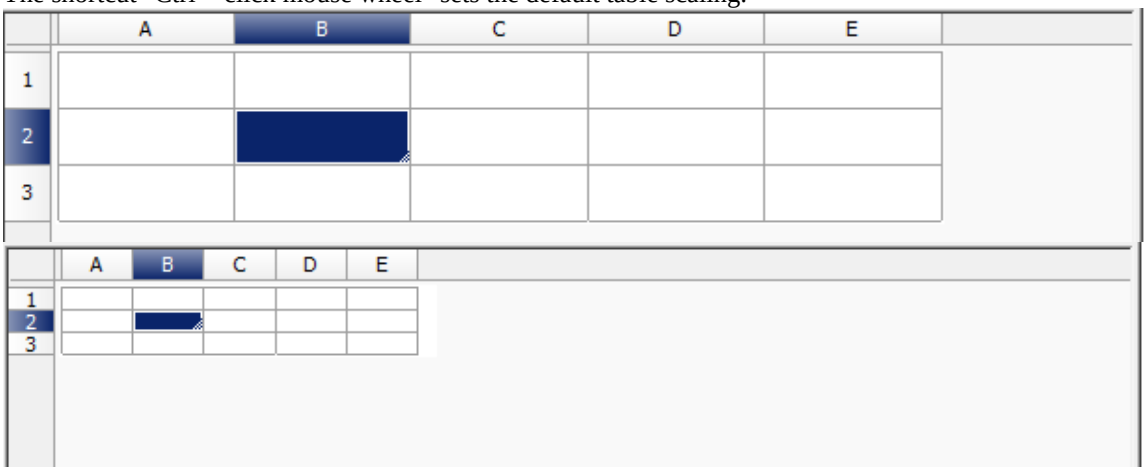




To select the entire table, select a rectangle at the intersection of the row and column names.

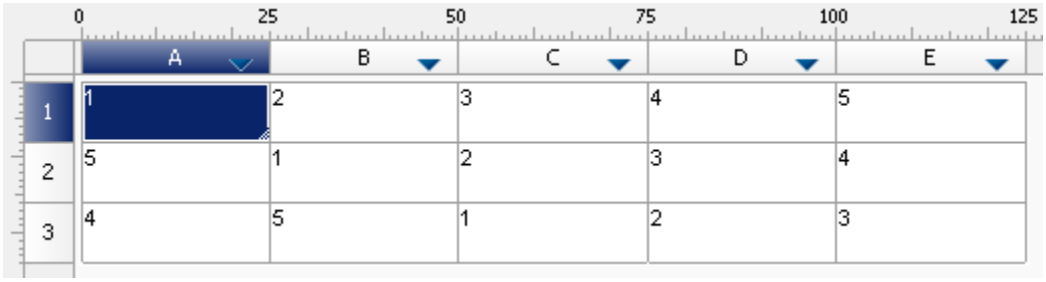


The shortcut "Ctrl + scrolling mouse wheel" allows you to change the scaling of the table.
The shortcut "Ctrl + click mouse wheel" sets the default table scaling.

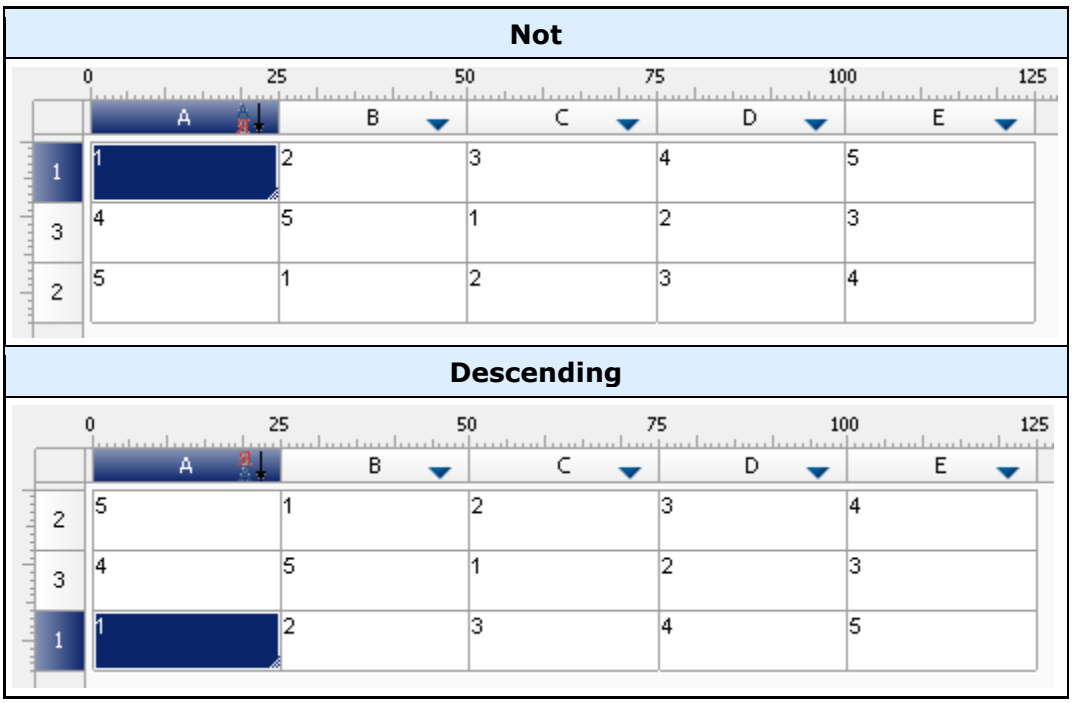


When *"View sorting"* is enabled, sort buttons appear on the right side of the column headings. This sorting is intended to improve the convenience of entering data in the spreadsheet editor and does not affect the state of the table in the drawing.
The button has three states: *"Not"*, *"Ascending"*, *"Descending"*.

Not



Ascending



Sorting features:

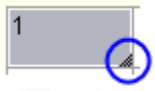
- Enabling sorting on one column disables sorting on other columns.
- Rows are sorted whole, not just the column.
- Strings are sorted within a section.
- The merging of cells is not broken.
- After enabling view sorting, the vast majority of commands are blocked. Allowed: exit, edit single cells and change sort mode.

Cell color

=Object.Name	- cell containing expression;
CSN M 12x120#	- non-editable cell, for example, in a report;
#ERR	- cell with an error in the expression;
60	- cell containing report section data;

Filling adjacent cells with data

To speed up data entry into the table, automatic data filling functions are used. The spreadsheet editor can automatically continue a series of numbers, combinations of numbers and text according to a given pattern. You can quickly fill in different types of data series by selecting multiple cells and dragging the fill handle.



Filling cells with a sequence of numbers or combinations of numbers and text according to a given pattern:

1. Select the first of the filled cells.

2. Enter an initial value for a range of values.

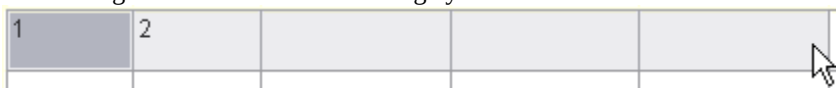
3. Enter a value in the next cell to set the fill pattern.

For example, if you want the sequence 1, 2, 3, 4, 5 ..., enter the values 1 and 2 in the first two cells. If you want the sequence 2, 4, 6, 8 ..., enter 2 and 4. If you need sequence 2, 2, 2, 2 ..., the second cell can be left blank.
As the sequence is completed, it continues as shown in the table below.

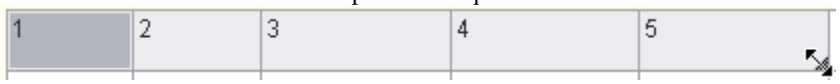
Initial value	Continuation of the series
1, 2, 3...	4, 5, 6 ...
1st period, 2nd period	3rd period ...
Product 1, Product 2	Product 3 ...

4. Highlight cell or cells containing seed values.

5. Drag the fill handle over the range you want to fill.



The cells will be filled with the specified sequence of numbers.



To fill in ascending order, drag the handle down or to the right. To fill in descending order, drag the handle up or left.

Basic tools

Main menu commands

File

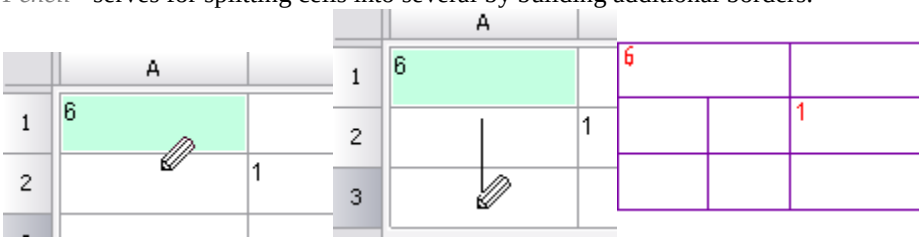
- *Open file...* - loads table data from a file. Files available for download: * .tbl, * .dat, * .mdb (only for 32-bit Windows), * .txt, * .csv, * .xml, * .xls, * .xlsx. When you select an Excel file, only the first sheet of the workbook is inserted.
- *Open database...* - opens a dialog of library to select a table saved in the database.
- *Save as file...* - writing a table to an external file *.dat, *.txt, *.cvs, *.xml, *.xls.
- *Save to database...* - writing the table to the base of elements.
- *Edit Form...* - opens the *form editor*. Using the form, you can filter data, control the composition and style of the table.
- *Show Form...* - opens the table filter form.
- *Imtort from Excel* - the command transfers the results of calculating tabular data from an open MS Excel workbook. The content of the active sheet or a selected range of cells is transferred.
- *Export to Excel* - the command is designed to transfer tabular data to MS Excel. After clicking the button, the *"Conversion options"* dialog opens. After specifying the options, a new Excel workbook is opened and all tabular data is transferred into it, while maintaining the cell formatting set in the table.
- *Close* - saves changes and closes the editor.

- *Close without saving* - closes the editor without saving changes.

Edit

- *Undo* - undo the last action.
- *Redo* - redo the last undone action.
- *Cut* - cuts data from selected cells to the clipboard.
- *Copy* - copies data from the selected cells to the clipboard.
- *Paste* - pastes data from the clipboard.

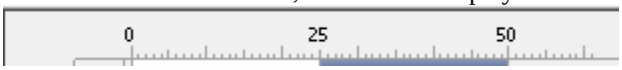
Pencil - serves for splitting cells into several by building additional borders.



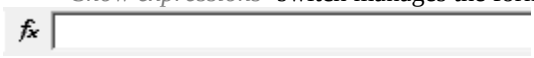
- *Select material...* - This tool is designed to insert into a table, technical requirements and technical characteristics of a formatted line-record for material and assortment. Clicking on the button brings up the *"Material"* dialog box.
- *Auto calculate expressions* - controls the mode of automatic recalculation of values of table cells. By default, the auto-recalculation mode is on.
- *Recalculate* - serves to recalculate data in a table after editing cell values or correcting formulas.
- *Auto update reports* - controls the mode of automatic recalculation of the report. The mode is on by default.
- *Reports* - update of the report.
- *Synchronize* - updates the table with data from a linked file. Available if the table was loaded from a file. For correct synchronization, the data changed in the file must be previously saved.

View

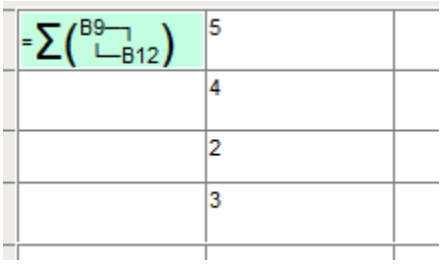
- *Show rulers* - switch, controls the display of the ruler.



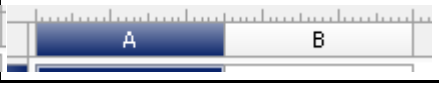
- *Show expressions* -switch manages the formula bar display.



- *Formula visualization* - switch, controls the display of the formulas in cells.



- *Zoom to objects* - centers the original report objects in the drawing.
- *View sorting* - controls the display of the sort buttons of the view.

Turned on	Turned off
	

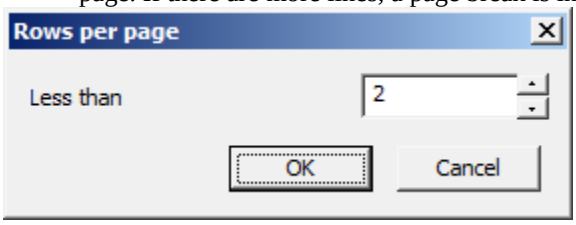
- *Calculator...* - the command calls calculator.
- *Notebook...* - the command calls notebook.

Column

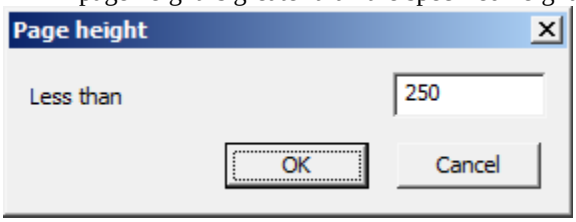
- *Add column* - adds a new empty column after the selected column.
- *Insert column* - adds a new empty column before the selected column.
- *Delete column* - deletes the selected column.
- *Hidden* - switch, hides the column. The column appears in the editor but does not appear in the drawing.
- *Properties...* - open properties of column.

Row

- *Add row* - adds a line below the selected one.
- *Delete row* - deletes the selected row.
- *Hide row* - hides the line. The line is not visible in both the editor and the drawing.
- *Show hidden rows* - shows hidden lines. To use the command, you must select several lines between which hidden lines are located.
- *Sort ascending* - sorts the table in ascending order relative to the selected column.
- *Sort descending* - sorts the table in descending order relative to the selected column.
- *Custom sort...* - opens a dialogue *"Collections editor"*.
- *Merging and Grouping...* - opens a dialogue *"Row grouping and merging"*.
- *Start new page* - switch, controls the page break before the selected line.
- *Limit rows per page...* - opens the *"Rows per page"* dialog, which indicates the maximum number of lines per page. If there are more lines, a page break is inserted.



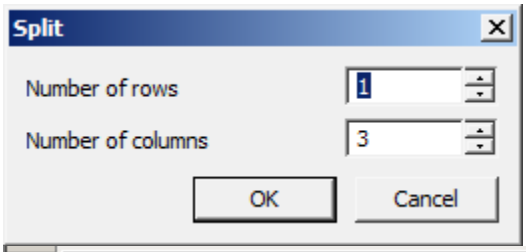
- *Limit page height...* - opens the "Page height" dialog in which the maximum page height is indicated. If the page height is greater than the specified height, a page break is inserted.

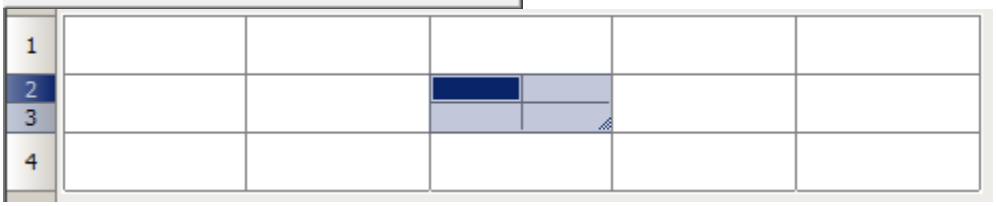


- *Single page* - the command removes all created page breaks.
- *Up to down* - switch, controls the data display sequence.
- *Properties...* - opens string properties.

Cell

- *Edit* - puts the cell into edit mode.
- *Expression...* - opens a dialogue *"Expression builder"*.
- *Properties...* - opens a dialogue *"Cell properties"*.
- *Group* - merges the selected cells.
- *Ungroup* - corrodes the selected cells.
- *Split...* - splits a cell into multiple rows and columns. When the command is called, the *"Split"* dialog appears.





- *Cut* - cuts data from the selected cells to the clipboard.
- *Copy* - copies data from the selected cells to the clipboard.
- *Paste* - pastes data from the clipboard.
- *Clear* - clears the selected cells.
- *Insert table* - pastes a range of cells copied from Excel.

Section

- *Header of first page* - adds a header for the first page.
- *Header of following pages* - adds a header for pages.
- *Header of last page* - add a header for the last page.
- *Insert Data section* - inserts a data section. If the selected cell is in the data section, then an empty row is added.
- *Insert Report section* - adds a report section. Used when building reports.
- *Footer of first page* - adds a footer for the first page.
- *Footer of following pages* - adds a footer for pages.
- *Footer of last page* - add footer for the last page.

Toolbar commands

- *Open* - allows you to choose one of two commands:

Open file- loads table data from a file. Files available for download: * .tbl, * .dat, * .mdb (only for 32-bit Windows) * .txt, * .csv, * .xml, * .xls, * .xlsx. When you select an Excel file, only the first sheet of the workbook is inserted.

Load from base- opens a dialog of standard elements to select a table saved in the database.

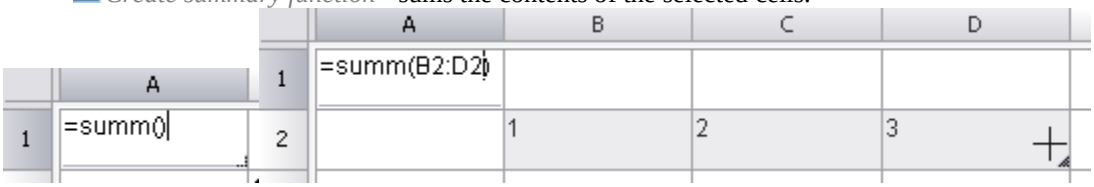
- *Save* - allows you to choose one of two commands:

Save to base- writing the table to the base of elements.

Save to file- writing a table to an external file * .dat, * .txt, * .cvs, * .xml, * .xls.

- *Import table from Excel* - the command transfers the results of calculating tabular data from an open MS Excel workbook. The content of the active sheet or a selected range of cells is transferred.
- *Export table to Excel* -the command is designed to transfer tabular data to MS Excel. After clicking the button, the *"Conversion options"* dialog opens. After specifying the options, a new Excel workbook is opened and all tabular data is transferred into it, while maintaining the cell formatting set in the table.
- *Update table from external source* - updates the table with data from a linked file. Available if the table was loaded from a file. For correct synchronization, the data changed in the file must be previously saved.
- *Pan and zoom view to show objects* - centers the original report objects in the drawing.
- *Cut selection* - cuts data from the selected cells.

- *Copy selection* - copies data from the selected cells.
- *Paste from clipboard* - pastes data from the clipboard.
- *Undo last change* - undo the last action.
- *Redo last change* - redo the previously undone action.
- *Move row down* - moves the selected line down.
- *Move row up* - moves the selected line up.
- *Move column left* - moves the selected column to the left.
- *Move column right* - moves the selected column to the right.
- *Page division* -the tool is designed to *split the table* into separate parts, without losing integrity. With this function it is possible to split a large table in order to fit on the format, and still retain the ability to edit the table as a single object.
- *Open form* - opens the table filter form.
- *Sort ascend* - sorts the table in ascending order.
- *Sort descend* - sorts the table in descending order.
- *Create summary function* - sums the contents of the selected cells.



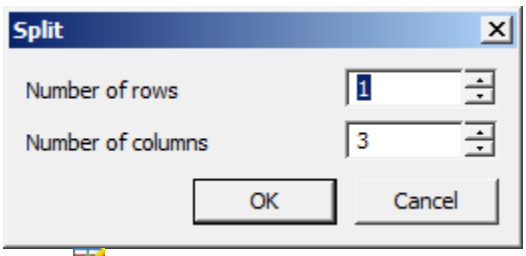
Procedure:

1. Select the cell in which you want to calculate the amount, press *"Create summary function"*.
2. Select the cells whose contents you want to summarize, press *"Enter"*.

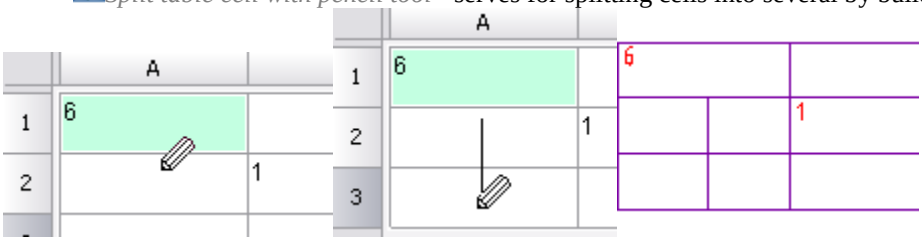
- *Open calculator* - the command calls calculator.
- *Open notebook* - the command calls notebook.
- *Insert material* - This tool is designed to insert into a table, technical requirements and technical characteristics of a formatted line-record for material and assortment. When you click on the button, a dialog box is called *"Material"*.
- *Special symbols* - inserting a special character from the list.
- *Recalculate table* - serves to recalculate data in a table after editing cell values or correcting formulas.
- *Automatic calculation* - controls the mode of automatic recalculation of values of table cells. By default, the auto-recalculation mode is on.
- *Update reports* - update of the report.
- *Automatic report update* - controls the mode of automatic recalculation of the report. The mode is on by default.
- *Group selection* - serves to combine the selected table cells into one.
- *Ungroup selected cells* - cancels the merge for the selected cells.
- *Change number of rows and columns* - serves to change the number of rows and (or) columns in the selected range of cells.

Procedure:

1. Select one or more cells and call the command.
2. In the *"Split"* dialog box that appears, enter the desired numeric values to stake.



- *Split table cell with pencil tool* - serves for splitting cells into several by building additional borders.



To split a cell with the mouse cursor, specify the start and end points of the segment that will divide each cell it crosses into two.
Right-clicking on a cell border removes that border.

- *Select cell style* - calls the window for applying the style to the selected cells.
- *Text alignment* - serves to control the placement of text in selected cells. Additional buttons are available by pressing the black triangle. The purpose of the buttons is clear from the figure on each of them.

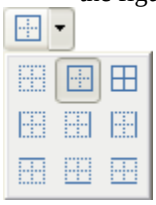
- *Fitting text horizontally* - controls the horizontal alignment of text in the cell. The following options are available in the drop-down list:

No horizontal tolerance
Shrink horizontally
Word wrap

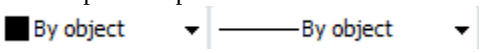
- *Fitting text vertically* - controls the vertical alignment of text in the cell. The following options are available in the drop-down list:

No vertical tolerance
Shrink text height
Expand row height
Add virtual rows

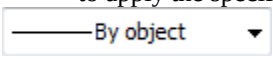
- *Borders* - is designed to control the display of the borders of the selected cells. When you click on the black triangle, additional buttons for displaying borders become available. The purpose of the buttons is clear from the figure on each of them.



- *Color of borders* - drop-down list for choosing border colors. To set the border color, you must first select a color and then select the border (using the *"Borders"* command) to which you want to apply the specified parameters.

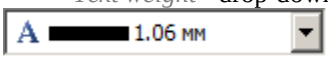


- *Lineweight of borders* -drop-down list of border thickness selection. To set the border thickness, you must first select the thickness and then select the border (using the *"Borders"*command) to which you want to apply the specified parameters.

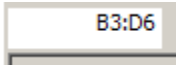


- *Text style* - drop-down list for choosing the text style of the cell.
- *Text height* - drop-down list for selecting the height of the cell text.

- *Text color* - drop-down list for choosing the color of the cell text.
- *Text weight* - drop-down list for choosing the thickness of the lines of the cell text.



- *Selected cell coordinate field* - displays the coordinates of the selected cell (s) and allows you to enter them.



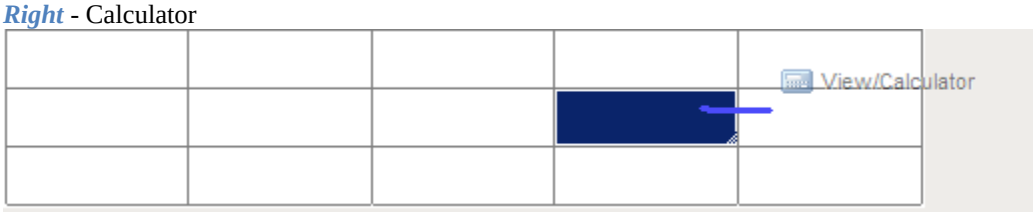
- *Row expressions* - formula input field. The line is displayed if the *"Main menu - View - Show expressions"* switch is active.

Using gestures for shortcuts

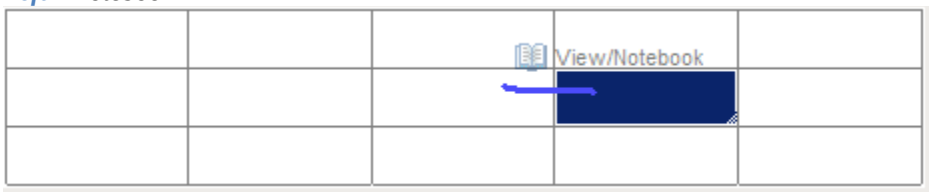
Gestures are designed for quick access to commands from the main menu of the table editor. Gestures made with pressed RMB in the table editor.

Note: If when the prompt appears gesture command being executed, you can omit the RMB.

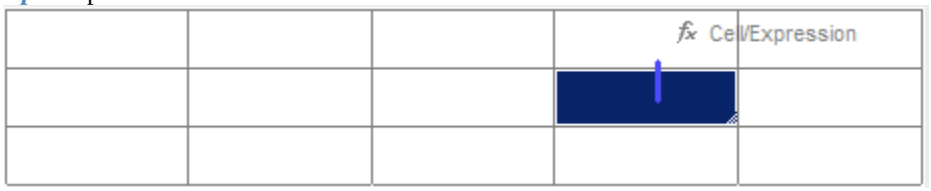
Right - Calculator



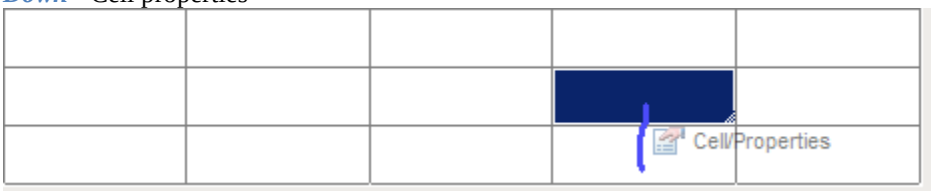
Left - Notebook



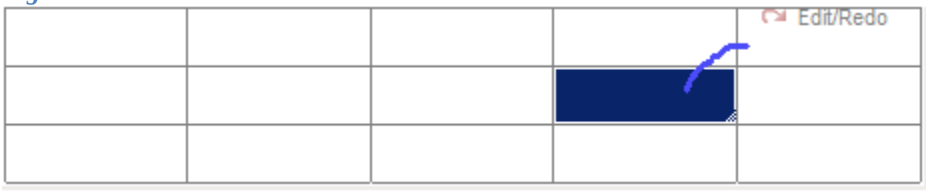
Up - Expression



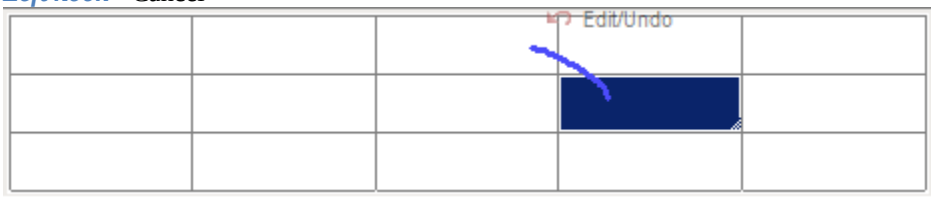
Down - Cell properties



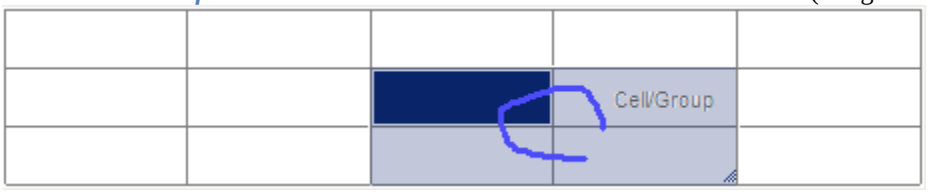
Right hook - Return



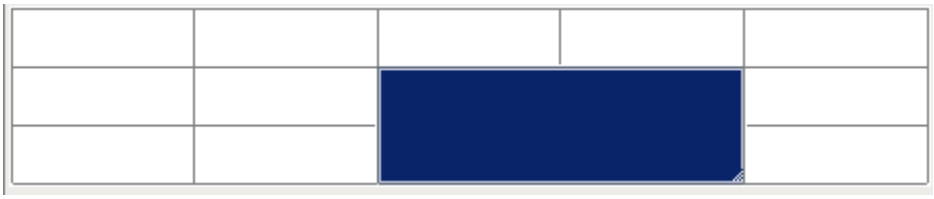
Left hook - Cancel



The closed area of the selected cells in a clockwise direction - Connection (merged cells pre-allocated)

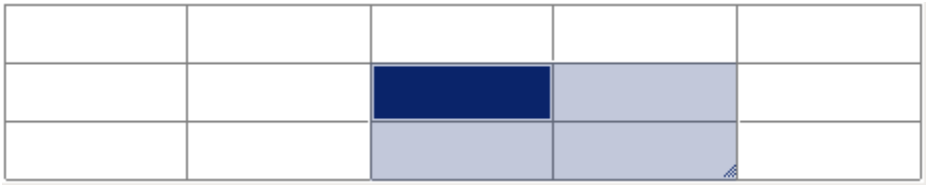


Result:

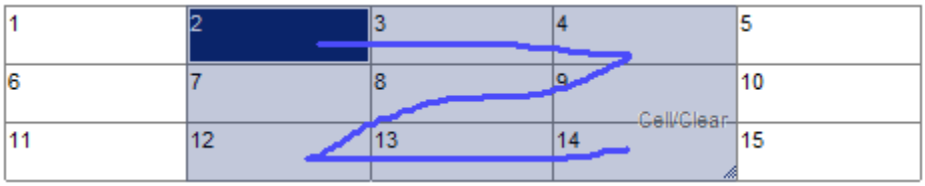


The closed area of the selected cells counterclockwise - Disconnection

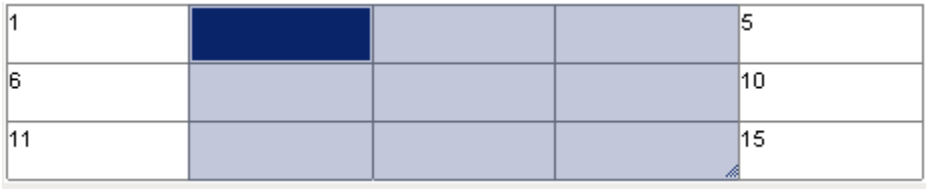
Result:



Symbol "z" on select cells - Clear (pre-clean cell is released)



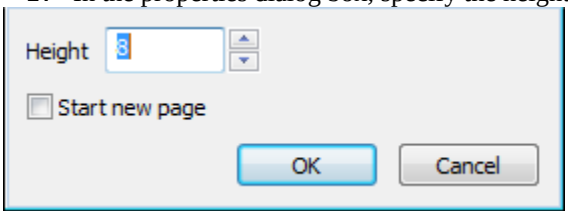
Result:



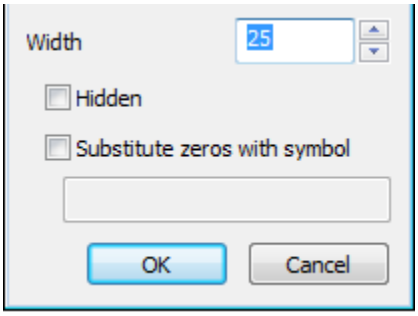
Editing the size of rows and columns

To modify the row height or column width, move your mouse to the border of the row or column. Click the left mouse button and holding the button down drag the border. The move will be highlighted by the current value of the row height or column width, as well as its new size.
Resize a row or column can also be done in the properties dialog box, selected by pressing the right mouse button on the header row or column.

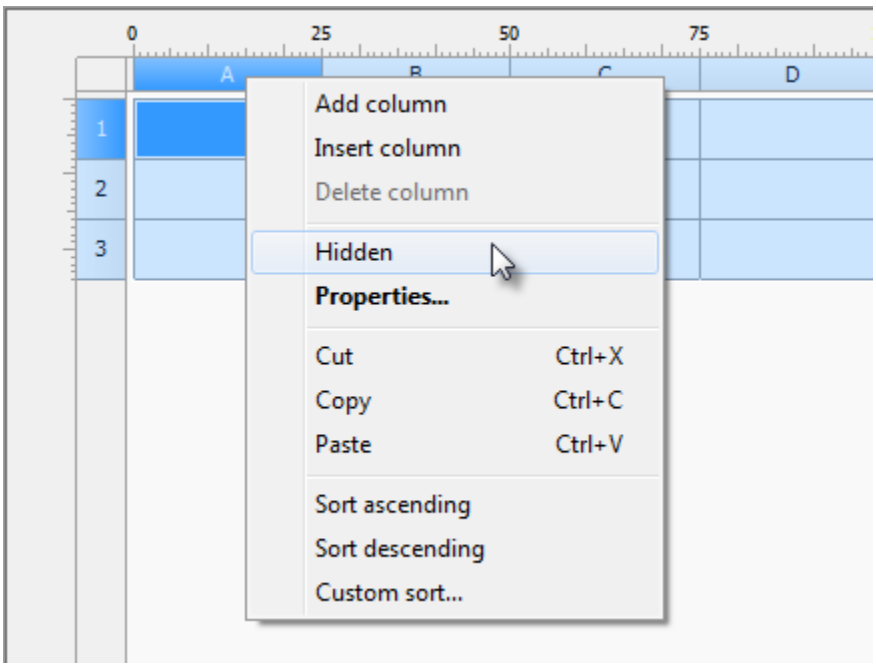
1. In the properties dialog box, specify the height (width) in millimeters.



2. In the Properties dialog box of the column switch is available *Hidden*. With it, you can disable the display of the selected column in the drawing, however, the Editor will be displayed.



You can also hide a column from the context menu for the column.



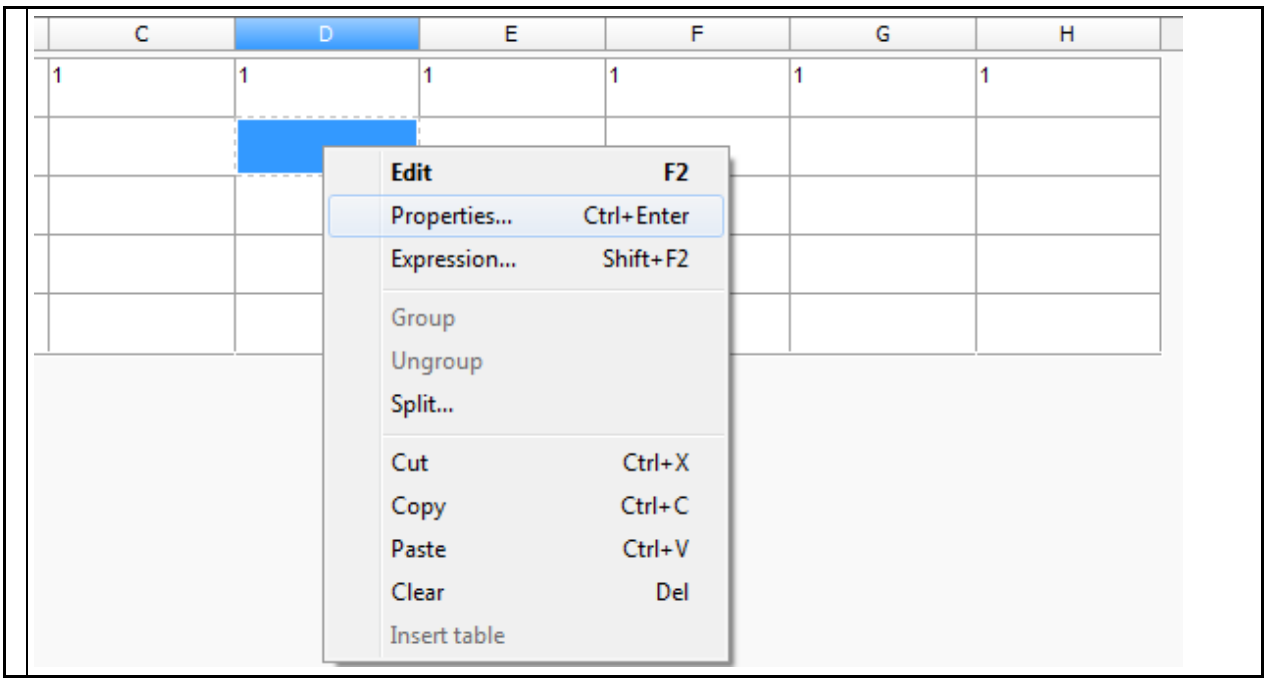
Command *string Replace null values* (for numeric cells with the value zero, the cell must be a numeric format), replaces the contents of the cells specified text.

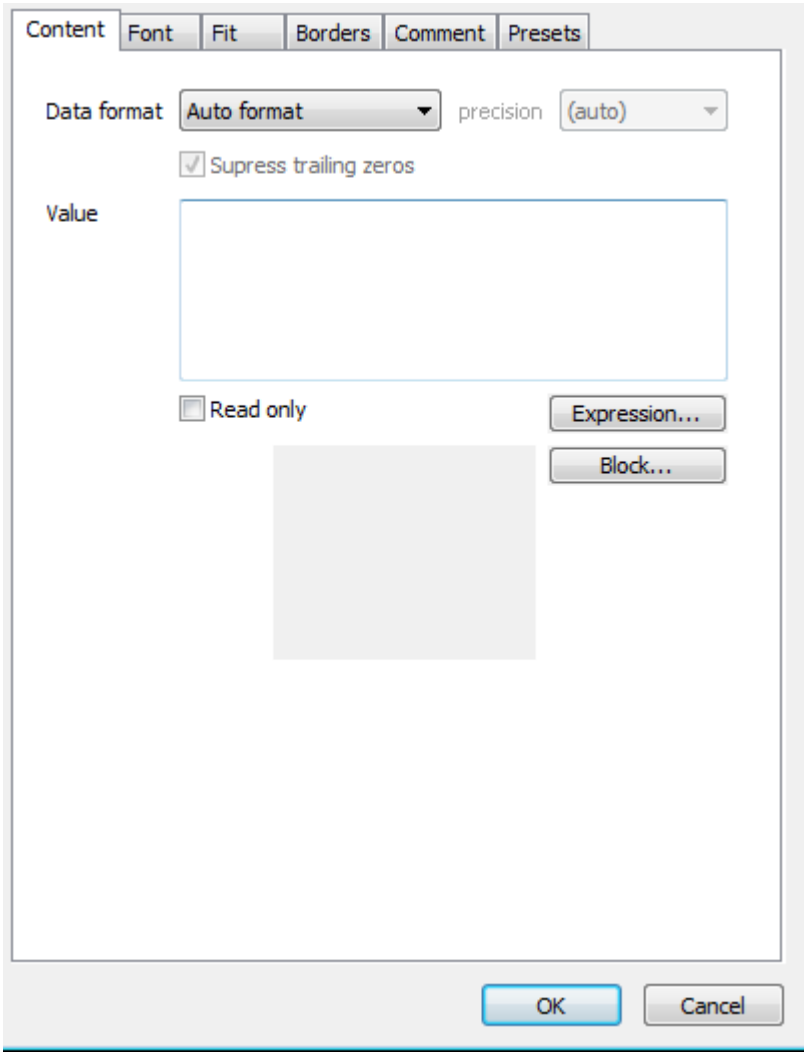
Important!	Format text in a cell value is set in the properties of the cell on the tab *Content* .

Close the dialog by clicking OK.

Cell Properties

In the dialog box *Properties* ***parameters are set for table cells.***
To edit the properties of a single cell in a table full editor, select the desired cell and issue *Properties* from the context menu or by pressing Ctrl + Enter, double click on the cell, as a gesture to "down" on the cell. The effect of this command is similar *Cell Properties* screen table editor.
To edit multiple table cells in the full editor, select the desired cells and issue the *Properties* from the context menu. The effect of this command is the same action as *Properties* screen editor table.


C
D
E
F
G
H
1
1
1
1
1
1
Edit F2
Properties... Ctrl+Enter
Expression... Shift+F2
Group
Ungroup
Split...
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Clear Del
Insert table

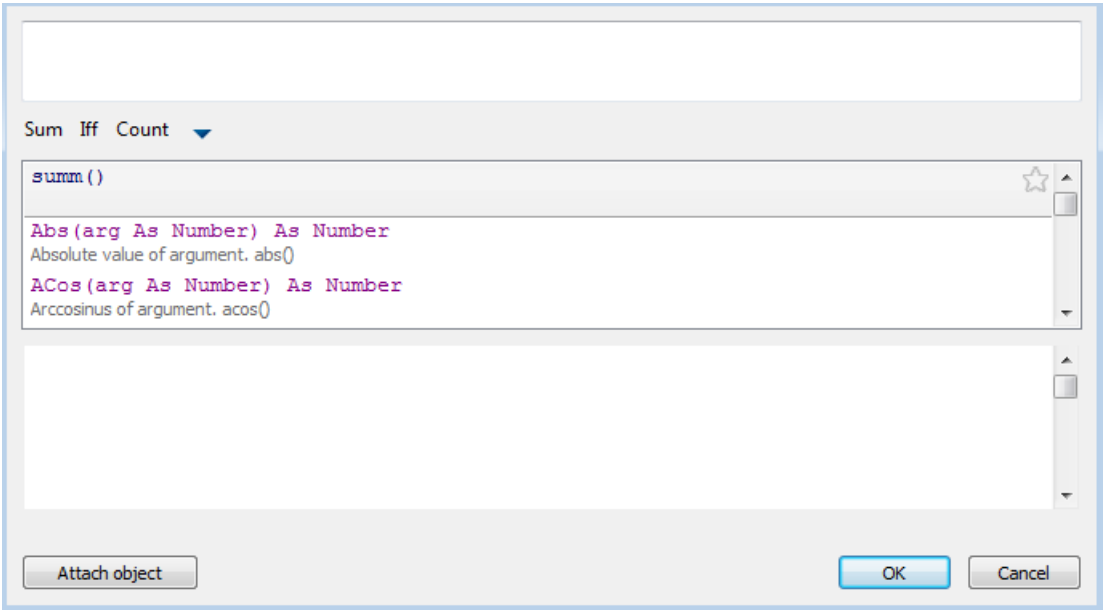


There are six tabs:

- Flag ***Content***. This tab specifies the type of data in a cell the user, set the formula for calculating values. In the properties of the cell can format values and enter the contents of a table cell. Checking ***unmodifiable*** disables the ability to edit the cell. Such a cell is highlighted in color.

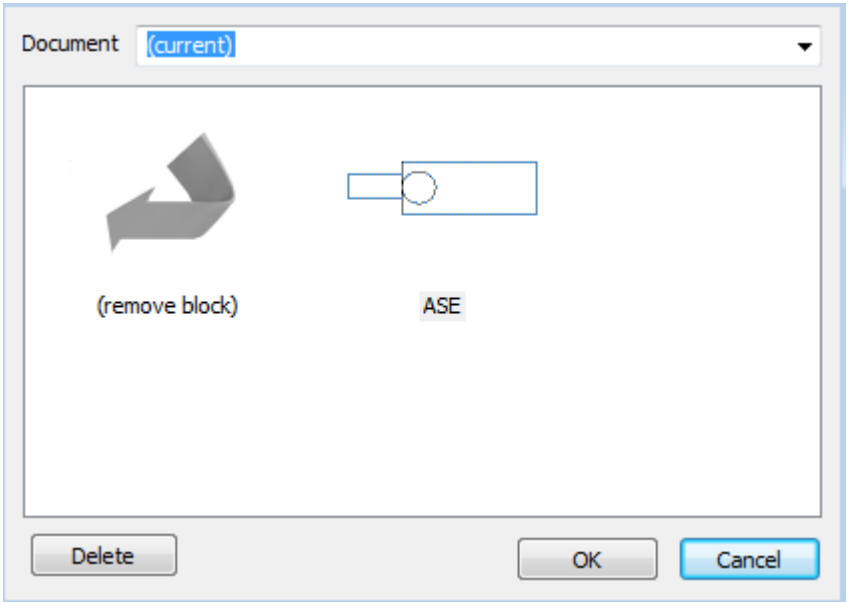
Expression...	- Opens the *Expression Builder*.

-

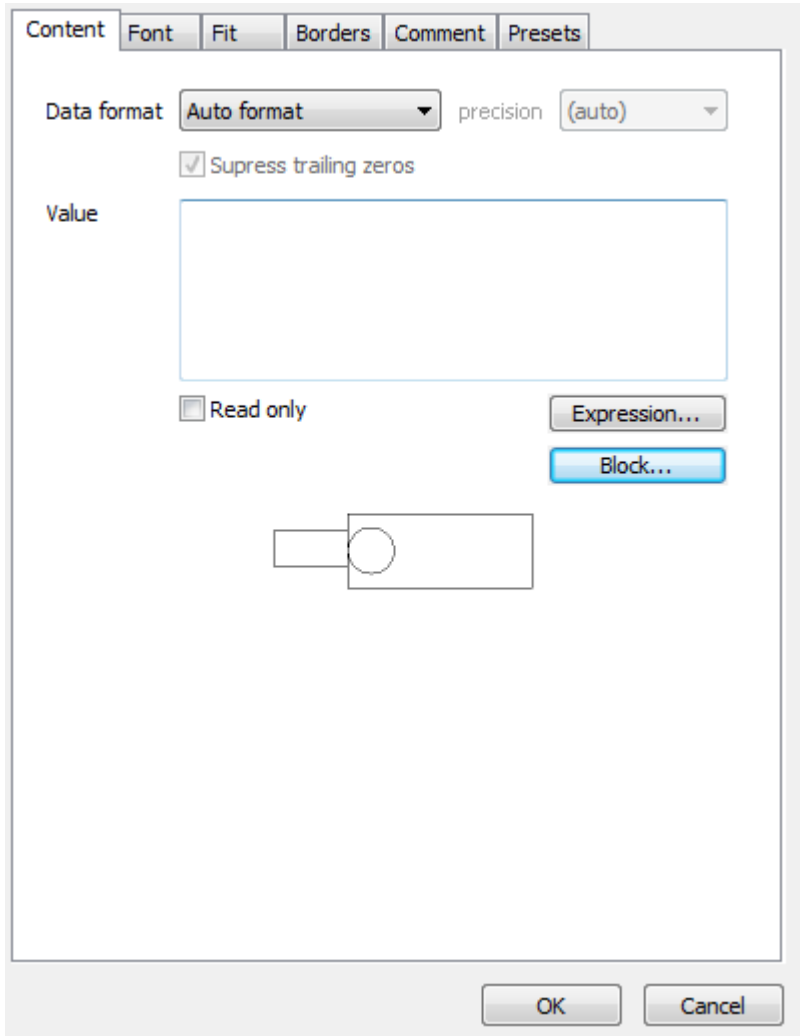


Block...	- The tool is used to insert the nanoCAD block the cell.

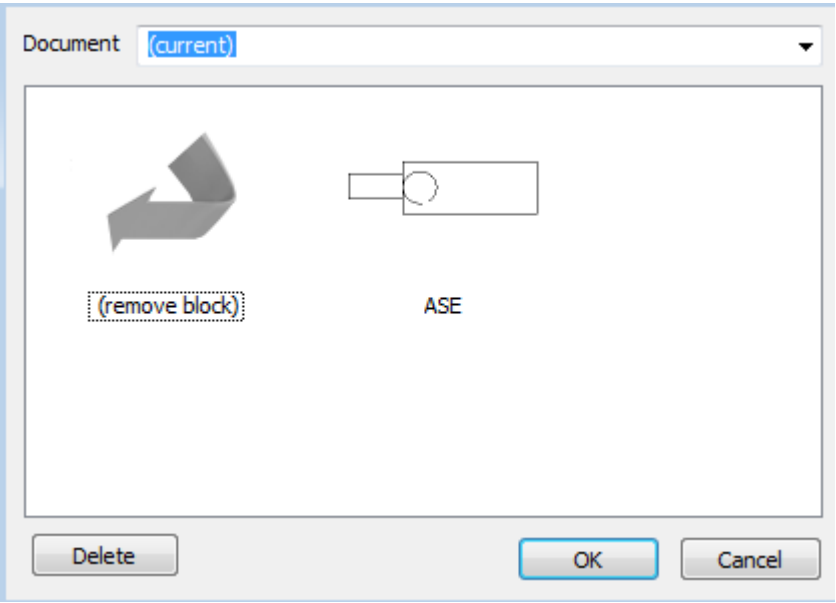
Select the list that appears in the desired unit from the current drawing file. If necessary, select another file using the menu *View*.



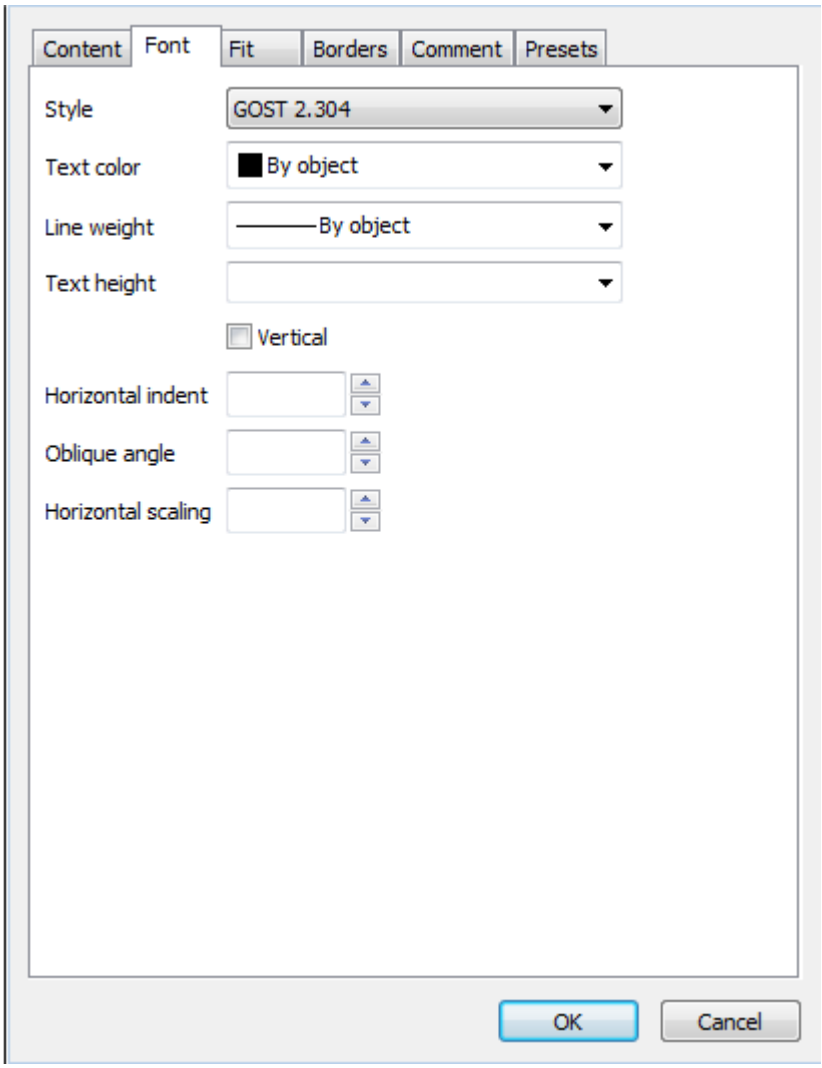
After selecting a unit, it is displayed in the properties window of the cell, as well as in a table cell.



To disconnect the unit, press the ***Remove block*** in the *block selection*.

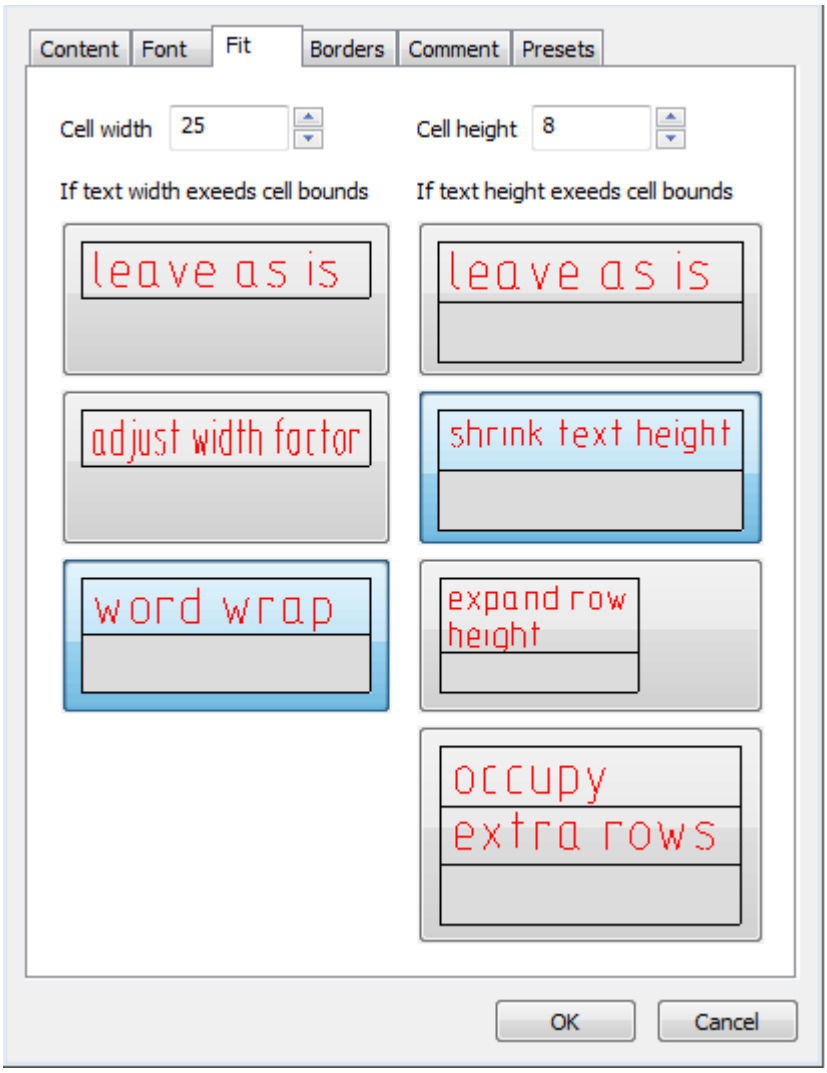


- Bookmark *Font*. This specifies the text font, color symbols, line thickness, font height, offset from the cell boundaries, the angle that defines the direction of the text, as well as the aspect ratio of the text.



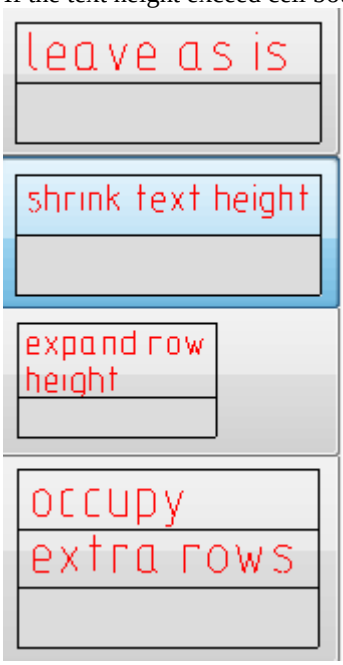
Switch *Vertical* change text direction to vertical.
Empty fields *oblique Angle* and *horizontal scaling* means that the values are taken from the text style and *horizontal indent* - that the value is taken from the table settings.

- Bookmark *fit*



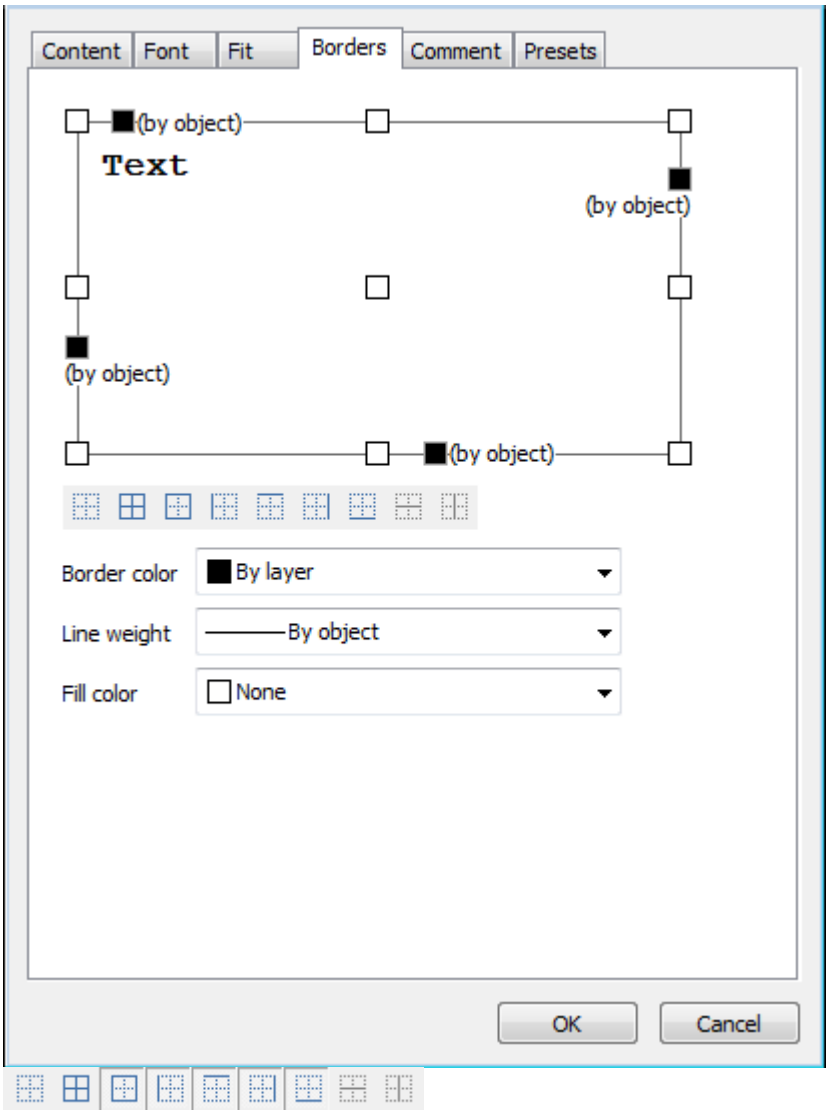
This specifies the *cell, cell* as well as parameters of inscribing text into the cell.
If the text width exceeds cell bounds

If the text height exceed cell bounds:

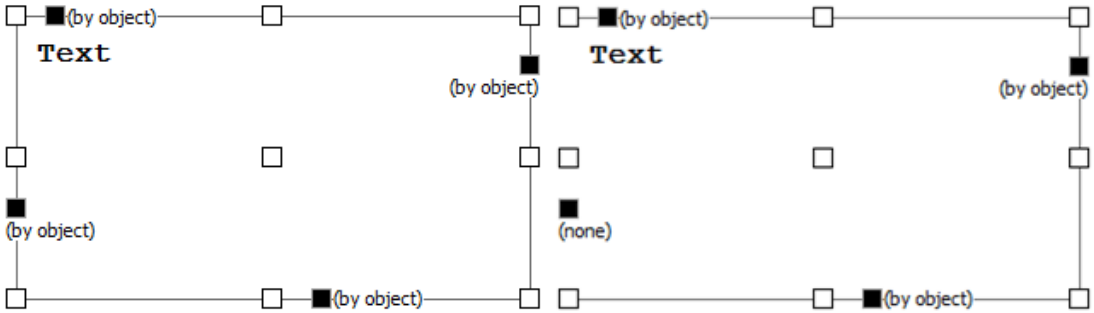


Mode *Occupy extra row* does not change the number of rows in the table, the desired row is made higher by the required number of times and line.

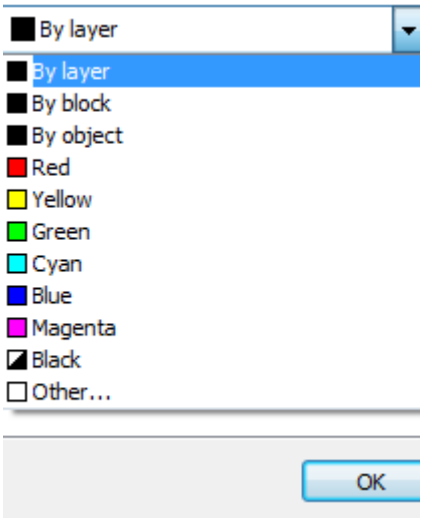
- Bookmark *borders*. Here you specify the type, color, thickness of the boundary line of the selected cell, you can manage the display of individual cell borders. Enable or disable the display of individual boundaries, you can use buttons



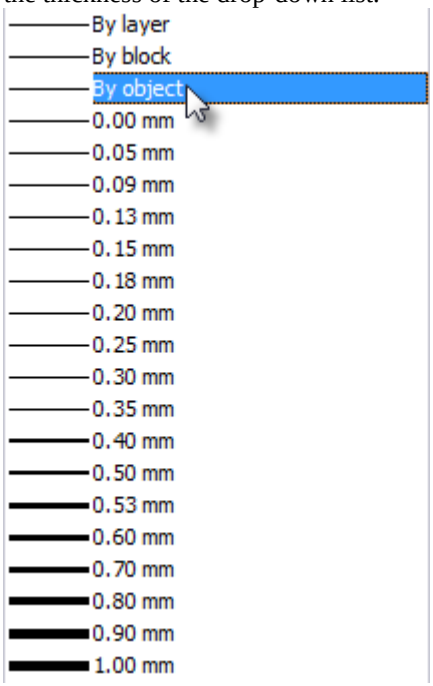
- or in the preview by clicking the left mouse button near one of the cell borders. Alignment of text within a cell preview is set by clicking the left mouse button.



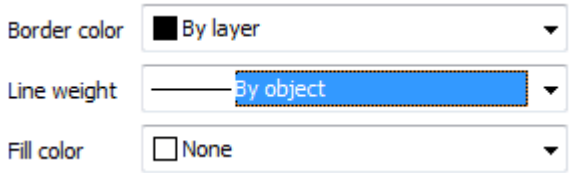
To set the color of individual borders, click the left mouse button on the icon	■	and select a color from the drop down list



To determine the weight of the individual borders, click the left mouse button on the line (***By layer***) and choose the thickness of the drop-down list.

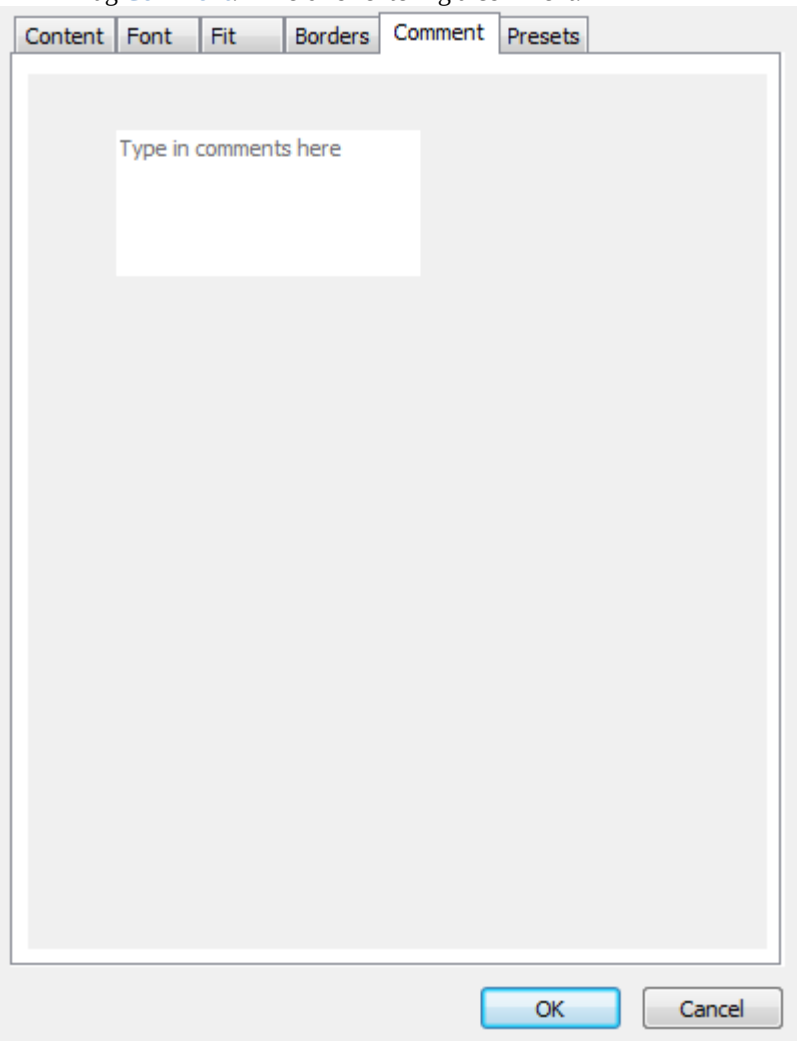


- Color, line thickness for the cell borders, and cell shading can be set from the menu.

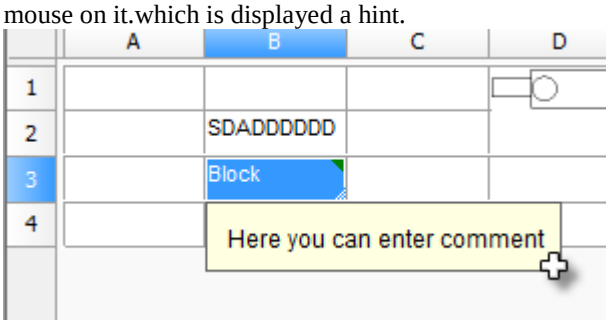


- To apply the changes in color and thickness, click on the desired boundary. Can also be selected by using the display boundaries.

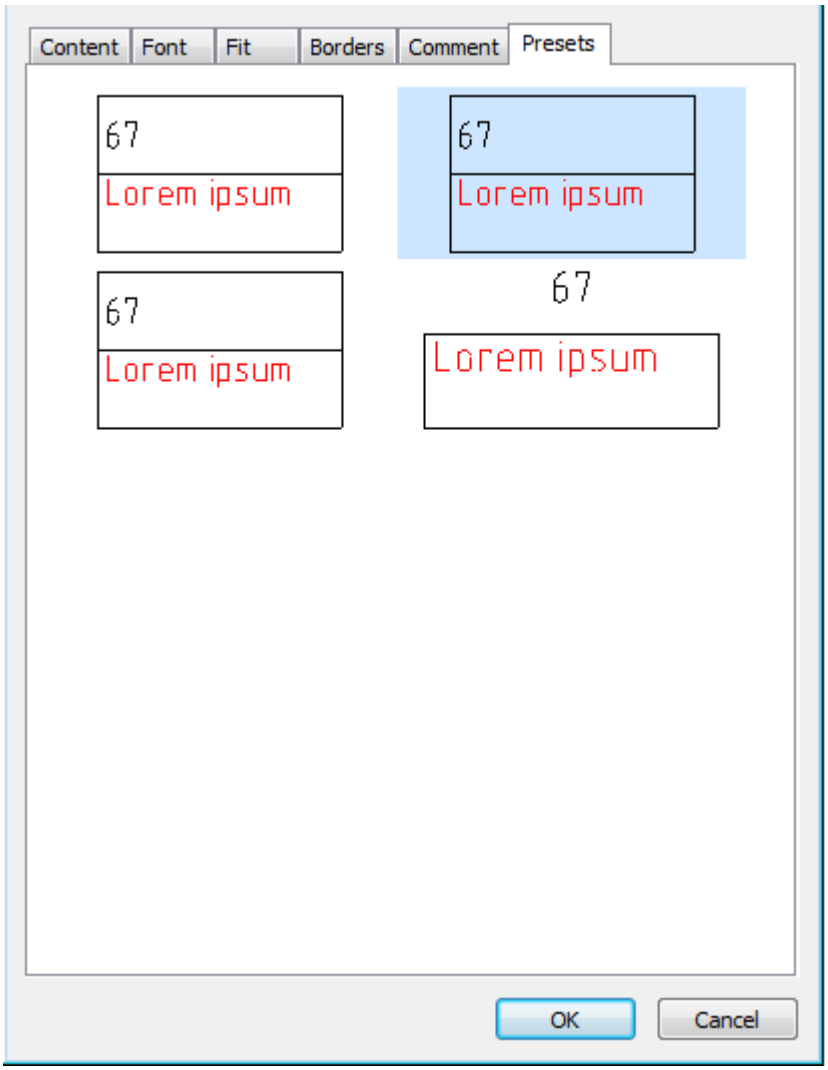
- Flag ***Comment***. A field for entering a comment.



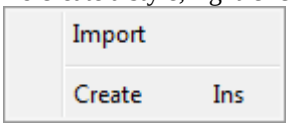
Cell with the comment is displayed in the editor by a green label, which displays a hint when you hover the mouse on it.which is displayed a hint.



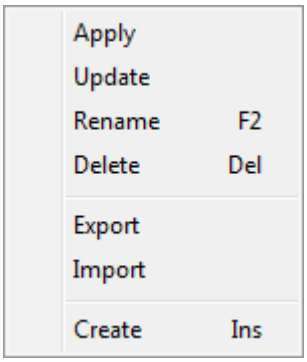
- Bookmark ***Styles***. Here you can set the style for the cell.



To create a style, right-click in the blank area of the ***Style*** tab, then on the shortcut menu, select ***Create***.



To work with a style template, open the context menu style.



Using styles

Style assignment to table cells is performed by the command *"Select cell style"* in the toolbar of the table editor or tab *"Fit"* in dialogue *"Cell properties"*. he choice of style is carried out in the graphics window. Also, styles can be created and edited using the context menu.

Context menu

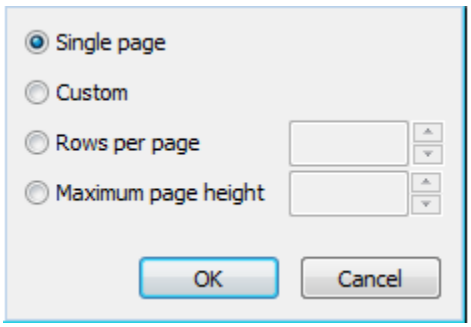
- *Apply* - the command applies the selected style on the cell (group of cells).
- *Update* - the command replaces the selected style with the style of the selected cell.
- *Rename* - command allows you to rename the name of the style. The name of the style is generated automatically during creation.
- *Delete* - command removes the selected style from the list.
- *Import* - command imports style in * .mcd format.
- *Export* - command exports the style in * .mcd format.
- *Create* - command allows you to create a new style.

Creation order:
1. Set the cell style.
2. Open the style window
3. Select the "Create" context menu command.
4. A new style will be created. The name of the style is generated automatically during creation.

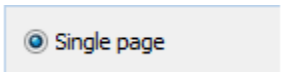
Division by page

Division by page. tool is designed to separate the table into separate parts, without losing its integrity. With this feature, you can split a large table in order to place a format, while preserving the ability to edit the table as a single object.

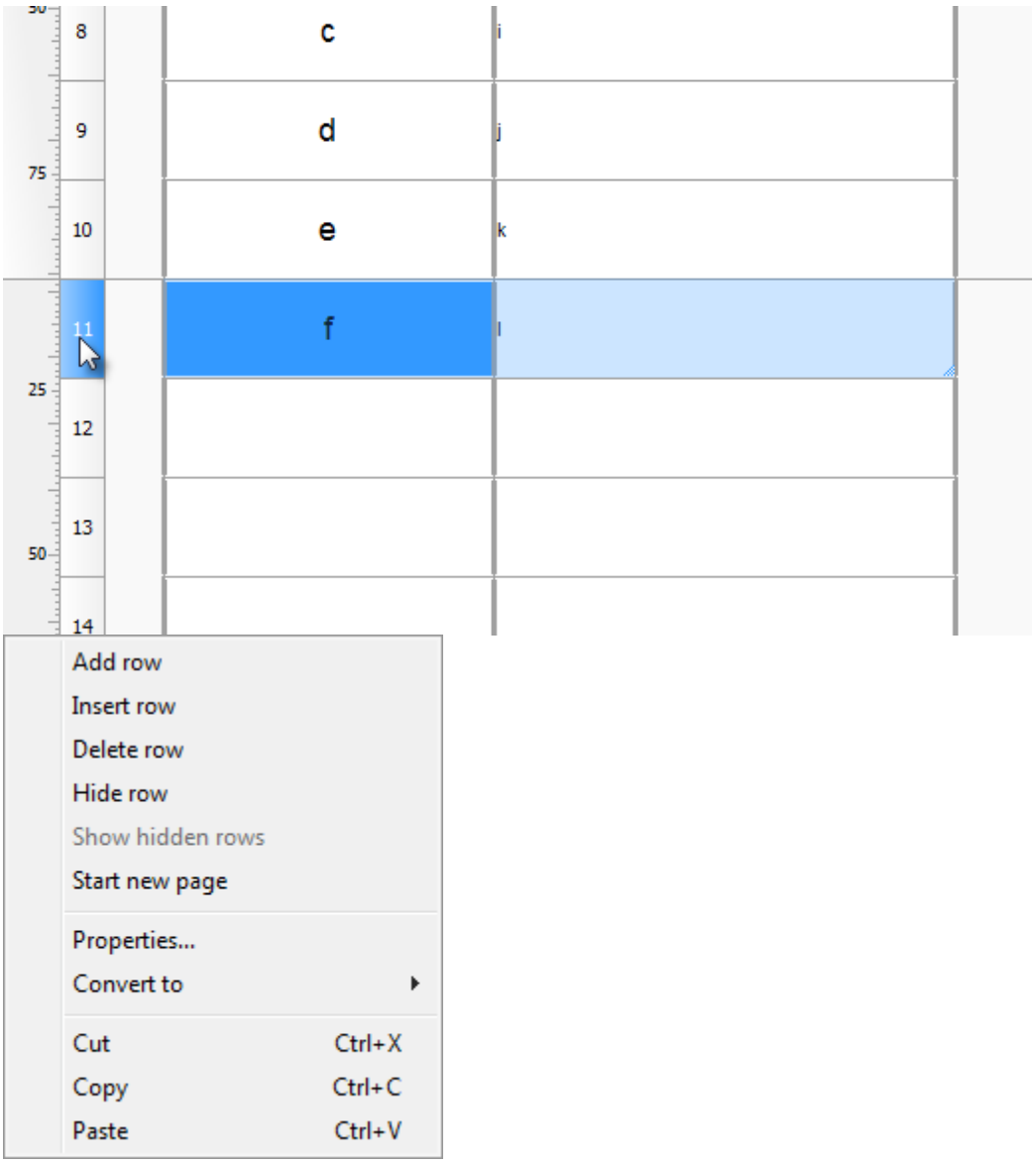
- When the command a dialog box appears *page division.*



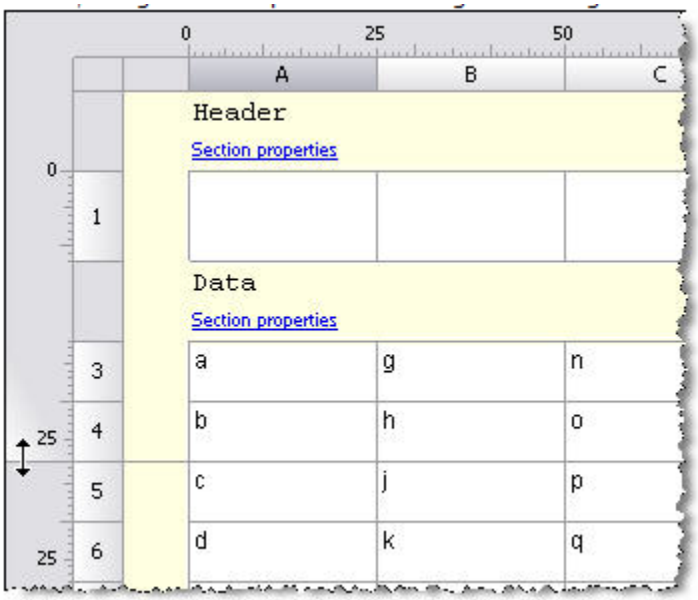
- Switch *single page* disables table division.
- Select the method of dividing the table:
    1. *Custom*- The table can divided into any parts.



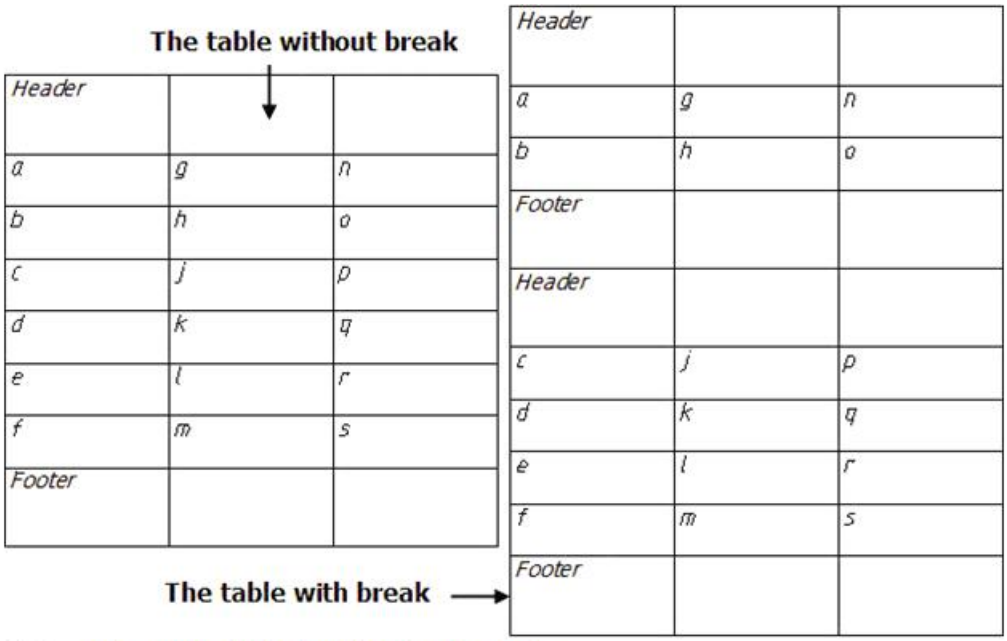
- Page break before a row is set by start new command from the context menu.

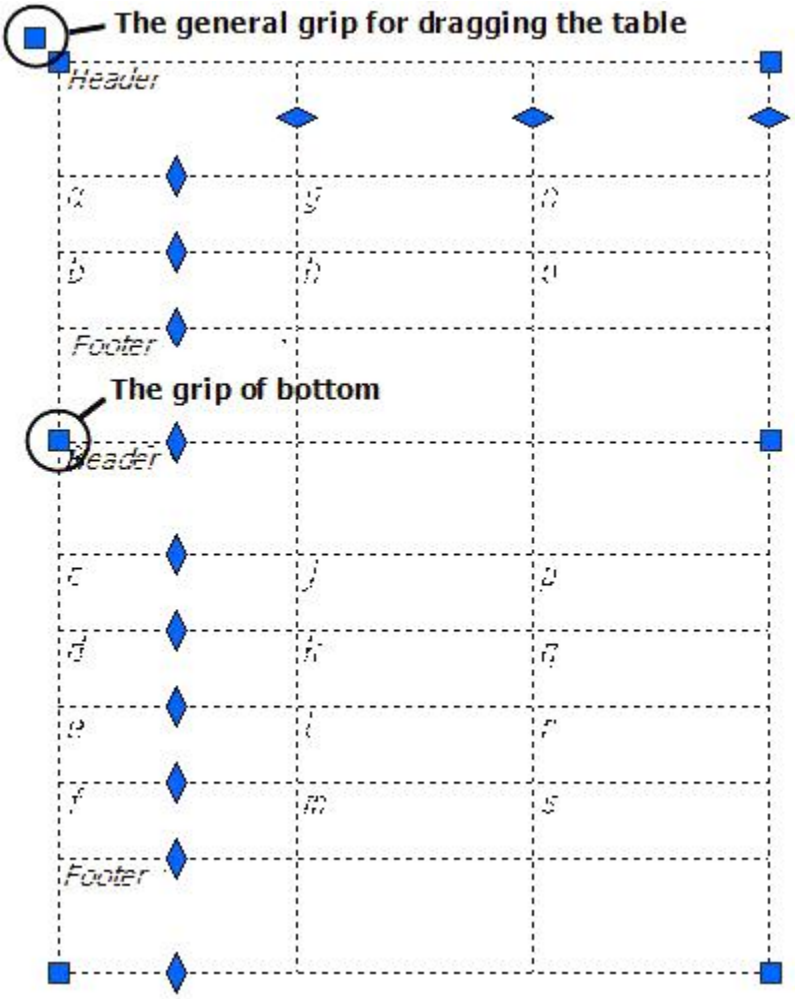


- Close the dialog. The editor of the table indicate the boundaries of the gap, with the help of it, you can change the row height.

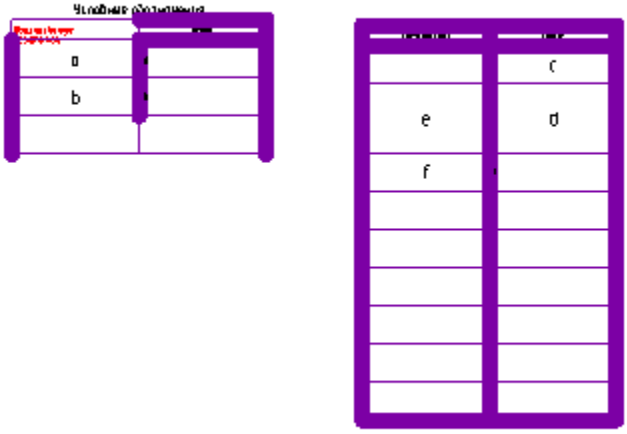


- On the drawing table with a gap will be as follows:

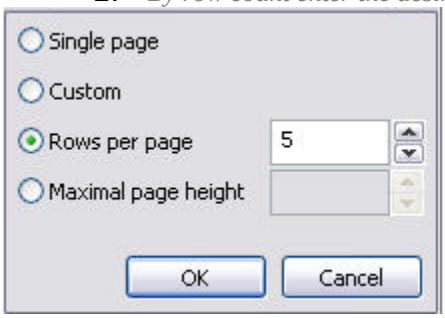

The table without break
Header
a
g
n
b
h
o
c
j
p
d
k
q
e
l
r
f
m
s
Footer
The table with break
Header
a
g
n
b
h
o
Footer
Header
c
j
p
d
k
q
e
l
r
f
m
s
Footer



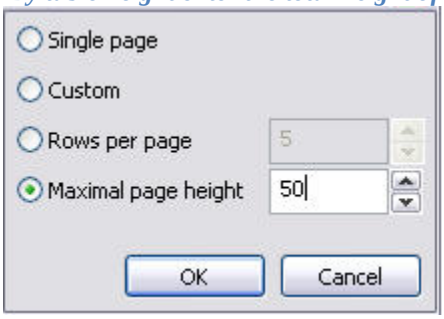
- Side of the split table can be moved in the drawing independently, double-clicking on any part of the full table editor opens.



2. *By row count enter the desired number of rows in a fragment. Headers do not count*.



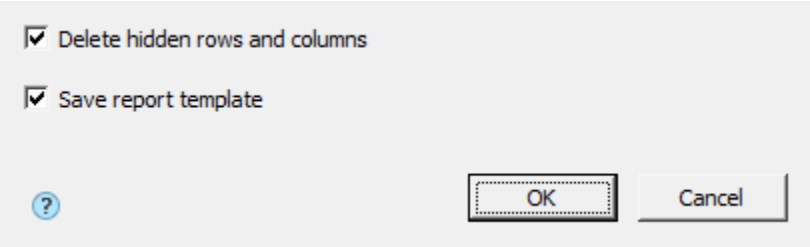
By table height enter the total height of a fragment, in millimeters, using the current scal.



- ***Dividing the table into parts allows each part tobe moved independently:***

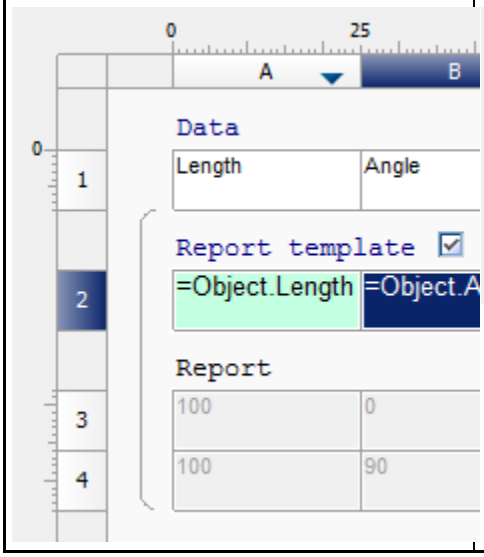
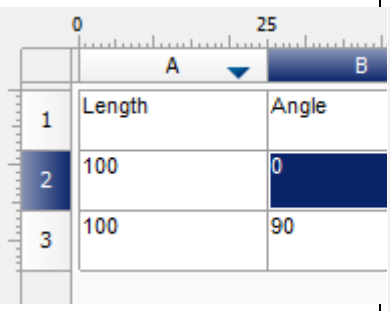
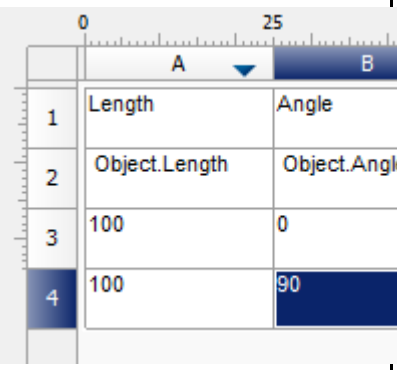
Conversion options

The *"Conversion options"* dialog is a filter that influences the set of transmitted data. Appears when converting a report to data and when exporting a table to Excel.



Converting a report to data

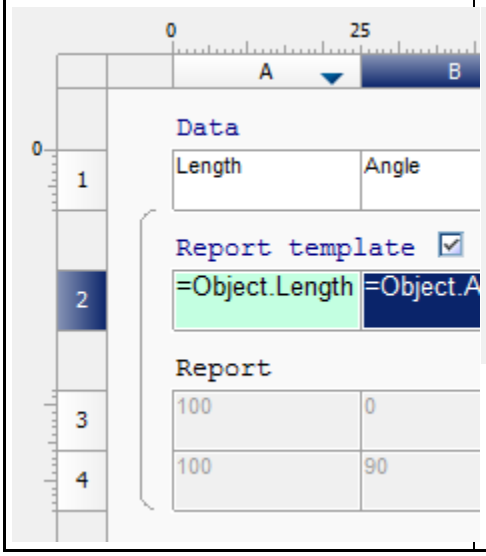
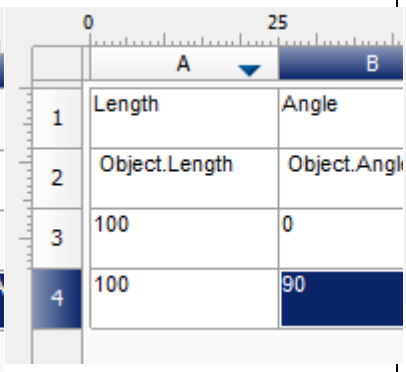
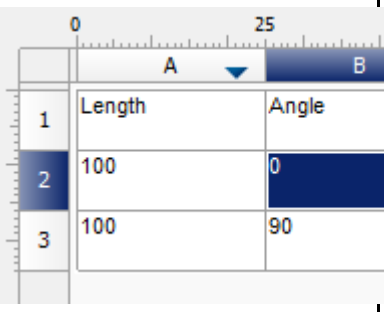
Delete hidden rows and columns - affects the conversion of hidden strings. If enabled, hidden rows are not passed to the data.

Table	Turned on	Turned off
line 4 is hidden	**no hidden lines**	**line 3 is hidden**
		

Note: The converted hidden lines remain hidden.

Save report template - affects the conversion of report template strings to data. If enabled, strings are transferred.

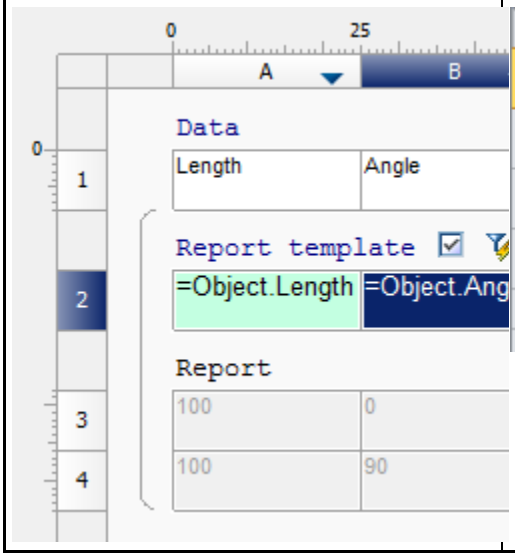
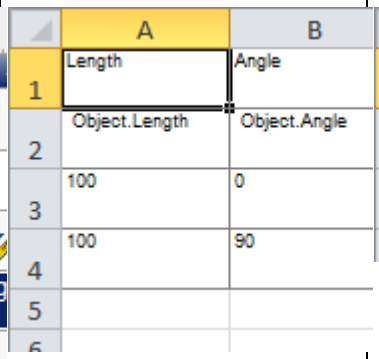
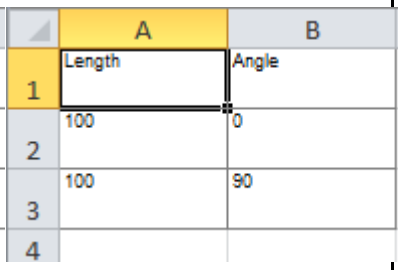
Table	Turned on	Turned off

Table	Turned on	Turned off
		

Export to Excel

Delete hidden rows and columns - the option does not affect the result, since hidden rows are not exported.

Save report template - affects the export of report template strings. If enabled, strings are transferred.

Table	Turned on	Turned off
		

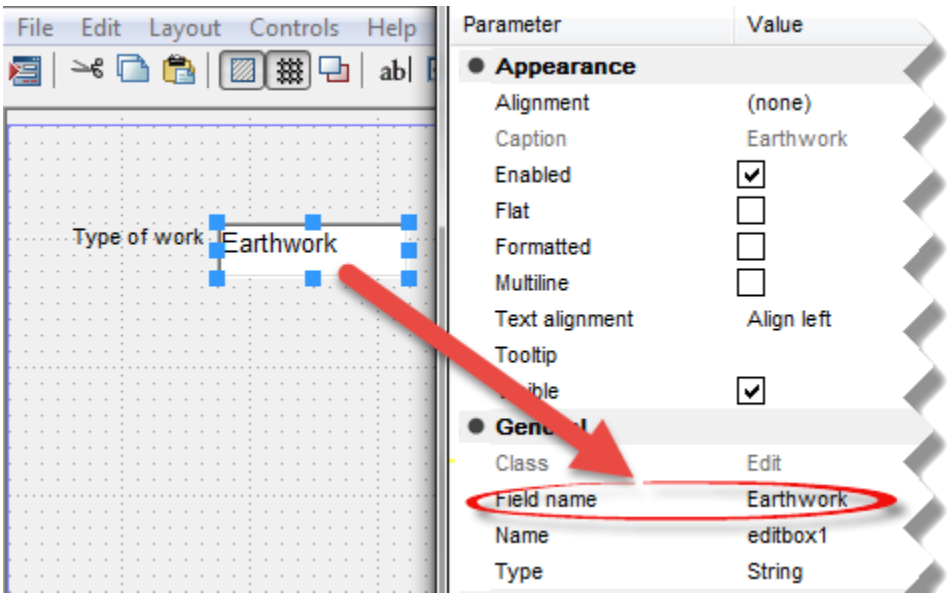
Using Variables

Important!	This function is available only in vertical applications based nanoCAD

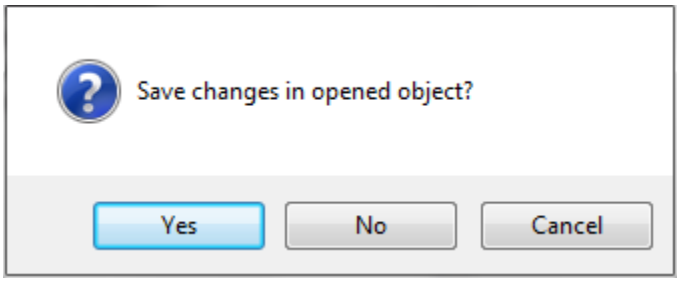
Tables BuildIndustry support the use of additional variables defined in a special dialog (form).

Procedure for creating a set of user-defined variables and their use in the table below:

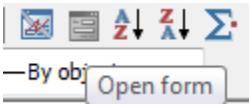
1. In *full table editor* from the menu ***Edit*** invoke the command ***Form Editor***.
2. Create a form with controls and bind each control to a field (variable). More information about the forms, see *Form Editor* .



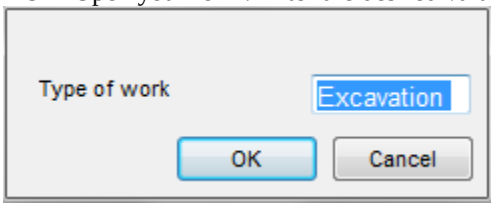
3. Close the form editor and save the form.



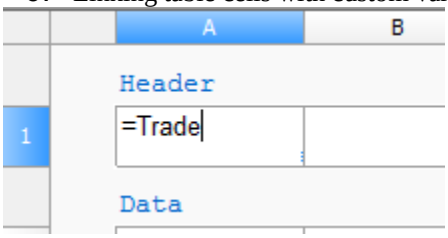
4. To set the variable, call the command ***form***.



5. Open your form. Enter the desired values. Click ***OK***.



6. Linking table cells with custom variables by using *formulas*.



Cell value will be associated with the variable.

Variables can also be used in complex formulations.

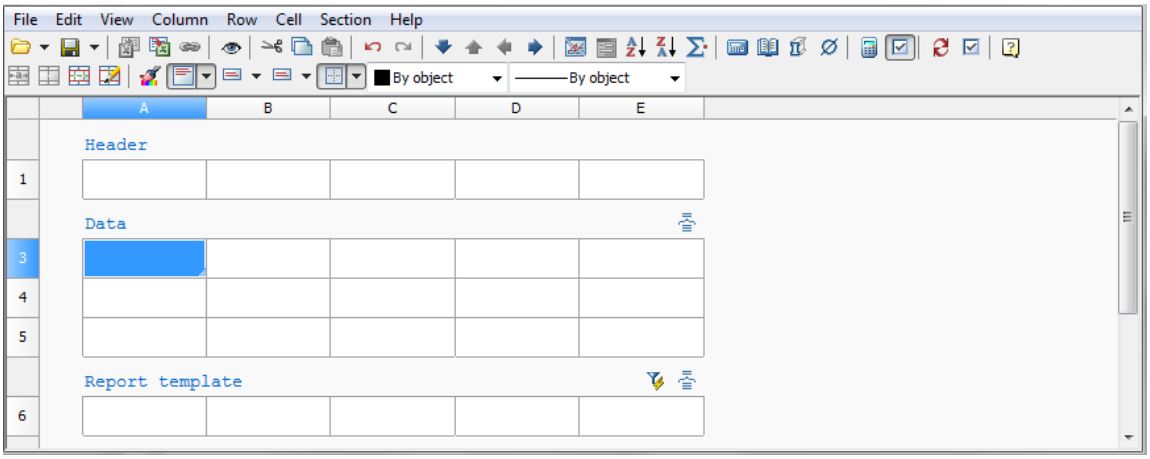
Create reports

Reports are necessary in order to organize the data of objects in the drawing.
Recommended report structure:

- Header of first page;
- Header of following page;
- Header of last page;
- Report title;
- Report template;
- Report;
- Report summary;
- Footer of first page;
- Footer of following page;
- Footer of last page;

Generating a report

To create a report in the table editor, call the command from *"Main menu - Section - Insert Report section"*. The command will create a subsection *"Report template"*.
Additional subsections are added using the *"Report template"* menu: *"Report title"*, *"Report Summary"*.
Header and footer sections are added from "Main menu - Section".

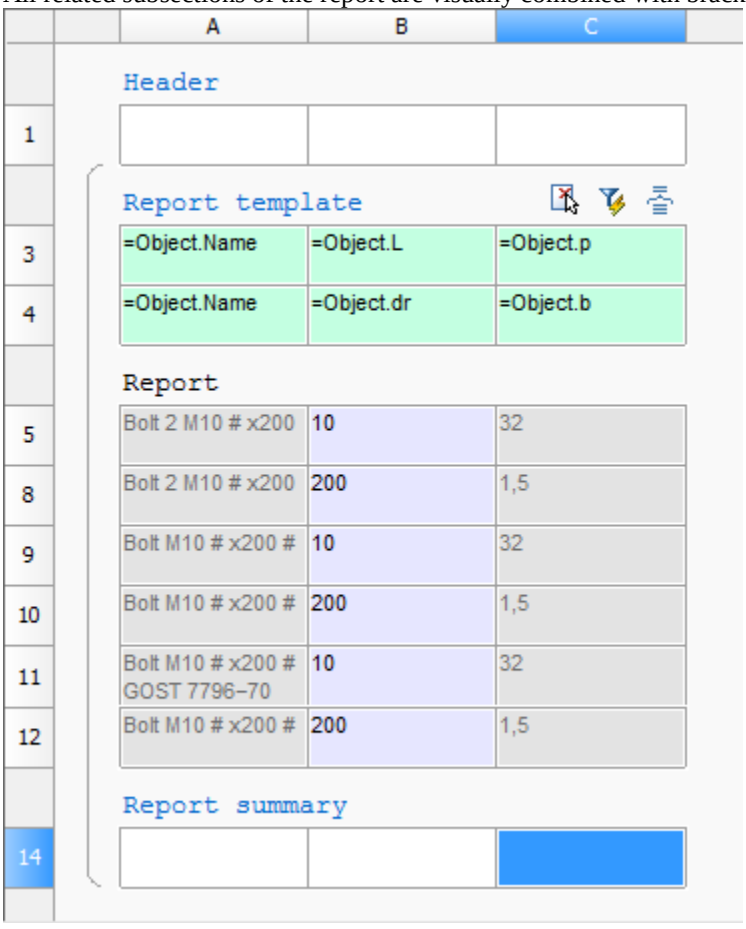


Report template

The report template defines the content and appearance of the report; it can consist of one or several lines. A separate copy of the report lines is created for each object. Formulas in the report are calculated using the properties of the selection objects.
The following commands are available in the "Report template" menu (click on the subsection name):

- *"Object filter..."* - opens the *"Quick Selection"* dialog for forming a selection of drawing objects. The command is duplicated by the *"Select source objects"* button on the right in the *"Report template"* section.
- *"Merging and Grouping..."* - opens the "Row grouping and merging" dialog in which the parameters of grouping and merging table cells are configured. The command is duplicated by the *"Group and merge"* button on the right in the *"Report template"* section.
- *"Freeze report"* - the switch controls the report refresh mode. If the checkbox is selected, the update is performed manually, and the *"Update report"* command becomes available.
- *"Update report"* - refreshes the values of report objects, the command is available in manual refresh mode.
- *"Report header"* - adds a *"Report Title"* subsection to the beginning of the report, displayed at the beginning of each part of the table.
- *"Append Data section"* - adds a data section below the report. If the section already exists, an additional line is appended.
- *"Append Report section"* - adds a new report below the current one.
- *"Remove Report"* - deletes the report with all associated subsections.
- *"Rebuild"* - completely rebuilds the report based on the report template. Data entered manually in the *"Report"* section will be overwritten.
- *"Convert to Data section"* - the command converts sections of the report to data. Blank lines are not converted. Calling the command opens the *"Conversion options"* dialog.
- *"Report footer"* - Adds the *"Report Summary"* subsection to the end of the report, displayed after each part of the table.

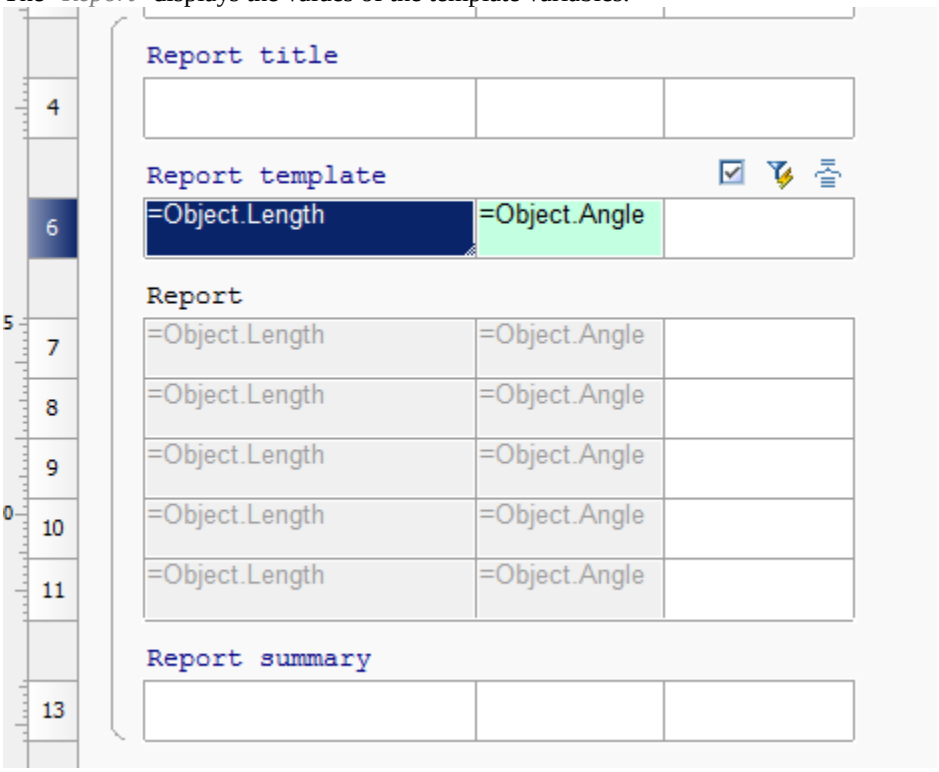
All related subsections of the report are visually combined with brackets.



Important!	The report template has absolute priority over custom changes. After the refresh, the report colored by the user will become the same as the specified report template. Automatic report lines corresponding to the collected objects will be automatically formatted strictly according to the report template.

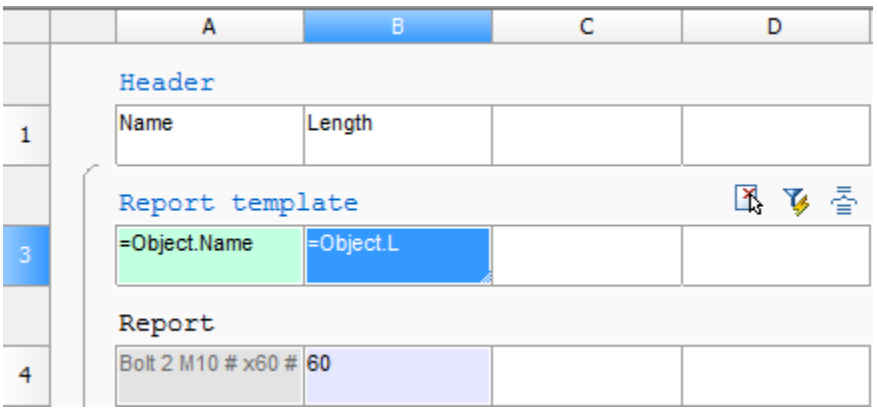
Report

The *"Report"* displays the values of the template variables.

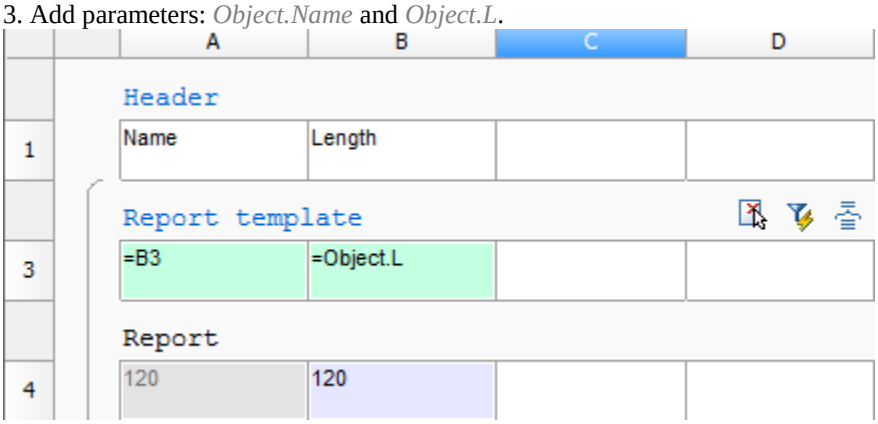


The report contains cells (with a lilac background) containing object data, marked with a special color. Selecting cells highlights the corresponding objects in the drawing.
When editing such cells, the changes will be applied to the very object to which they refer. For instance:
1. Take a Bolt GOST 7795-70 with a length of 50 mm and place a leader on it.

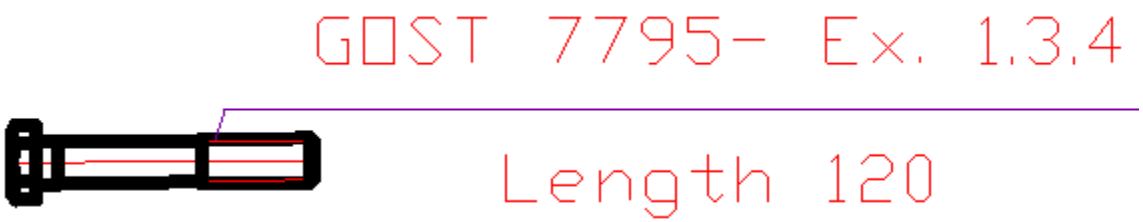
2. Create table, report template and tie to bolt (parametric object).



3. Add parameters: *Object.Name* and *Object.L*.



4. Enter the new length, for example 120. Close the table editor.
5. The length of the bolt in the drawing has changed.



Report title

The following commands are available in the *"Report Title"* menu (click on the subsection name):

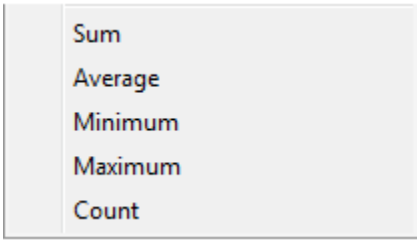
- *"Remove section"* - deletes the *"Report Title"* section.

Report summary

The following commands are available in the *"Report Summary"* menu (click on the subsection name):

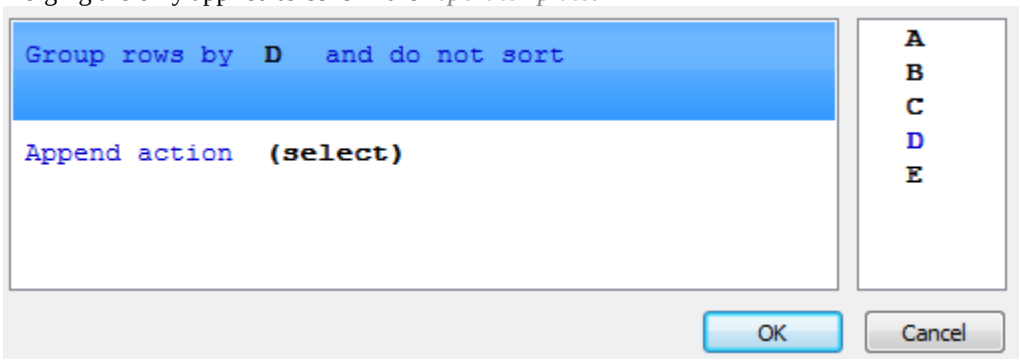
- *"Remove section"* - deletes the *"Report Summary"* section.

In the context menu of the *"Report Summary"* cells, there are the following additional commands: "Sum", "Average", "Minimum", "Maximum", "Count".
The commands insert into the selected cell corresponding to the name of the expression. The action is applied to a fragment of a column within one section.

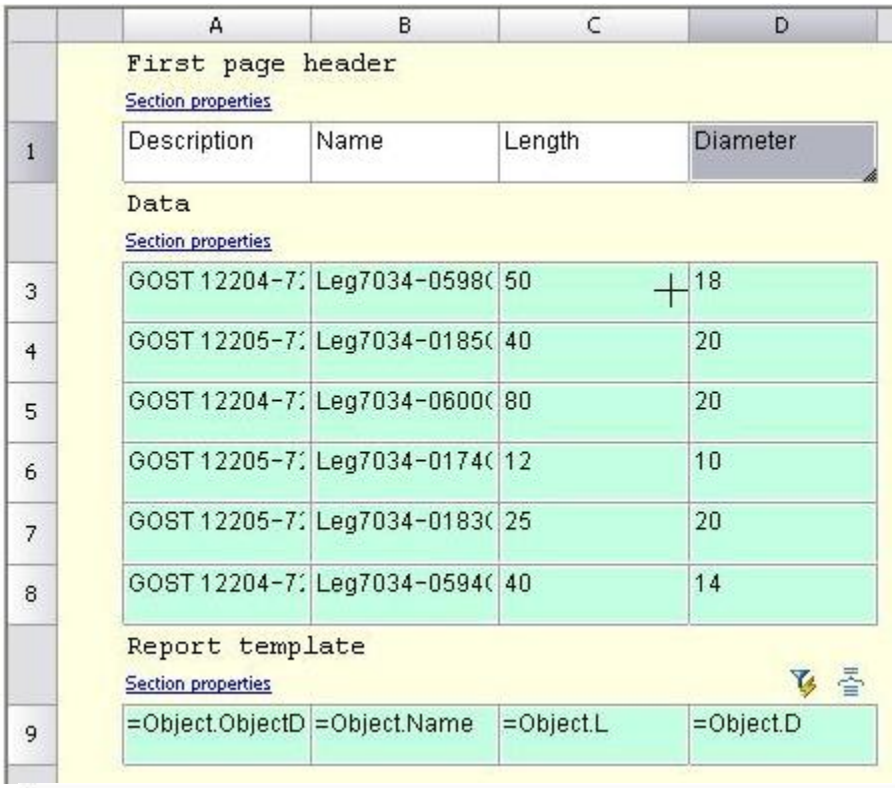


Grouping and merging cells

Button ***Grouping and merging***. Report template.
In the *Grouping and merging* dialog you can specify settings for grouping and merging table cells. Grouping and merging are only applied to cells in the *report template*.



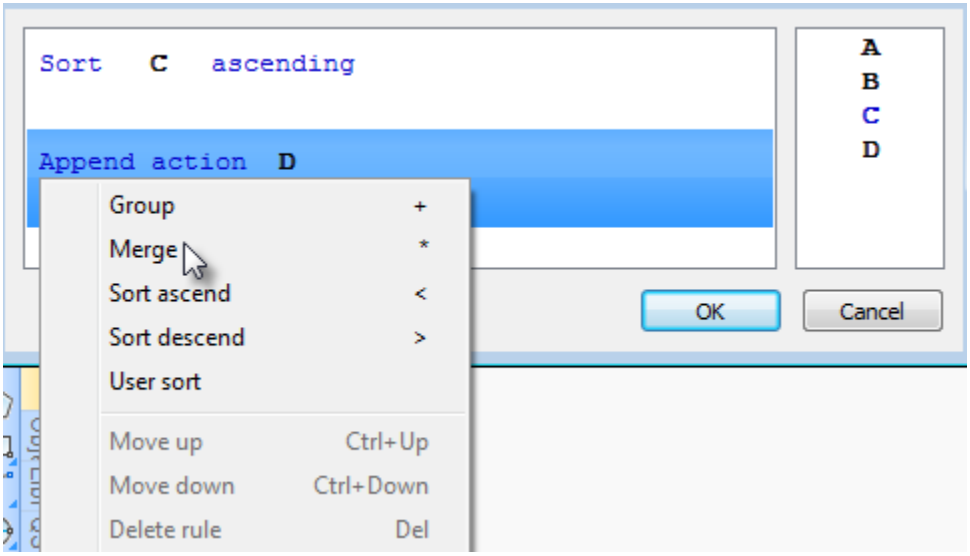
Study the example for how to merge cells.



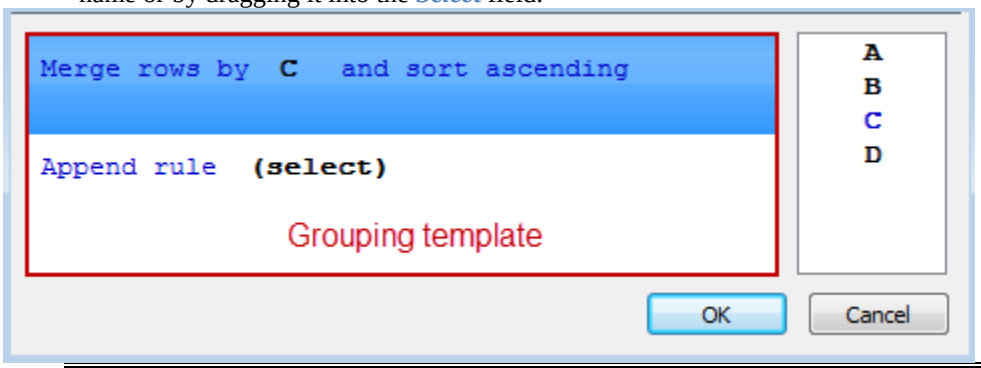
Description	Name	Length	Diameter
GOST 12204-72	Leg7034-0598GOST 12204-72	50	18
GOST 12205-72	Leg7034-0185GOST 12205-72	40	20
GOST 12204-72	Leg7034-0600GOST 12204-72	80	20
GOST 12205-72	Leg7034-0174GOST 12205-72	12	10
GOST 12205-72	Leg7034-0183GOST 12205-72	25	20
GOST 12204-72	Leg7034-0594GOST 12204-72	40	14

Merging

1. In the ***Grouping and merging*** dialog box select ***Append rule*** and select ***Merge***.



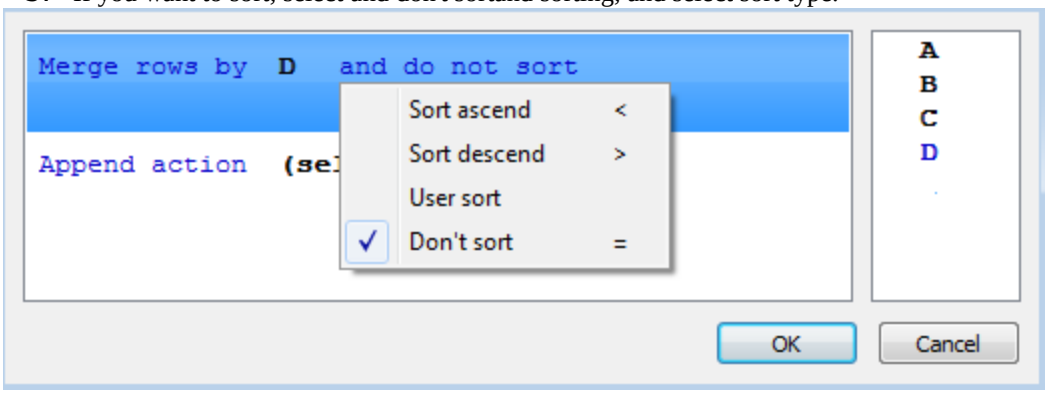
2. Select the column name for the cells you want to merge. Selection is made by double clicking on the column name or by dragging it into the ***Select*** field.



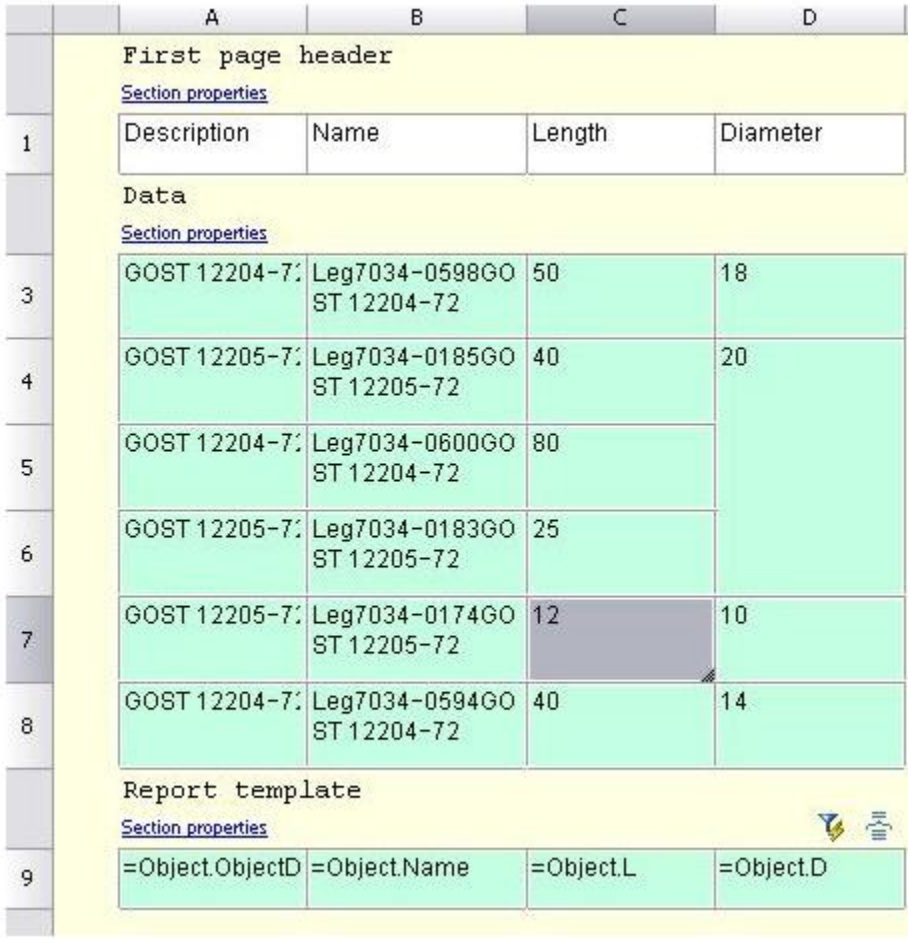
Note:	To select another column, you must drag it into the column list and then select it.You can select several columns then place them in the required order.

The columns are processed according to the rules of the ***grouping remplate***.

3. If you want to sort, select and don't sortand sorting, and select sort type.

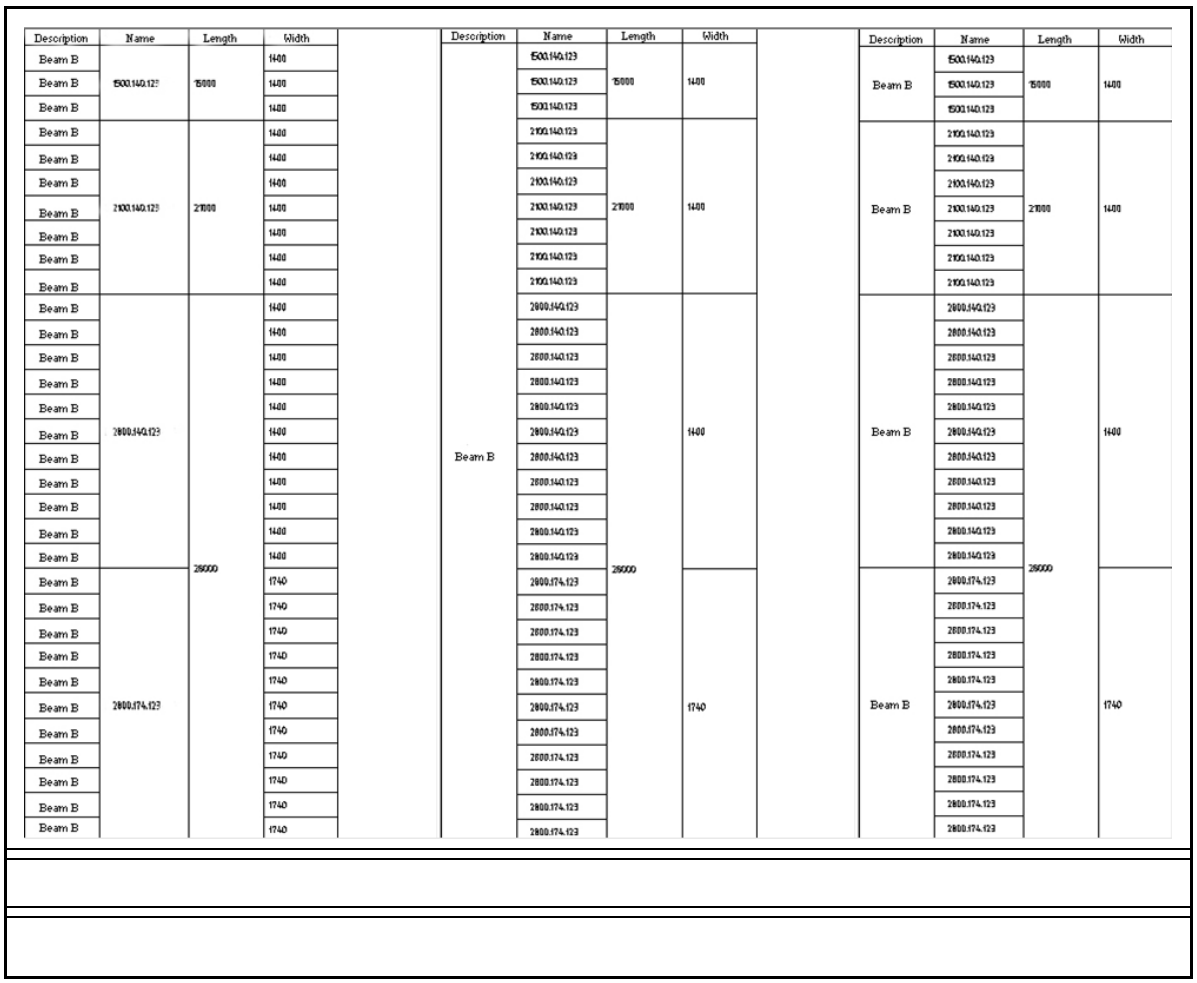


4. Click *OK* to check the result in the table editor. Cells having identical values in the D column will be merged.



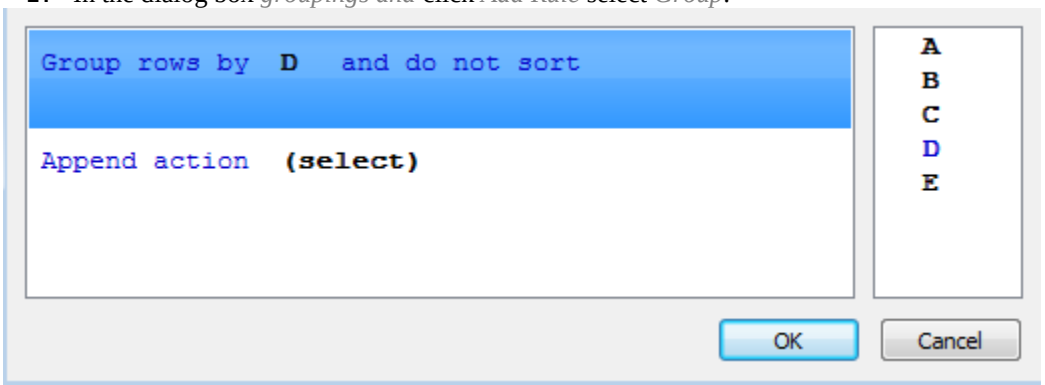
	A	B	C	D
	First page header			
	Section properties			
1	Description	Name	Length	Diameter
	Data			
	Section properties			
3	GOST 12204-7:	Leg7034-0598GOST 12204-72	50	18
4	GOST 12205-7:	Leg7034-0185GOST 12205-72	40	20
5	GOST 12204-7:	Leg7034-0600GOST 12204-72	80	
6	GOST 12205-7:	Leg7034-0183GOST 12205-72	25	
7	GOST 12205-7:	Leg7034-0174GOST 12205-72	12	10
8	GOST 12204-7:	Leg7034-0594GOST 12204-72	40	14
	Report template			
	Section properties			
9	=Object.ObjectD	=Object.Name	=Object.L	=Object.D

C, B Rows, which have identical values in B and identical values in C columns, are merged. The order of specifying columns is unimportant.	A C, D Rows, which have identical values in A are merged, then cells in C and D are merged. The order of specifying columns is unimportant.	C, D A Rows, which have identical values in C, are merged, then cells in D and A are merged. The order of specifying columns is unimportant.

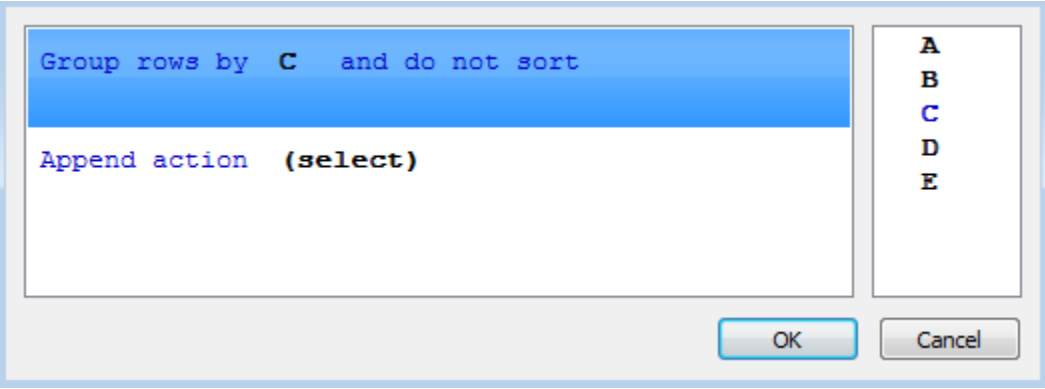


Grouping

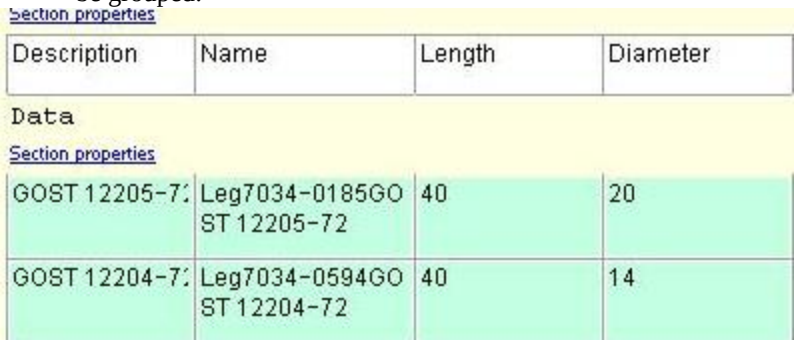
1. In the dialog box *groupings and* click *Add Rule* select *Group*.



2. Select the name of the column whose cells you want to group. Choice is either by double-clicking on the column name, or by dragging it into the *Select* .



3. If you want to sort, click on the link and sorting, and select the type of sort.
4. Click *OK* to view the results in the table editor. Table rows that contain the same values ??in column C, will be grouped.



Section properties

Description	Name	Length	Diameter
Data			
Section properties			
GOST 12205-7:	Leg7034-0185GOST 12205-72	40	20
GOST 12204-7:	Leg7034-0594GOST 12204-72	40	14

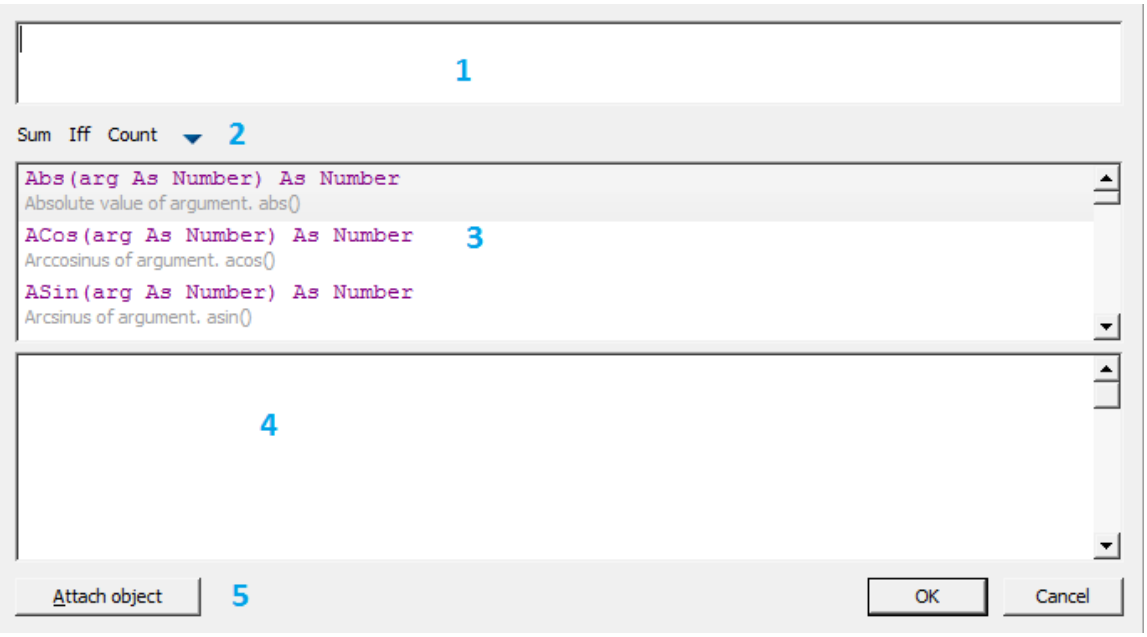
Interface Expression Editor

The Formula Editor (Expression Builder) allows you to set parameters, arithmetic expressions, and object property references for a selected table cell.
The formula editor is invoked:

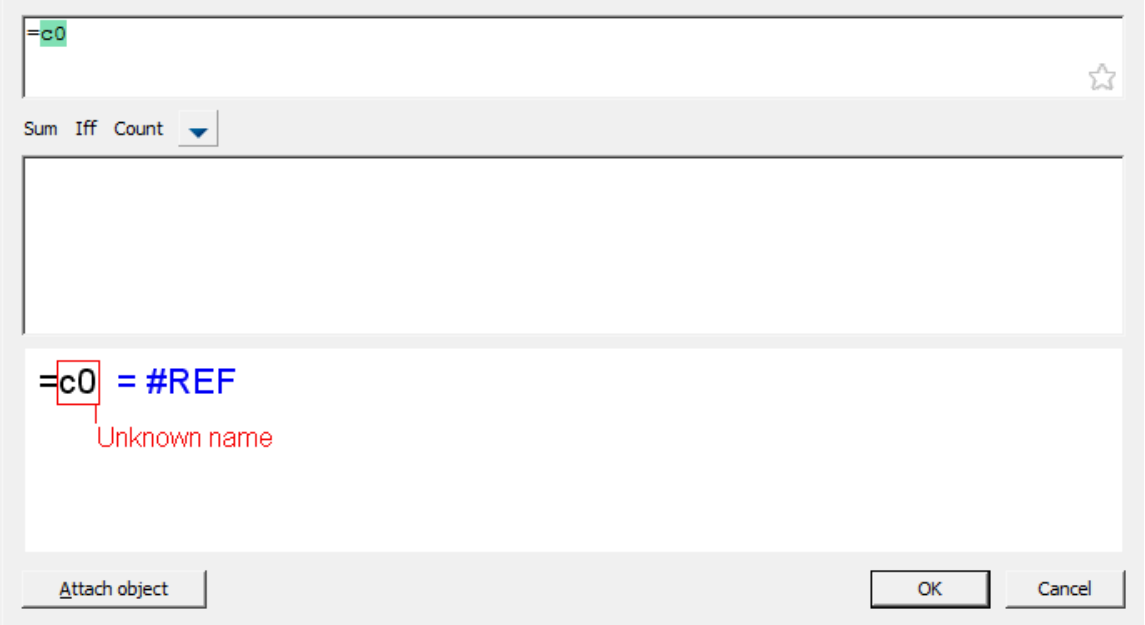
- By clicking on the *"fx"* button in the formula bar;
- By pressing Shift + F2;
- Right-click upward;
- From the context menu command *"Expression .."*.

Dialog

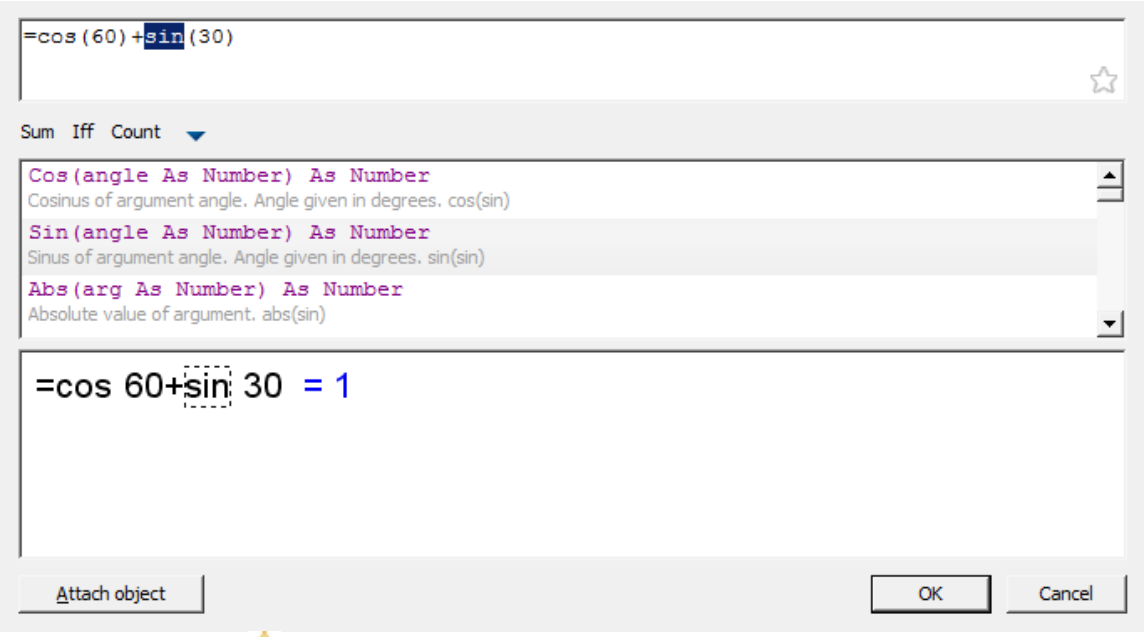
The dialog box consists of the *"Expression Input Field"*, *"Formula Templates"*, *"List of Registered Functions"*, *"Quick View Fields"* and the *"Attach object"* button.



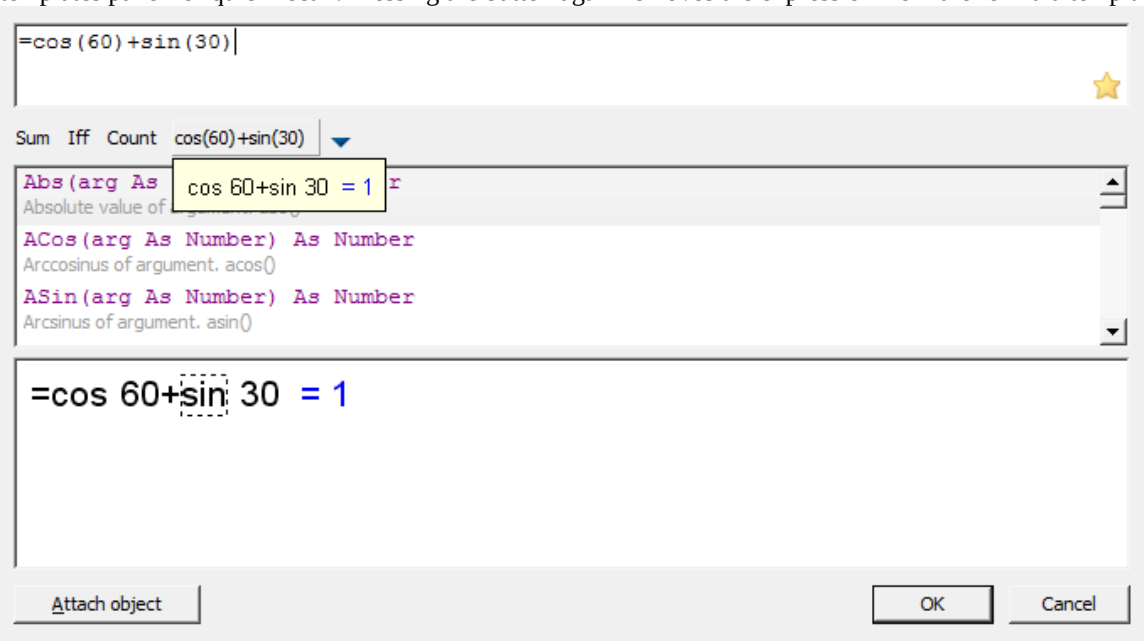
The "Attach Object" button allows you to attach to objects.
At the top of the dialog there is an expression input field, with which you can manually enter any arithmetic expressions, as well as use reserved variables and object properties.
When entering manually, a list of functions and variables containing the word being entered appears in the list of reserved functions; syntax is also checked. If the formula is incorrect, an error warning or hint will appear in the Quick View.



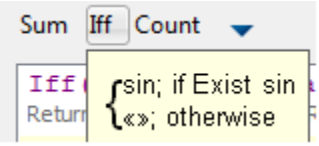
In the quick view field, by clicking the mouse on an expression, the text in the input field related to this expression is selected.



When you click on the ☆ button in the *"Expression input field"*, the entered expression is saved on the formula templates panel for quick recall. Pressing the button again removes the expression from the formula template.



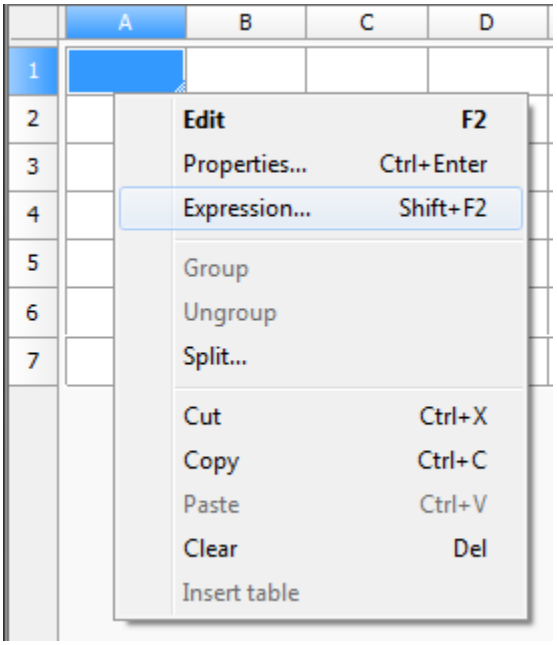
The *"Reserved Functions List"* contains recently used expressions that are highlighted in yellow. When you click on the ☆ button, expressions are saved in the Formula Templates panel for quick recall. Expressions that are in the Formula Template Bar in the Reserved Functions list will be marked with an asterisk. Pressing the ☆ button again removes the expression from the formula template.
When you hover the cursor over the button of the saved template, a tooltip will appear with the contents of this template.



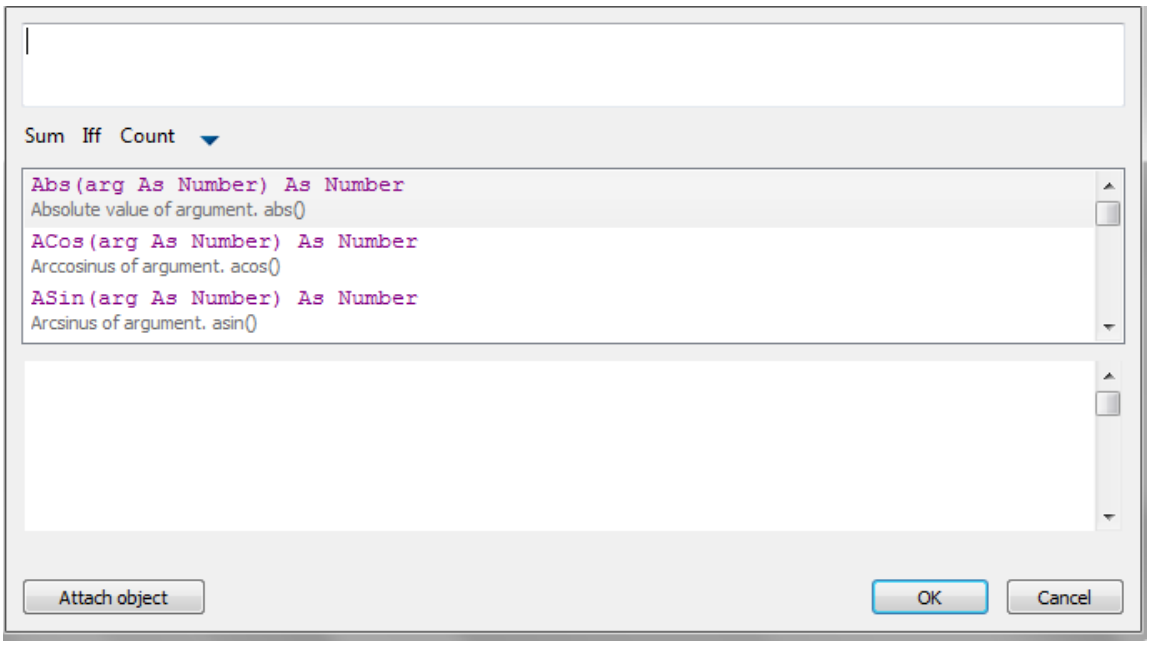
In order to insert an expression from the *"Formula template"*, place the cursor in the required position, press the button with the template.
To delete an expression template, click the triangle button and uncheck the required template in the list that appears.

Binding to a table cell

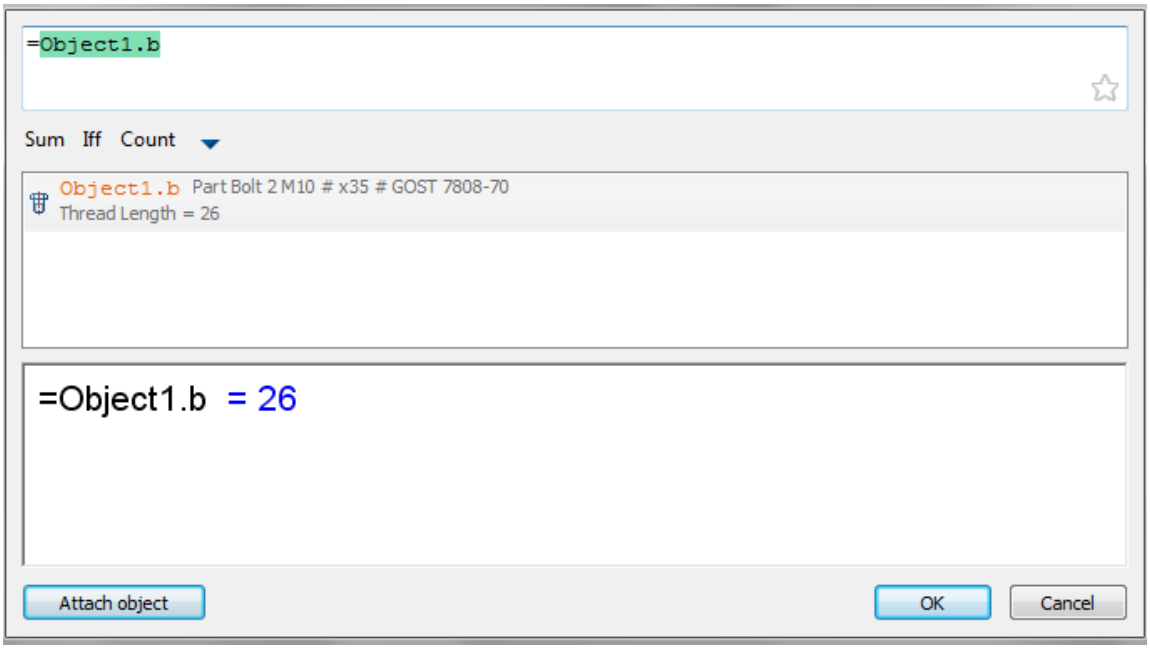
In the formula, you can use any cell values of the properties of another object. If using automatic recalculation of the table, the formula object is automatically recalculated. Each cell can attach one or more objects. Objects are named Object1, Object2, and Object3, … sequentially numbered in the table. If the object is not used in a formula, it will detach from the table at the next calculation, and the object references are renumbered.
Binding properties for drawing objects to a single cell then use the ***expression*** or the keys *Shift+F2*. Available in the context menu of the selected cell.



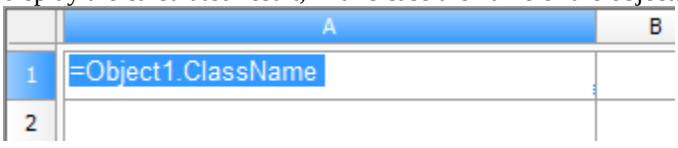
After selecting the command window opens *the Expression Builder*.



Click *Attach it*.
Select the object whose data you want to send to the table (eg, Bolt M52). Now the list of expressions appears in the properties of the selected object. Double-clicking on the property (select Object.Name), adds it to the text in a cell.
Click ***OK***.



After connection of the facility to the cell the cell color changes (it says that the formula in the cell) and it will display the calculated result, in this case the name of the object:



Functions in the Formula Editor

Mathematical operations:

+	Addition
-	Subtraction
*	Multiplication
/	Division
^	Involution
sqrt()	Rootsquaring

Logical comparison and selection:

==	equally
>	more
<	less
!	not
|	or
!=	not equal
>=	not less
<=	no more
||	or
&&	and

Functions:

math:

Cos, Sin, Tg - trigonometric functions, the argument given in degrees.
Acos, Asin, Atg - inverse trigonometric functions, the result is given in degrees.
Abs - absolute value of (module).
Int - rounding to whole number.
Summ - returns the sum of the values of the variables in a grouped cells.

When you type a function should pay attention to the opening and closing brackets.

For example:
Summ(Row) - returns the sum of the numbers of grouped rows..
Summ(Standartnaya_detal.L) - returns the sum of the values ??of the L parameter object "standard part" in the grouped rows.
data conversion:
Str - Data conversion to a string type.
Num - conversion of data into a numeric type.
Frm - convert a numeric value to a string formatted according to the settings of a table column.
For example:

Frm(0.001230) returns a string 0,0012, if the column containing the cell is set to zero suppression and accuracy of 0,0000
selection and comparison:
FmtText - Addition formatted strings.
FmtSub - the creation of a subscript.
FmtSuper - create superscript.
FmtDigit - transfer number in hard copy.
FmtRaw - text without formatting.
DmtDiv - creation of fractions.
If/Iff - The logical choice. Recording Format:
if (Logich_Uslovie; Esli_Istina; Esli_Lozh)
where:
Logich_Uslovie - logical condition using logical comparisons (see above);
Esli_istina - the return value when the logical condition;
Esli_Lozh - return value at default logical condition.
For example:
if(object == Marker_universalny; Marker_universalny.position; "Undefined") . If an object type, which is connected with the cell column has Marker_universalny value, the function returns the position value of the referenced object. Another type of object it returns a string *Undefined* .
Exist - Checks whether there is a constant: =IF(EXIST(Object.Name);Object.Name;0)
Min/Max - returns the minimum / maximum value listed in parentheses
For example:
Min(*maskimalnoe_znachenie; MinValue*) - nourishes *MinValue;*
Max(*maskimalnoe_znachenie; MinValue*) - returns *maskimalnoe_znachenie.*
Count - returns the number of objects (for grouped rows in the table).
For example:
Count() - returns the number of objects associated with cells grouped rows.
Avg - Calculates the arithmetic mean of the arguments. Takes an arbitrary number of arguments understand ranges. Empty arguments are ignored.
Off - returns the value of the cell, given the relative index. The index is written in the format:
Off(string; column)
For example:
Off(-1;2) - returns the value of cell in the table on one line above (-1) and two columns to the right (2) of the current
Cell - the cell returns the value specified absolute index.
Merge (Cells,Expression) - brings together a range of Cells, if Expression is not equal to 0. Returns the value of Expression.
For example:
=merge(A5:C5; "Title") – integrates cell A5 to C5, in the summary displays the text "Title".
SetHeight() – reference line height. In brackets the height value.
Val() - computes the value of the argument.
For example:
=val("A"+"1") - calculates the "A1", and then uses it as another expression (obtained by the value in cell A1). If the argument - not a string, then he comes back, ie =val(10+2) - the same thing =10+2.
=val("summ(A"+Str(off(0;-1))+":D"+Str(off(0;-1))+")") - the amount of cells from rows A to D, the number which is introduced into the cell to the left of the current one. In most cases you can do without val. The classic example, when without it - no way: in a custom form, enter the cell address, then to take the value out of it. Create a variable Addr, associate it with an input field on a form and in the table do so: = val (Addr) function allows recursion: = val (val ("A" + "1")) - to take the value of the cell whose address is recorded in cell A1. Nesting depth is limited to 64.
Geometry(Object) - works as ***Implement object***. Height embedded object shrinks to the line height. Object is taken from the report, or attached to the cell.
SUMM(Section(-1)) - Sum of cells in the current column of the section next to the current.
COUNT(Section(A3)) - The number of rows in the section containing the cell A3.

Reserved variables:

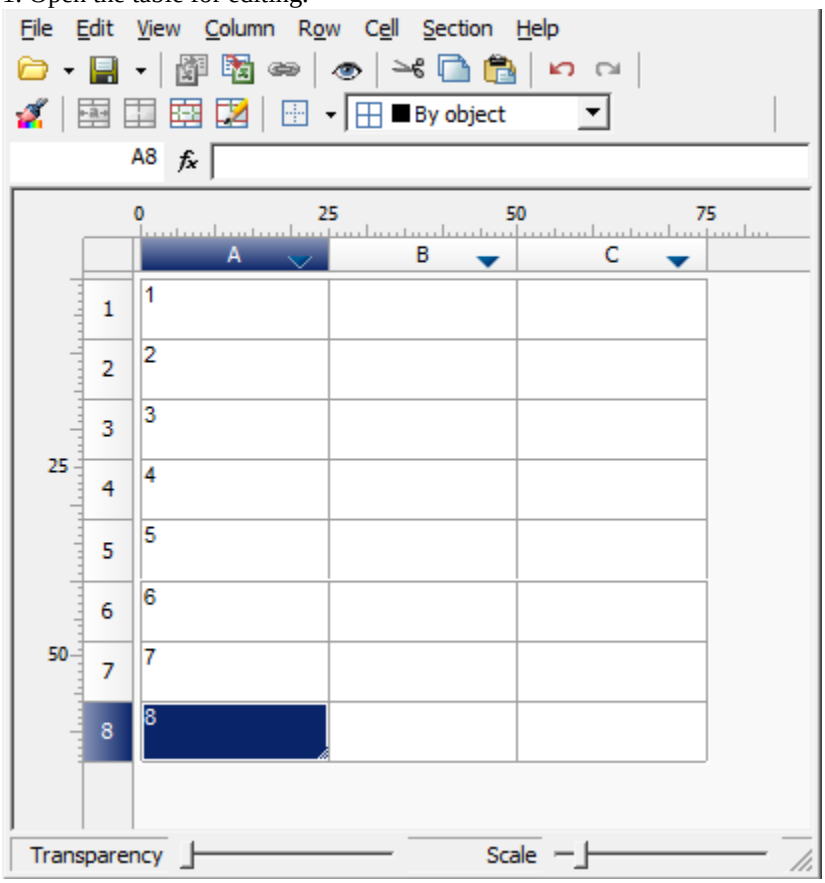
Pi - pi.
Row - for each cell in the column it returns the number of rows (rows are numbered starting with 1, excluding the header lines).
Col - for all cells in a column returns the column number of the table (column "A" is number 1).
Object - returns the type of the object, which is associated with the cell line.
Object1, Object2, … - objects attached to the cell.
Title - the name of the table.
RowCount - total number of rows.
ColCount - the total number of columns.

Split the table by pages

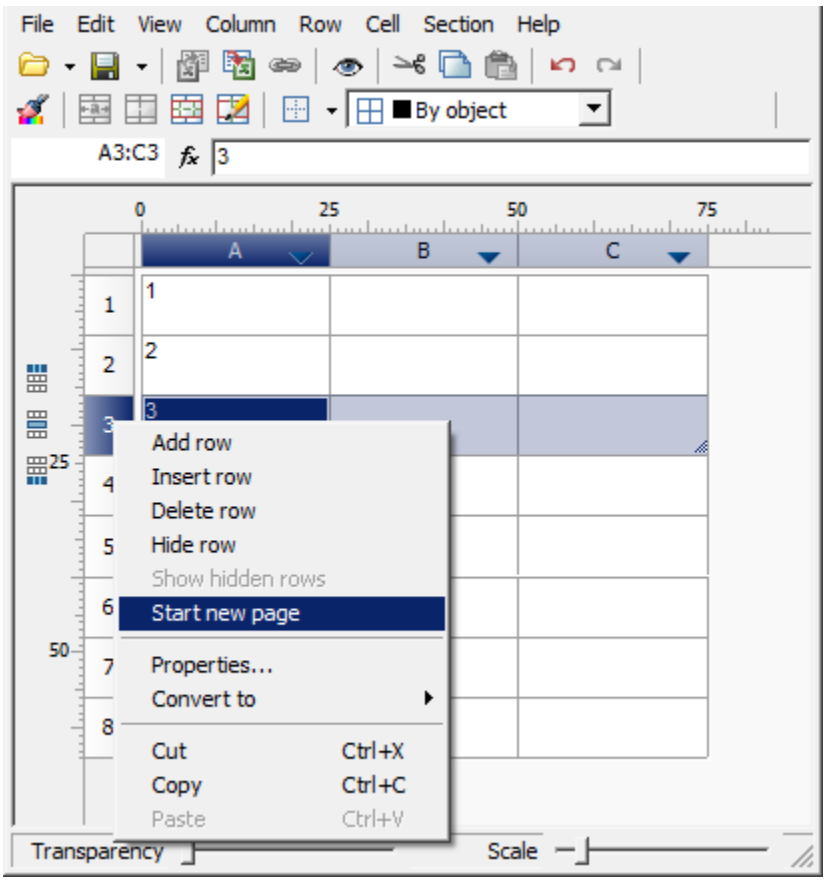
The *"Split the table by pages"* tool is similar to the "Division by page" editor tool, but the result is not a split single table, but several separate tables.

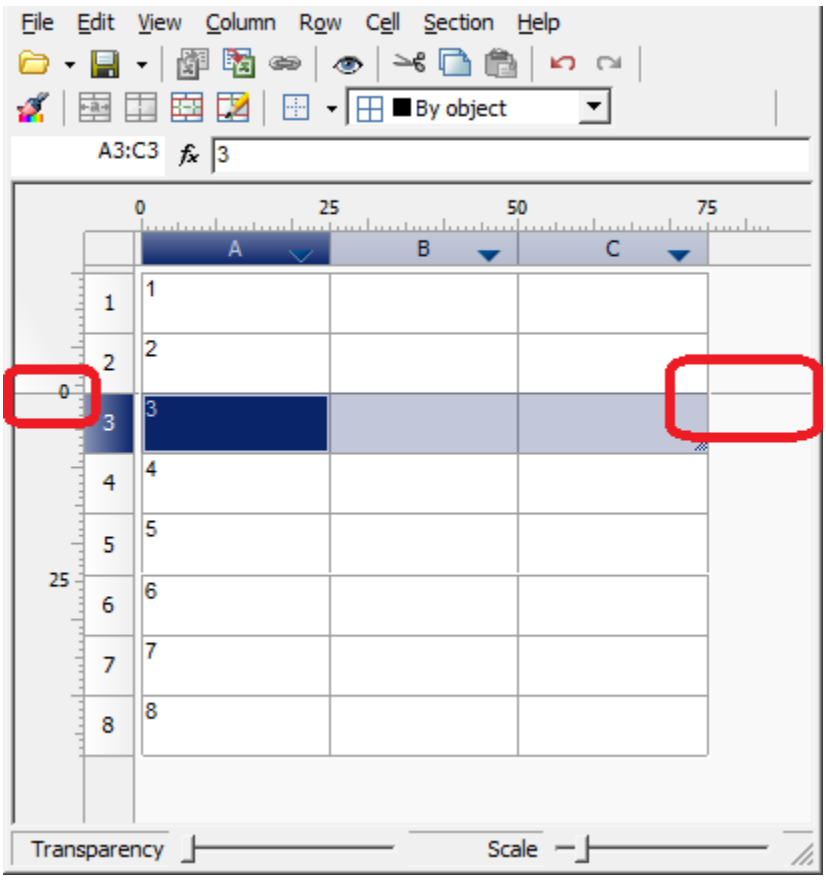
Procedure

1. Open the table for editing.

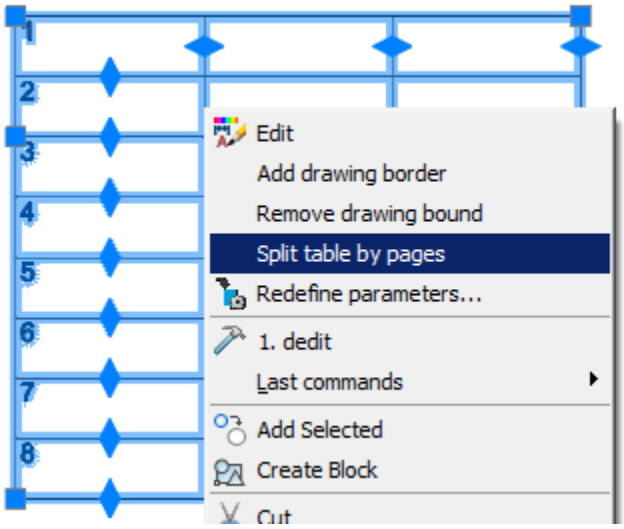


2. In the places where the break will be, set the break with the command *"Start new page"*.

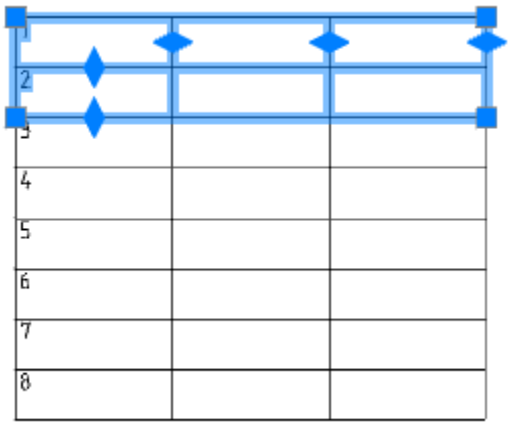

File Edit View Column Row Cell Section Help
By object
A3:C3
3
0 25 50 75
A B C
1
2
3
Add row
Insert row
Delete row
Hide row
Show hidden rows
Start new page
Properties...
Convert to
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Transparency
Scale



3. Close the editor.
4. Select a table.
5. From the context menu, select the *"Split table by pages"* command.



6. The table will be split into separate tables. The separation will occur at the places of the established breaks.



1		
2		

3		
4		
5		
6		
7		
8		

Note:	If the command is applied to a report, the following destructive actions are performed: 1. The report is converted into static tables. 2. All connections with objects are broken. 3. All formulas are removed, and the cells begin to display the previously formed text. 4. Rows with report titles are hidden so that the display of the table in the drawing is not changed.

Dimensions

Dimension Style Manager

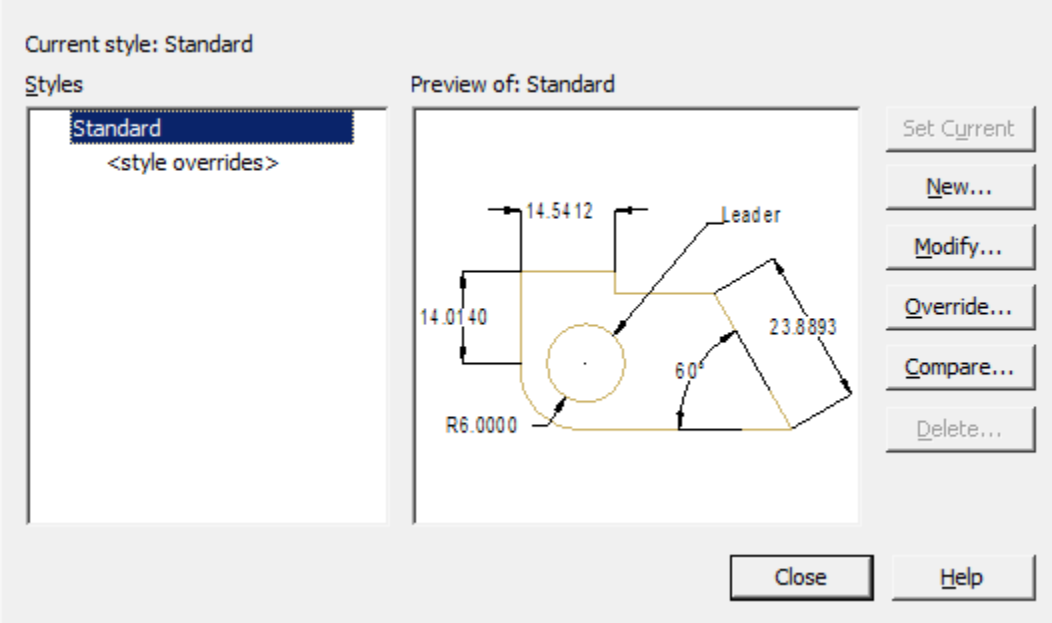
Main menu: ***Format -*** ***Dimension Styles...***.

Main menu: ***Dimensions -*** ***Dimension Styles...***.

Ribbon: ***Home - Annotation -*** ***Dimension Style Manager***.

Toolbar: ***Settings -*** ***Dimension Style Manager***.

Command line: ***DIMSTYLE***.
The software defaults to a government industry standard dimension style.
Dimension styles allow you to control the appearance of dimensions and quickly format them.



Dialog box composition

The inscription *"Current style: ..."* - information about the current dimension style, which is used when applying dimensions.
"Styles" list - a list of all dimension styles in the document.
The *"Preview of: ..."* window is a graphical representation of the style selected in the *"Styles"* list.
Buttons:

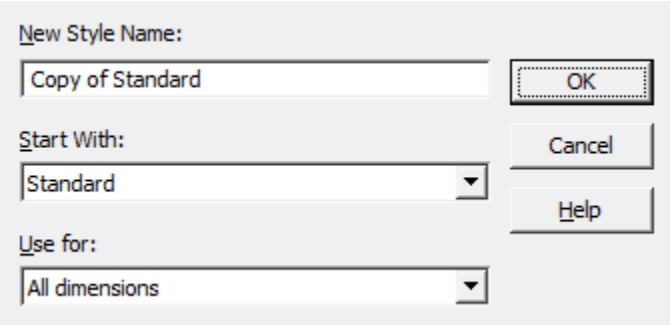
- *Set current* – sets the selected dimension style current.
- *New...* – opens a dialogue *"Creating style"*.
- *Modify...* – change the parameters of the style selected in the list. Opens dialog *"Modify dimension style"*.
- *Override...* – change the parameters of the current dimension style. Opens dialog *"Modify dimension style"*.
- *Compare...* - compare two dimension styles, or view a list of all the properties of a style. The comparison results can be copied to the clipboard. Opens dialog *"Comparing Dimension Styles"*.
- *Delete...* – deletes the selected dimension style.

Restoring the Dimension Style

Command line: ***MCSETGOST, SETGOST***.
This command is necessary to create (restore) the standard dimensional style of the environment nanoCAD Mechanica.
Dimension style settings are stored within drawing files. If the file was not created in nanoCAD Mechanica, then it may not have the required dimension style for correct work in the nanoCAD Mechanica environment.

Creating a dimension style

Button *"New..."* in the dialog box *"Dimension style manager"*.
The new style is based on the existing style.

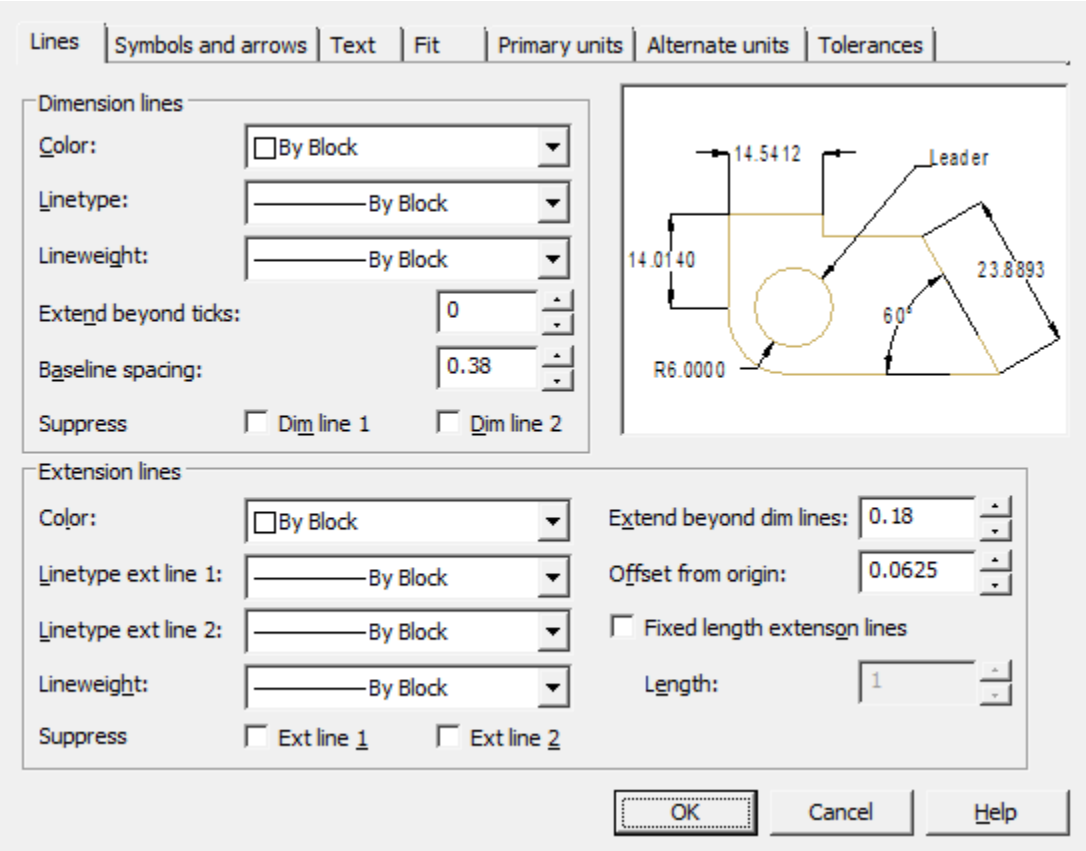


Dialog box composition

Field *"New Style Name"* – style name.
List *"Start With"* – the style on the basis of which the new one will be created.
List *"Use for"* – a dimension sub-style is selected from the list (applicable only to certain types of dimensions).
Button *"OK"* – opens a dialogue *"Modify dimension style"*.

Modify dimension style

In the "Modify dimension style" dialog, properties for a new style are set and parameters of an existing one are edited.
This dialog is called when you click the *"Modify..."* or *"Override..."* buttons from the *"Dimension Style Manager"*, as well as from the *"Creating style"* window.
The name of the edited style is reflected in the title of the dialog.



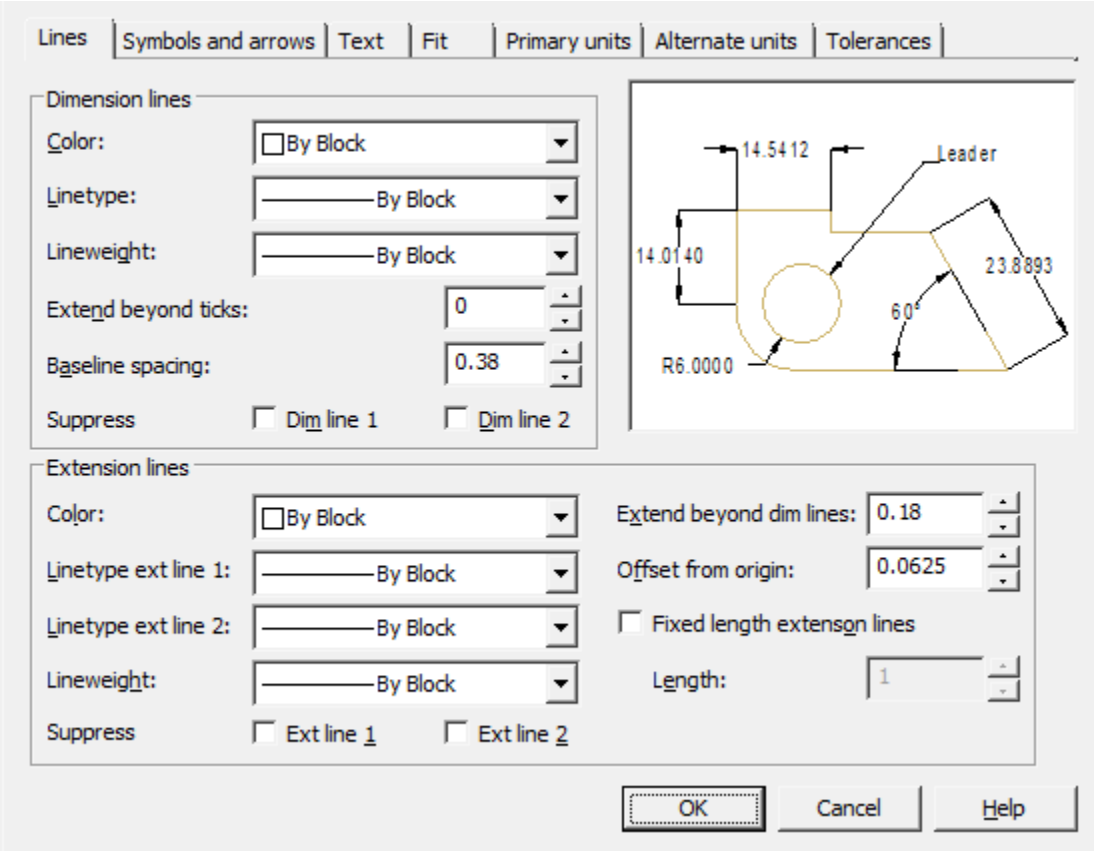
Dialog *"Modify dimension style"* contains tabs:

- Lines
- Symbols and arrows
- Text
- Fit
- Primary units
- Alternate units
- Tolerances

The model image on each tab shows a preliminary result of the property changes.

Tab Lines

Sets the format and properties of dimension lines, extension lines, arrows, and center marks.



Simension lines

Color – setting the color of dimension lines.
Linetype – setting the dimension line type.
Lineweight – setting weights for dimension lines.
Extend deyond ticks – when using serifs as dimension arrows, setting the distance the dimension line should extend beyond the extension lines.
Baseline spacing - setting the distance between dimension lines in dimensions from a common base.
Suppress – sets the suppression of the display of dimension lines if they go beyond the extension lines.

- *Dim line 1* – suppress the first dimension line.
- *Dim line 2* – suppress the second dimension line.

Extension lines

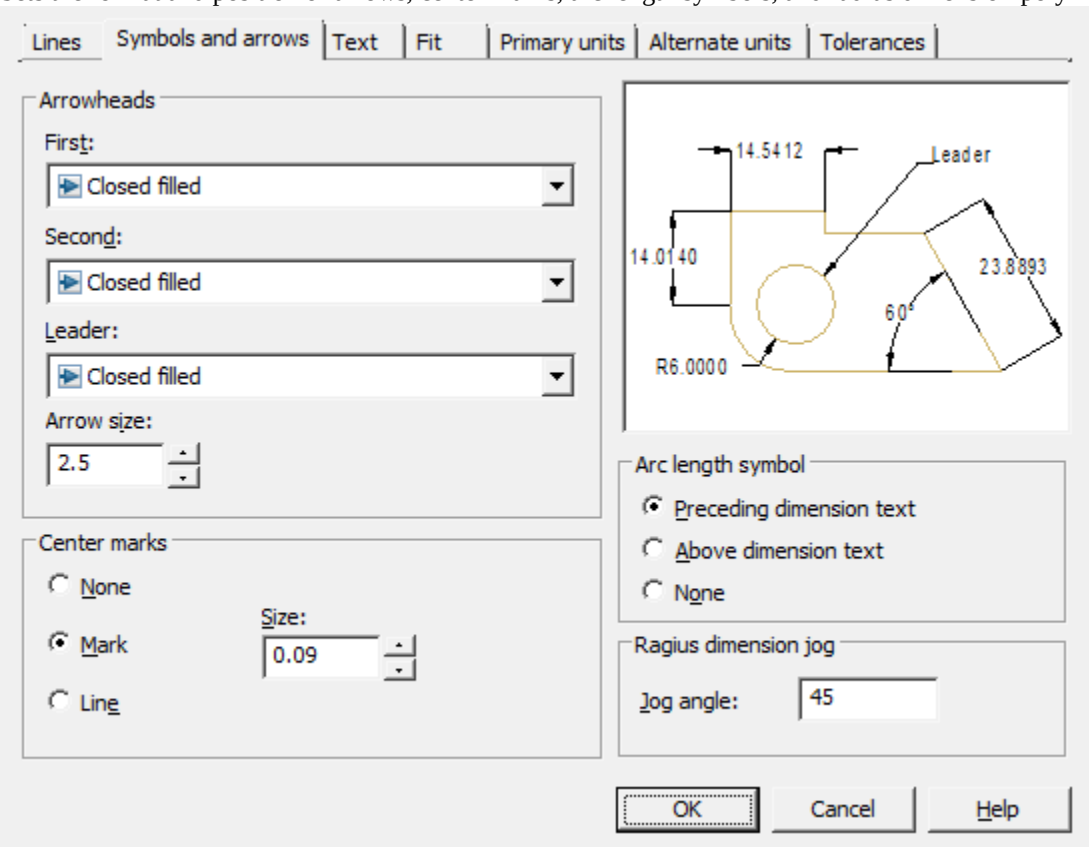
Color - setting the color of the extension lines.
Linetype ext line 1 – setting the type of the first extension line.
Linetype ext line 2 – setting the type of the second extension line.
Lineweight – setting weights for extension lines.
Suppress – set to suppress extension lines:

- *Ext line 1* – suppress the first extension line.
- *Ext line 2* – suppress the second extension line.

Extend beyond dim lines – setting the distance that extension lines should protrude beyond the dimension line.
Offset from origin – sets the distance by which extension lines deviate from object points.
Fixed length extenson lines - sets the full length of the extension lines from the dimension line to the origin of the dimension.

Tab Symbols and arrows

Sets the format and position of arrows, center marks, arc length symbols, and radius dimension polylines.



Arrowheads

Controls the appearance of dimension arrows.

First – selection of the arrow type for the first dimension line. When you change the type of the first arrow, the type of the second is automatically changed.

Second – selection of the arrow type for the second dimension line.

Leader – setting the arrow type for the leader.

Arrow size – displaying the existing one and setting a new size for the arrows.

Center marks

Controls the placement of center marks and centerlines when placing a diameter or radius.

None - do not create center mark and centerlines.

Mark - create a center mark.

Line - create centerlines.

Size - displays an existing one and sets a new size for the center mark or centerline.

Arc length symbol

Controls the display of the arc symbol in the arc length dimension.

Preceding dimension text - inserts arc length characters before dimension text.

Above dimension text - inserts arc length characters above dimension text.

None - prohibiting the display of arc length symbols.

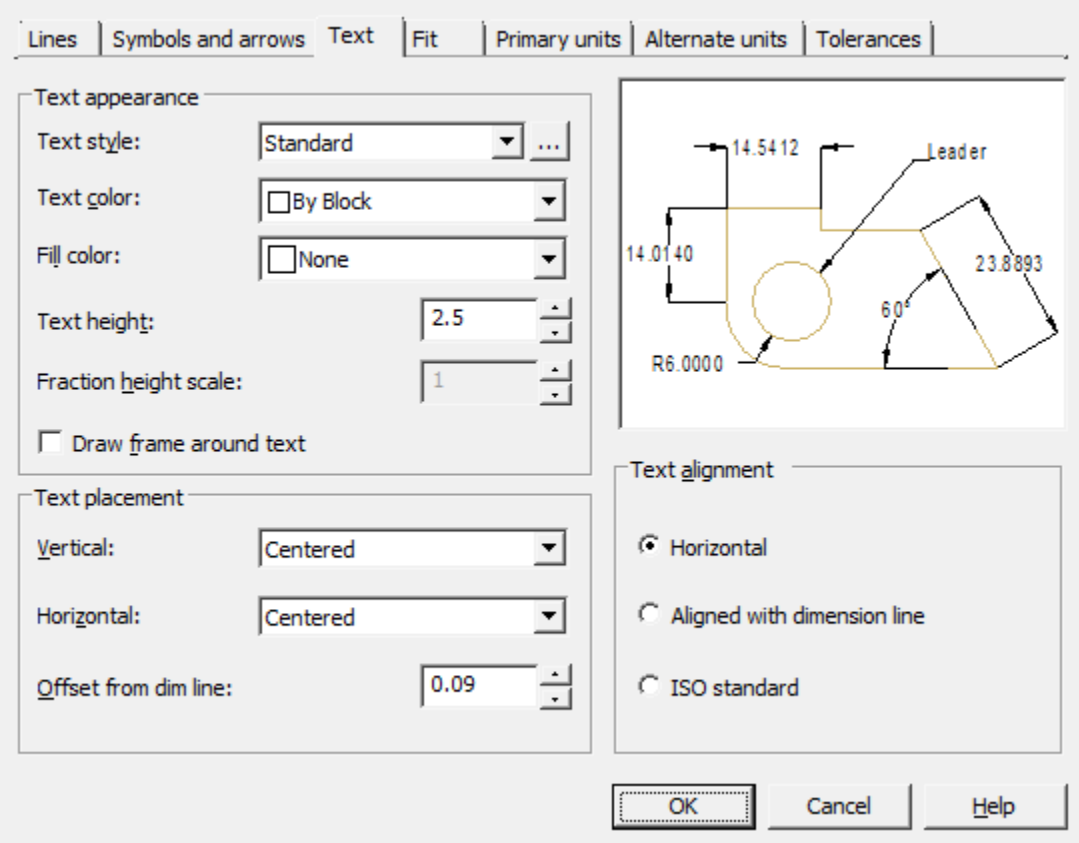
Radius dimension jog

Controls the display of broken lines (zigzags) of radius dimensions, in cases where the center is located outside the page.

Jog angle - sets the angle of the perpendicular line connecting the extension and dimension lines of the radius dimension.

Tab Text

Setting the format, placement and alignment of dimension texts.



Text appearance

Text style – selects the current dimension text style from the list.

Text color – setting the color of the dimension text.

Fill color – setting the background color of the text in dimensions.

Text height - sets the current height of the dimension text. If the text style specifies a fixed height (that is, the style's height value is not 0), then the height entered here is ignored.

Fraction height scale – setting the scale of fractional values relative to the rest of the text. This option is available if the *"Unit format"* is set to *"Fractional"* on the *"Primary units"* tab. The height of fractional texts is determined by multiplying the height of the regular text by a given factor.

Draw frame around text – sets the display of dimension text in a frame.

Text placement

Vertical – select the option to align the dimension text vertically relative to the dimension line:

- *Centered* - centered dimension text between extension lines.
- *Above* - positioning the dimension text above the dimension line. The distance from the dimension line to the bottom border of the text is equal to the value specified in the *"Offset from dim line"* option.
- *Outdise* - positioning the dimension text next to the dimension line on the side farthest from the first fit point.
- *JIS* - placement of dimension text in accordance with the requirements of Japanese industrial standards JIS (Japanese Industrial Standards).
- *Below* - the location of the dimension text below the dimension line. The distance from the dimension line to the top border of the text is equal to the value specified in the *"Offset from dim line"* option.

Horizontal – select an option to position the dimension text horizontally along the dimension line relative to the extension lines:

- *Centered* - centered dimension text between extension lines along the dimension line.
- *At Ext Line 1* - the location of the text to the left of the first extension line. The distance from the extension line to the text is equal to the sum of twice the dimension arrow (*"Symbols and arrows"* tab) and the text gap (the *"Offset from dim line"* option).
- *At Ext Line 2* - the distance from the extension line to the text is equal to the sum of twice the dimension arrow (*"Symbols and arrows"* tab) and the text gap (the *"Offset from dim line"* option).
- *Over Ext Line 1* - positioning text above or along the first extension line.
- *Over Ext Line 2* - positioning text above or along the second extension line.

Offset from dim line – setting the size of the current text gap (the width of the empty space around the text located at the break in the dimension line). This value is also used as the minimum length of the broken dimension line fragments.

The text is located between the extension lines only if the lengths of these fragments are not less than the size of the text gap. If text is above or below the dimension line, it will only fit between the extension lines if there is enough space for the dimension arrows and dimension text to be spaced apart by the text gap.

Text alignment

Sets the orientation of the dimension text between and outside dimension lines.

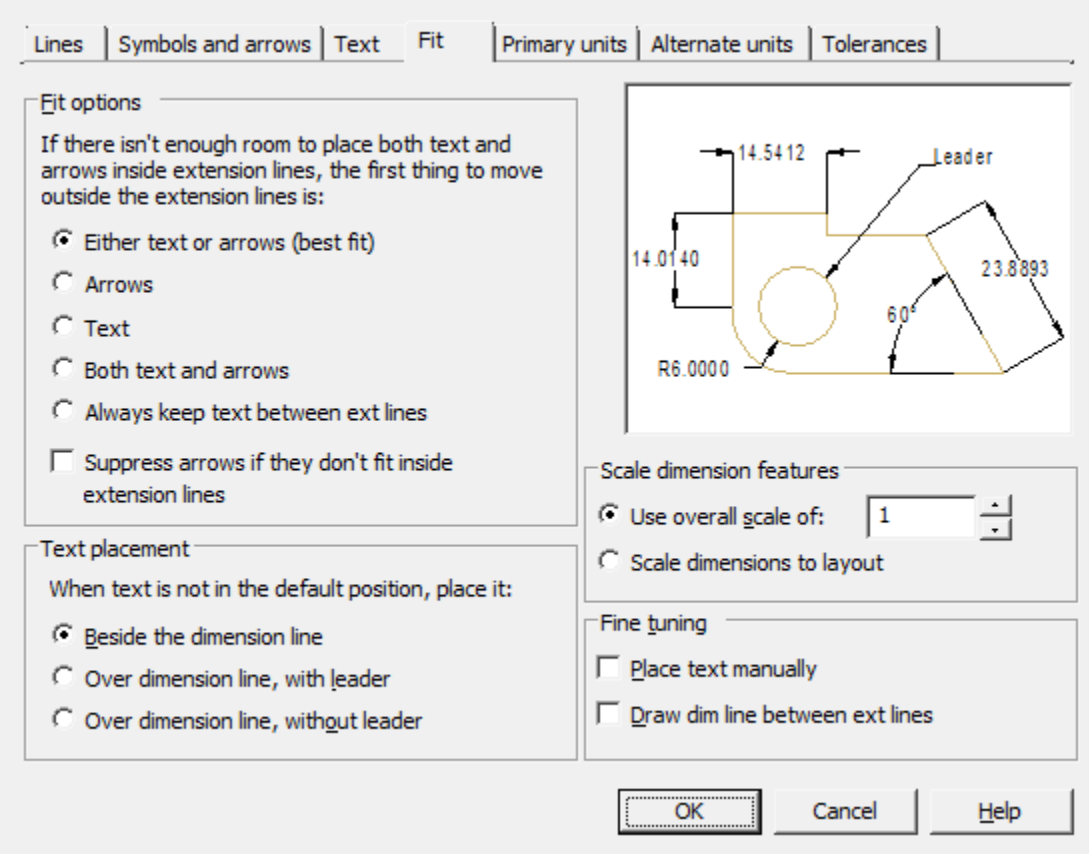
Horizontal – place text horizontally.

Aligned with dimension line – place text along the dimension line.

ISO standart - place along the dimension line if the text is inside the extension lines, or horizontally if the text is outside.

Tab Fit

Controls the position of dimension text, arrows, leaders, and dimension lines.



Fit options

Set the layout of text and arrows if there is not enough space between the extension lines to place them together:
Either text or arrows (best fit) - move either the text or arrows outside the dimension lines, whichever is the best location.
Arrows - move outside the extension lines, first arrows, then text.
Text - move outside the extension lines, first the text, then the arrows.
Both text and arrows - if there is not enough space, then both text and arrows are located outside the extension lines.
Always keep text between ext lines – always place text between extension lines.
Suppress arrows if they don't inside extension lines - if there is not enough space for arrows between the extension lines, they are not displayed.

Text placement

Selecting an action when the dimension text is moved from the default position (specified by the dimension style):
Beside the dimension line – the dimension line moves with the text.
Over dimension line, with leader – dimension lines do not move, but a leader is drawn that connects them and the text. The leader is not drawn if the text is too close to the dimension line.
Over dimension line, without leader – no connections are made between the text and the dimension line.

Scale dimension feature

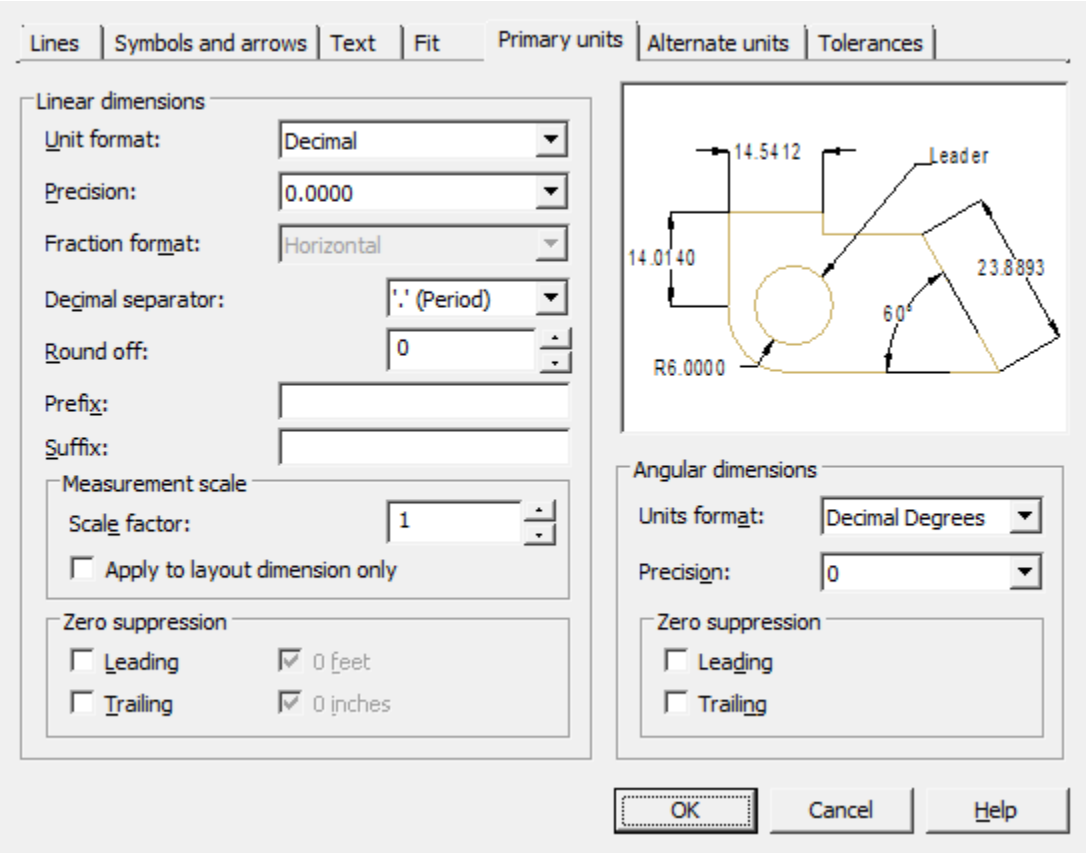
Use overall scale of – setting the scale factor for all dimension style parameters that specify dimensions, distances and indents, including the height of the text and the size of the arrows.
Scale dimensions to layout - sets a scale factor based on the ratio of units in the current model space viewport and paper space.

Fine tuning

Place text manually – places the dimension at the point specified when prompted for Dimension line position. All horizontal placement modes are ignored.
Draw dim line between ext lines – draw dimension line between extension lines, even if dimension arrows are positioned outside of them.

Tab Primary units

Sets the format and precision of base units, and dimension text prefixes and suffixes.



Linear dimensions

Unit format - controls the display and format of tolerances in dimension text.
Precision – setting the number of decimal places in the dimension text.
Fraction format – setting format for fractional texts.
Decimal separator – selection of separator type for sizes expressed in decimal units.
Round off – setting rounding rules for all types of dimensions, except for angular dimensions. Example: for a given accuracy of 0.25, all measured distances are rounded to the nearest multiple of 0.25. When you enter a value of 1.0, all measured distances are rounded to the nearest whole value. The number of digits in fractional parts of distances depends on the precision specified in the "Precision" field.
Prefix – defining the dimension text prefix. You can enter special characters or use control codes to display special characters. For example, the control code "%% c" matches the diameter character. An explicit prefix takes precedence over default prefixes such as those used for diameter and radius.
Suffix – definition of a dimension text suffix. You can enter special characters or use control codes to display special characters. For example, typing the suffix mm will add it to the dimension text. The suffix entered here takes precedence over the default suffixes.
Measurement scale
Scale factor – setting the scale factor for linear dimensions. It is recommended not to change the default value of 1.00. The factor has no effect on angular dimensions, rounding accuracy, positive and negative tolerances.
Apply to layout dimension only – applies a scale factor only to dimensions plotted in layout viewports. It is recommended to disable the setting.

Important!	Changing the value of the scale of the dimensions of a style in a customization changes the value in the dimensions of the same style that are already assigned. This bug is unrecoverable for technical reasons.

Zero suppression - Controls the suppression of leading and trailing zeros, as well as feet and inches zero.

- *Leading* – suppress leading zeros in all decimal sizes. Example: 0.3000 is written as .3000.
- *Trailing* – suppress trailing zeros in all decimal sizes. Example: 11.5000 is represented as 11.5 and 30.0000 is represented as 30.

Angular dimensions

Sets the current corner format for angular dimensions.

Units format – selection of units of measure for angular dimensions.

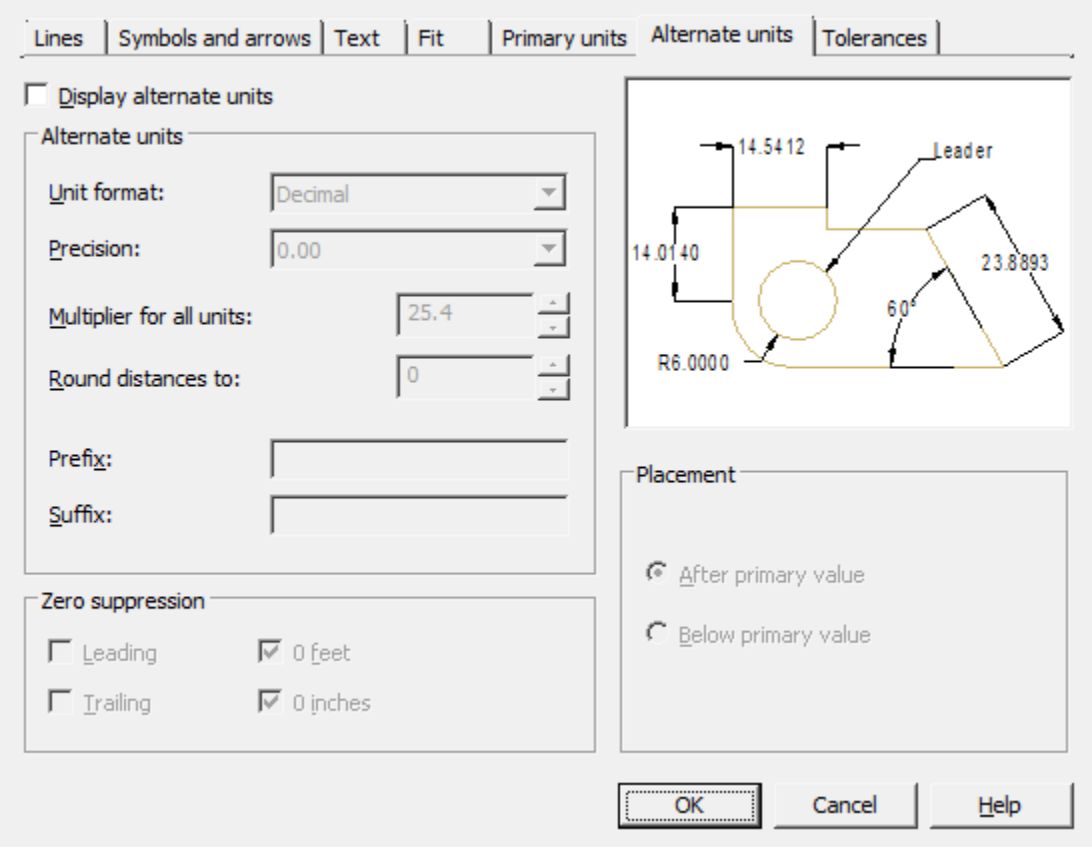
Precision – setting the number of decimal places in angular dimensions.

Zero suppression - control of suppression of leading and trailing zeros.

- *Leading* – suppress leading zeros in all decimal sizes. Example: 0.3000 is written as .3000.
- *Trailing* – suppress trailing zeros in all decimal sizes. Example: 11.5000 is represented as 11.5 and 30.0000 is represented as 30.

Tab Alternate units

Setting the format and precision for alternative dimension units.



Display alternate units – inclusion of the possibility of using alternative dimensional units.

Alternate units

Unit format - setting the format of alternative units.

Precision – setting the number of decimal places after the decimal point.

Multiplier for all units - setting the conversion factor from basic units to alternative ones. Example: To convert inches to millimeters, enter 25.4. The value has no effect on angular dimensions and does not apply to rounding precision or positive and negative tolerances.

Round distances to – sets the rounding rule for all types of dimensions, except for angular dimensions. Example: when setting the accuracy to 0.25, all measured distances are rounded to the nearest multiple of 0.25. When you

enter a value of 1.0, all measured distances are rounded to the nearest whole value. The number of digits in fractional parts of distances depends on the precision specified in the "Precision" field.
Prefix – defining the prefix of the alternate dimension text. You can enter special characters or use control codes to display special characters. For example, the control code "%% c" matches the diameter character. An explicit prefix takes precedence over default prefixes such as those used for diameter and radius.
Suffix – defining the suffix of the alternate dimension text. You can enter special characters or use control codes to display special characters. For example, entering the suffix mm will add it to the text. The suffix entered here takes precedence over the default suffixes.

Zero suppression

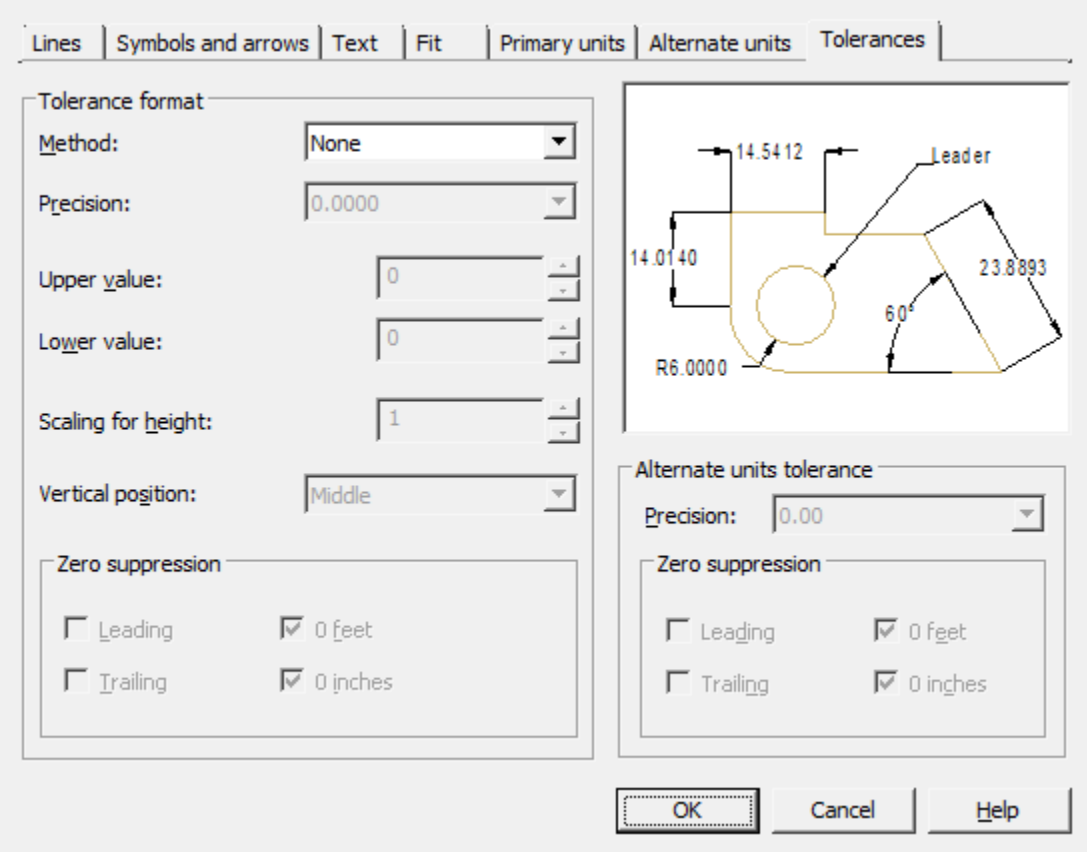
Controls the suppression of leading and trailing zeros and feet and inches zero.
Leading – suppress leading zeros in all decimal sizes. Example: 0.3000 is written as .3000.
Trailing – suppress trailing zeros in all decimal sizes. Example: 11.5000 is represented as 11.5 and 30.0000 is represented as 30.

Placement

Sets the position of alternate units in dimension text.
After primary value – display the value in alternate units immediately following the value in base units.
Below primary value - display the value in alternate units below the value in base units.

Tab Tolerances

Controls the display and format of tolerances in dimension text.



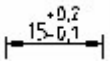
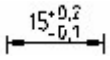
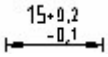
Tolerance format

Method – choice of method for determining tolerances.

None- disable display of tolerances.

Symmetrical – application of two limit deviations, with which the dimension text is displayed with the same upper and lower limit devi from the dimension by the "±" symbol. The value is entered in the *"Upper value"* field.
Deviation – application of two limiting deviations. Dimension text is displayed with different upper and lower tolerance limits. When a put in front of the upper limit deviation and a minus sign in front of the lower one.
Limits – displays dimension text as limit dimensions. The maximum size limit is above the minimum size limit. The largest limit is obt deviation to the nominal size; the smallest - by subtracting the lower limit deviation from the nominal size.
Basic – displays the dimension text as a nominal boxed dimension.

Precission - setting the number of decimal places after the decimal point.
Upper value - setting the value of the upper limit deviation. When the *"Symmetrical"* method is turned on, both deviations are assigned this value.
Lower value - setting the value of the lower limit deviation.
Scaling for height - setting the current text height for deviations.
Vertical position - alignment of deviation texts:

- *Bottom* - alignment of deviation and nominal size on the bottom. 
- *Middle* - aligning deviation and nominal size to the middle of the dimension text. 
- *Top* - alignment of deviation and nominal size at the top. 

Zero suppression - control of suppression of leading and trailing zeros.

- *Leading* – suppress leading zeros in all decimal sizes. Example: 0.3000 is written as .3000.
- *Trailing* – suppress trailing zeros in all decimal sizes. Example: 11.5000 is represented as 11.5 and 30.0000 is represented as 30.

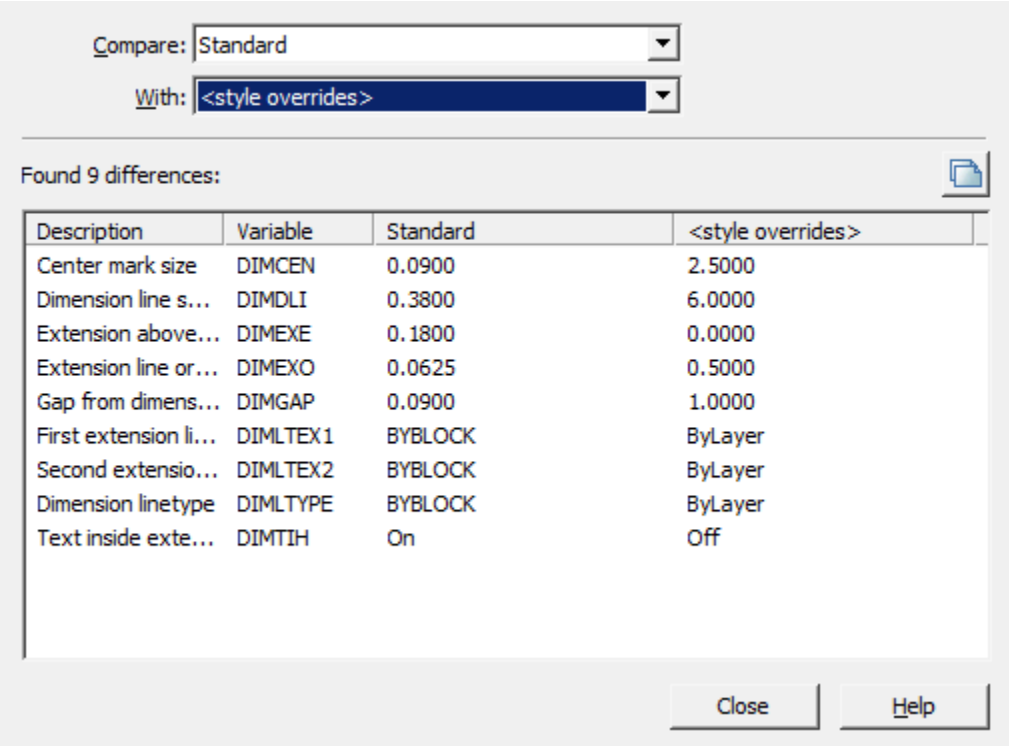
Alternate units tolerance

Alternative unit tolerance formatting.
Precision - setting the number of decimal places after the decimal point.
Zero suppression - control of suppression of leading and trailing zeros.

- *Leading* – suppress leading zeros in all decimal sizes. Example: 0.3000 is written as .3000.
- *Trailing* – suppress trailing zeros in all decimal sizes. Example: 11.5000 is represented as 11.5 and 30.0000 is represented as 30.

Comparing Dimension Styles

The dialog for comparing two dimension styles or viewing a list of all style properties is invoked by the *"Compare..."* button in the *"Dimension Style Manager"* dialog.



"Compare:" field - selects the first dimension style to compare.
"With:" field - selects the second dimension style for comparison.
"Difference table" - the table contains data on non-matching properties.
Table columns:

- *Description* - parameter name.
- *Variable* - parameter system name.
- *Variable value columns* - parameter values in compared styles.

Button *"Copy to clipboard"* - allows you to copy comparison results to the clipboard.

Auto

Main menu: ***Dimensions -*** ***Auto***.

Ribbon: ***Annotate - Dimensions -*** ***Auto***.

Ribbon: ***Mechanical - Symbols -*** ***Dimensions***.

Toolbar: ***Dimensions -*** ***Dimensions***.

Command line: ***MCMDIM, MDIM***.

When inserting dimensions, context menu commands are available to switch the dimension view:

- Auto
- Aligned
- Horizontal
- Vertical
- Radius
- Diameter

- Large arc radius
- Arc
- Ordinate
- Angle
- Chain
- Base
- Properties (Opens the size edit dialog)
- Group

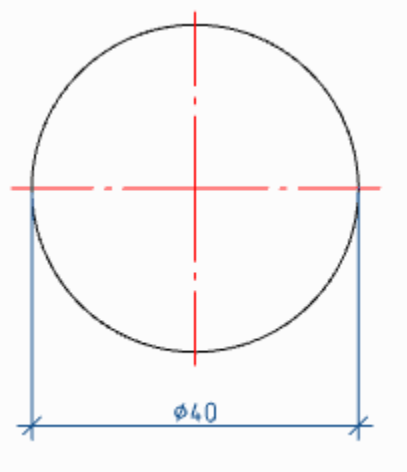
	If you hold down the CTRL key while placing linear dimensions at the time you select the dimension line, you can skew the dimension extension lines.
Note:	If you hold down the SHIFT key while placing linear or angular dimensions while selecting the position of the dimension line, you can move the dimension text along the dimension line.

Linear dimensioning

To apply the dimension you need:
1. Select a segment;
2. Place the dimension number.

Dimensions can change to horizontal, vertical, or parallel, depending on the position of the drop point.

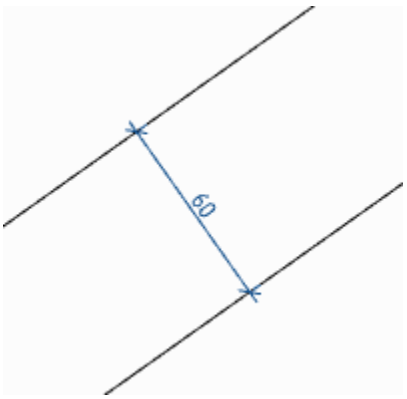
Dimensioning by selecting characteristic (anchor) points



To apply dimensions, you must:
1. Select the first node;
2. Select the second node;
3. Place the dimension number.

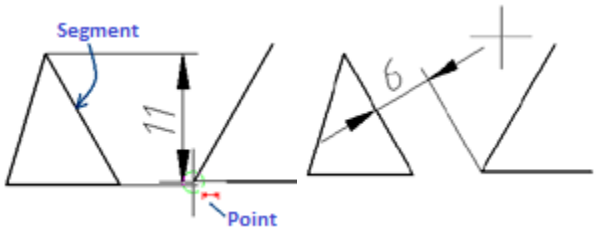
When selecting anchor points of the circle, the diameter sign is automatically set.

Dimensioning between two parallel lines



To apply the dimensions shown above, you must:
1. Select the first segment;
2. Select the second segment;
3. Place the dimension number.

Dimensioning through a point perpendicular to the selected line



To apply the dimensions shown above, you must:
1. Specify the segment, perpendicular to which the dimension will be applied;
2. Without interrupting the commands, point to the end point of the line segment (a circle of small diameter should appear around the point);
3. Place the dimension number.

Undo an erroneously inserted dimension without exiting the command

Dimensioning is performed in a cyclic mode, i.e. when applying one dimension, the team does not stop its work, but suggests setting the next dimension or choosing a different type of dimension. In some cases, the previous dimension may not be set correctly. To remove an erroneous dimension, there is the *"Undo"* context menu command. You can also call the command with the *"Ctrl + Z"* hotkeys. The command is available when setting the dimension after specifying the dimension number.

Auxiliary markers

Auxiliary autodimension markers appear when you hover over a drawing object and show what type of dimension will be built in this case:

- Indicates that the diametral dimension will be drawn.
- Shows what a linear dimension will be drawn (horizontal, vertical or parallel).
- Shows that the angular dimension will be drawn.

- Shows that the chain of dimensions will be built.
- Shows that the slanted dimension will be drawn.
- Indicates that the radial dimension will be drawn.

Linear dimensions

Aligned dimension

Main menu: ***Dimensions -*** ***Aligned***.

Ribbon: ***Annotate - Dimensions -*** ***Aligned***.

Toolbar: ***Dimensions -*** ***Aligned***.

Context menu: ***aLigned - when calling any dimension***.

Command line: ***MCMDIMALI, MDIMALI***.

Horizontal dimension

Context menu: ***Horizontal - when calling any dimension***.

Command line: ***MCMDIMHOR, MDIMHOR***.

Vertical dimension

Context menu: ***Vertical - when calling any dimension***.

Command line: ***MCMDIMVER, MDIMVER***.

Parallel, horizontal, vertical dimensions

To apply dimensions, you must:
1. Choose dimension;
2. Select the type of binding:
   - "nEarest snap gets object" - dimensions the entire selected object;
   - "nEarest snap gets point" (default) - sets dimension nodes at a selected location on an object.
3. Specify the end nodes of the dimension based on the selected snap type:
   - "nEarest snap gets object" - specify an object, the end nodes of the dimension will be the end points of the object;
   - "nEarest snap gets point" - specify the start and end nodes on the selected object.
4. Place the dimension number. The dimension will be built.

Auxiliary markers

Construction markers appear when you select a certain linear dimension:

- Parallel dimension is selected.
- Horizontal dimension is selected.
- Vertical dimension is selected.

Linear dimension

Main menu: ***Dimensions - Linear***.

Ribbon: ***Annotate - Dimensions - Linear***.

Toolbar: ***Dimensions - Linear***.

Command line: ***DIMLINEAR***.

Linear dimensioning

To apply dimensions, you must:
1. Call the *"Linear"* command;
2. Specify the first node;
3. Indicate the second node;
4. Place the dimension number. Linear dimension will be created.

Radius

Main menu: ***Dimensions - Radius***.

Ribbon: ***Annotate - Dimensions - Radius***.

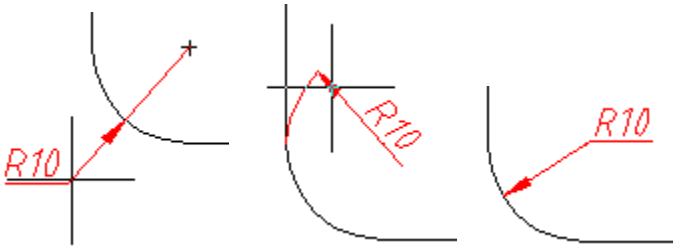
Ribbon: ***Mechanical - Symbols - Radius***.

Toolbar: ***Dimensions - Radius*** .

Context menu: ***Radial - when calling any dimension***.

Command line: ***MCMDIMRAD, MDIMRAD***.

Procedure



To apply a radial dimension, you must:
1. Select the "Radius" dimension;
2. Select an arc (circle), the dimension will snap to the arc;

3. Specify the insertion point.

Diameter

Main menu: ***Dimensions -*** ***Diameter***.

Ribbon: ***Annotate - Dimensions -*** ***Diameter***.

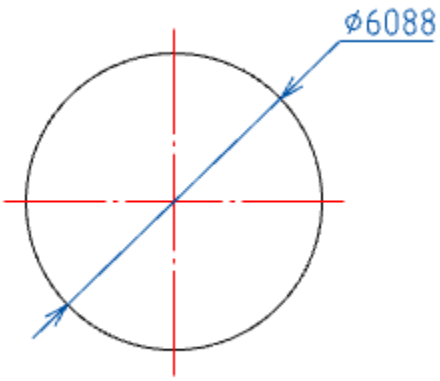
Ribbon: ***Mechanical - Symbols -*** ***Diameter***.

Toolbar: ***Dimensions -*** ***Diameter***.

Context menu: ***Diametral - when calling any dimension***.

Command line: ***MCMDIMDIA, MDIMDIA***.

Procedure



To apply the dimension of the circle, you must:
1. Select the dimension "Diameter";
2. Select the circle, the dimension snaps to the circle;
3. Pick an insertion point.

Large arc radius

Main menu: ***Dimensions -*** ***Large arc radius***.

Ribbon: ***Annotate - Dimensions -*** ***Large arc radius***.

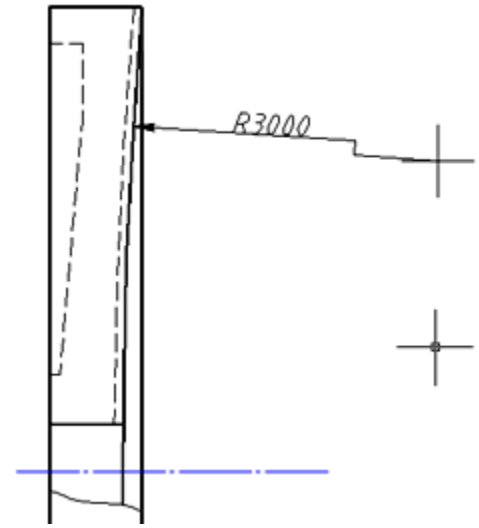
Ribbon: ***Mechanical - Symbols -*** ***Large arc radius***.

Toolbar: ***Dimensions -*** ***Large arc radius***.

Context menu: ***big RAdius - when calling any dimension***.

Command line: ***MCDIMHRAD, DIMHRAD***.

Procedure



To dimension a large arc radius, you need to:
1. Select the "Large arc radius" dimension;
2. Select an arc (circle), the dimension will snap to the arc;
3. Specify the insertion point of the arrow;
4. Specify the insertion point of the value on the arrow.

Arc length dimension

Main menu: ***Dimensions -*** ***Arc***.

Ribbon: ***Annotate - Dimensions -*** ***Arc length***.

Ribbon: ***Mechanical - Symbols -*** ***Arc length dimension***.

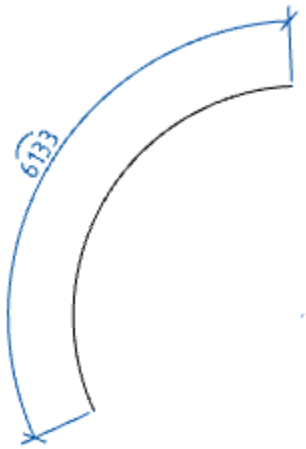
Toolbar: ***Dimensions -*** ***Arc length***.

Context menu: ***arC - when calling any dimension***.

Command line: ***MCMDIMARC, MDIMARC***.

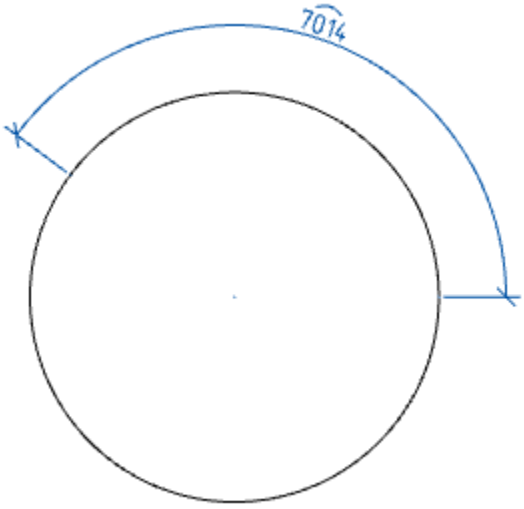
Procedure

Arc dimension



To apply the dimension you need:
1. Select the arc dimension;
2. Select an arc (the arc dimension will snap to the arc);
3. Specify the insertion point.

Circle dimension



To apply the dimension you need:
1. Select the arc dimension;
2. Specify the starting point on the circle;
3. Specify the end point on the circle;
4. Specify the insertion point.

Ordinate dimension

Main menu: ***Dimensions -*** ***Ordinate dimension***.

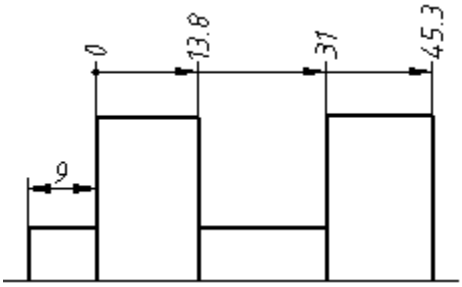
Ribbon: ***Annotate - Dimensions -*** ***Ordinate***.

Ribbon: ***Mechanical - Symbols -*** ***Ordinate dimension***.

Toolbar: ***Dimensions -*** ***Ordinate dimension***.

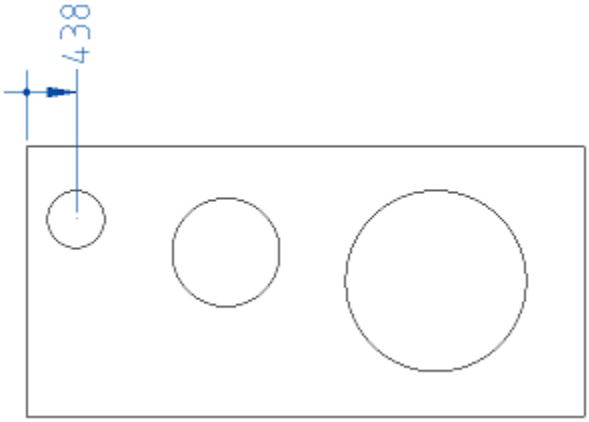
Context menu: ***Ordinate - when calling any dimension***.

Command line: ***MCMDIMORD, MDIMORD***.

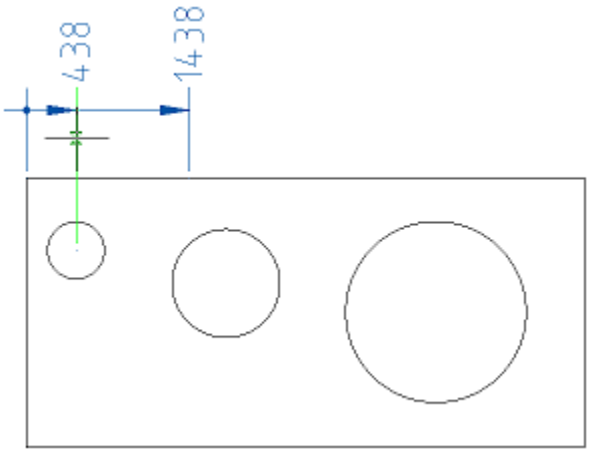


Procedure

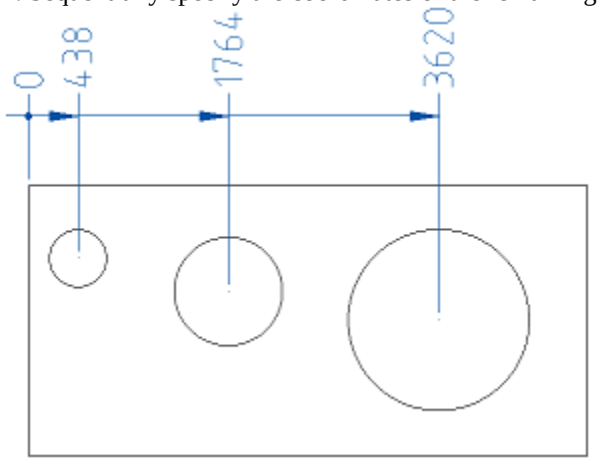
1. Call the "Ordinate dimension" command.
2. Pick the location of the first dimension of the ordinate chain.



3. Select the first dimension you set. All subsequent dimensions will be snapped to it. If for some reason dimensioning has been completed (after the first dimension has been added), select the ordinate dimension (or autosize) again and specify the dimensioned.



4. Sequentially specify the coordinates of the remaining dimensions.



5. Press the "Esc" key to end the command.

Angle dimension

Main menu: ***Dimensions - Fill angle***.

Ribbon: ***Annotate - Dimensions - Angular***.

Ribbon: ***Mechanical - Symbols - Angle dimension***.

Toolbar: ***Dimensions - Angle dimension***.

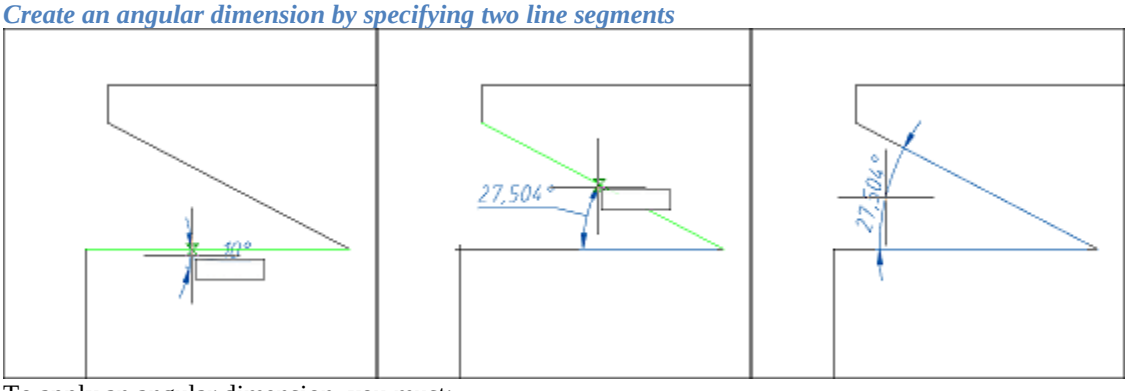
Toolbar: MCS Dimensions - ***Angle dimension***.

Context menu: ***aNguLar - when calling any dimension***.

Command line: ***MCMDIMANG, MDIMANG***.

Procedure

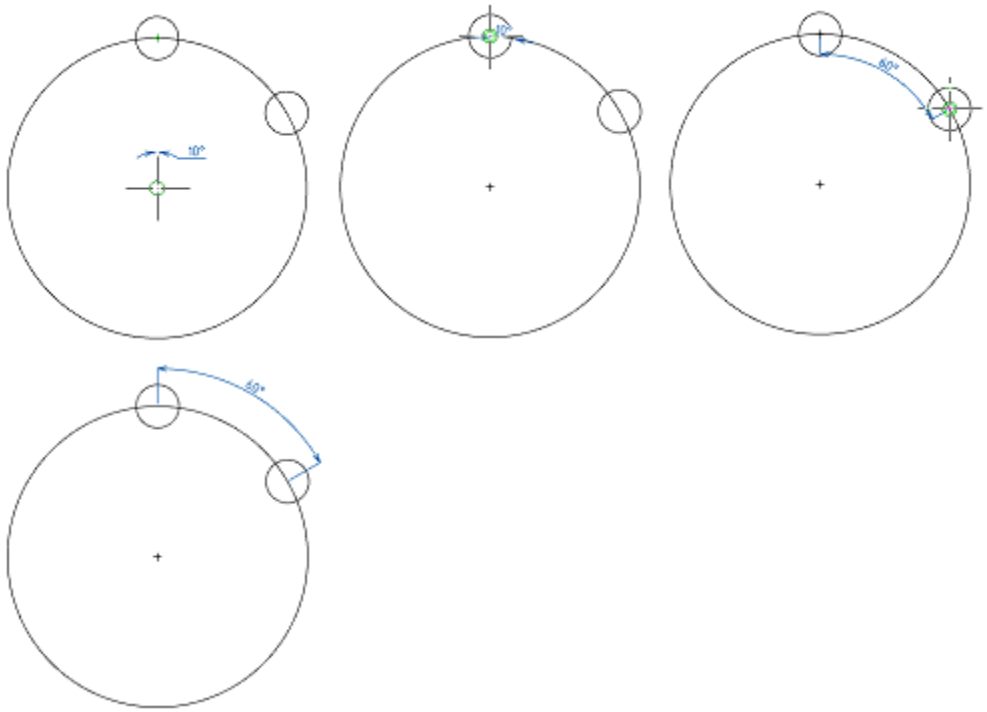
Create an angular dimension by specifying two line segments



To apply an angular dimension, you must:
1. Select an angular dimension;
2. Select the first segment;
3. Select the second segment;

4. Pick an insertion point.

Angular dimension of hole centers



To apply the angular dimension with the indication of the centers, you must:
1. Select an angular dimension;
2. Specify the center of the axial circle (on which the holes lie);
3. Pick the center of the first hole;
4. Pick the center of the second hole;
5. Pick an insertion point.

Dimensions Chain

Main menu: *Dimensions -* *Dimensions Chain*.

Ribbon: *Annotate - Dimensions -* *Continue dimension*.

Ribbon: *Mechanical - Symbols -* *Continue dimension*.

Toolbar: *Dimensions -* *Dimensions Chain*.

Command line: *MCCHAINCONT, CHAINCONT*.

A chain of dimensions is a sequence of dimensions counted from the end of the previous dimension (the beginning of each new dimension coincides with the end of the previous one).
Dimension chains can be linear, ordinate, or angular. Before building the base dimensions, at least one linear, ordinate or angular dimension must be applied to the object.

Procedure

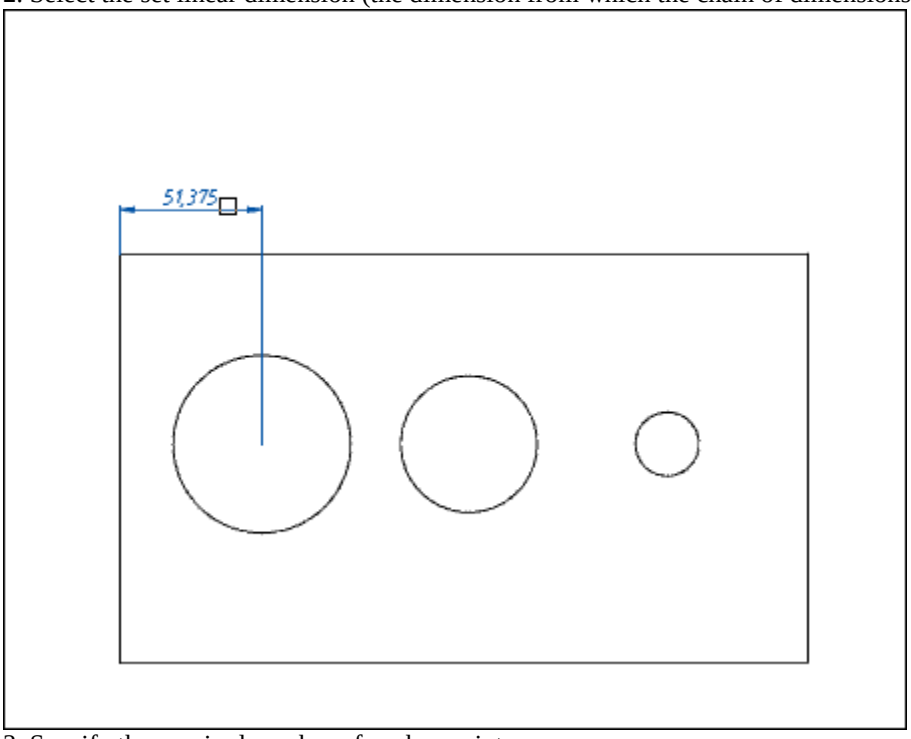
Ordinate chain of dimensions

Note:	Constructing an ordinate chain of dimensions is similar to constructing ordinate dimensions.

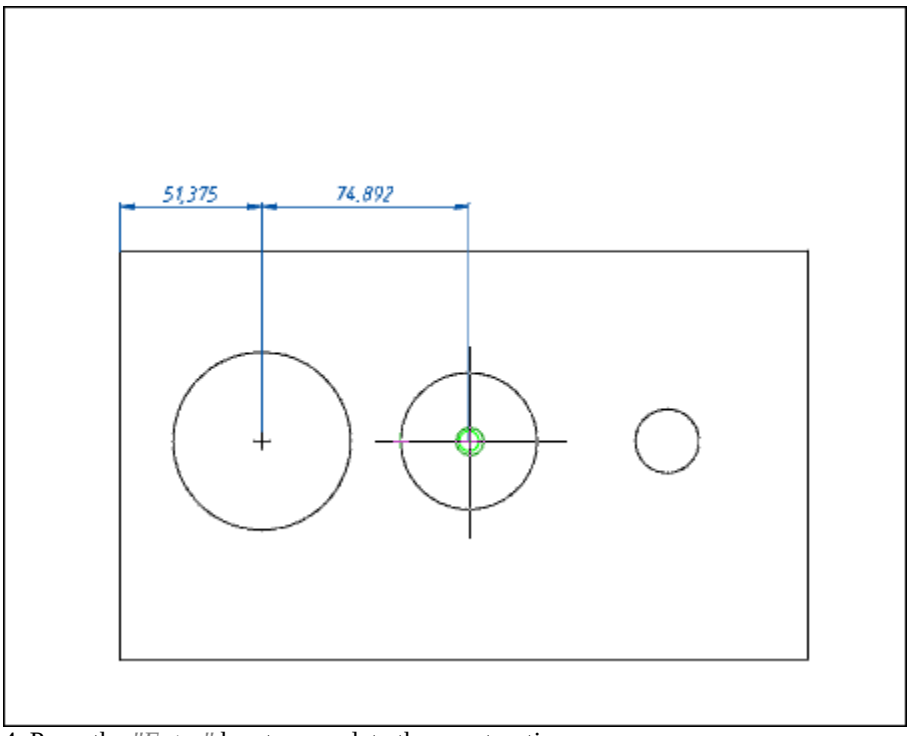
1. Call the command;
2. Select the set ordinate dimension (the dimension from which the ordinate sizes will begin to form);
3. Construct the ordinate dimensions in sequence;
4. Press the *"Enter"* key to complete the construction.

Linear dimension chain

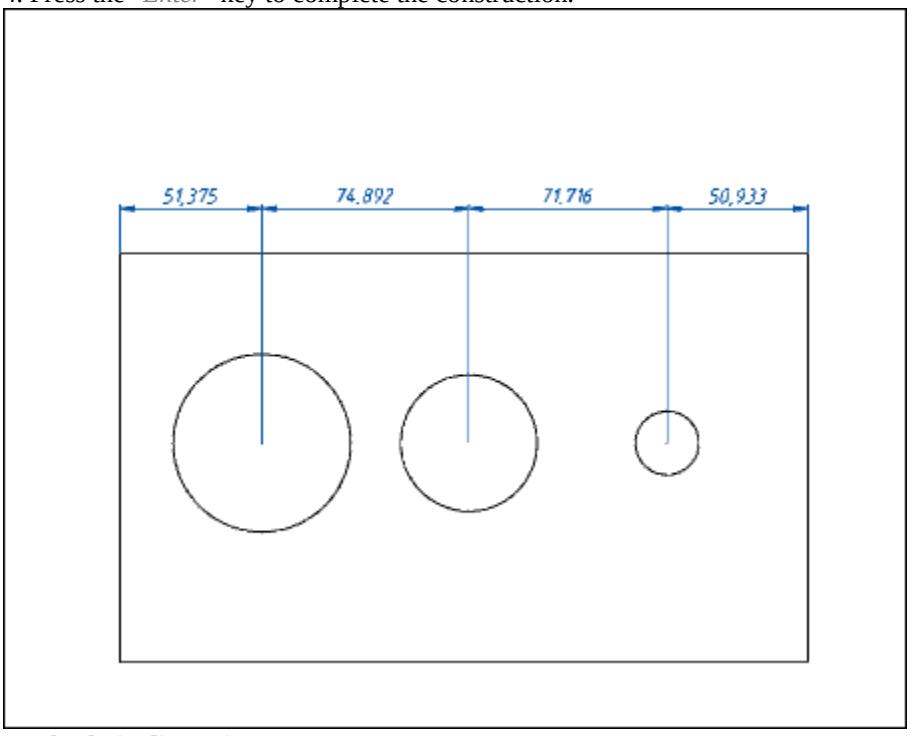
1. Call the command;
2. Select the set linear dimension (the dimension from which the chain of dimensions will begin to form);



3. Specify the required number of anchor points;

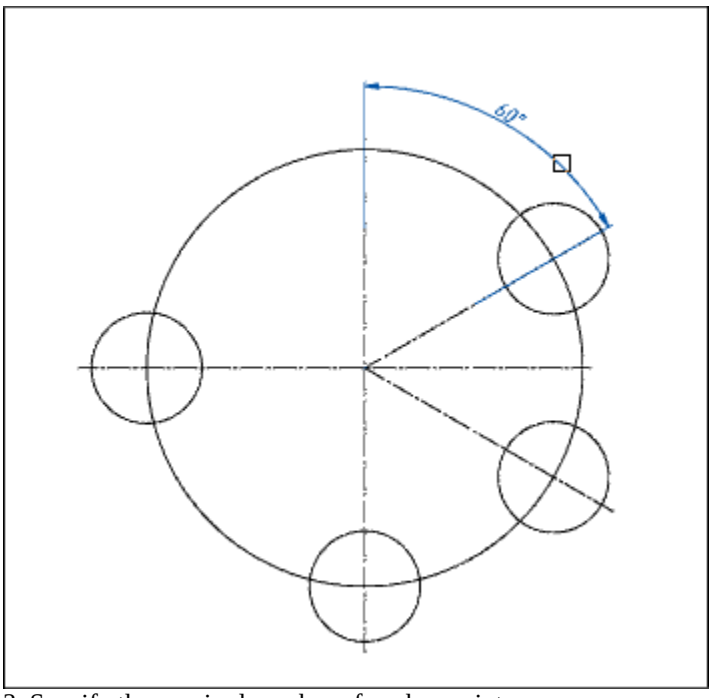


4. Press the *"Enter"* key to complete the construction.

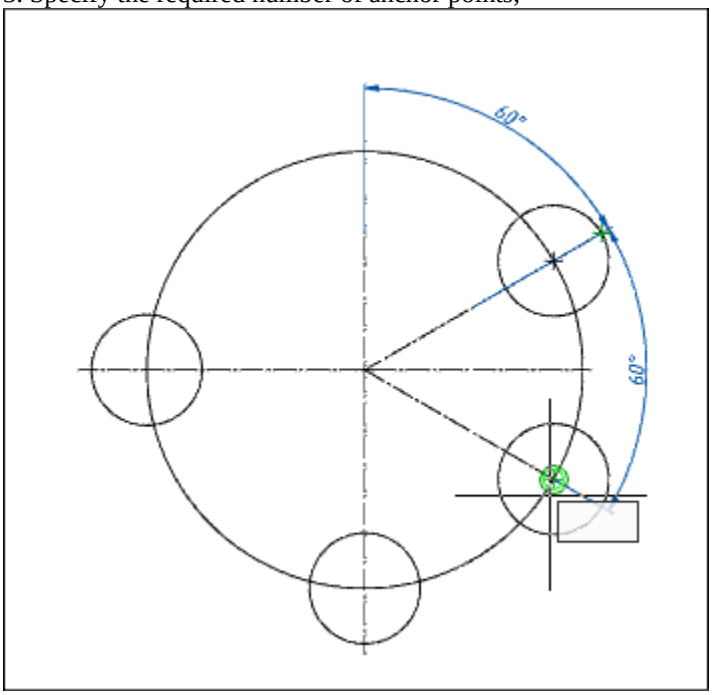


Angle chain dimensions

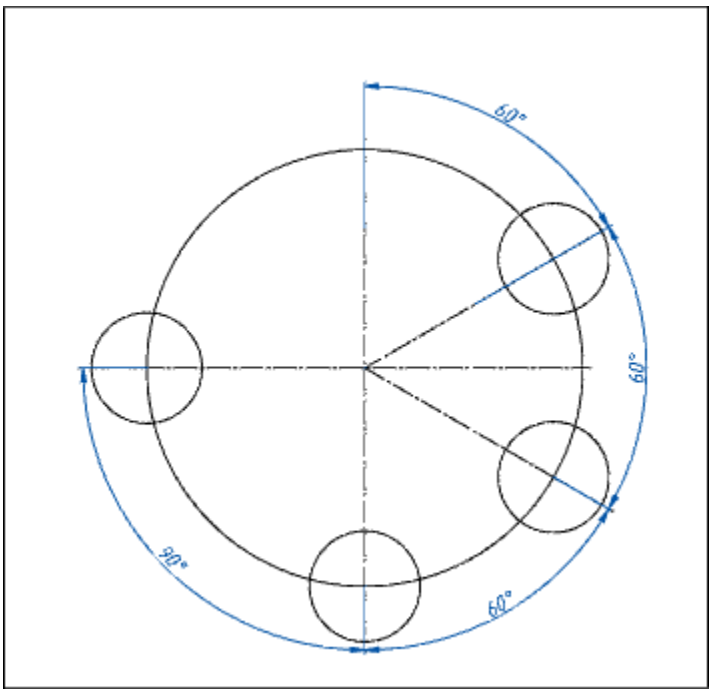
1. Call the command;
2. Select the preset angular dimension (the dimension from which the chain of dimensions will begin to form);



3. Specify the required number of anchor points;



4. Press the *"Enter"* key to complete the construction.



Baseline dimension

Main menu: ***Dimensions -*** ***Base dimension***.

Ribbon: ***Annotate -Dimensions -*** ***Baseline dimension***.

Ribbon: ***Mechanical - Symbols -*** ***Baseline dimension***.

Toolbar: ***Dimensions -*** ***Base dimension***.

Context menu: ***Base - when calling any dimension***.

Command line: ***MCCHAINBASE, CHAINBASE***.

Base dimensions are a sequence of dimensions measured from one base point.
Base dimensions can be linear, ordinate, or angular. Before building the base dimensions, at least one linear, ordinate or angular dimension must be applied to the object.

Procedure

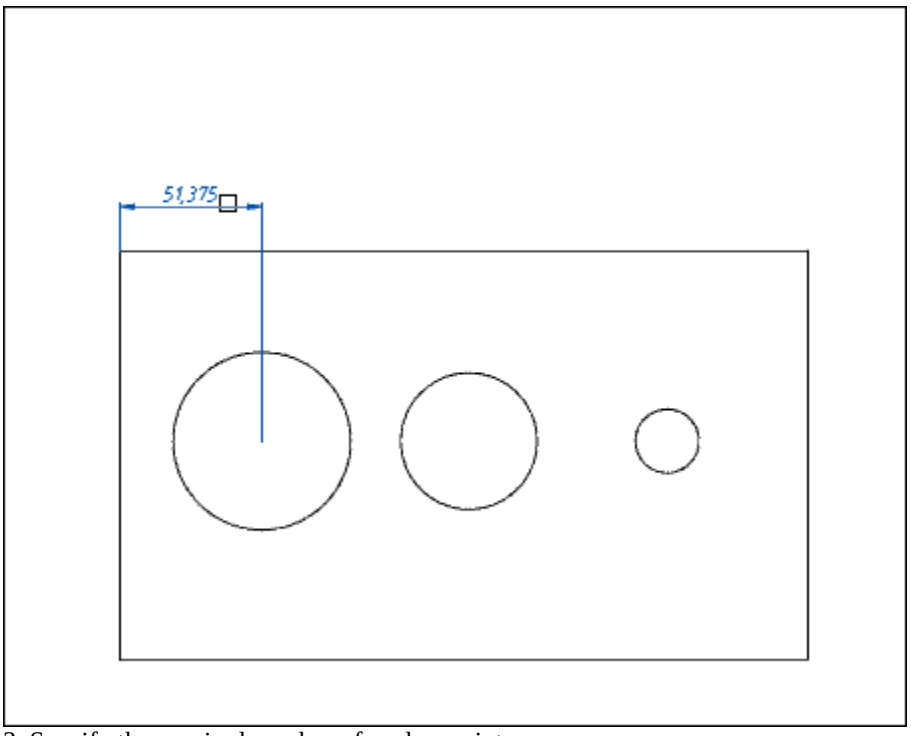
Ordinate chain of dimensions

Note:	The construction of an ordinate chain of dimensions is similar to construction ordinate dimension.

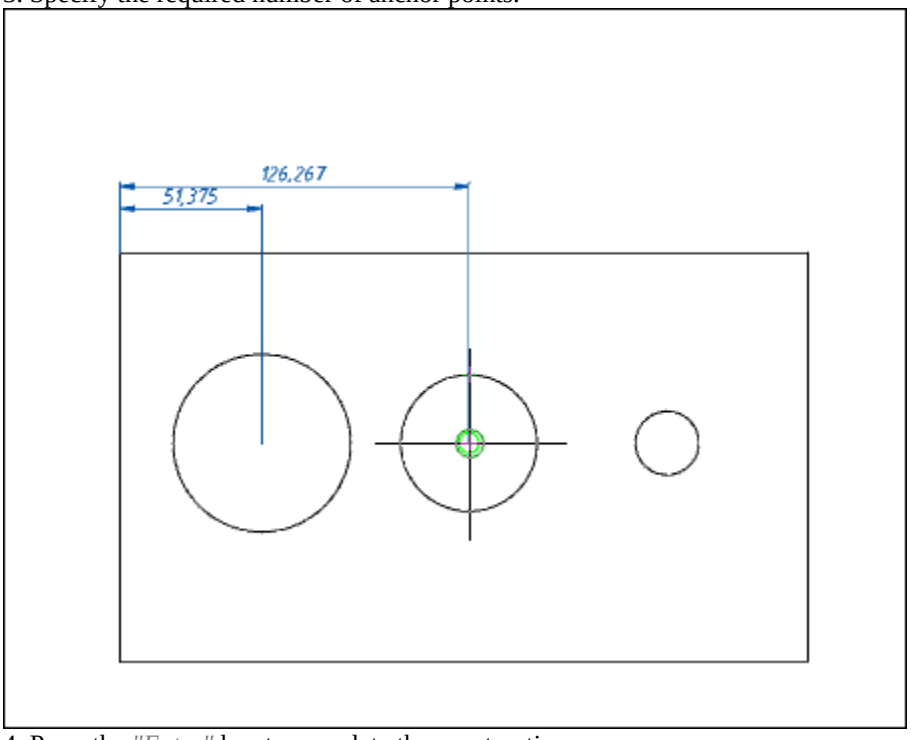
1. Call the command;
2. Select the set ordinate dimension (the dimension from which the ordinate sizes will begin to form);
3. Construct the ordinate dimensions in sequence;
4. Press the *"Enter"* key to complete the construction.

Linear base dimensions

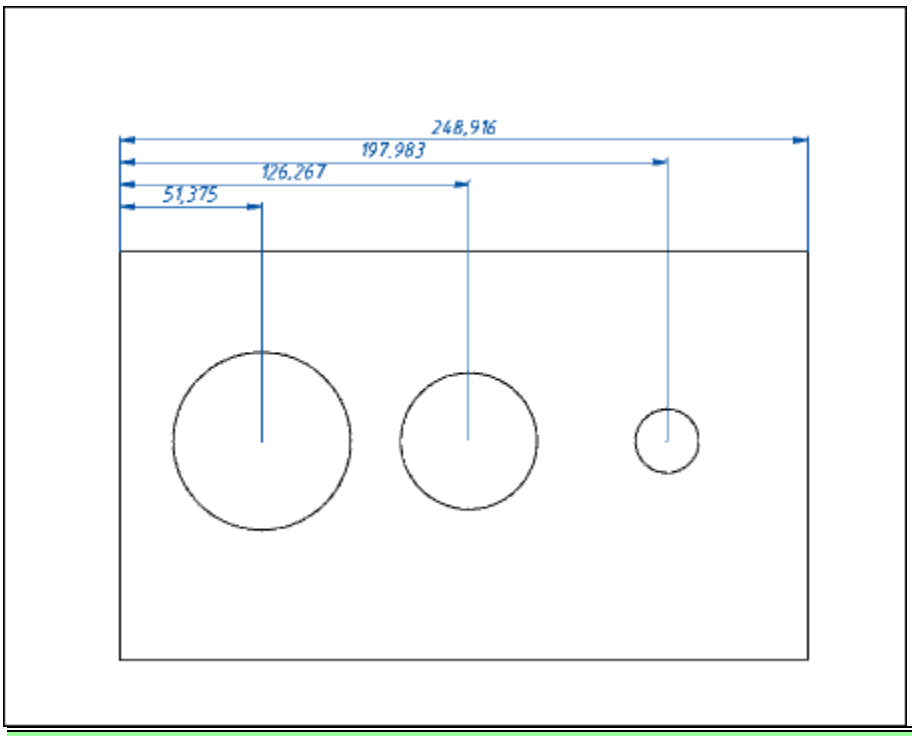
1. Call the command;
2. Select the set dimension as the base (the dimension from which the base sizes will begin to form);



3. Specify the required number of anchor points.



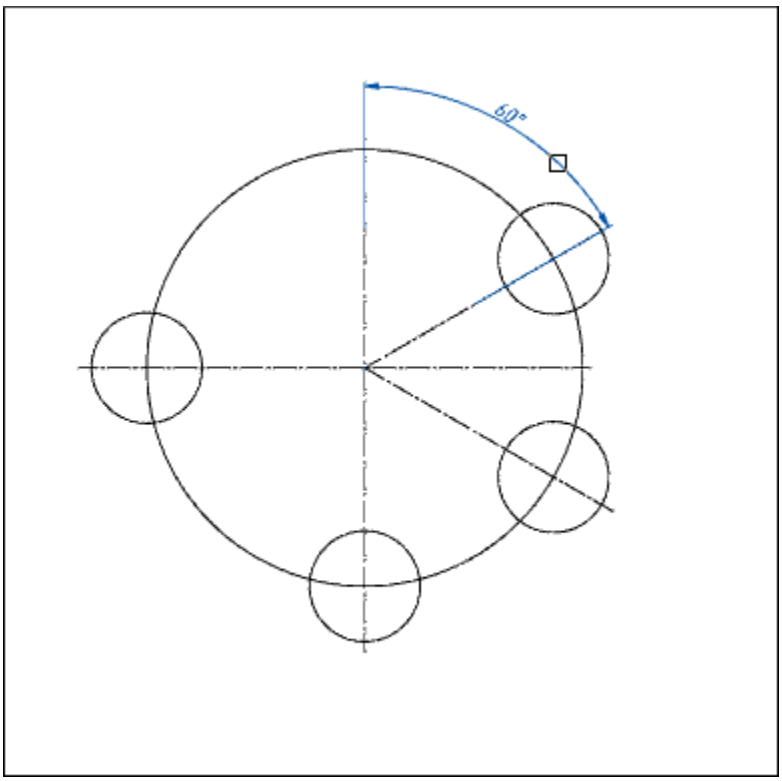
4. Press the *"Enter"* key to complete the construction.



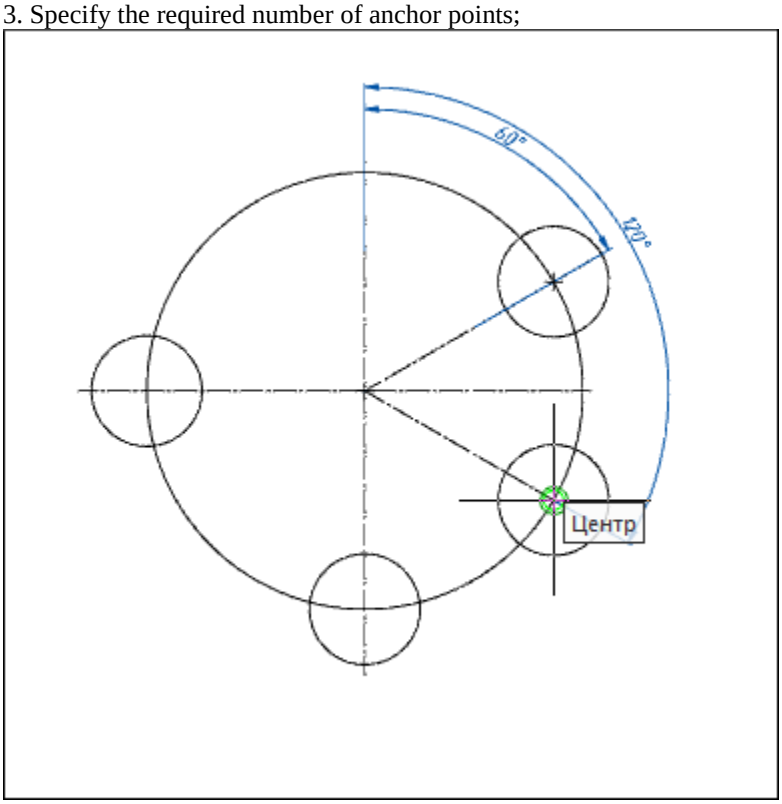
Note:	Each new dimension is set at a specified distance from the previous one. The value of the offset of the dimensions is set by the *"Step"* parameter in the base dimensions in the *"Lines"* tab of the "Modify dimension style" dialog.

Corner base dimensions

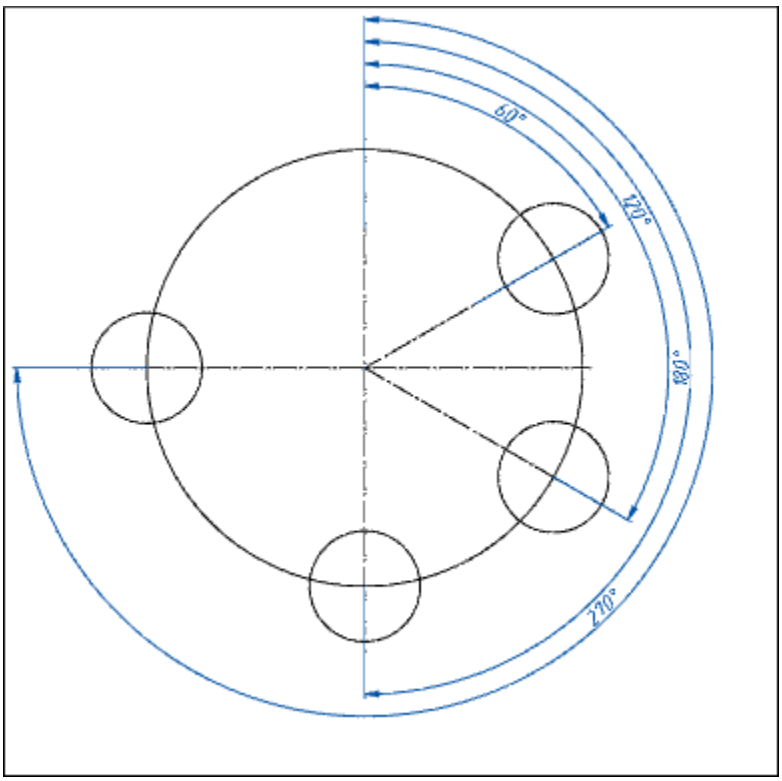
1. Call the command;
2. Select the preset angular dimension (the dimension from which the base dimensions will begin to form);



3. Specify the required number of anchor points;



4. Press the *"Enter"* key to complete the construction.



Group dimension

Main menu: ***Dimensions - Group dimension***.

Ribbon: ***Annotate - Dimensions - Group dimension***.

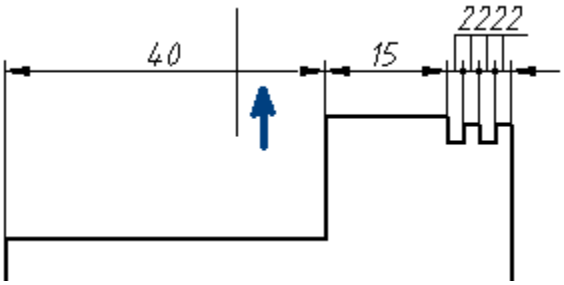
Ribbon: ***Mechanical - Symbols - Group dimension***.

Toolbar: ***Dimensions - Group dimension***.

Context menu: ***Grouped - when calling any dimension***.

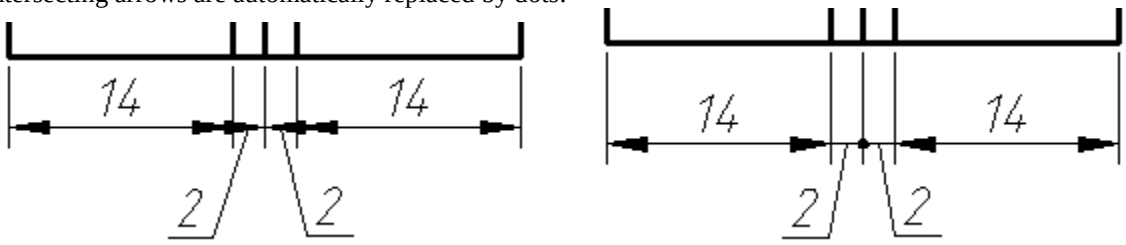
Command line: ***MCDIMGROUP, DIMGROUP***.

Procedure



1. Call the command;

2. Select all the objects that will participate in creating the group dimension. To complete the selection press the *"Enter"* key;
3. Specify the location of the chain of dimensions.
If the setting "Arrows in chains" is set to "Point", then when dimensioning a chain of small numerical dimensions, intersecting arrows are automatically replaced by dots.



Offset dimension

Main menu: ***Dimensions -*** ***Offset dimension***.

Ribbon: ***Mechanical - Symbols -*** ***Offset dimension***.

Toolbar: MCS Dimensions - ***Offset dimension***.

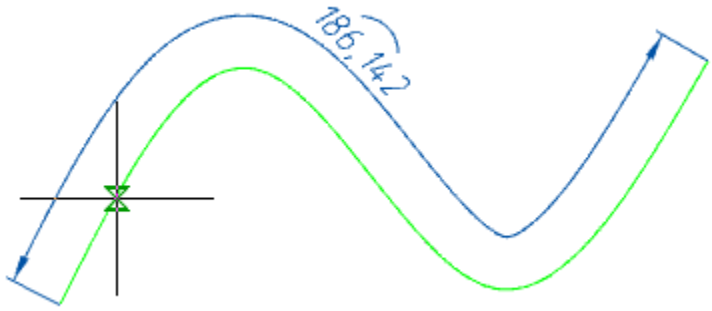
Command line: ***MCDIMOFFSET, DIMOFFSET***.

The command allows you to build a size similar to the contour of the selected object. Similarity size is indicated by an arc above the value.

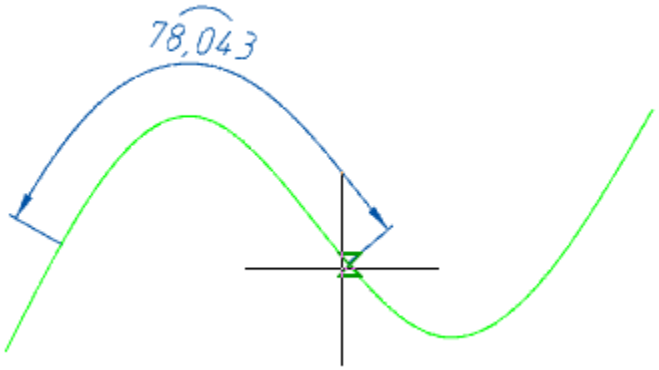
Procedure

1. Call the command *"Offset dimension"*.
2. Select the method for specifying the object: "sElection" or "frEe". The "sElection" method is used in paper space when you need to dimension an object in an inactive viewport. In all other cases, the "frEe" method is used.
3. Specify the object from which the dimension will be built. The selected object can be specified: line, arc, spline, polyline, circle or ellipse.

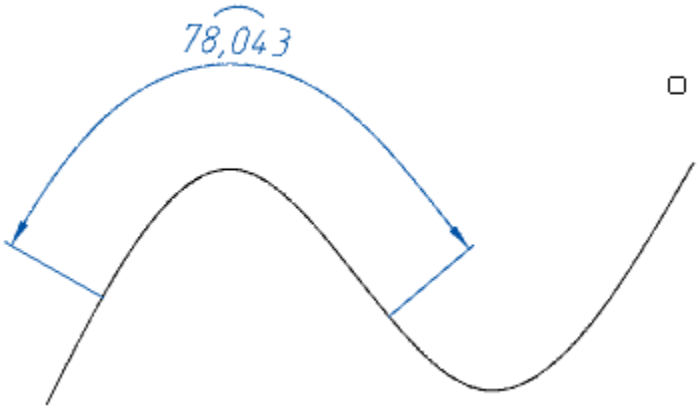
4. Specify the first insertion point (starting point of origin of the dimension) or press the "Space" (or RMB) to automatically add endpoints to the entire length of the object (does not work for closed objects).



5. Specify the second insertion point (end point of origin of the dimension), if you have not pressed the "Space" (or RMB) key.



6. Place the dimension leader in the drawing.

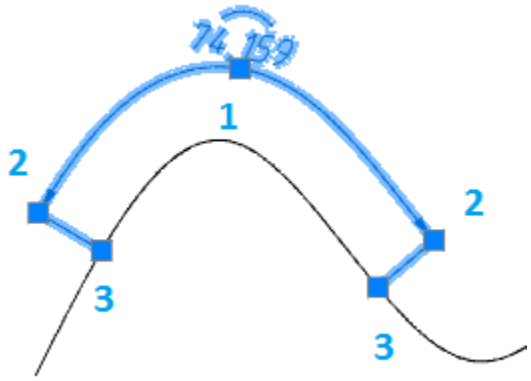


7. Offset dimension will be built. The command will continue to work in a cyclic mode. Press the *"Esc"* key to exit cyclic mode.

Grips

1. Dimension value placement grip.
2. Extension line position grips.

3. Dimension endpoint grips.



Edit dimension

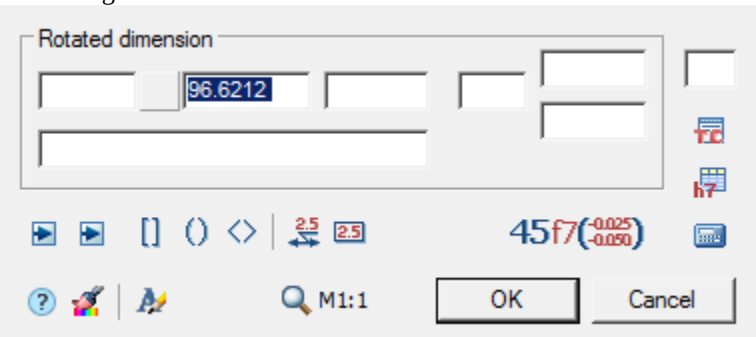
LMB double click on the dimension.

Context menu: ***Properties - when calling any dimension***.

Context menu: ***Edit - at the selected dimension***.

Command line: ***MCFEDIT, FEDIT***.

The dialog box looks like this:



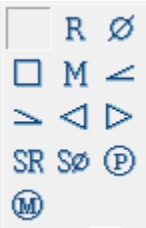
The window is divided into an area of text input fields with the name of the dimension type and a set of dimension design commands.

Additional auxiliary commands are available in the context menu of the dialog box text fields:

- History - the list contains the values previously entered in this field;
- Recent - the list contains frequently used values when filling in this type of field;
- Insert division- allows you to insert a division;
- Pick from drawing - allows you to copy values from drawing objects;
- Symbols - allows you to insert symbols from a list or from a symbol table;
- etc.

The presence of this or that command in the context menu depends on the purpose of the field.

The button in front of the dimension value input field allows you to add special characters to the dimension text.



Buttons *"First arrow"* and *"Second arrow"* allow you to select the type of arrows. When you click on them, a list with images of arrow options appears.

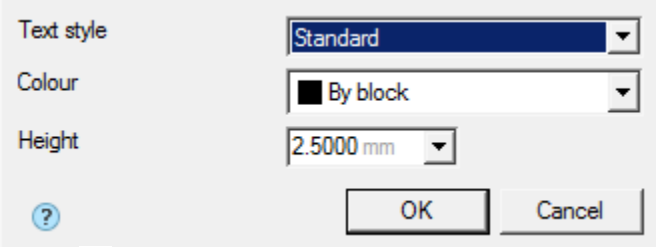
Button *"Test on leader"* - toggles the display mode of the text on the leader.

Button *"Text in rectangle"*- toggles the display mode of dimension text in a rectangle.

Button M1:100 *"Dimension value display scale"*- provides a list of display scales for the size value. Default 1:1.

Button *"Calculate value"* - calls the calculator to calculate the value.

Button *"Text style"* - calls the text settings dialog box.

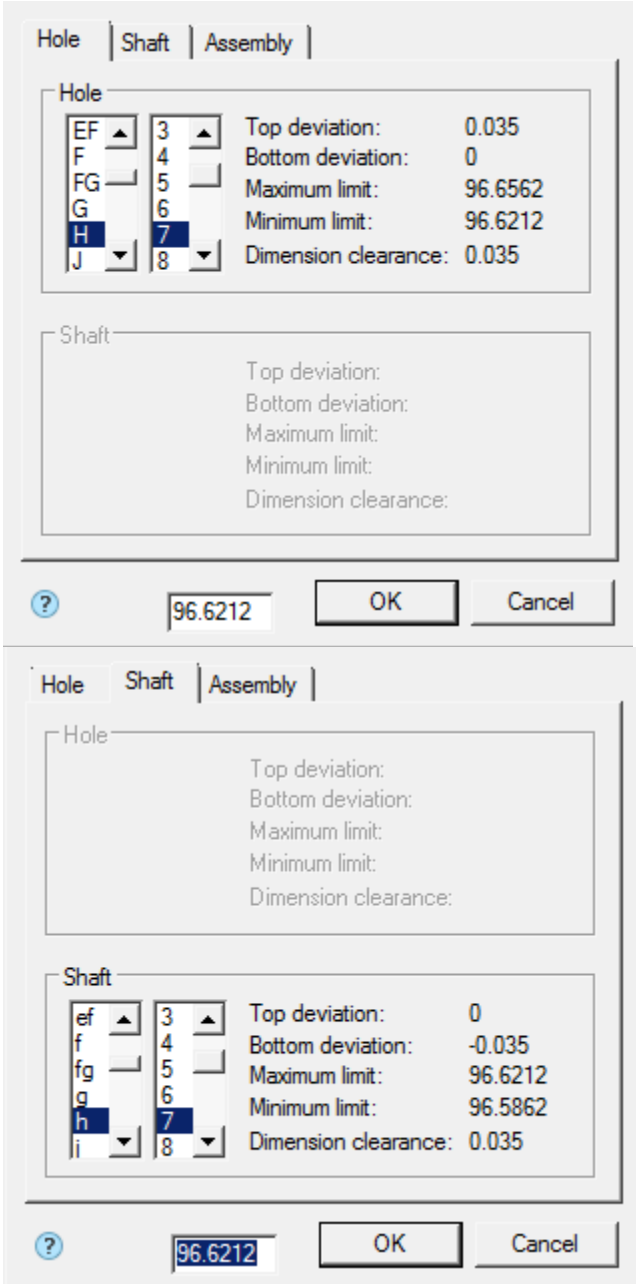


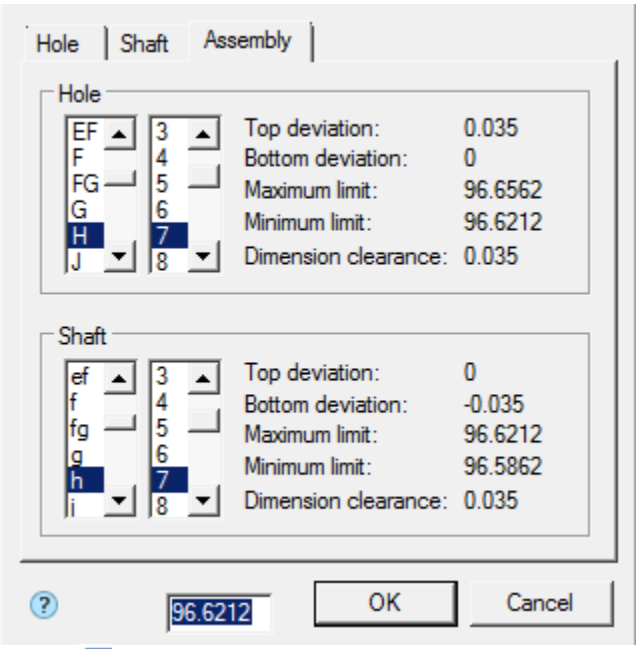
Button *"Match properties"*- allows you to copy properties from an existing dimension in the drawing.

Buttons *selection of brackets* [] () <> - allow you to set the dimension value in brackets.

Button *"Method of recording tolerances"* - provides a list of options for displaying tolerances.

Button *"Fit table"* - provides the ability to customize tolerance.


Hole
Shaft
Assembly
Hole
EF
F
FG
G
H
J
3
4
5
6
7
8
Top deviation: 0.035
Bottom deviation: 0
Maximum limit: 96.6562
Minimum limit: 96.6212
Dimension clearance: 0.035
Shaft
Top deviation:
Bottom deviation:
Maximum limit:
Minimum limit:
Dimension clearance:
96.6212
OK
Cancel
Hole
Shaft
Assembly
Hole
Top deviation:
Bottom deviation:
Maximum limit:
Minimum limit:
Dimension clearance:
Shaft
ef
f
fg
g
h
i
3
4
5
6
7
8
Top deviation: 0
Bottom deviation: -0.035
Maximum limit: 96.6212
Minimum limit: 96.5862
Dimension clearance: 0.035
96.6212
OK
Cancel



Button *"Technical conditions"*. Connects technical conditions and dimension.

Break and Restore dimension line

Break dimension line

Main menu: ***Dimensions - Break dimension***.

Main menu: ***Mechanical - Utilities - Breaks - Break dimension line***.

Ribbon: ***Annotate - Dimensions - Break dimension line***.

Ribbon: ***Mechanical - Utilities - Break dimension line***.

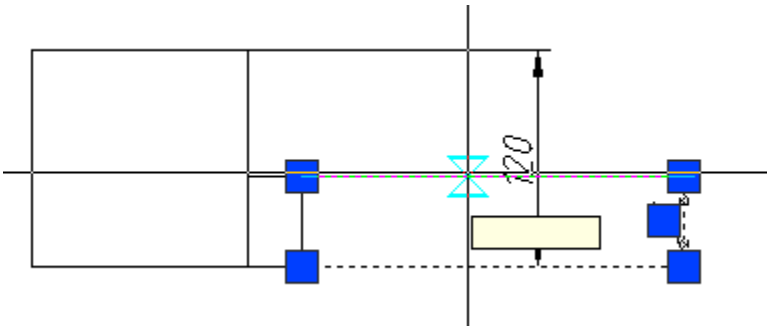
Toolbar: ***Dimensions - Break dimension line***.

Toolbar: MCS Breaks - ***Break dimension line***.

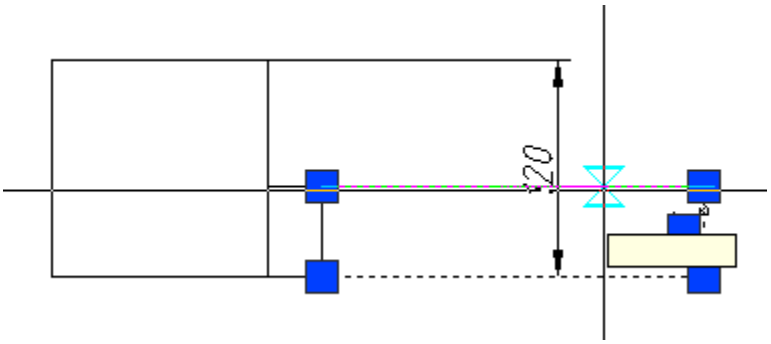
Command line: ***MCDIMBREAK, DIMBREAK***.

To create a dimension line break, you must:

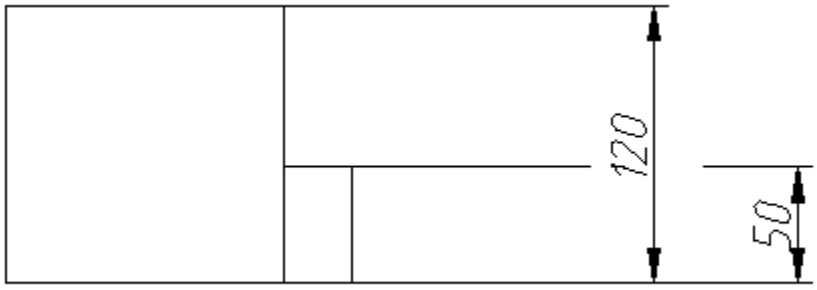
- Call the command;
- Select the dimension. If the size was selected before calling the command, you do not need to select it again;
- Select two points by specifying a section of the dimension line to create a break;
- The break will be made.



First point selection



Second point selection



Result

Restore dimension line

Main menu: ***Dimensions - Restore dimension***.

Main menu: ***Mechanical - Utilities - Breaks - Unbreak dimension line***.

Ribbon: ***Unnotate - Dimensions - Unbreak dimension line***.

Ribbon: ***Mechanical - Utilities - Unbreak dimension line***.

Toolbar: ***Dimensions - Unbreak dimension line***.

Toolbar: MCS Breaks - ***Unbreak dimension line***.

Command line: ***MCDIMUNBREAK, DIMUNBREAK***.

To restore the dimension line, you must:

- Call the command;
- Select the dimension. If the size was selected before calling the command, you do not need to select it again;;
- The restoration will be done.

Oblique dimension

Main menu: .

Ribbon: .

Command line: ***MCDIMOBLIQUE, DIMOBLIQUE***.

The command allows you to skew the dimension.

Procedure

1. Call the command *"Oblique Dimension"*.
2. Select the required dimensions. Press the *"Enter"* key to complete the selection.
3. Enter the angle of inclination and press the *"Enter"* key.
4. The selected dimensions will change the angle of inclination.

Arrange dimensions

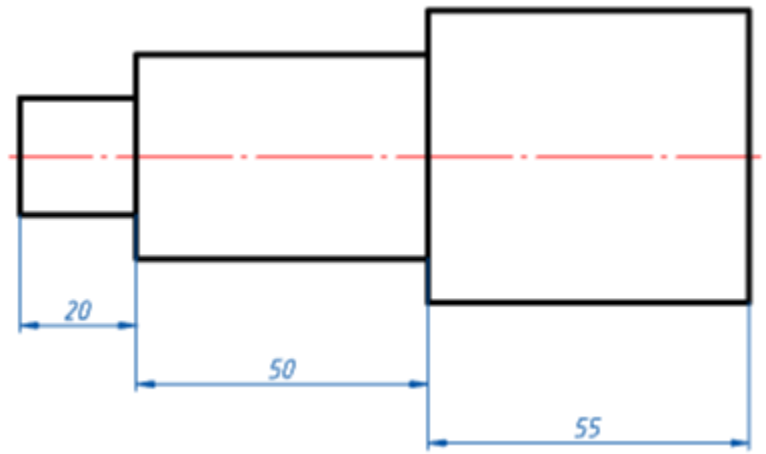
Toolbar: "MCS Dimensions" - ***Arrange dimensions***.

Command line: ***MCDIMARRANGE***.

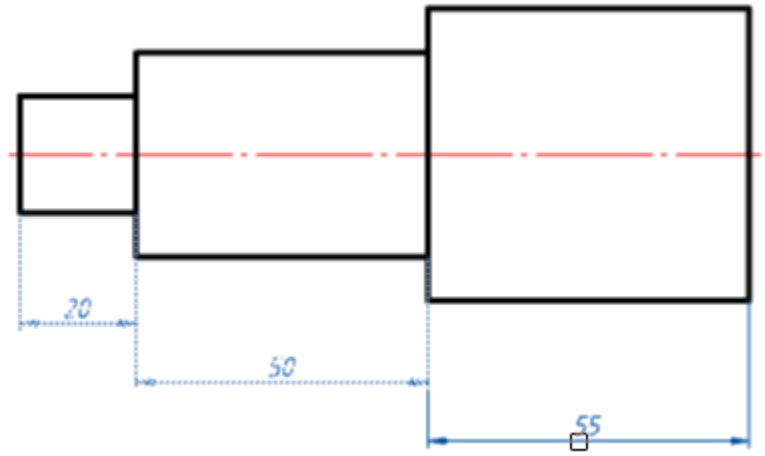
Procedure

To arrange, you need:

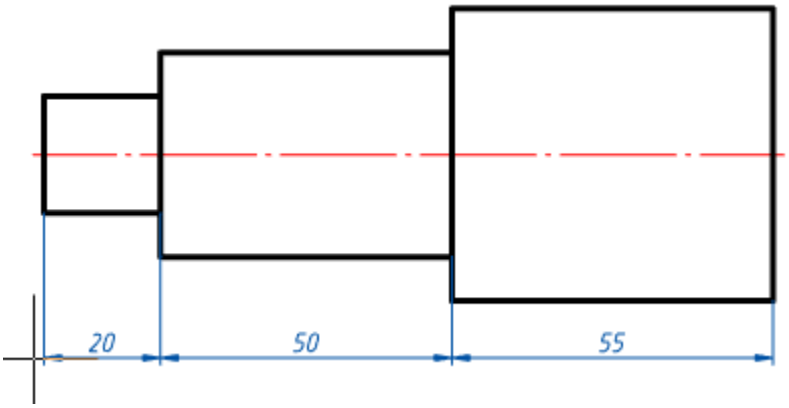
1. Call the command;



2. Indicate the dimensions to be aligned;



3. Confirm the selection with the *"Enter"* key;
4. Pick a new location for the dimension group.



Fit list

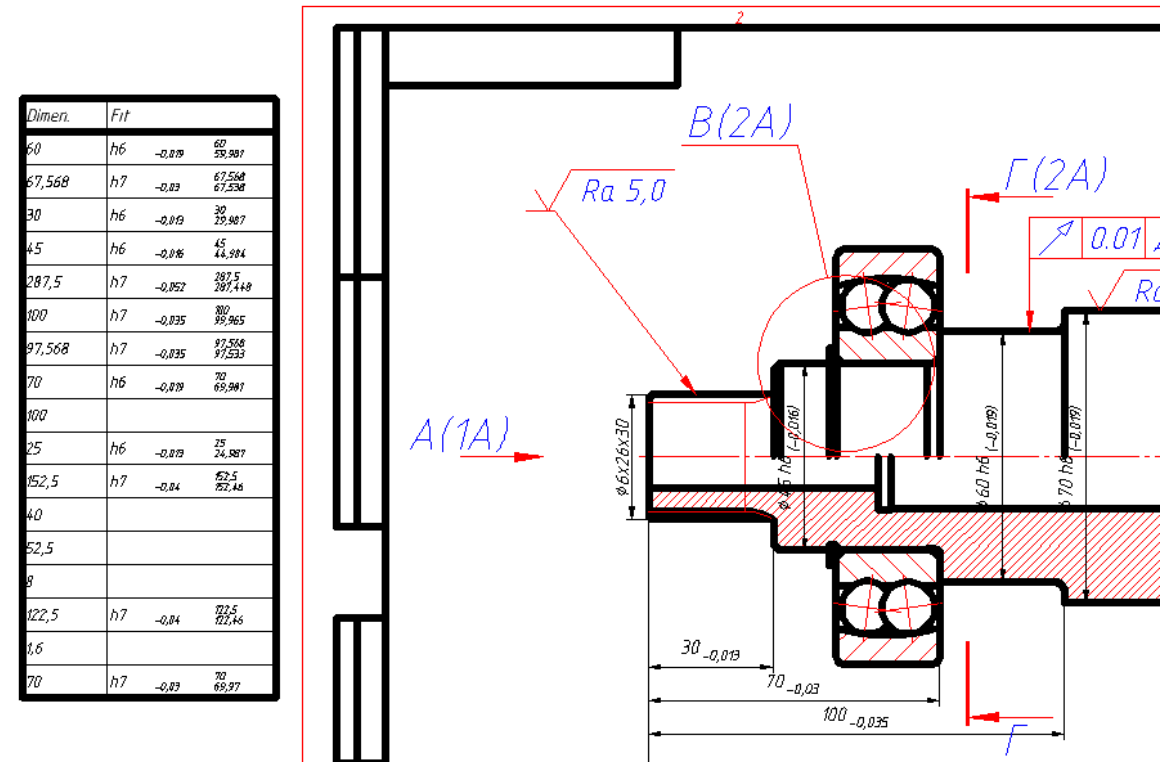
Toolbar: MCS Dimensions - ***Fit list***.

Command line: ***MCFITLIST***.

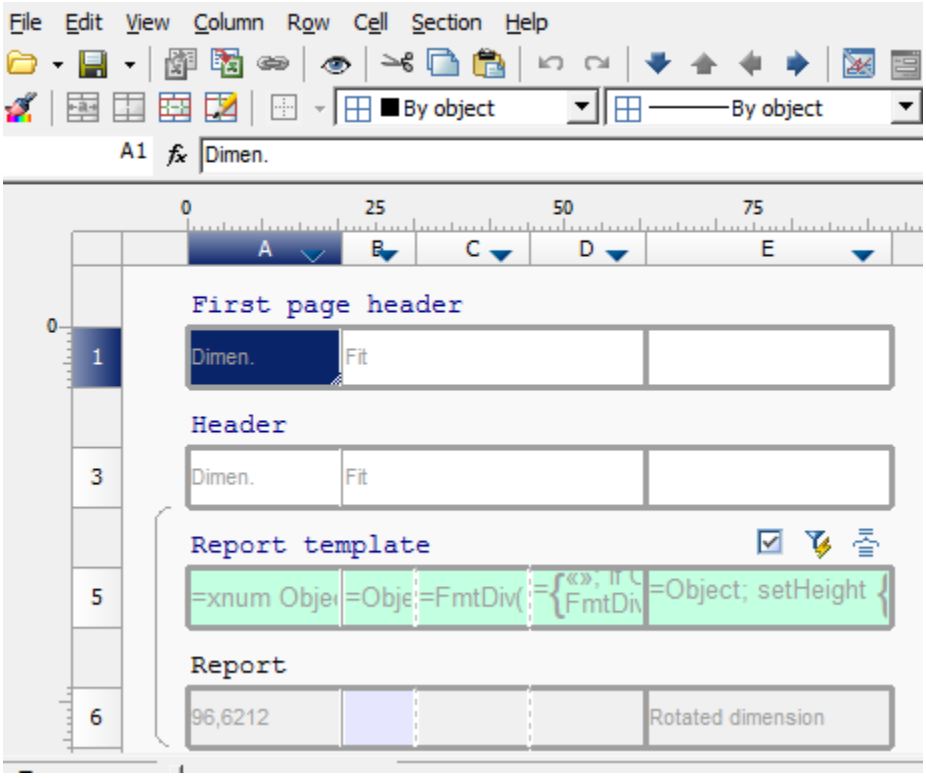
The command is designed to create an automatically updated table by dimensions in the drawing.

Procedure

1. Call the command;
2. Indicate the position of the table of maximum deviations in the drawing;
3. The table will be automatically filled in. The table is updated automatically when you delete, add or edit dimensions.



By double clicking, you can edit the tolerance table. For example, specify an alphanumeric designation for a size tolerance.



Symbols

Designation of surface texture

Main menu: *Mechanical - Symbols - Surface Texture - Surface texture*.

Ribbon: *Mechanical - Symbols - Surface texture*.

Toolbar: *Surface texture (toolbar* "MCS Symbols"*)*.

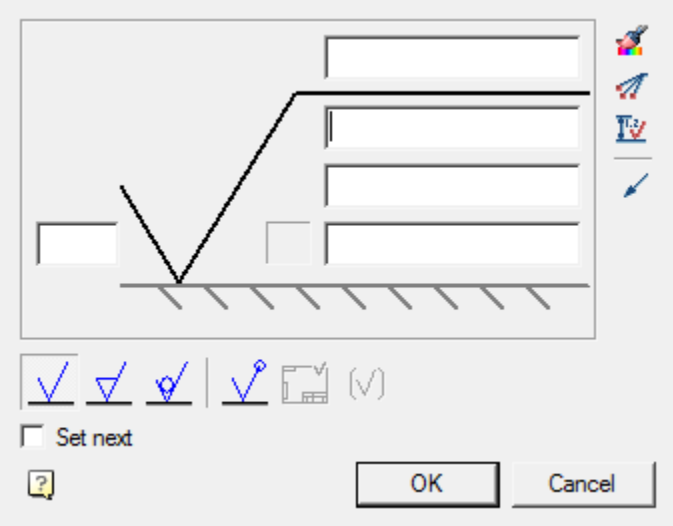
Command line: *MCSURF*.

This command sets the sign of surface texture.

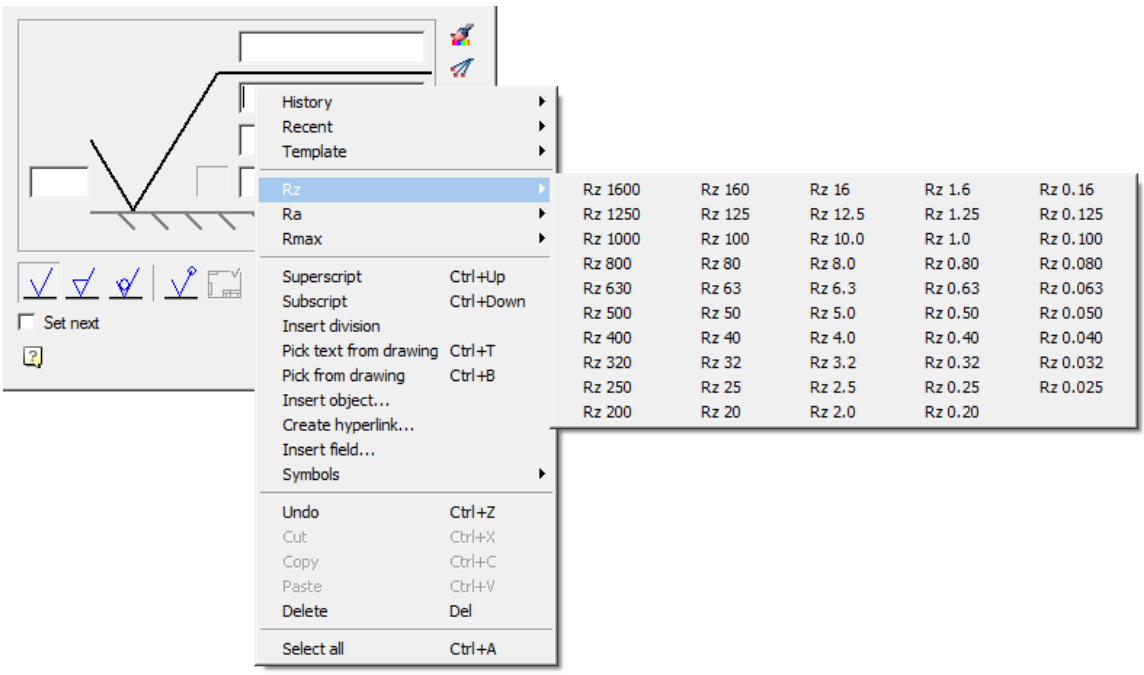
Procedure

1. Call command;
2. Pick a point on the object. When specifying the location of the specified segment is highlighted;
3. Specify the size and position of the callout. The sign is positioned normal to the surface. To place a character without a callout, press the spacebar;
4. Indicate the position of the roughness mark on the shelf. The editing dialog "Surface texture";
5. In the "Surface texture" dialog, specify the necessary parameters and click the "OK" button;

Dialog edit



Fields (4 field) - Allows you to enter surface texture values and text above and below the shelf.
In the input field through the context menu, you can select the roughness of the series:



Button ***"Surface lay and orientation"*** - allows you to choose the surface texture orientation.

Button ***"Match properties"*** - allows you to copy design properties from another surface texture.

Button ***"Add extension line"*** - allows you to add an additional extension line.

Button ***"Dimension dependent"*** - allows you to select a size for setting dependencies. The amount of roughness can be set automatically depending on the quality of the specified size. To do this, click on the "Dimension dependent" icon in the "Surface texture" dialog box and select the size with the quality mark.

Button *"Arrow type"* - allows you to set the appearance of the callout arrow.

Processing switch - controls the display of the processing method: Any manufacturing process permitted, Material should be removed, Material should not be removed.

Character switch ***"By contour"*** - controls the display of the symbol *"By contour"*.

Character switch ***"Surface texture not specified"*** - controls the display of the symbol *"Surface texture not specified"*.

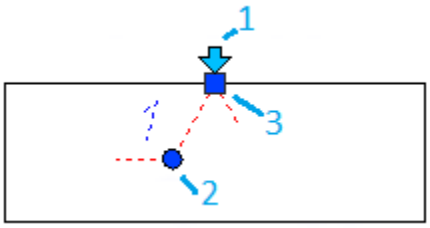
Character switch ***"Braked sign"*** - controls the display of the symbol *"Braked sign"*.

Switch "Set next" - Active when building a roughness symbol. When turned on, the switch goes into the mode of continuous construction of roughness symbols with values equal to the first.

Editing with grips

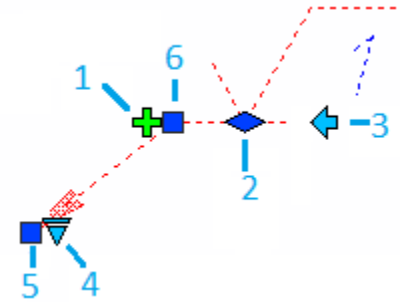
Designation surface texture without callouts

1. Grip flip;
2. Grip turn;
3. Grip movement.



Designation surface texture with callouts

1. Grip add callout line;
2. Grip of moving the position of surface texture marking along the shelf;
3. Shelf reflection grip;
4. Arrow type grip;
5. Grip move callouts;
6. Grip moving shelves.



Designation of surface texture (old)

Main menu: ***Mechanical - Symbols - Surface Texture-*** ***Surface texture (old)***.

Ribbon: ***Mechanical - Symbols -*** ***Surface texture (old)***.

Toolbar: ***Surface texture (old) (toolbar*** "MCS Symbols"***)***.

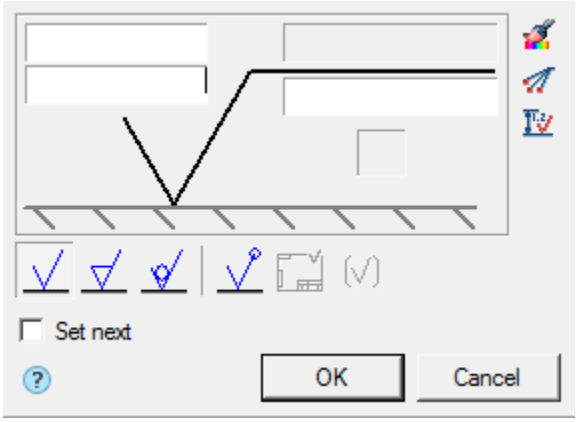
Command line: ***MCSURFOLD***.

The command sets the sign of the surface texture of previous versions.

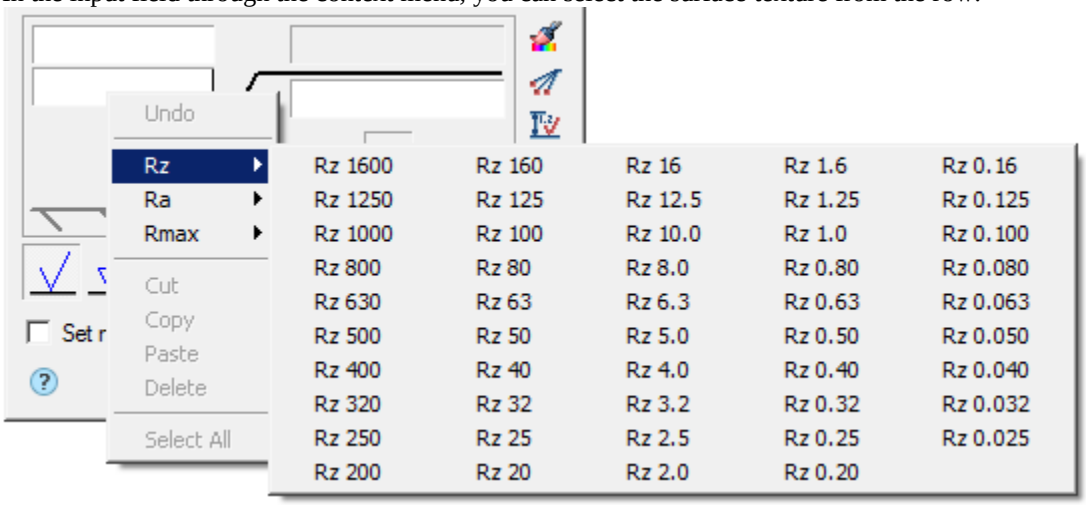
Procedure

1. Call command;
2. Pick a point on the object. When specifying a location, the specified segment is highlighted;
3. Specify the size and position of the callout. The sign is positioned normal to the surface. To place a character without a callout, press the "Space";
4. Specify the position of the surface texture sign on the shelf. The "Surface texture" edit dialog opens;
5. In the dialog "Surface texture" specify the necessary parameters and click "OK";

Dialog edit



Fields (4 fields) - allows you to enter roughness values and text above and below the shelf.
In the input field through the context menu, you can select the surface texture from the row:



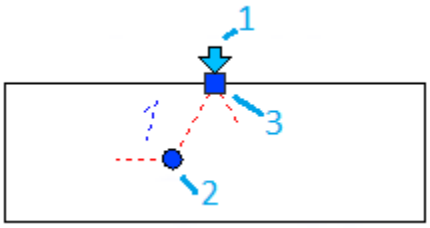
Button ***"Surface lay and orientation"*** - allows you to choose the surface texture orientation.
Button ***"Match properties"*** - allows you to copy design properties from another surface texture.
Button ***"Add extension line"*** - allows you to add an additional extension line.
Button ***"Dimension dependent"*** - allows you to select a size for setting dependencies. The amount of roughness can be set automatically depending on the quality of the specified size. To do this, click on the "Dimension dependent" icon in the "Surface texture" dialog box and select the size with the quality mark.

Processing switch - controls the display of the processing method: Any manufacturing process permitted, Material should be removed, Material should not be removed.
Character switch ***"By contour"*** - controls the display of the symbol *"By contour"*.
Character switch ***"Surface texture not specified"*** - controls the display of the symbol *"Surface texture not specified"*.
Character switch ***"Braked sign"*** - controls the display of the symbol *"Braked sign"*.
Switch "Set next" - Active when building a roughness symbol. When turned on, the switch goes into the mode of continuous construction of roughness symbols with values equal to the first.

Grips

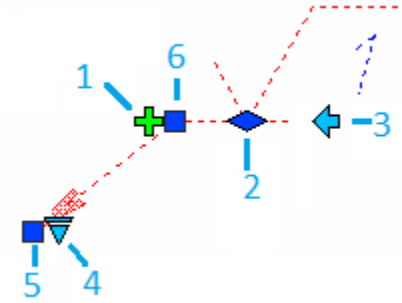
Designation surface texture without extension line

1. Grip flip;
2. Grip turn;
3. Grip moving.



Designation surface texture with extension line

1. Grip add extension line;
2. Grip of moving the position of surface texture marking along the shelf;
3. Shelf reflection grip;
4. Arrow type grip;
5. Grip relocation extension line;
6. Grip moving shelves.



Convert old surface texture symbol

Main menu: *Mechanical - Symbols - Surface Texture -* *Convert old surface texture symbol*.

Ribbon: *Mechanical - Symbols -* *Convert old surface texture symbol*.

Toolbar: *Convert old surface texture symbol (toolbar* "MCS Symbols"*)*.

Command line: *MCSURCONV*.

The command converts the old character surface texture to a new one.

Procedure

1. Call command;
2. Specify all the required old surface texture characters for the transformation. To complete the selection, press the "Enter" key;
3. The command converts old surface texture symbols to new ones.

Feature control frame

Main menu: *Mechanical - Symbols -* *Feature control frame*.

Ribbon: ***Mechanical - Symbols -*** ***Feature control frame***.

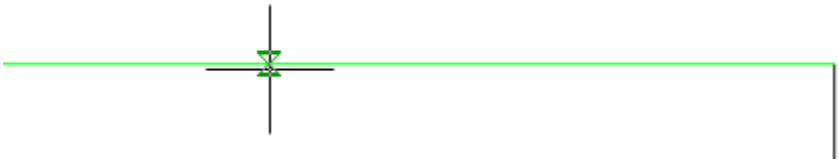
Toolbar: ***Feature control frame (toolbar*** "MCS Symbols"***)***.

Command line: ***MCTOL***.

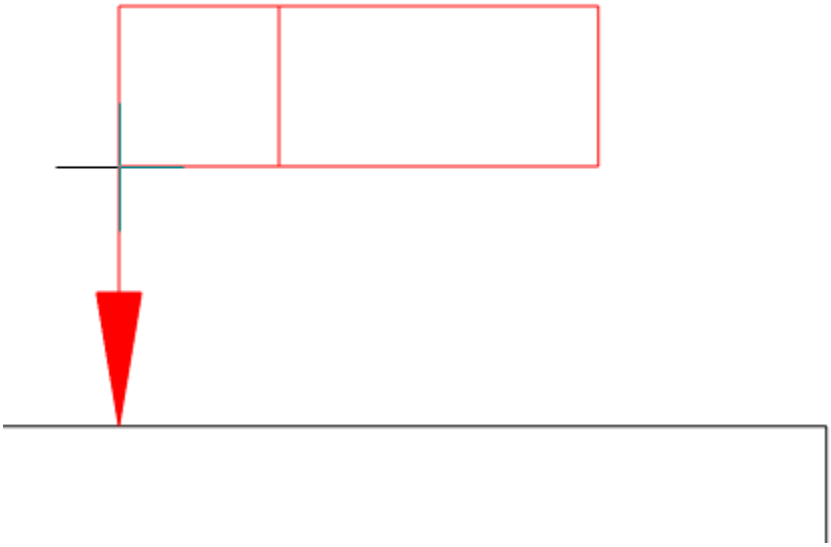
The tolerance can be set depending on the quality of the specified size.

Procedure

1. Call command.
2. Pick a point on the object. When you select a base surface, the graphic element is highlighted in green.

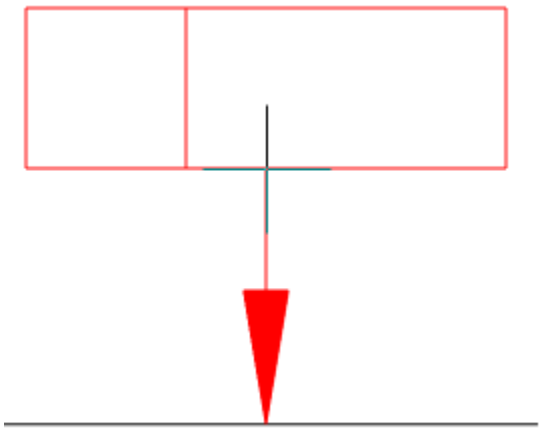
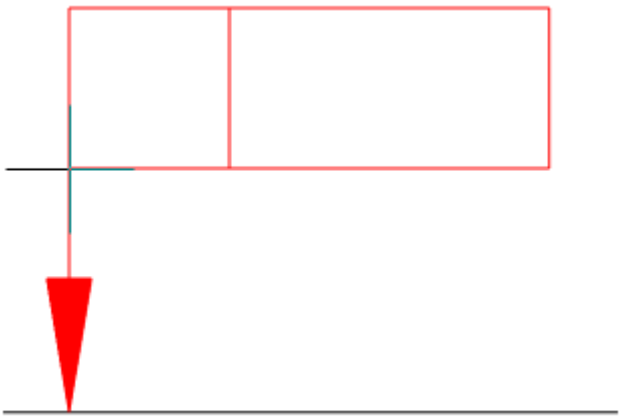


3. Point to the next point to pin the line of the mark. The sign line is drawn along the normal to the specified surface.

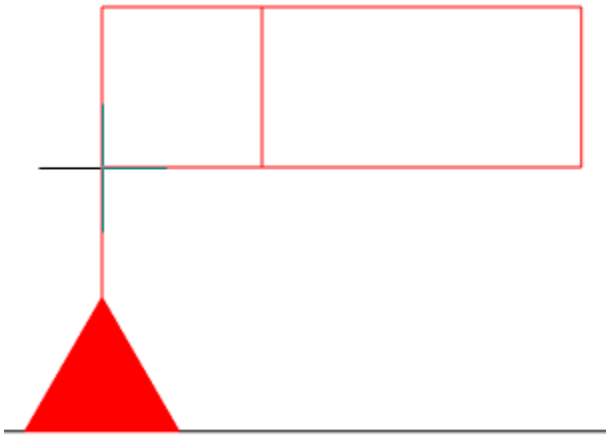


The sign line may extend from the corner of the frame or from the middle of its side. To control the display of these elements, you must call the context menu and select "Sides" or "Angles".

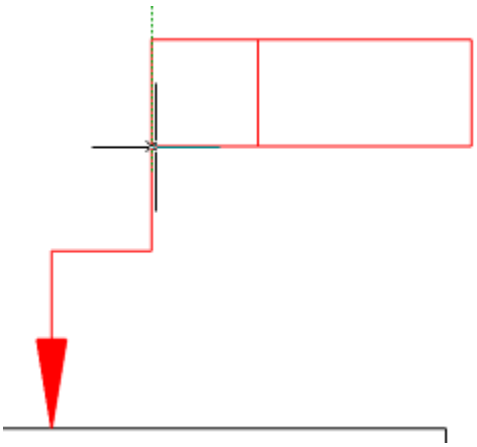
Sides	Angles

Sides	Angles
	

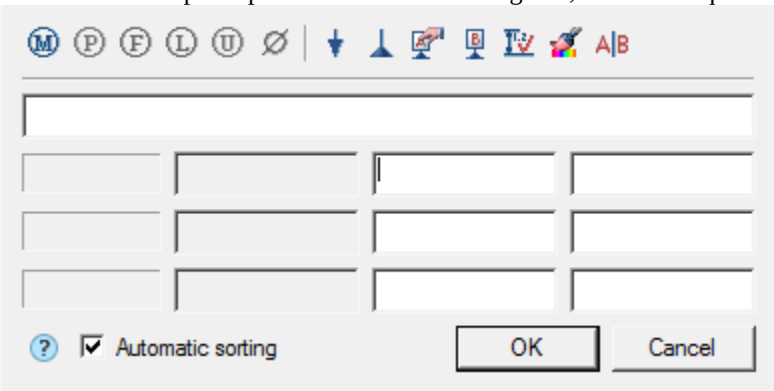
When you select the "Base" line in the context menu, a triangle symbol is drawn on the leader.
When you select the "Tolerance" line in the context menu, an arrow is drawn on the callout.

Base	Tolerance
	

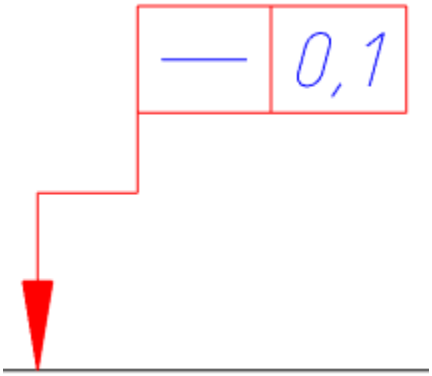
4. Next, place a tolerance mark in the place you need. To do this, using the mouse cursor, specify new points in the drawing. To undo the last action, use the "Back" command from the context menu. To finish, press the "Enter" key, a dialog box will appear *"Form&plane position tolerance"*.



5. In the "Form&plane position tolerance" dialog box, enter the required parameters and click the "OK" button.

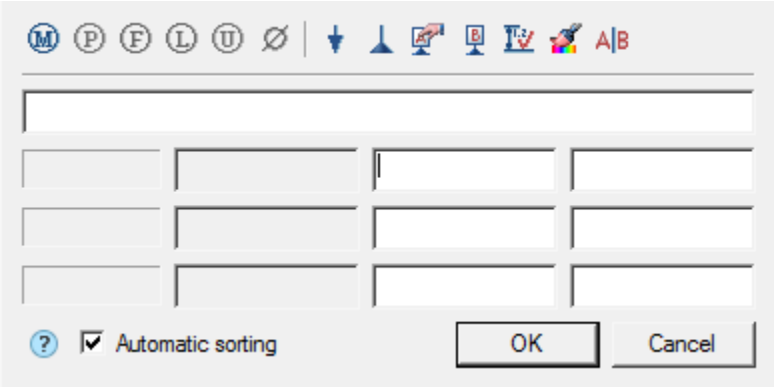


6. The tolerance of the form and plane will be built.



Dialog edit

The editing dialog consists of a toolbar and a workspace.



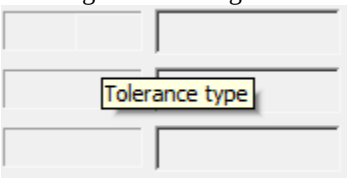
Workspace

The working area of the dialog box is a list of 4 lines, allowing add header and form 3 rows of values in the tolerance.

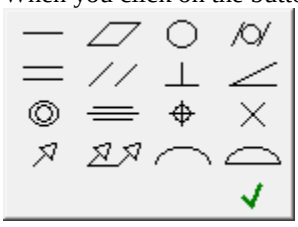
Lines are divided into 4 columns:

- *Type of tolerance column*

The field of this column allows you to add a type of tolerance. In each field, you can enter 2 types of tolerance by selecting the left or right button of the field.

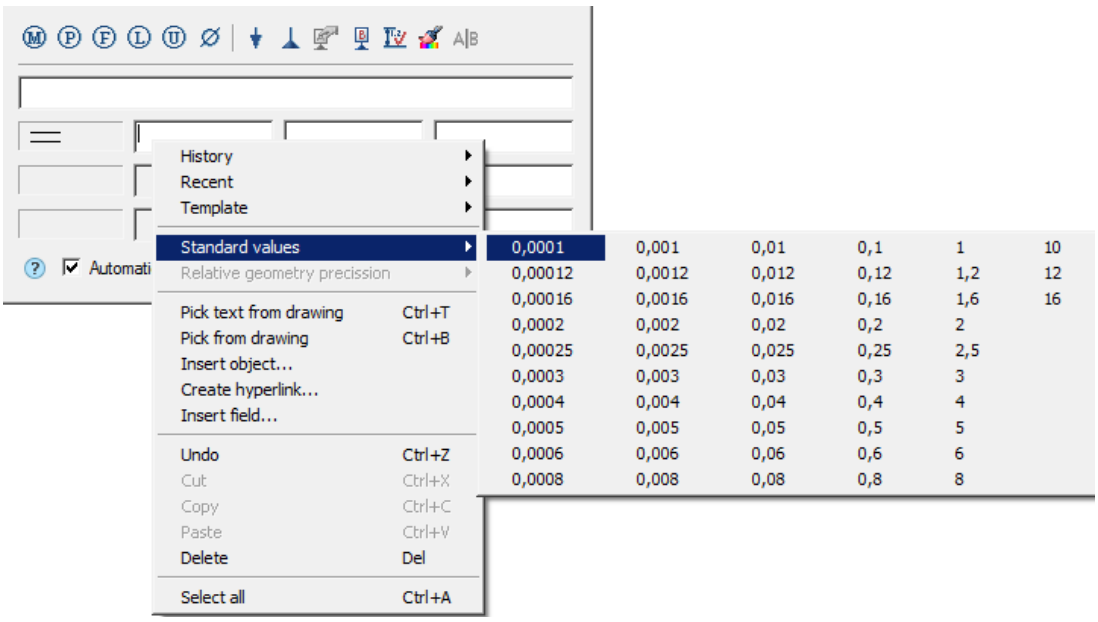


When you click on the buttons, a form for selecting the type of tolerance appears.



- *Column of tolerance deviation*

The tolerance deviation value is entered in the fields of this column. The tolerance deviation value can be taken from the "Standard values" when the context menu is opened in the input field.



- *Base designation column*

In the fields of this column, the notation for bases is entered.

- *Column notes*

Notes are entered in the fields in this column.

Dialog Toolbar

Dependent tolerance - the command inserts the designation "Dependent tolerance" in the columns "tolerance deviation values" and "bases designation".

Extended tolerance range - the command inserts the symbol "Extended tolerance range" in the column "tolerance deviation values".

Extended tolerance range - the command inserts the symbol "Non-rigid part" in the column "tolerance deviation values".

Extended tolerance range - the command inserts the symbol "Least material requirement" in the column "tolerance deviation values".

Extended tolerance range - the command inserts the symbol "Unequally-disposed modifier" in the column "tolerance deviation values".

Diameter - the command inserts the designation "Diameter" in the column "tolerance deviation values".

Set additional leader - the command allows you to set an additional leader for an editable tolerance.

Set additional base - the command allows you to set an additional base for an editable tolerance.

To install additional callouts, you must:

1. Call command;
2. Select object;
3. Bring the callout line to the tolerance mark.

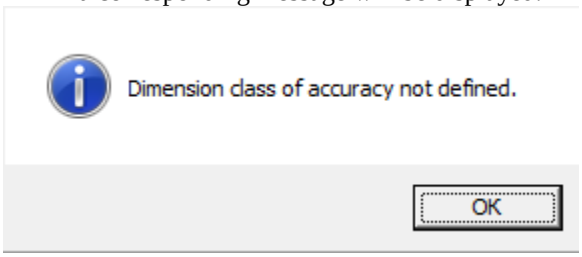
Specify base - the command allows you to specify the base, to do this, call the command and point to the previously placed base symbol on the drawing. The letter designation of the base will be automatically applied in the current input field. When linking the base designation in the drawing, the cursor must be in the base designation input field.

Insert base denotation - allows you to insert a base symbol when it is missing in the drawing.

Linked to dimensions - the command serves to fill the deviation value depending on the quality of the specified size.

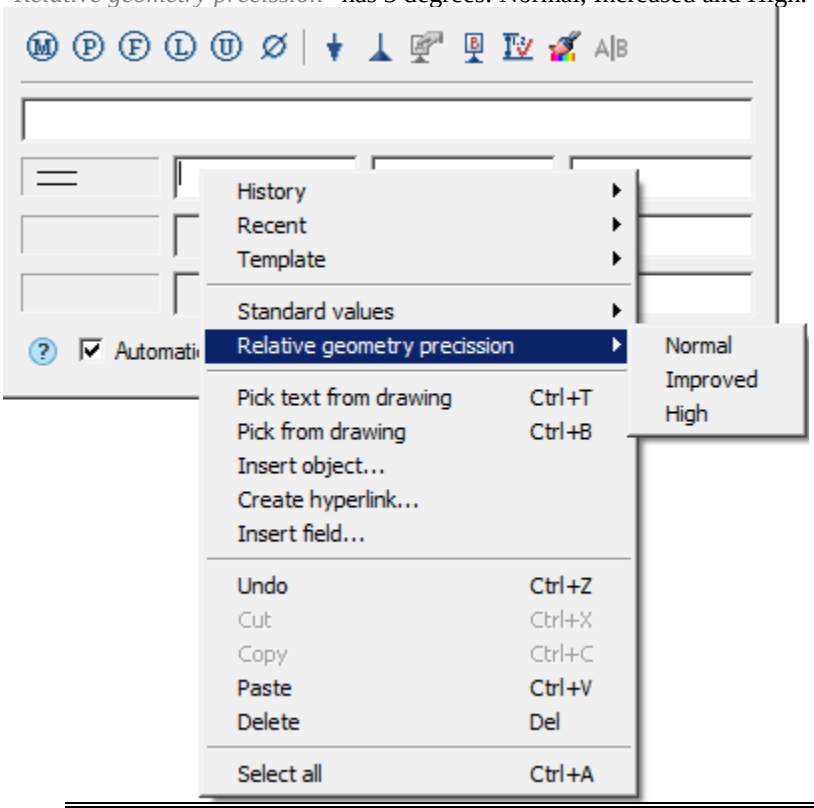
To fill the value of the deviation is necessary:

1. Select the type of deviation.
2. Call command *"Linked to dimensions"*.
3. Indicate the size with the assigned quality. A link will be created, otherwise, if the qualification is not affixed, a corresponding message will be displayed.



4. From the context menu of the deviation value input field, select "Relative geometry precission".

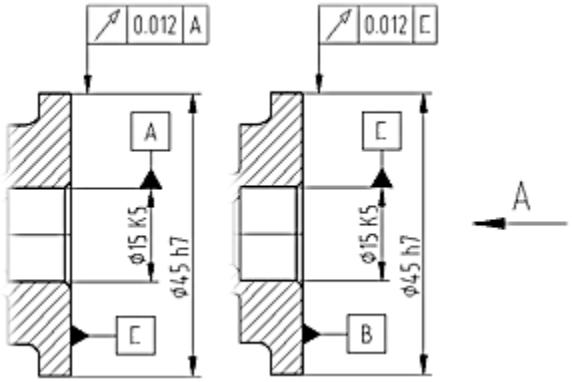
"Relative geometry precission" has 3 degrees: Normal, Increased and High.



Note:	If a mark for the shape deviation mark has been applied to the size with the specified quality, the context menu bar of the tolerance value "Relative geometry precission" will be active. The relationship with the size is NOT associative: the tolerance field established relative to the size will NOT change along with the size value. Moreover, all information about the connection is lost after closing the edit edit form admission dialog. Thus, if it is necessary to select the value for admission again, the connection will have to be imposed again.

Match properties - this command allows you to copy values from the tolerance already entered.
Insert separator - the command inserts a separator into the database designation field.
Switch *Automatic sorting* - manages automatic database sorting. When you turn on the designation of bases are put alphabetically, taking into account the previously used letter designations of species, sections, sections.

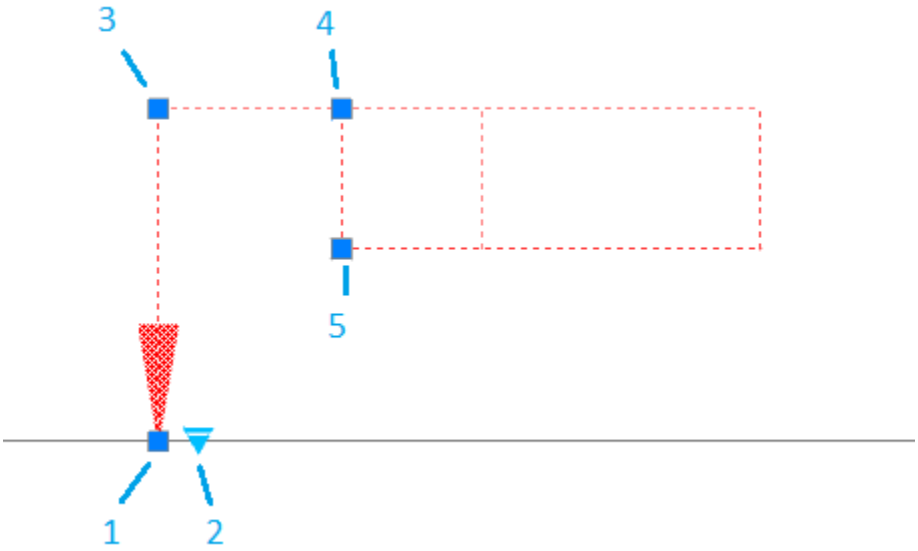
Note:	Deleting symbols of basic surfaces or drawing additional designations of types, sections, sections will lead to automatic re-sorting of all associated text symbols, including references to them in the technical requirements.



Put down two designations of the basic elements: "A", "B". Added view "A" to drawing. The designation of bases was adjusted automatically.

Grips

1. Grip moving arrows.
2. Arrow type grip.
3. Grip move the nodes of the arrow (maybe several).
4. Grip joining the sign. Allows you to change the point of attachment of the leader line to the sign both in the corners and along the sides.
5. Sign move grip.



Datum identifier

Main menu: ***Mechanical - Symbols -*** ***Datum identifier***.

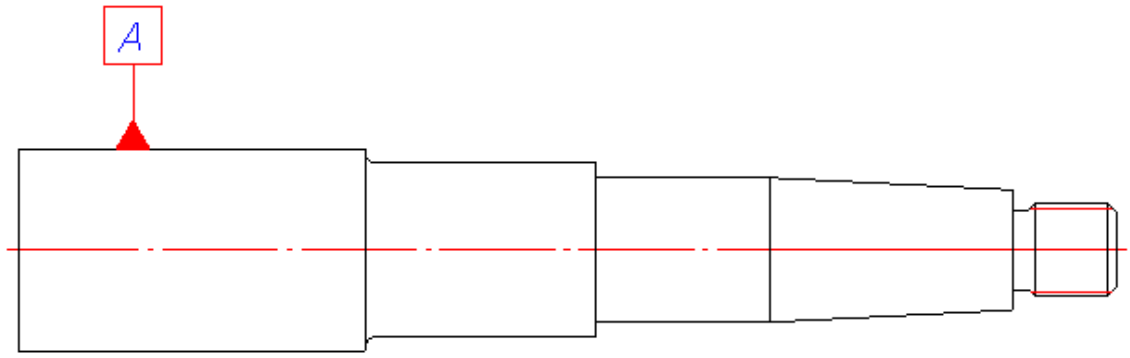
Ribbon: ***Mechanical - Symbols -*** ***Datum identifier***.

Toolbar: ***Datum identifier (toolbar*** "MCS Symbols"***)***.

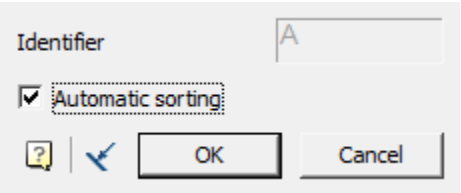
Command line: ***MCDATID***.

Procedure

1. Call command;
2. Pick a location point. When you select a base surface, the graphic element is highlighted in green;
3. Indicate the position of the mark. The designation of the base will automatically select a free letter.



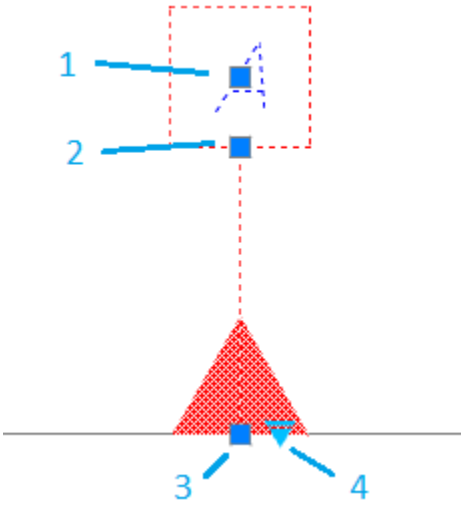
Dialog edit



Field *"Identifier"* - input letter field notation.
Switch *"Automatic sorting"* - manages auto sort identifier. If the switch is on, the letter is taken automatically from the free and the "Identifier" field is inactive.
Switch *"Attach"* - allows you to attach the base to the object. To attach the base to the object, you must enable the "Attach" switch, select the anchor object and set the new position of the base character.

Editing with grips

1. Grip Rotate - allows you to rotate the sign left or right 90 degrees.
2. Grip direction of the callout or handle move the base. If the attachment base, the knob moves the entire base relative to the attached object. If the base is detached, then the knob controls the direction of the leader line.
3. Grip disconnection - when using this handle, the base is disconnected from the object.
4. Grip select the type of arrow - allows you to select the type of arrow from the drop-down list.



Views, cuts, sections

Views

Main menu: ***Mechanical - Symbols - Views, Sections -*** ***View***.

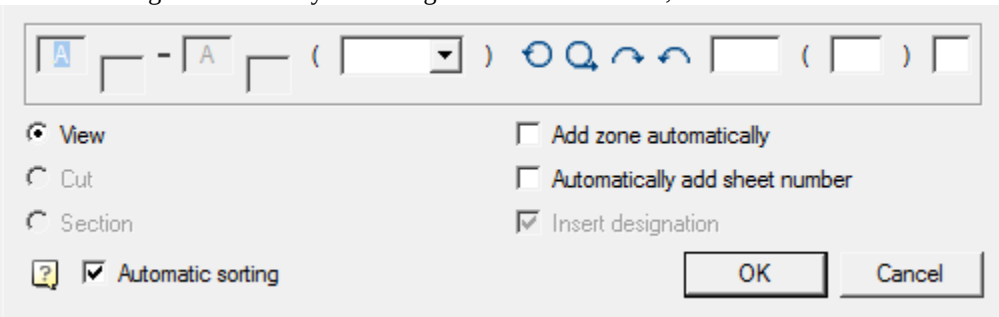
Ribbon: ***Mechanical - Symbols -*** ***View***.

Toolbar: ***View (toolbar*** "MCS Symbols"***)***.

Command line: ***MCVIEW***.

Procedure

1. Call command;
2. Specify the insertion point of view;
3. Specify the direction of the view arrow. A dialog box appears *"View"*;
4. In the dialog *"View"* enter your settings. Press button *"OK"*;



5. Place the view symbol in the drawing.

Note:	When you transfer the designation by means of nanoCAD, the integrity of the data in terms of specifying the mutual location and specifying the zones of the drawing can be violated. In this case, you must run the "Update" command.

Editing

Editing is done by double-clicking the LMB on any element of the symbol. This opens a dialog box "Views, cuts, sections", where you can change the required parameters.

Section

Main menu: ***Mechanical - Symbols - Views, Sections - Sections***.

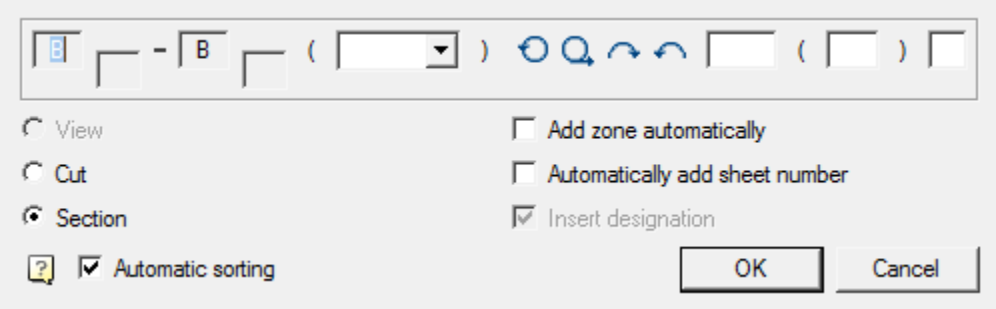
Ribbon: ***Mechanical - Symbols - Sections***.

Toolbar: ***Sections (toolbar*** "MCS Symbols"***)***.

Command line: ***MCSECT***.

Procedure

1. Call command;
2. Select the path type: Arc or Segment. Each type has its own order of construction;
   - *Type Segment.* Allows you to create a section as a polyline;
     - Specify the first point of the section;
     - Specify the subsequent cut points. To finish entering points, press *"Enter"*;
   - *Type Arc.* Creates an arcuate cut;
     - Select the desired arc construction method: Arc (3 points), Arc (2 points and bend), Arc (2 points and center);
     - Draw an arc in accordance with the selection.
3. Select the direction of the cut arrows. A dialog box appears *"Section"*;
4. In the dialog box, enter the required parameters. Press the button *"OK"*;



5. Insert the section symbol in the drawing.

Note: When you transfer the designation by means of nanoCAD, the integrity of the data in terms of specifying the mutual location and specifying the zones of the drawing can be violated. In this case, you must run the "Update" command.

Editing

Editing is done by double-clicking the LMB on any element of the symbol. This opens the "Views, cuts, sections" dialog box, where you can change the required parameters.

Scale fragment view

Main menu: ***Mechanical - Symbols - Views, Sections - Scale fragment view***.

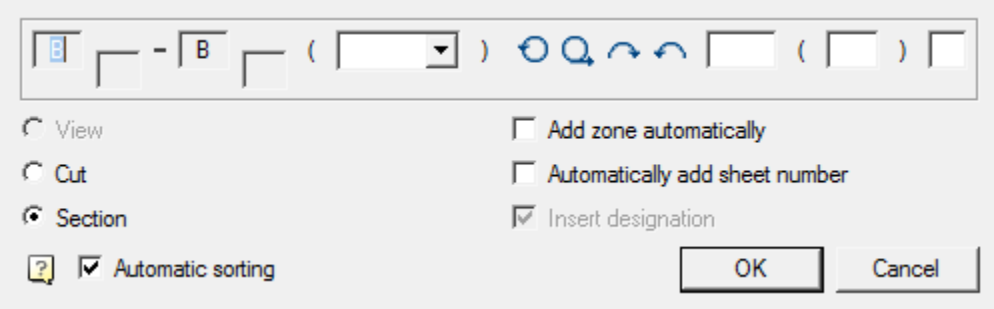
Ribbon: ***Mechanical - Symbols - Scale fragment view***.

Toolbar: ***Scale fragment view (toolbar*** "MCS Symbols"***)***.

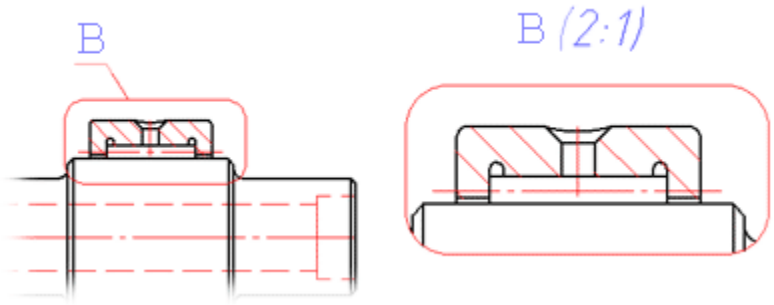
Command line: ***MCEVIEW***.

Procedure

1. Call command;
2. Specify the center of the circle (rectangle);
3. Select the type of contour: Circle or Rectangle;
4. Specify the size of the circle (rectangle);
5. Specify the position of the symbol shelf. A dialog box appears *"Views, cuts, sections"*;
6. In the dialog box, fill in the required fields. Press *"OK"*;



7. Specify the insertion point of the symbol.



Placing an enlarged view of the selection

Note:	When you transfer the designation by means of nanoCAD, the integrity of the data in terms of specifying the mutual location and specifying the zones of the drawing can be violated. In this case, you must run the "Update" command.

Editing

Editing is done by double-clicking the LMB on any element of the symbol. This opens the "Views, cuts, sections" dialog box, where you can change the required parameters.

View and section label

Main menu: ***Mechanical - Symbols - Views, Sections- View and section label***.

Ribbon: ***Mechanical - Symbols - View and section label***.

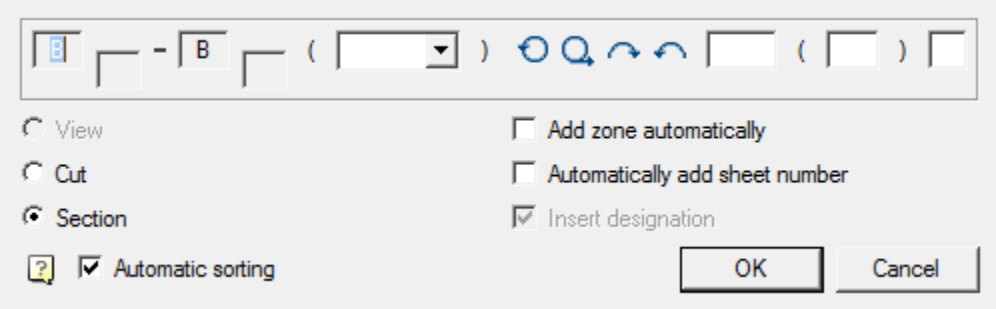
Toolbar: *View and section label (toolbar* "MCS Symbols"*)*.

Command line: ***MCVSDESC***.

Important!	You cannot insert a view (section) symbol separately from the element itself in automatic sorting mode.

Procedure

1. Call command;
2. Specify the insertion point of the symbol. A dialog box appears *"Views, cuts, sections"*;
2. In the dialog box, fill in the required fields. Press the button *"OK"*;



Editing

Editing is done by double-clicking the LMB on any element of the symbol. This opens the "Views, cuts, sections" dialog box, where you can change the required parameters.

Sort views and sections...

Main menu: ***Mechanical - Symbols - Views, Sections - Sort views and sections...***.

Ribbon: ***Mechanical - Symbols - Sort views and sections...***.

Toolbar: *Sort views and sections... (toolbar* "MCS Symbols"*)*.

Command line: ***MCSORT***.

Important!	For the command to work, all symbols in the drawing must be bound to objects.

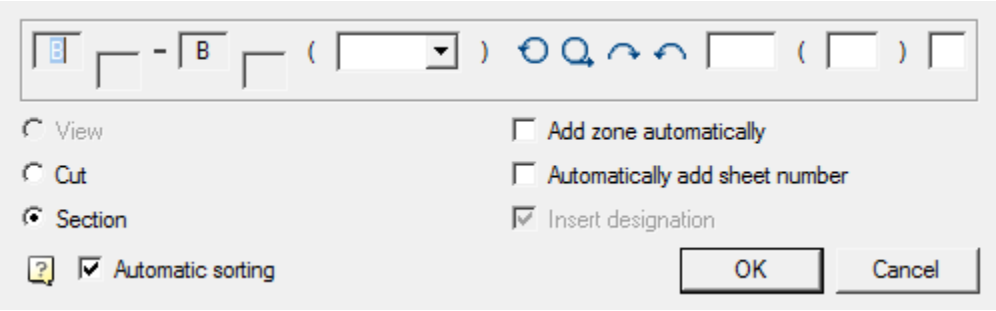
This command is used if views, sections, and sections were built without auto-sorting.

Procedure

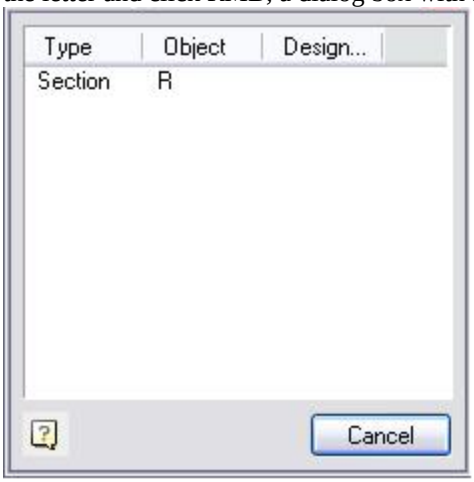
1. Call command.
2. Will be sorting types, cuts and sections (especially designated species, then the cuts-section).

Dialog for editing views, cuts, sections

One of the features of nanoCAD Mechanica 20.1 in the application of symbols of types, sections, sections is to support the functions of design control. We have implemented alphabetical arrangement of the letter designation, automatic generation of the zone designation and the sheet in the drawing.
The view, section, section, and detail symbol command allows you to dynamically link a symbol to information in technical requirements.



Letter and index input fields - allows you to enter letters and indexes of views, cuts, and sections.
When auto sort is enabled, fields are not editable.
With auto sort enabled, it is possible to see previously used letters. To do this, place the cursor in the input field of the letter and click RMB, a dialog box with a list of the presented letters will appear.



List "Scale" - allows you to assign a scale.
Switch "Turned" and "Unfolded" - allow to insert to the symbol characters are rotated and deployed.
Field "Zone" - allows you to specify a reference to a zone.
Field "Sheet" - allows you to specify a reference to the sheet number.
You can configure the type of the sheet number reference designation record in the setup "Sheet template".
Group switch designation type - switches the symbol type from "Cut" to "Section" and back.
Switch "Automatic sorting" - controls the automatic designation of views, sections, and sections.
If the switch "Automatic sorting" is enabled, first of all the views are indicated, then the sections.
To comply with the automatic sorting rule, you must specify whether the specified symbol belongs to a particular type of symbol.
In the global settings of the system, the user can enable the function of automatic placement of zones in the drawing - in this case, when you move the designation to another place in the drawing or change the format of the frame, the designation of zones in all sheets will be automatically changed.
Switch "Add zone automatically" - automatic zone detection.
Switch "Automatically add sheet number" - automatic detection of the sheet.
Switch "Insert designation" - controls the ability to insert a text symbol for a view, cut, section .

Welds

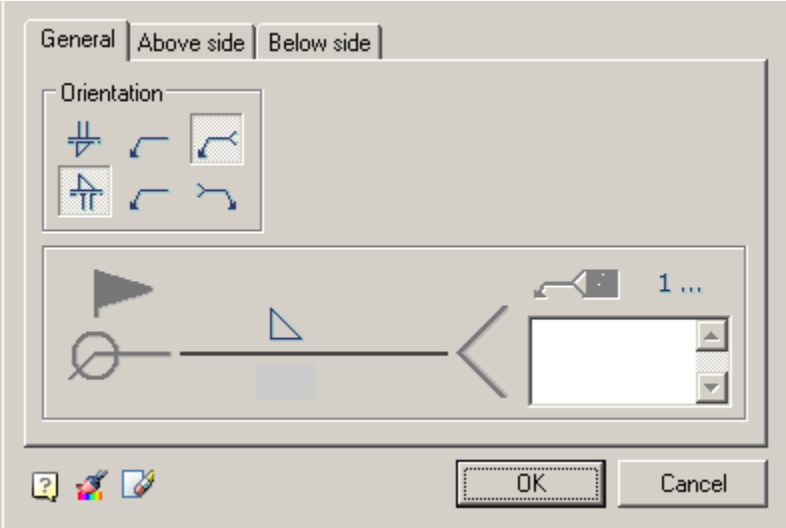
Welding...

Main menu: ***Mechanical - Symbols - Permanent connection - Welding...***.

Toolbar: "Symbols" - ***Welding...***.

Command line: ***MCISOWELD***.

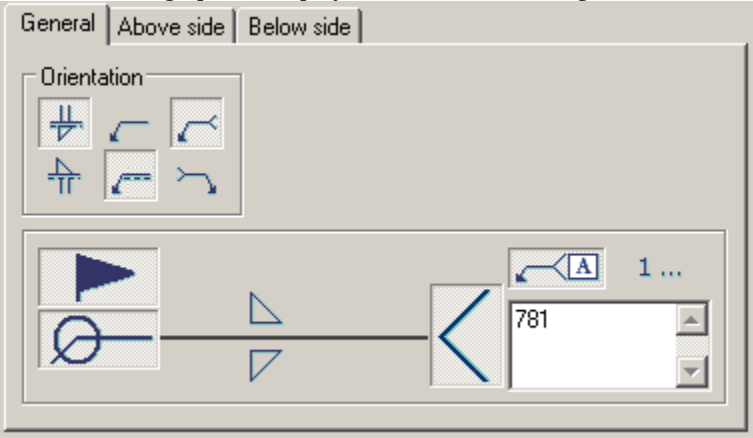
Setting notes refer to welding.



The dialog box consists of three tabs:

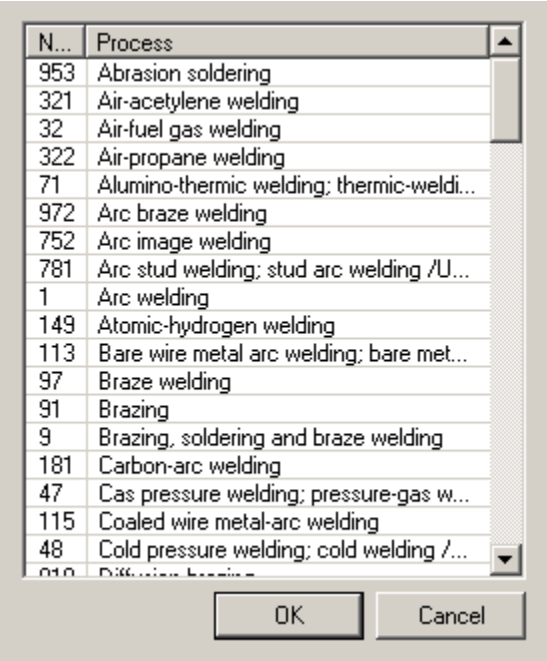
General

On the tab, the graphical display and fill in the drawing note.

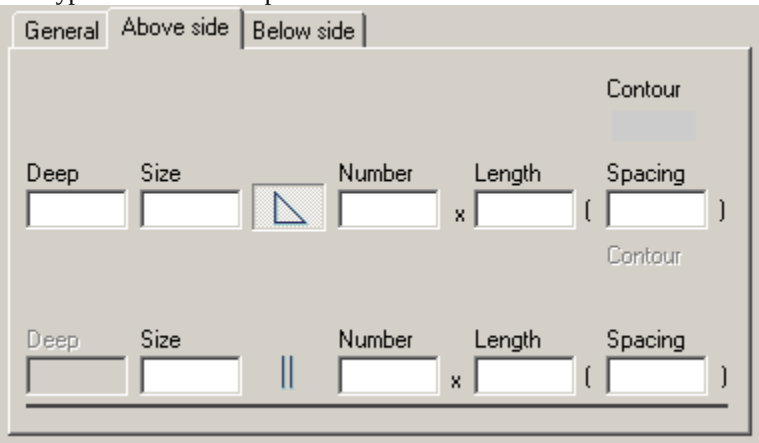


For convenient entry process, you can use Process list.

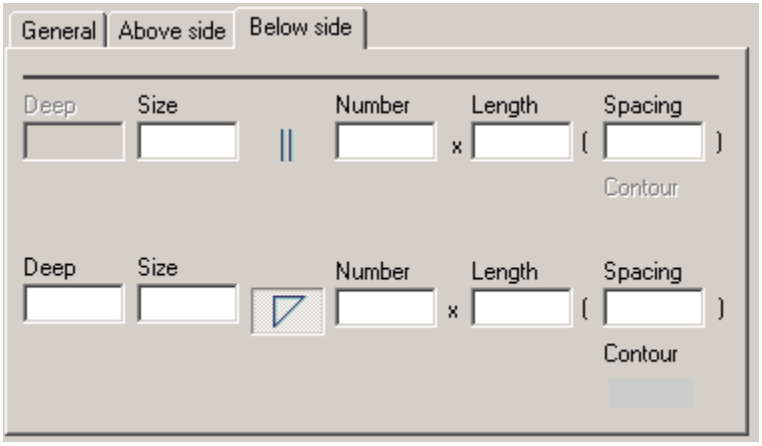
Process list poddrerzhivaet multiple selection processes.



Above side
Tab necessary to fill in the values above the side
The type of weld and its parameters.



Below side
Tab necessary to fill in the values below the side
The type of weld and its parameters.



Simple welding

Main menu: ***Mechanical - Symbols - Permanent connection -*** ***Simple welding***.

Toolbar: "Symbols" - ***Simple welding***.

Command line: ***MCWLEG***.

This command is used to render the weld leg between two intersecting segments.

1. Enter *the first rib*.
2. Enter *the second rib*.
3. Press *"Enter"* or right mouse button to exit the cyclic command.

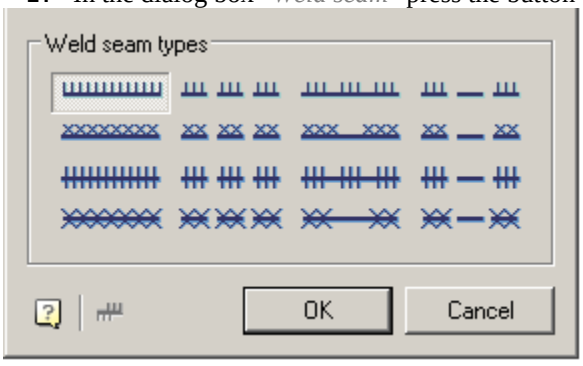
Weld seam

Main menu: ***Mechanical - Symbols - Permanent Connection-*** ***Weld seam***.

Toolbar: "Symbols" - ***Weld seam***.

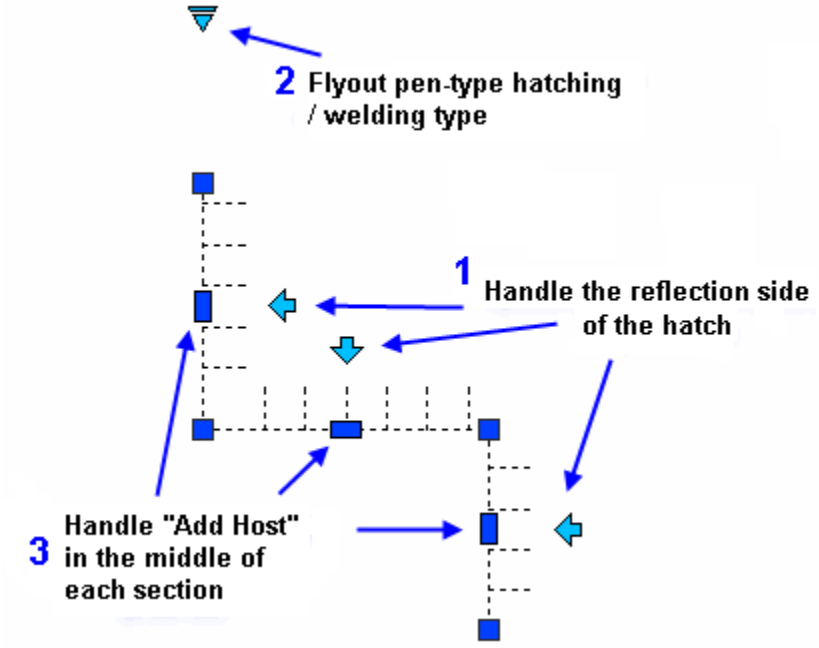
Command line: ***MCWELD***.

1. In the dialog box *"Weld seam"* press the button corresponding to the desired type of weld. Click OK.



2. Specify *the first point of the circuit* (or P - polyline, C - circle M - magnet).
3. Specify *the next point of the contour* (or L - line A - arc, C - close, U - rollback, M - a method for constructing the arc).
4. Press *Enter* or right mouse button to exit the cyclic command.

In the dialog box are available 16 types of welds. Each type of weld corresponds to the button with his picture. To select the desired type of seam, press the button and confirm with the OK button.
Construction of the circuit is made by the following methods:
L - via line: construction of straight segments of the weld points introduced;
A - by arc: construction of the weld arc segments in three points;
C - close: automatic generation of the last segment, closing the main line of the weld;
U - recoil: undo the last segment of the weld;
M - a method for constructing the arc: T - tangentially, S - by three points.
Button *Side hatch* allows you to select the direction of the hatch.
Change the direction of the hatch, you can select the "handle" the reflection side hatch:



Fixed joint

Main menu: ***Mechanical - Symbols - Permanent connection -*** ***Fixed joint***.

Ribbon: ***Mechanical - Symbols -*** ***Fixed joint***.

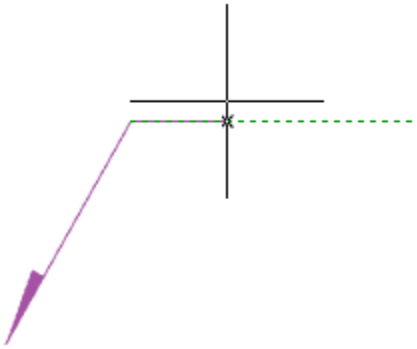
Toolbar: "MCSSymbols" - ***Fixed joint*** .

Command line: ***MCFIX***.

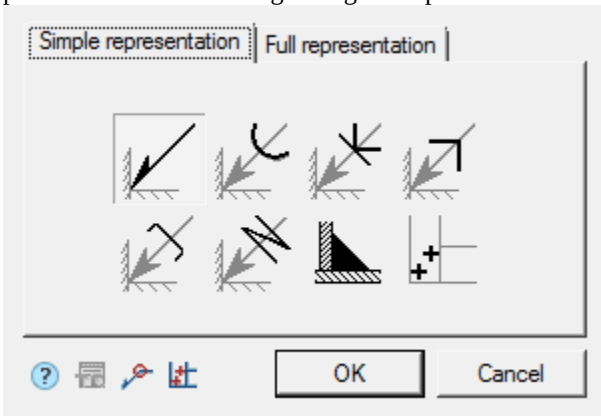
Procedure

1. Call the command *"Fixed joint"*.
2. Indicate the place of the fixed joint.

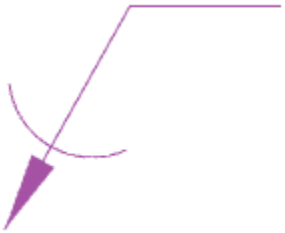
3. Pick a break point.



4. Specify the end point of the leader or press the *"Enter"* key, then the end point of the leader will be the break point. The "Joints" editing dialog will open.

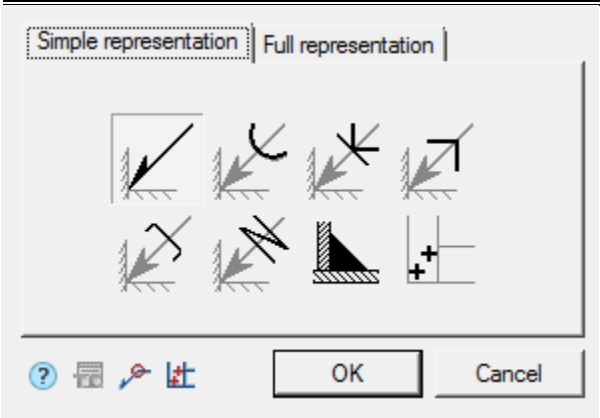


5. Set the type of joint in the *"Joints"* editing dialog and confirm the setting with the *"OK"* button.
6. The Fixed joint will be built.



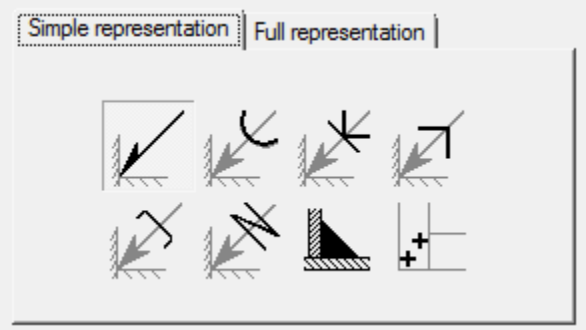
Dialog

Important!	When you select the fixed joint *"Soldired Joint Leg"* or *"Weld point"*, you cannot call the dialog from the object for editing.

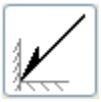
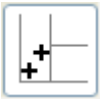


The editing dialog contains additional commands and two tabs: "Simple representation" and "Full representation".

- *Technical conditions* - the command associates a designation with a item of technical conditions.
- *Place spot welds* - the command is intended for insertion of weld point elevations in the drawing.
- *Along closed contour* - the command is intended to add the symbol *"Joint along closed contour"* to the connection designation.
- The "Simple representation" tab allows you to select the type of fixed joint.

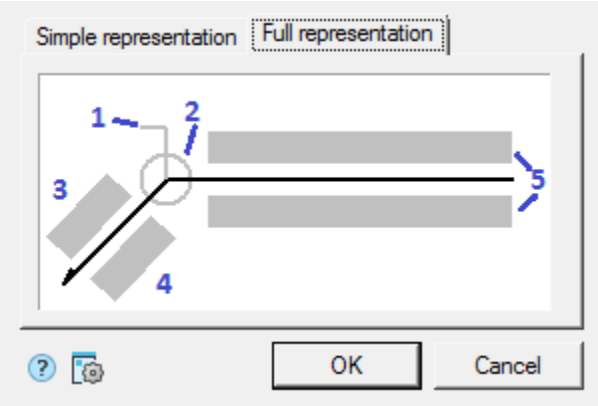


The following designations for fixed joints are available:

Seam denotation	*Soldering*	*Pasting together*	*Clamps, type 1*
			
Clamps, type 2	***Lacing***	***Soldired Joint Leg***	***Weld point***
			

Depending on the selected type, the "Full representation" tab will be available.

- The "Full representation" tab allows you to customize the fixed joint symbol.



Input fields and graphic elements are edited on the tab. To edit input fields and turn on / off graphic elements, click LMB on them. When editing input fields, dialogs for editing fields appear, containing an input field and additional commands.

1. Graphic element *"Field joint"*.

2. Graphic element *"Joint along closed contour"*.

3. Number designation input field.

The editing dialog contains:

- Value input field.
- Command *Technical conditions* - associates value with a technical conditions item.
- Command *"Sum"* - the number of welded seams with the same number is counted. Deleting or changing a seam number designation is automatically displayed in the full designation for that joint number.
- Command *"Copy properties"* - allows you to copy properties from an existing dimension in the drawing.
- Command *"OK"* - closes the dialog, saving changes.
- Command *"Cancel"* - closes the dialog without saving changes.

4. Input field for the designation of the control complex or the seam control category.

The editing dialog contains:

- Value input field.
- Command *"Copy properties"* - allows you to copy properties from an existing dimension in the drawing.
- Command *"OK"* - closes the dialog, saving changes.
- Command *"Cancel"* - closes the dialog without saving changes.

5. Input fields for the symbol.

The editing dialog contains:

- Value input field.
- Buttons for quick insertion of special symbols for fixed joint:

- Soldered leg;
- Remove weld reinforcement;
- Treat seam overflows and unevenness with graded junction to parent metal;
- Discrete double unfaced seam;
- Discrete double faced seam;
- Opened contour seam.

- Command *"Copy properties"* - allows you to copy properties from an existing dimension in the drawing.
- Command *"OK"* - closes the dialog, saving changes.
- Command *"Cancel"* - closes the dialog without saving changes.

Properties

Seam denotation:

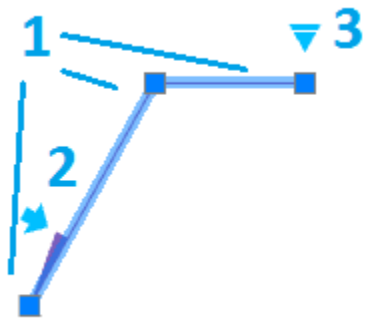
- Text above self of leader
- Text under self of leader
- Text above leader
- Text under leader
- Enable calculate joints
- Count of joints
- Field joint
- Joint along closed contour

Soldering, Lacing, Clamps type 1, Clamps type 1, Pasting together:

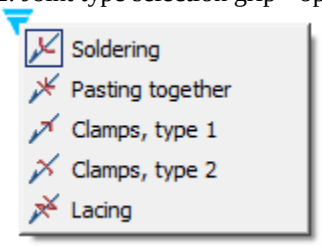
- Joint along closed contour
- TR item number

Grips

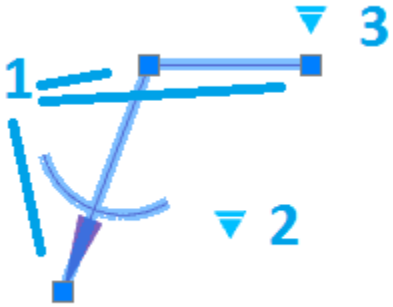
Seam denotation:
1. Move grips - Move the endpoints of the parts of the weld symbol.
2. "Mirror arrow side" grip - changes the side of the arrow.
3. "Weld seam properties" grip - opens a list of available graphic elements. In the list, you can enable or disable the display.



Soldering, Lacing, Clamps type 1, Clamps type 1, Pasting together:
1. Move grips - Move the endpoints of the parts of the weld symbol.
2. Joint type selection grip - opens a drop-down list for selecting a joint type.



3. "Weld seam properties" grip - opens a list of available graphic elements. In the list, you can enable or disable the display.



Soldered Joint Leg:
1. Move grip - allows you to move the leg.
2. Grips "Change leg size" - allow you to change the size of the leg.

Weld point:

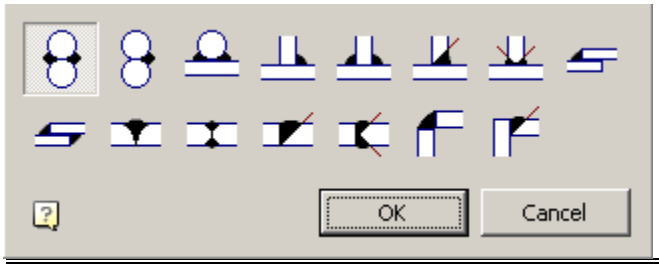
1. Move grip - Allows you to move the weld point in the drawing.

Simplified weld joint

Main menu: ***Mechanical - Symbols - Permanent connection -*** ***Simplified weld joint***.

Toolbar: "Symbols" - ***Simplified weld joint***.

Command line: ***MCWJ***.



Important!	As the rods and plates can be used primitives nanoCAD: circle and polyline (a special case polylines: rectangles, and polygons). To use the unit as a plate or an object from the database, you must use when choosing an instrument Magnet plate for constructing the circuit plate. Primitives must be tangent to each other.

Type: two rods overlap joints.

To add an overlap joint:

1. Call the command.
2. Select the drawing ***first rod*** .
3. Select the ***second rod***.
4. To select the direction of one-way seam weld placement.

Two-way joint.

One-way joint.

Two-way joint.

Type: T-connection between two rods..

To add a T-joints:

1. Call the command.
2. Select the drawing ***first plate*** .

3. Select the ***second plate***.
4. To select the direction of one-way seam weld placement.

Two-way joint.

One-way joint.

Two-way joint.

Two-way joint.

Type: two plates overlap joints.

To add an overlap joint:

1. Call the command.
2. Select the drawing ***first plate***.
3. Select the ***second plate***.
4. To select the direction of one-way seam weld placement.

Corner seam-sided.

Corner sided seam.

Type: Butt joint of two plates.

To add a butt joint:

1. Call the command.
2. Select the drawing ***first plate*** .
3. Select the ***second plate***.
4. To select the direction of one-way seam weld placement.

One-way butt weld.

Sided butt weld.

One-way butt weld.

Sided butt weld.

Type: Angular connection of two plates.

To add a butt joint:

1. Call the command.
2. Select the drawing ***first plate***.

3. Select the ***second plate***.
4. To select the direction of one-way seam weld placement.

Corner seam-sided.

Corner seam-sided.

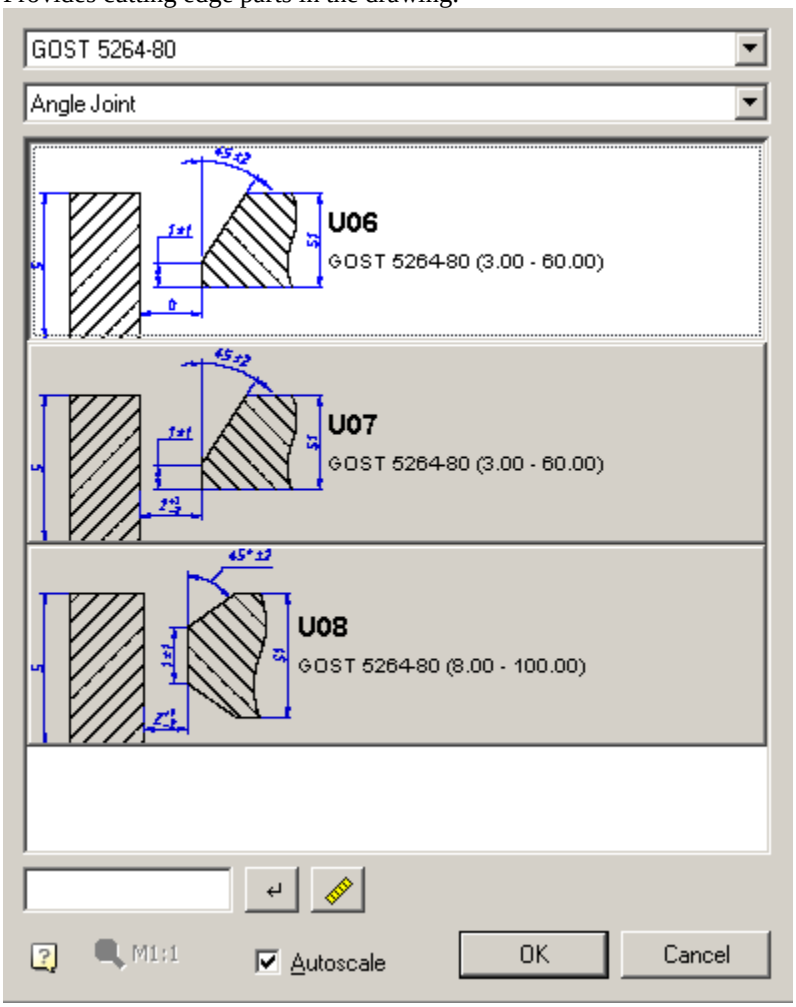
Edge preparation

Main menu: ***Mechanical - Symbols - Permanent connection-*** ***Edge preparation***.

Toolbar: "Symbols" - ***Edge preparation***.

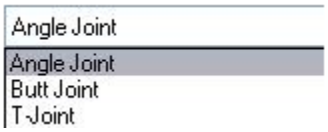
Command line: ***MCEDGEPREPARATION***.

Provides cutting edge parts in the drawing.

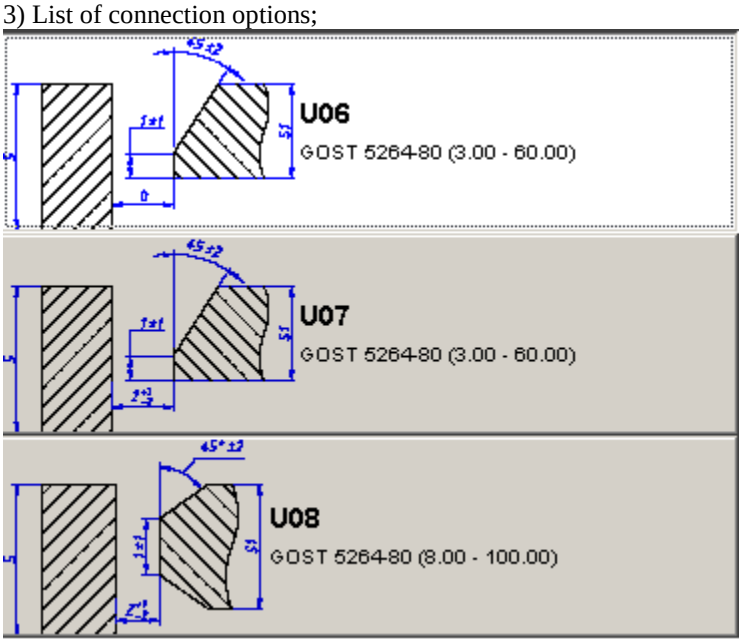


Elements of the dialogue:
1)The list of standard cutting edges for welding;

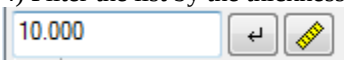
2) List of types of connection plates welded: Butt, T-bar, angle;



3) List of connection options;



4) Filter the list by the thickness of the plates;



When you enter a numeric value thickness dialogue filters only those types of cutting edges, matching the thickness of the part. The filter is made by pressing the ***"Apply filter"***, or by pressing Enter. The value of the thickness in the filter can be measured by the meter to be launched on the ***"Measure"***.

5) Zoom menu created design elements

Menu is disabled if ***Auto*** is enabled check box. In this case, the scale created by the size and design elements set to the current scale decoration.

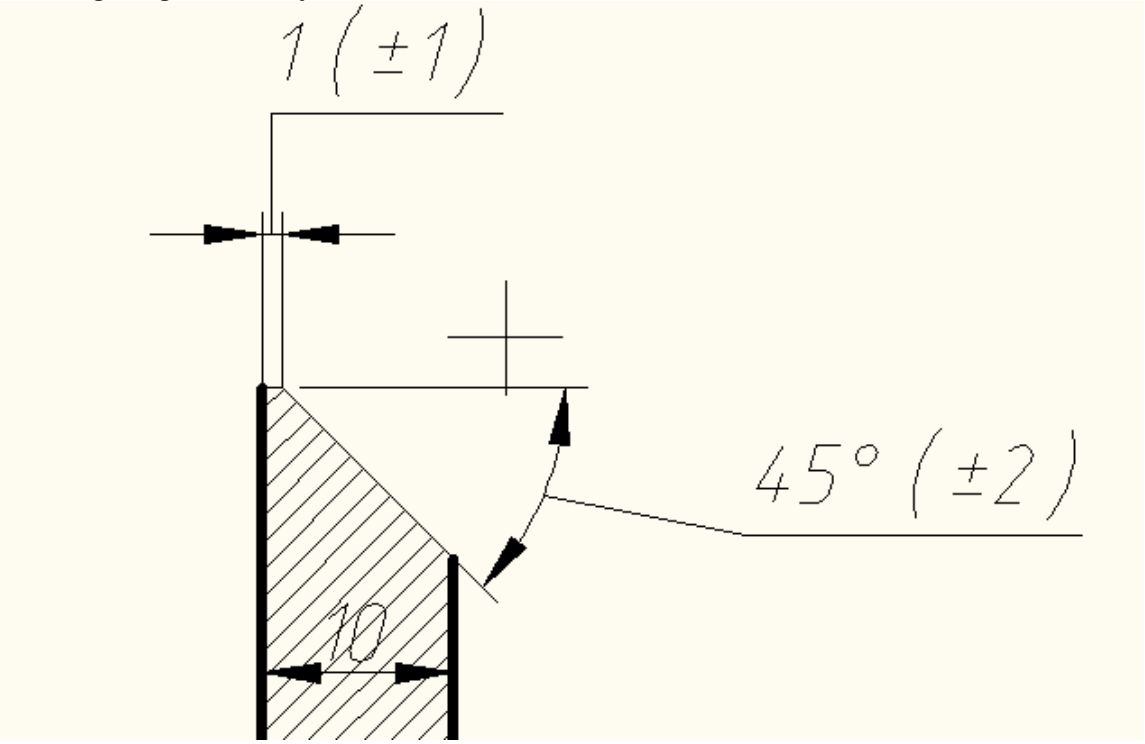
If you turn off ***Auto*** box, it will be possible to create a drawing cutting edges in arbitrary scale is selected in this dialog.

Operation

For the cutting edges must be vyponit following:

1. Select the type of connection: butt, T-bar or angle;
2. If necessary, filter preparation method edges, depending on their thickness. This can be done directly by entering the minimum thickness of the edges, or by measuring the drawing;
3. Determine the scope of dimension lines;
4. Double-click method of preparing the edges;

5. Specify the edge of the weld joint. Depending on the method of preparation is required to select the left and right edges alternately.



When you create a cutting edge bevel one team will cycle to offer select sites for the creation of such ramps.
When you create a cutting edge bevel two team provides consistently indicate bevel for one, and then to other parts of the joint.

Important!	The plates can be used as primitives nanoCAD, as well as objects nanoCAD Mechanica. Turn inserted edge welded connection can hold down Shift. It is necessary to include the "Object Snap" mode and "nearest" or in the settings nanoCAD Mechanica against the item "Connections" Nearest "automatically when you insert objects" put "Yes".

Marking symbol

Main menu: ***Mechanical - Symbols - Marking symbol***.

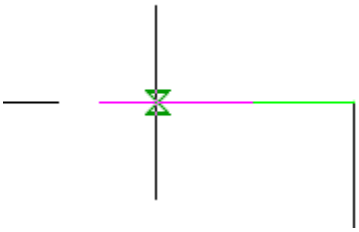
Ribbon: ***Mechanical - Symbols - Marking symbol***.

Toolbar: ***Marking symbol (toolbar*** "MCS Symbols"***)***.

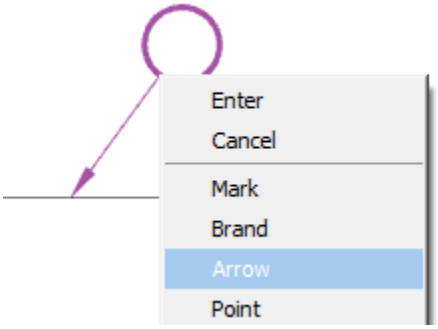
Command line: ***MCMARK***.

Procedure

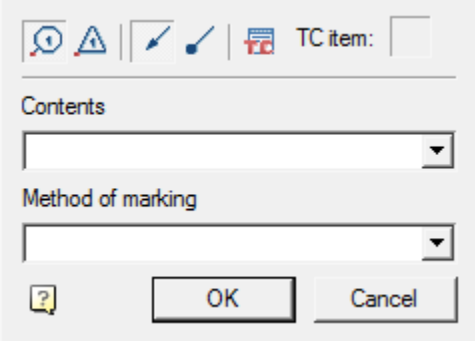
1. Call command *"Marking symbol"*.
2. Specify the place of marking.



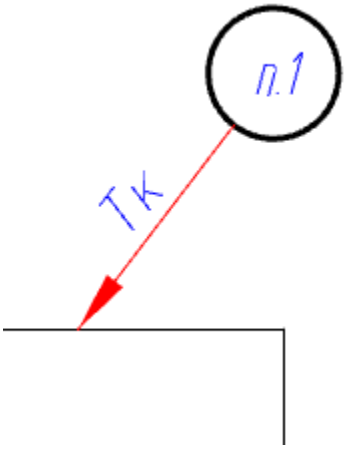
3. If necessary, select the arrow type in the context menu: Arrow or Point.
4. If necessary, select the type of designation in the context menu.: Mark or Brand.



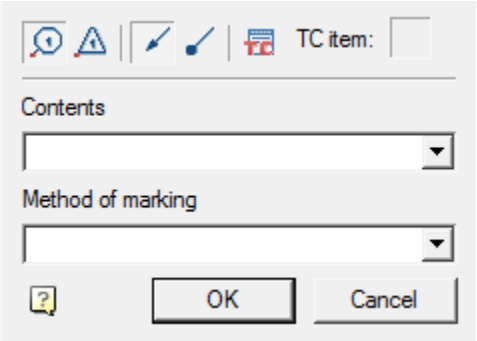
5. Indicate the position of the mark. Open dialog *"Marking and branding"*.



6. Adjust the settings and click "OK."
7. A marking or branding mark will be created.



Dialog edit



The "Marking and branding" dialog contains:

- Graphic switch sign type: *Marking* or Branding.
- Graphic switch arrow type: *Arrow* or *Point*.
- Button *Technical conditions*. Allows you to add technical conditions. The last item added will automatically bind to the marking.

The number of the selected item is displayed in the field next to the technical conditions button.

> ***Note:*** The item number in the technical conditions will match the number in the designation in the drawing, even after changing the order of the items in the editor.

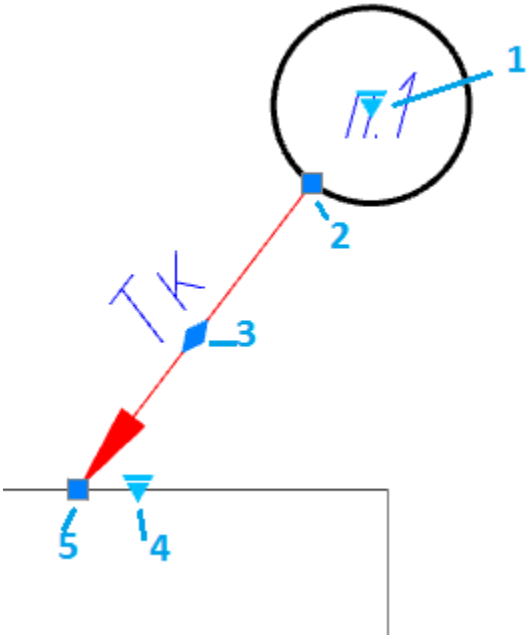
- Drop-down list *"Contents"*. Allows you to specify the content type of the mark marking or branding.
- Drop-down list *"Method of marking"*. Allows you to specify the method of applying the mark marking or branding.

Properties

- *Type* - Type of mark: Marking or Branding.
- *Arrow type* - Arrow type: Arrow or Point.
- *Content*
- *Method of marking*
- *Text position* - The position of the text content and method of applying to the extension line. The value changes from 0 to 1. The report starts from the place of marking.
- *TR item number*.
- *Coeff. of compression of TR item text*.

Grips

1. Grip type designation: Marking or Branding.
2. Grip position of the mark.
3. Grip position text on the extension line.
4. Grip select the type of arrow.
5. Grip position of the arrow.



Notes

Mechanical note

Main menu: ***Draw - Notes -*** ***Mechanical note***.

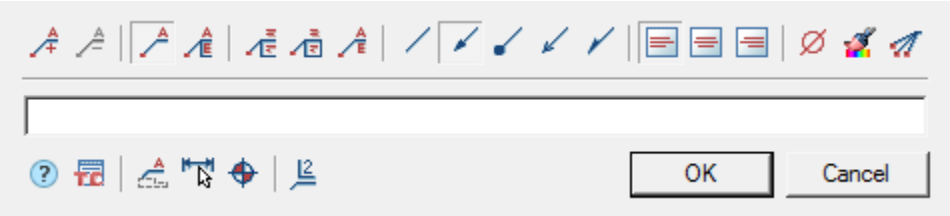
Main menu: ***Mechanical - Symbols - Notes -*** ***Mechanical note***.

Ribbon: ***Annotate - Leaders -*** ***Mechanical note***.

Ribbon: ***Mechanical - Symbols -*** ***Mechanical note***.

Toolbar: ***Notes -*** ***Mechanical note***.

Command line: ***MCNOTE, NOTE***.



Procedure

1. Call the command.
2. If the option "Show dialog before inserting objects" is active, the *"Mechanical note"* dialog opens. In the *"Mechanical note"* dialog box, enter text and define the note options. Click *"OK"*.
3. Pick a point on the object (position of the leader arrow).
4. Place a note leader in the drawing.

5. If the "Show dialog before inserting objects" option is disabled, the *"Mechanical note"* dialog will open.In the *"Mechanical note"* dialog box, enter text and define the note options. Click *"OK"*.
6. Note will be biuld.

Menu

- *Add string* - command adds an extra input field;
- *Remove string* - command deletes the input field where the cursor is located;
- *Simple note* - command toggles the display of the note;
- *Multiline note* - command toggles the display of the note;
- *Multi-line text* - the switch controls the display of multi-line text on the shelf;
- *Frame* - text under the shelf is framed;
- Type arrows:
  - *None*,
  - *Arrow*,
  - *Point*,
  - *Open arrow*,
  - *Half-arrow*,
- Align text horizontally;
  - *By left edge*
  - *By center*
  - *By right edge*
- *Insert special symbol* - *the command allows you to insert a special character into the input field;*
- *Match properties* - command copies appearance parameters from another object;
- *Add extension line* - the command allows you to add an additional note line;
- *Technical conditions*;
- Designations:
  - *Surface*,
  - *Dimension*,
  - *Others*;
- *Dulling dimension*.

Context menu

The context menu opens in the input field. View Context menu entry fields.

Fields

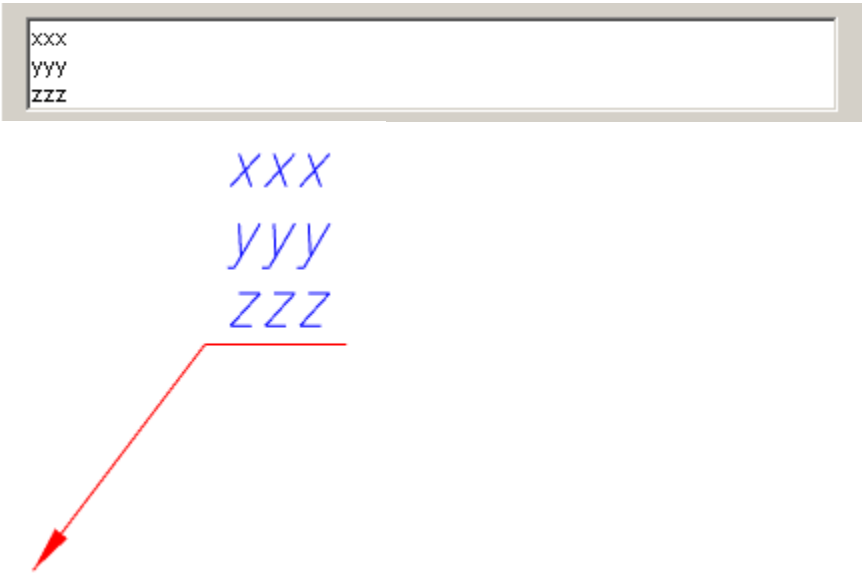
By default, *"Mechanical note"* contains one text box for the label above the leader flange.

Using the *"Simple note"* and *"Multiline note"* menu commands, you edit the presence of additional shelves.

Use the "Add string" and "Remove string" menu commands to edit the number of leader ledges. When using a simple note, these commands control the presence of text under the shelf.

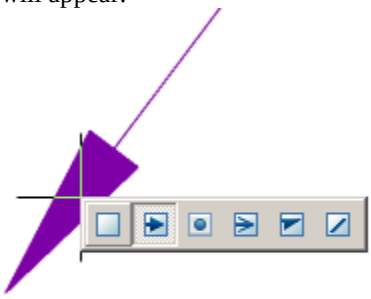
Note:	Hotkeys are available for the *"Add string"* command. *"Ctrl + Enter"* - adds a shelf below the selected shelf. *"Shift + Enter"* - adds a shelf above the selected shelf.

The *"Multi-line text"* option allows you to display multiline text on the shelf. When this parameter is enabled, the transition to another line is performed by the *"Ctrl + Enter"* key combination.



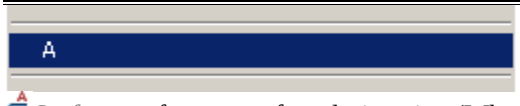
Arrow type selection context menu

When you call the context menu on the note arrow (without selecting the note), a dialog box for selecting the arrow type will appear.



Auto sort modes

Note:	The submenu (in our example - the letter A) is activated in cases of using the auto sort mode when applying letters to the drawing.



Surface - reference surface designation (When automatic lettering sorting is enabled).

Dimension -dimension designation.

Others - designation of other elements, such as holes.

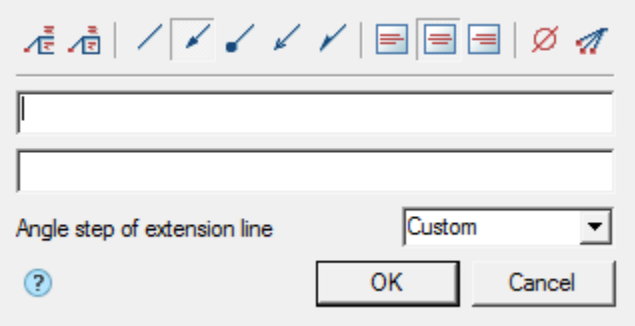
Dulling dimension - special note to indicate blunt edges.

Construction note

Main menu: ***Draw - Notes -*** ***Construction note***.

Ribbon: ***Annotate - Leaders -*** ***Construction note***.

Command line: ***MCNOTEP, NOTEP***.



Procedure

1. Call the command *"Construction note"*.
2. If "Show dialog before inserting the object" is active, the *"Construction Note"* dialog opens. In the *"Construction Note"* dialog box, enter text and define the leader options. Click the *"OK"* button.
3. Pick a point on the object (select the object) to which the leader arrow will point. To select an object, use the *"Select"* context menu command; to freely specify a point in the drawing, use the *"Free"* context menu command.
4. Place the leader shelf in the drawing.

Note:	If the parameter *"Angle step of extension line"* is arbitrary, it is possible to place the leader ledge orthogonally in the ORTO mode (F8). When you turn on the ORTO mode via the "SHIFT" key, Object Snap (F3) must be turned on.

5. If the "Show dialog before inserting the object" option is disabled, the *"Construction Note"* dialog will open. In the *"Construction Note"* dialog box, enter text and define the leader options. Click the *"OK"* button.
6. The note is built.

Menu

- *Multi-line text* - the switch controls the display of multi-line text on the shelf;
- *Frame* - text under the shelf is framed;
- Type arrows:
  - *None*,
  - *Arrow*,
  - *Point*,
  - *Open arrow*,
  - *Half-arrow*,
- Align text horizontally:
  - *By left edge*,
  - *By center*,
  - *By right edge*;
- *Insert special symbol* - the command allows you to insert a special character into the input field;
- *Add extension line* - the command allows you to add an additional note line;

Context menu

The context menu opens in the input field. View Context menu entry fields.

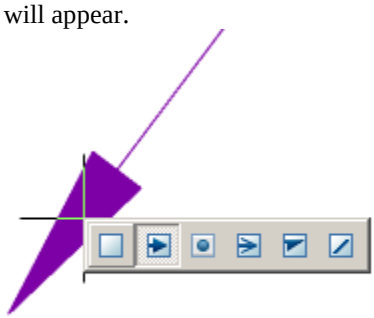
Fields

The default positional leader contains two input lines and a list for specifying the pitch of the bevel angle.
The first line is for the label above the leader shelf, the second is under the leader shelf.
In the *"Angle step of extension line"* list box, the slope angle of the leader lines is selected:

- *Custom* - a leader line is placed arbitrarily (default);
- *step 15°* - the leader line is put down with a step multiple 15°;
- *step 30°* - the leader line is put down with a step multiple 30°;
- *step 45°* - the leader line is put down with a step multiple 45°;
- *step 90°* - the leader line is put down with a step multiple 90°.

Arrow type selection context menu

When you call the context menu on the note arrow (without selecting the note), a dialog box for selecting the arrow type will appear.



Comb leader note

Main menu: ***Draw - Notes - Comb leader note***.

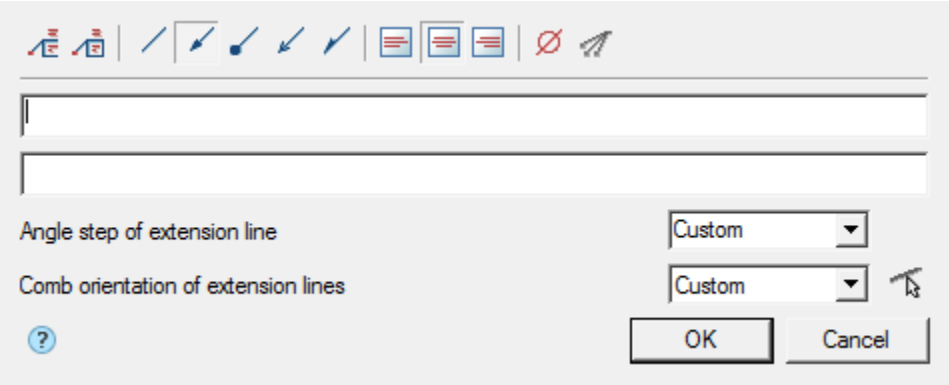
Main menu: ***Mechanical - Symbols - Notes - Comb leader note***.

Ribbon: ***Annotate - Leaders - Comb leader note***.

Ribbon: ***Mechanical - Symbols - Comb leader note***.

Toolbar: ***Notes - Comb leader note***.

Command line: ***MCNOTEC, NOTEC***.



Procedure

1. Call the command.
2. If the option "Show dialog before inserting objects" is active, the *"Comb leader note"* dialog opens. In the *"Comb leader note"* dialog box, enter text and define the note options. Click *"OK"*.
3. Insert the required number of leader lines. When you specify the location of the leader line, the specified line segment is highlighted. Press the *"Enter"* key to complete the paste.
4. Specify the slope of the leader lines. Commands for switching the ridge direction are available in the context menu: H - horizontal, V - vertical, P - parallel (parallel to the selected line).
5. Place a note leader in the drawing.
6. If the "Show dialog before inserting objects" option is disabled, the *"Comb leader note"* dialog will open.In the *"Comb leader note"* dialog box, enter text and define the note options. Click *"OK"*.
7. Note will be build.

Menu

- *Multi-line text* - the switch controls the display of multi-line text on the shelf;
- *Frame* - text under the shelf is framed;
- Type arrows:
  - *None*,
  - *Arrow*,
  - *Point*,
  - *Open arrow*,
  - *Half-arrow*,
- Align text horizontally;
  - *By left edge*
  - *By center*
  - *By right edge*
- *Insert special symbol* - the command allows you to insert a special character into the input field;
- *Add extension line* - the command allows you to add an additional note line;

Context menu

The context menu opens in the input field. View Context menu entry fields.

Fields

Comb leader note by default contains: two lines of input, a list for specifying the pitch of the angle of inclination and the orientation of the ridge.
The first line is for the label above the leader shelf, the second is under the leader shelf.
In the *"Angle step of extension line"* list box, the slope angle of the leader lines is selected:

- *Custom* - a leader line is placed arbitrarily (default);
- *step 15°* - the leader line is put down with a step multiple 15°;
- *step 30°* - the leader line is put down with a step multiple 30°;
- *step 45°* - the leader line is put down with a step multiple 45°;
- *step 90°* - the leader line is put down with a step multiple 90°.

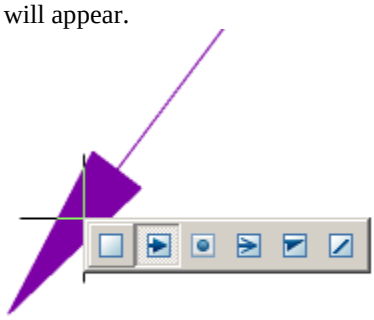
In the *"Comb orientation of extension lines"* list box, you can select the ridge orientation of the leader lines:

- *Custom*;
- *Horizontal*;
- *Vertical*.

The list is equipped with an additional button "Select line", which allows you to select the direction of the ridge of the leader lines parallel to any line segment in the drawing.
The *"Multi-line text"* option allows you to display multiline text on the shelf. When this parameter is enabled, the transition to another line is performed by the *"Ctrl + Enter"* key combination.

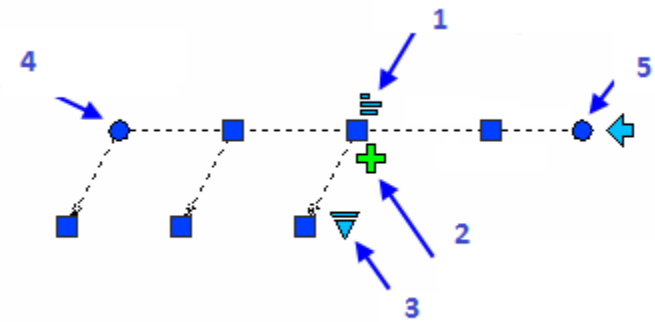
Arrow type selection context menu

When you call the context menu on the note arrow (without selecting the note), a dialog box for selecting the arrow type will appear.



Grips

1. Text alignment toggle grip.
2. Leader lines grip.
3. Arrow type switch-grip.
4. Ridge line rotation grip.
5. Shelf rotation grip.

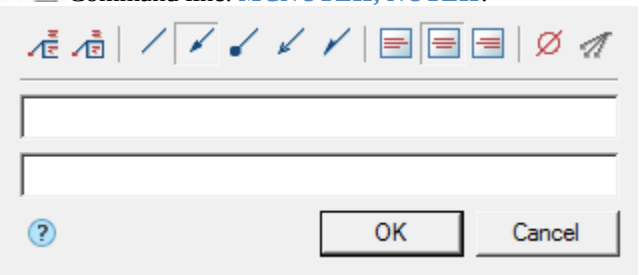


Chain note

Main menu: ***Draw - Notes - Chain note***.

Ribbon: ***Annotate - Leaders - Chain note***.

Command line: ***MCNOTEH, NOTEH***.



A chain note has several coaxial (collinear) series-connected pointers.
If the first node of the note is placed on a line segment, then the note will be located perpendicular to this segment.

Procedure

1. Call the command.
2. If the option "Show dialog before inserting objects" is active, the *"Chain note"* dialog opens. In the *"Chain note"* dialog box, enter text and define the note options. Click *"OK"*.
3. Specify note nodes, at least two. To complete the selection of nodes, press *"Enter"*.
4. Specify the position of the leader where the text will go.
5. If the "Show dialog before inserting objects" option is disabled, the *"Chain note"* dialog will open.In the *"Chain note"* dialog box, enter text and define the note options. Click *"OK"*.
6. Note will be build.

Menu

- *Multi-line text* - the switch controls the display of multi-line text on the shelf;
- *Frame* - text under the shelf is framed;
- Type arrows:
  - *None*,
  - *Arrow*,
  - *Point*,
  - *Open arrow*,

- *Half-arrow*,
- Align text horizontally;
  - *By left edge*
  - *By center*
  - *By right edge*
- *Insert special symbol* - the command allows you to insert a special character into the input field;
- *Add extension line* - the command allows you to add an additional note line;

Context menu

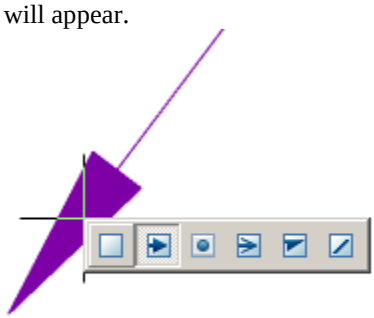
The context menu opens in the input field. View Context menu entry fields.

Fields

Chain note contains two lines of input by default.
The first line is for the label above the leader shelf, the second is under the leader shelf.
Input fields are expandable. Switching to another line of the input field is performed by the *"Ctrl + Enter"* key combination.

Arrow type selection context menu

When you call the context menu on the note arrow (without selecting the note), a dialog box for selecting the arrow type will appear.

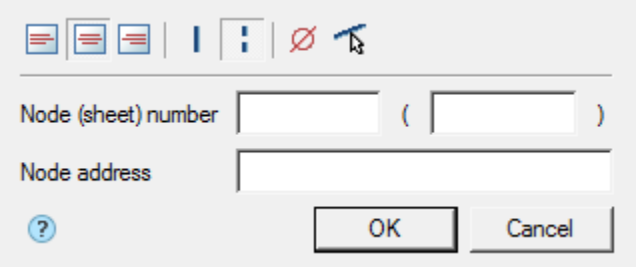


Section note

Main menu: ***Draw - Notes -*** ***Section note***.

Ribbon: ***Annotate - Leaders -*** ***Section note***.

Command line: ***MCNOTES, NOTES***.



Procedure

1. Call the command.
2. If the option "Show dialog before inserting objects" is active, the *"Section note"* dialog opens. In the *"Section note"* dialog box, enter text and define the note options. Click *"OK"*.
3. Select the insert mode *"Free"* or *"Selection"*:
   - *"Selection"* - you need to specify the first and second lines to place the note, the cursor is in the form of a square,
   - *"Free"* - no lines are required, can be placed anywhere by 2 points, the cursor is in the form of a cross.

To switch from the *"Selection"* mode to the *"Free"* mode, press the RMB and select the *"Enter"* command in the context menu. To switch from the *"Free"* mode to the *"Selection"* mode, press the RMB. Also, switching modes is carried out with the *"Space"* key.

4. Pick the first point (select the first line) in the drawing.
5. Pick the second point (select the second line) in the drawing.
6. Specify the position of the leader.
7. If the "Show dialog before inserting objects" option is disabled, the *"Section note"* dialog will open.In the *"Section note"* dialog box, enter text and define the note options. Click *"OK"*.
8. Section note will be build.

Menu

- Align text horizontally;
  - *By left edge*
  - *By center*
  - *By right edge*
- Lines;
  - *Single-stroked line*
  - *Double-stroked line*
- *Insert special symbol* - the command allows you to insert a special character into the input field;
- *Select line* - the command allows you to relocate the location of the leader line by specifying a line in the drawing;

Context menu

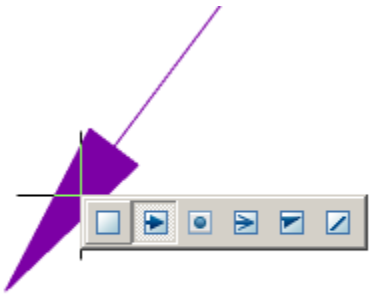
The context menu opens in the input field. View Context menu entry fields.

Fields

Section note contains two lines of input by default.
The first line of input consists of two fields for specifying the *"Node (sheet) number"*.
Second line of input for specifying the *"Node address"*.

Arrow type selection context menu

When you call the context menu on the note arrow (without selecting the note), a dialog box for selecting the arrow type will appear.

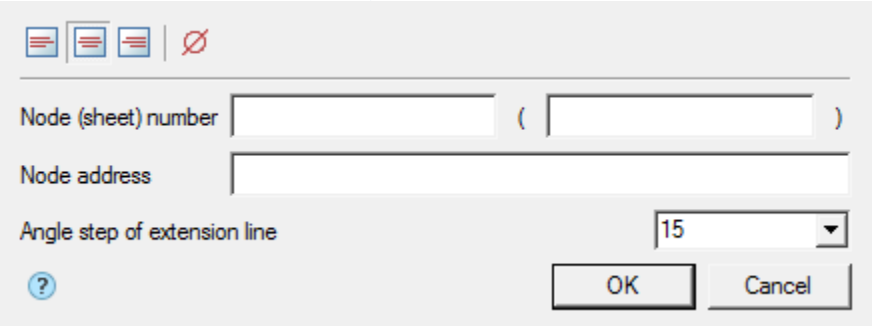


Node note

Main menu: ***Draw - Notes -*** ***Node note***.

Ribbon: ***Annotate - Leaders -*** ***Node note***.

Command line: ***MCNOTEK, NOTEK***.



Procedure

1. Call the command.
2. If the option "Show dialog before inserting objects" is active, the *"Node note"* dialog opens. In the *"Node note"* dialog box, enter text and define the leader options. Click *"OK"*.
3. Pick the center of the leader oval (circle).
4. Specify the size of the leader oval (circle).
5. Show the position of the shelf or enter in advance in the command line the step of the angle of inclination of the leader line.
6. If the "Show dialog before inserting objects" option is disabled, the *"Node note"* dialog will open.In the *"Node note"* dialog box, enter text and define the leader options. Click *"OK"*.
7. Node note will be build.

Menu

- Align text horizontally;
  - *By left edge*
  - *By center*
  - *By right edge*
- *Insert special symbol* - the command allows you to insert a special character into the input field;

Context menu

The context menu opens in the input field. View Context menu entry fields.

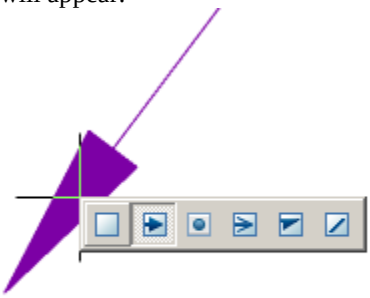
Fields

Node note by default contains: two input lines and a drop-down list for specifying the pitch of the slope.
The first line of input consists of two fields for specifying the *"Node (sheet) number"*.
Second line of input for specifying the *"Node address"*.
In the drop-down list *"Angle step of extension line"*, the slope angle of the leader lines is selected:

- *Custom* - a leader line is placed arbitrarily (default);
- *step 15°* - the leader line is put down with a step multiple 15°;
- *step 30°* - the leader line is put down with a step multiple 30°;
- *step 45°* - the leader line is put down with a step multiple 45°;
- *step 90°* - the leader line is put down with a step multiple 90°.

Arrow type selection context menu

When you call the context menu on the note arrow (without selecting the note), a dialog box for selecting the arrow type will appear.

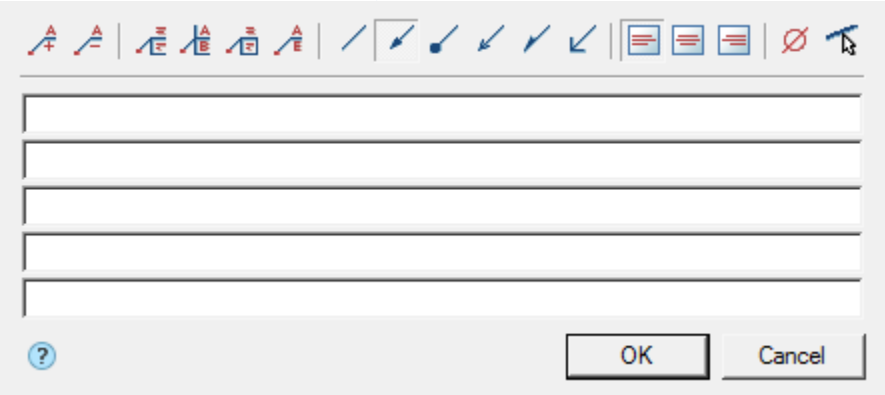


Note for multilayered constructions

Main menu: ***Draw - Notes - Note for multilayered constructions***.

Ribbon: ***Annotate - Leaders - Note for multilayered constructions***.

Command line: ***MCNOTEM, NOTEM***.



Procedure

1. Call the command.
2. If the option "Show dialog before inserting objects" is active, the *"Note for multilayered constructions"* dialog opens. In the *"Note for multilayered constructions"* dialog box, enter text and define the leader options. Click *"OK"*.
3. Pick a point on an object (drawing).
4. Specify the position of the shelf.
5. If the "Show dialog before inserting objects" option is disabled, the *"Note for multilayered constructions"* dialog will open. In the *"Note for multilayered constructions"* dialog box, enter text and define the leader options. Click OK.
6. The note will be built.

Menu

- *Add string* - command adds an extra input field;
- *Remove string* - command deletes the input field where the cursor is located;
- *First line* - the command adds a border (vertical line before the line) for the first line;
- *Multi-line text* - the switch controls the display of multi-line text on the shelf;
- *Frame* - text under the shelf is framed;
- Type arrows:
  - *None*,
  - *Arrow*
  - *Point*,
  - *Open arrow*,
  - *Half-arrow*,
  - *Right angle*;
- Align text horizontally;
  - *By left edge*
  - *By center*
  - *By right edge*
- *Insert special symbol* - the command allows you to insert a special character into the input field;
- *Select line* - the command allows you to relocate the location of the leader line by specifying a line in the drawing;

Context menu

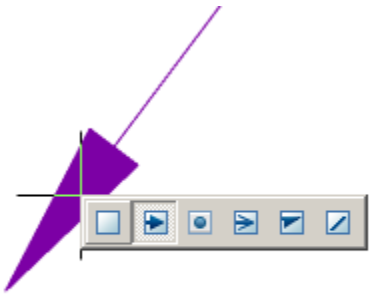
The context menu opens in the input field. View Context menu entry fields.

Fields

By default, *"Note for multilayered constructions"* contains 5 input fields.
The number of fields is edited using the menu commands *"Add string"* and *"Remove string"*.
With the help of the "*First line"* menu command, the presence of a border on the first line is edited.
Using the *"Multi-line text"* menu command, you edit the presence of shelves in multiline notes.

Arrow type selection context menu

When you call the context menu on the note arrow (without selecting the note), a dialog box for selecting the arrow type will appear.

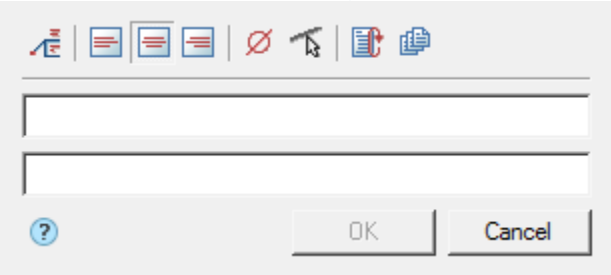


Linear aligned note

Main menu: ***Draw - Notes -*** ***Linear Aligned note***.

Ribbon: ***Annotate - Leaders -*** ***Linear Aligned note***.

Command line: ***MCLINM, LINM***.



Procedure

1. Call the command. A dialogue will open *"Linear aligned note"*
2. In the *"Linear aligned note"* dialog box, enter text and define the leader options. Click the *"OK"* button.
3. Specify the subassembly line on which the line subassembly will be located.
4. Specify text position.
5. *"Linear aligned note"* will be build.

Menu

- *Multi-line text* - the switch controls the display of multi-line text on the shelf;
- Align text horizontally;
  - *By left edge*
  - *By center*
  - *By right edge*
- *Insert special symbol* - the command allows you to insert a special character into the input field;
- *Select line* - the command allows you to relocate the location of the leader line by specifying a line in the drawing;
- *Auto repeat* - when the switch is on, the *"Linear aligned note"* command, upon completion of the previous command, is restarted;

- *Multiple insert* - when the switch is on, the insert is looped. To exit the cycle, press the *"Enter"* key.

Context menu

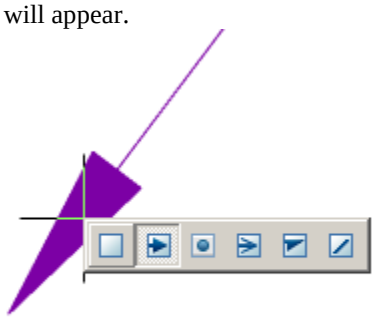
The context menu opens in the input field. View Context menu entry fields.

Fields

By default *"Linear aligned note"* contains 2 input fields.
The first field for the label is above the linear structure, the second - under the linear structure.
The *Multi-line text* parameter allows you to display multi-line text on the shelf. When this parameter is enabled, the transition to another line is performed by the *"Ctrl + Enter"* key combination.

Arrow type selection context menu

When you call the context menu on the note arrow (without selecting the note), a dialog box for selecting the arrow type will appear.



Editing Notes

There are 3 commands for editing leaders:

Append leader

Main menu: ***Mechanical - Symbols - Notes - + Append leader***.

Ribbon: ***Mechanical - Symbols - + Append leader***.

Toolbar: ***+Append leader (*** "Notes"***)***.

Command line: ***MCPLADD***.

1. Call the team.
2. Pick a point on the object (point of arrow position).
3. Indicate the required number of intermediate points that form a break in the leader line.
4. Point to the desired leader inserted in the drawing to complete the command.

Detach leader

Main menu: ***Mechanical - Symbols - Notes - − Detach leader***.

Ribbon: ***Mechanical - Symbols - − Detach leader***.

Toolbar: *Detach leader (* "Notes"*)*.

Command line: ***MCPLDEL***.

1. Call the team.
2. In the drawing, select the leader line you want to delete.

Edit leader

Main menu: ***Mechanical - Symbols - Notes - Edit leader***.

Ribbon: ***Mechanical - Symbols - Edit leader***.

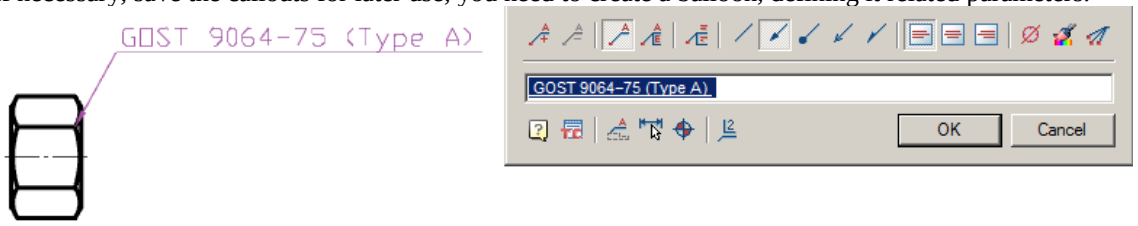
Toolbar: *Edit leader (* "Notes"*)*.

Command line: ***MCPLREC***.

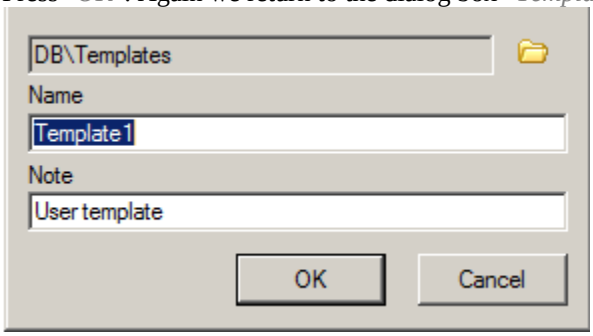
1. Call the team.
2. Highlight on the drawing a leader line that should be chained.
3. Pick a point on the desired leader in order to fix the new position of the leader line.

Save template note

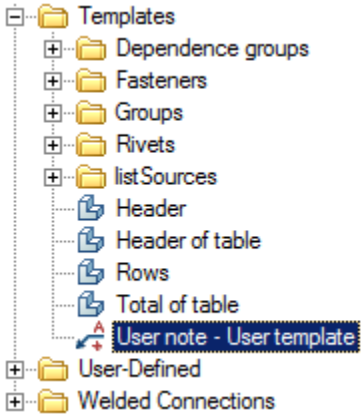
If necessary, save the callouts for later use, you need to create a balloon, defining it related parameters.



Further, by selecting it, specify the context menu - *"Save object as template"*.
In the *"Template wizard"* press button *"Create template"*.
In the dialog box *"Create element"* specify the storage folder in the base elements, Name and Note (if necessary).
Press *"OK"*. Again we return to the dialog box *"Template wizard"*.



Press button *"Update template"* and close *"Template wizard"*.
Update the tree base elements, the template will be created by the note.



Subsequently, this note can be inserted into a drawing for new objects. Adaptive communication with new objects and their settings will be saved.

Gradient symbol

Main menu: ***Mechanical - Symbols -*** ***Gradient symbol***.

Ribbon: ***Mechanical - Symbols -*** ***Gradient symbol***.

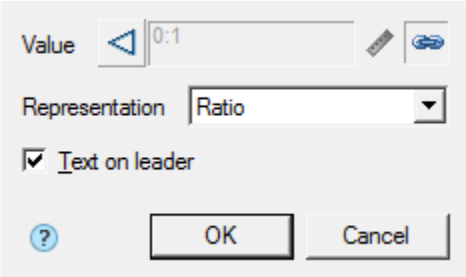
Toolbar: ***Symbols -*** ***Gradient symbol***.

Command line: ***MCGRAD***.

Procedure

1. Call the command.
2. Using the *"Properties"* context menu command, open the *"Gradient symbol"* dialog box. Make the gradient symbol settings and confirm the changes with the *"OK"* button.
3. Select the line in the drawing for which you want to set the gradient symbol.
4. Place the gradient symbol on the selected line or leader flange.
5. The gradient symbol is drawn and associated associatively with the specified line.

Dialog

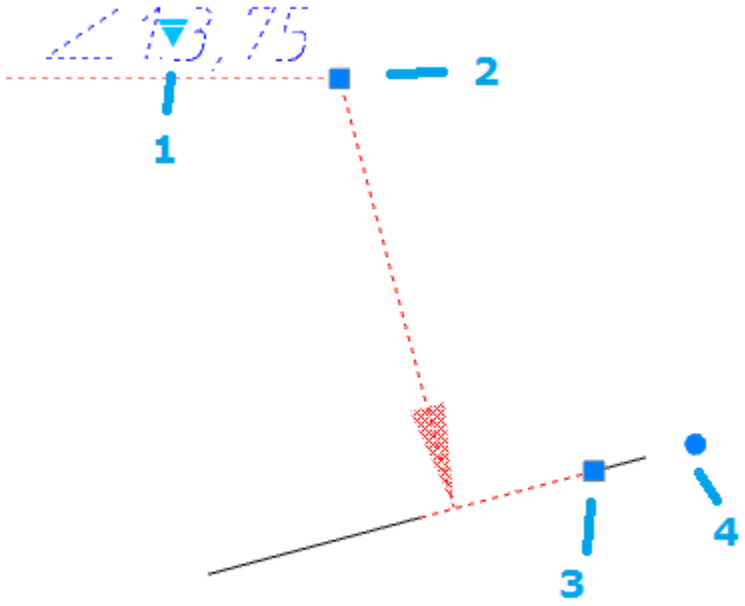


The editing dialog contains:

- Selection button type of designation: Gradient without designation, Left gradient and Left taper.

- Input field for gradient symbol value. The input field is active if the associativity mode is disabled. If necessary, turn off the associative connection mode by turning off the *"Associative"* switch (when setting, the associativity mode is turned on automatically).
- *"Pick from driwing"* button. When the associativity mode is off, the button is active and allows you to measure the gradient symbol value in the drawing.
- Switch *"Associative"*. Controls the display of the gradient symbol value. If the switch is on, the value is taken from the associated line, if it is off, it is entered manually.
- Drop-down list *"Representation"*. Allows you to select how the slope units are represented:
  - As is
  - Ratio
  - Decimal
  - Percent
  - Permille
  - Degree
- *"Text on leader"* checkbox. Toggles whether the gradient symbol is on a shelf or on a line.

Grips



1. Grip for selecting the slope symbol representation.
2. Shelf movement grip.
3. Base move grip. When you move, the associativity with the line is lost and an additional fourth grip appears.
4. Grip for changing the angle of inclination of the base.

Position

Main menu: ***Mechanical - Symbols - Position***.

Toolbar: ***Position (toolbar*** "MCS Symbols"***)***.

Command line: ***MCPOS***.

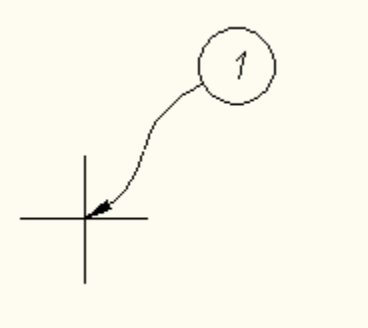
1. Specify *the insertion point to* the first marker position or call the context menu command "P-properties."
2. In the dialog box *Tagging* enter text.
3. Select an arrow. Click OK.
4. Insert the desired number of drawing markers. Subsequent node number is automatically incremented from the previous.

Fundamental rules

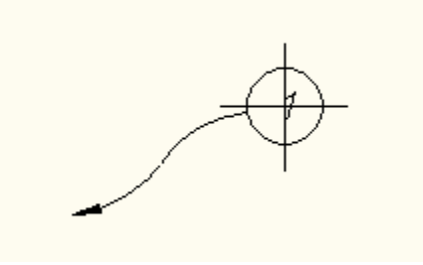
This command draws the drawing reference numerals used in the drawings.

Sequencing

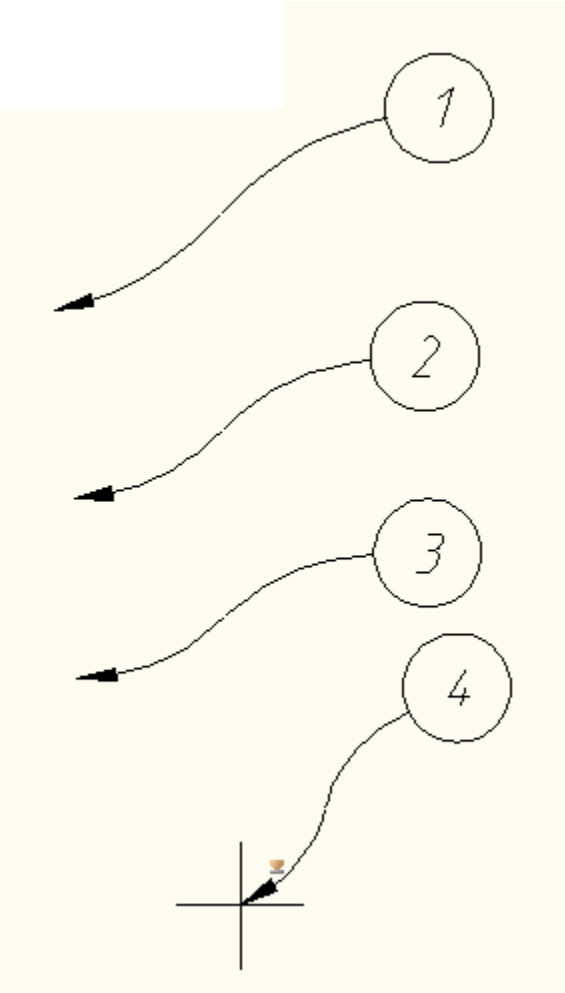
Insert position by specifying the datum marker



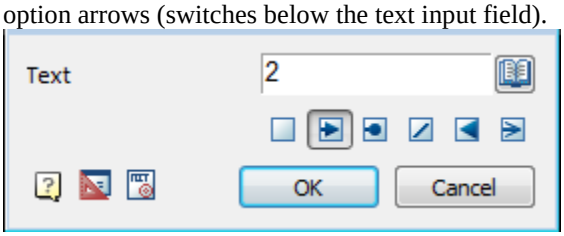
Next, mark the position statement of the position:



Inserted into a single command cycle positions are automatically numbered



To specify additional options position must call dialog "Position", where you can specify the text position display option arrows (switches below the text input field).



Hole group

Main menu: ***Mechanical - Symbols -*** ***Hole group***.

Ribbon: ***Mechanical - Symbols -*** ***Hole group***.

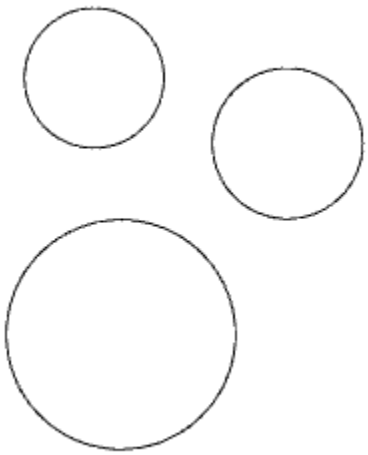
Toolbar: "MCSSymbols"***) -*** ***Hole group***.

Command line: ***MCFILL***.

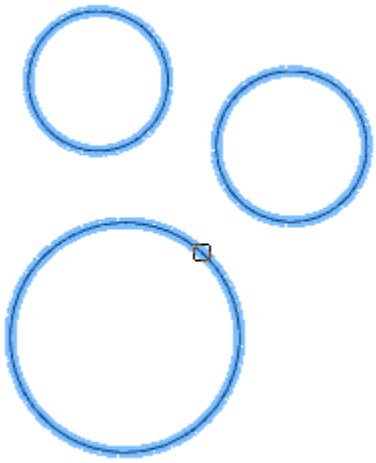
Universal command for drawing holes. With its help, you can draw a new hole with centerlines, or by specifying a group of holes to assign them general properties with filling of a sector.

Procedure

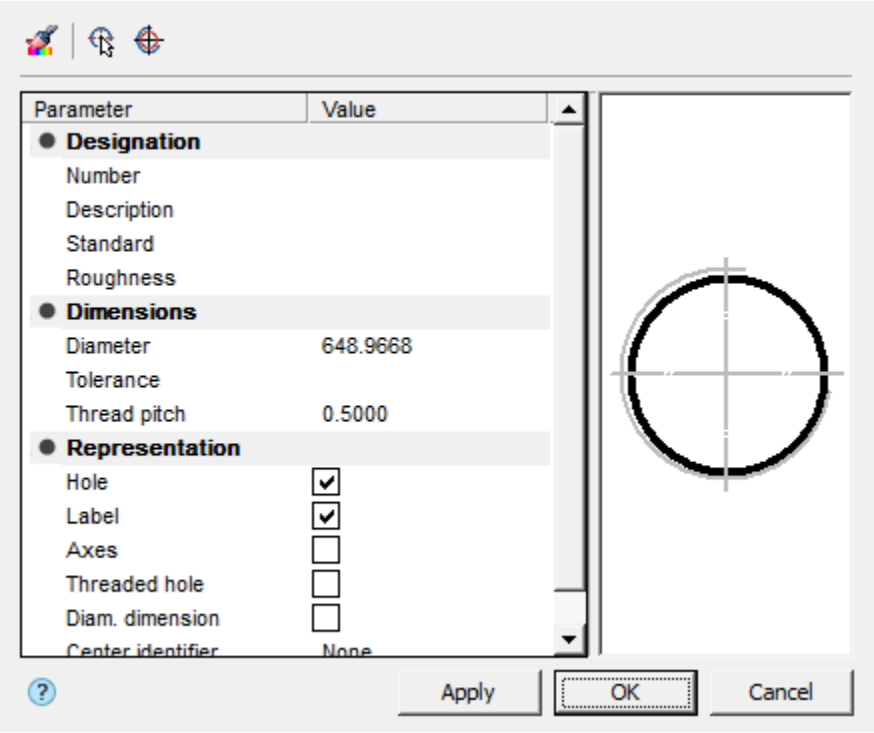
1. Draw holes from circles.



2. Call the command *"Hole group"*.



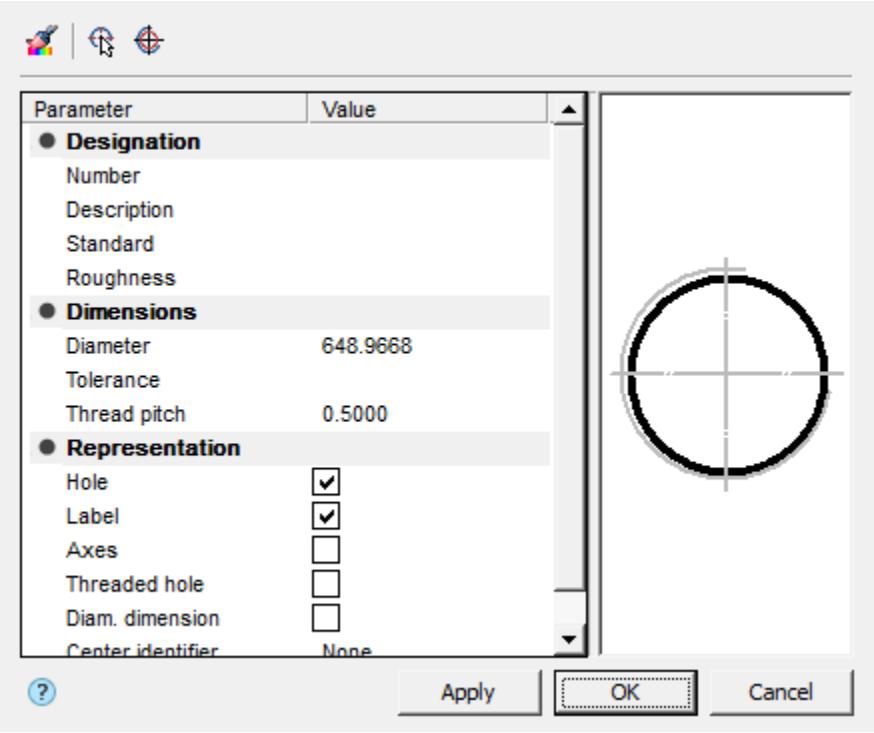
3. Select the required circles. Press the *"Enter"* key to complete the selection. The *"Holes"* dialog will open.



4. Adjust the hole settings and click *"OK"*. The parameters will be applied to the holes. The holes will be created.

Dialog

The *"Holes"* editing dialog is opened by double-clicking. The dialog consists of: Toolbar, Parameter List, Graphics Window.



Toolbar

Match properties - this command allows you to copy parameters from holes already installed in the drawing.

Select holes - the command allows you to select additional holes from the drawing.

Create holes - the command creates holes of the specified diameter. To dynamically set the radius on the screen, hold down the left mouse button and move the cursor.

Parameter List

Designation

- Number - specifies the number of the hole in the group.
- Description - specifies the hole table description field.
- Standard - sets the field to the standard of the hole table.
- Roughness - allows you to specify the roughness of the hole (displayed in the hole table).

Dimensions

- Diameter - hole diameter.
- Tolerance - the "..." button opens the "Tolerances" dialog, from which you can select a tolerance.
- Thread pitch - hole thread pitch.

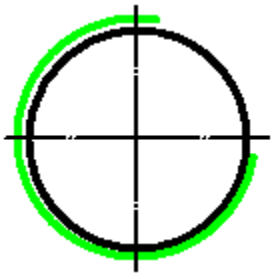
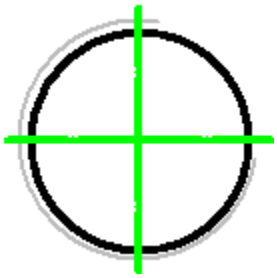
Representation

- Hole - hole display switch.
- Label - label display switch.
- Axes - axis display switch.
- Threaded hole - thread display switch.
- Diam. dimension - diametrical dimension display switch.
- Center identifier - drop-down list for selecting the hole center designation.

Note:	Also, the parameters of the *"Representation"* section can be configured through the graphics window.

Graphics Window

The graphics window allows you to see the resulting hole representation and customize the representation by directly specifying elements.

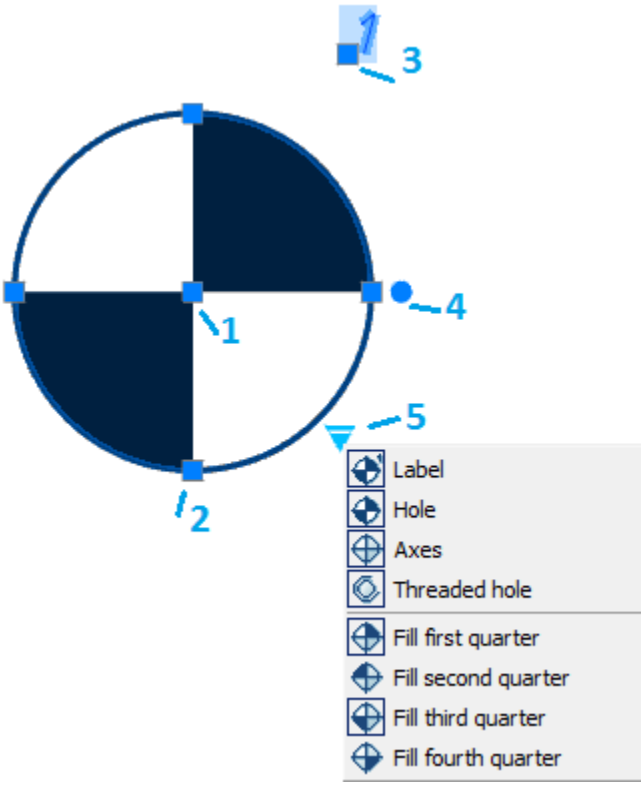
Threaded hole	Axes	Fill first quarter
		
Fill second quarter	**Fill third quarter**	**Fill fourth quarter**

Properties

The parameters on the *"Properties"* functional panel are similar to the parameters from the list of parameters of the *"Holes"* dialog.

Grips

1. Hole position change grip.
2. Diameter change grips.
3. Grip for changing the position of the hole number.
4. Hole rotation grip.
5. View adjustment grip.



Hole chart

Main menu: ***Mechanical - Symbols -*** ***Hole chart***.

Ribbon: ***Mechanical - Symbols -*** ***Hole chart***.

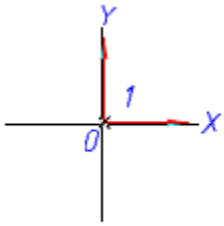
Toolbar: ***Hole chart (on toolbar*** "MCS Symbols"***)***.

Command line: ***MCHOLECHART***.

The command is designed to create automatically updated statements for a group of holes.

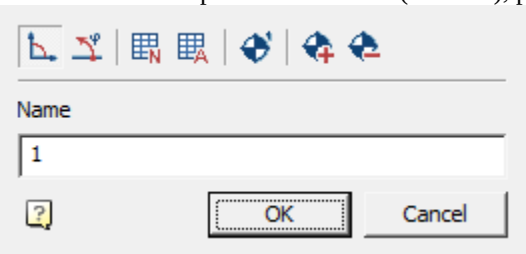
Procedure

1. Call command *"Hole chart"*.
2. Specify the insertion point of the coordinate system.



3. Set the angle of rotation of the coordinate system.

4. Specify the required holes and circles in the drawing, or create new ones using the "Create N" context menu command. To complete the selection (creation), press the "Enter" key. The "Hole chart" dialog opens.

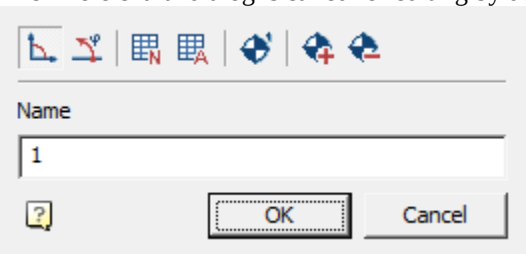


5. In the "Hole chart" dialog, specify the name and type (dekatrov or polar) of the coordinate system, add (or delete), if necessary, holes.
6. Insert into the drawing the necessary tables: List of coordinates, Table of holes, Table of values.
7. Click "OK." Tables will be built. Holes will be associated with tables and coordinate system.

Note:	To see the holes in the drawing associated with this coordinate system, double-click on the anchor.

Edit dialog

The "Hole chart" dialog is called for editing by double-clicking on the coordinate system.

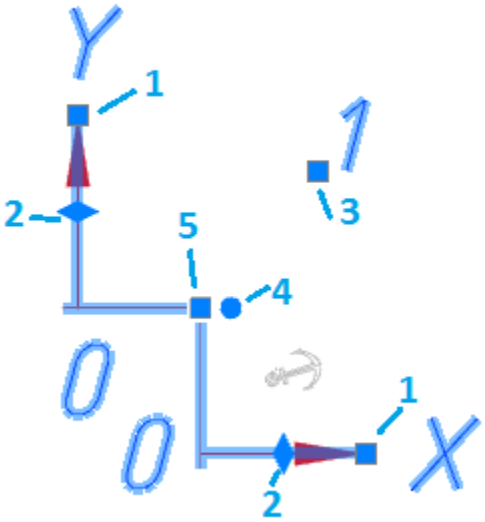


The "Hole chart" dialog contains:

- Graphic switch type coordinate system: Cartesian or Polar.
- Hole table creation buttons:
  - *List of coordinates* - an automatically updated table is formed with the coordinates of each hole.
  - *Hole table* - a table is formed, grouped by hole diameter. Each group of identical holes is assigned a letter.
- Button *Number holes* - the command automatically assigns consecutive numbers or letters to the holes (according to the setting for Hole Designation).
- Button *Add holes* - this command allows you to add holes to the report.
- Button *Remove holes* - this command allows you to remove holes from the report.
- Field "Name" - the name of the coordinate system. It is added to the hole designation as a prefix.

Grips

The handles of the report tables are similar to the handles of ordinary tables.
Coordinate system:
1. Axle Strength Handles.
2. Handles parallel transfer axes.
3. Handle move the designation of the coordinate system.
4. Handle rotation coordinate system.
5. Handle transfer coordinate system.



Text

Technical conditions

Main menu: ***Mechanical - Drawing title/ Borders -*** ***Technical conditions***.

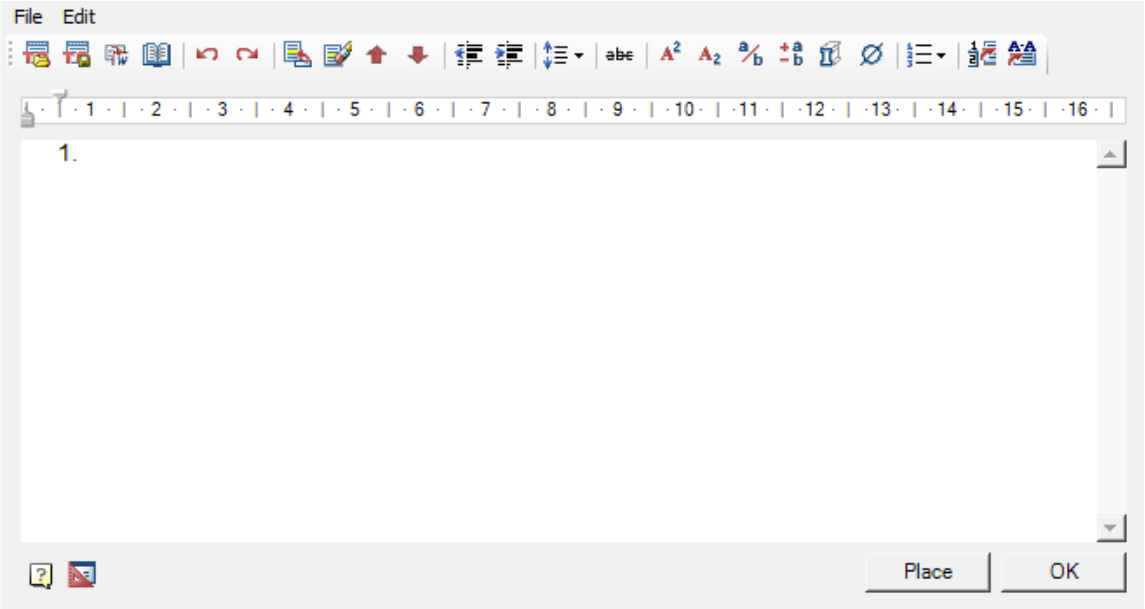
Ribbon: ***Mechanical - Title blocks -*** ***Technical conditions***.

Toolbar: ***Technical conditions (on toolbar*** "Drawing title/ Borders"***)***.

Command line: ***MCTT***.

A distinctive feature of nanoCAD Mechanica 20.1 is the possibility of linking the designation of textual information on the drawing with the text in the technical requirements.
When writing paragraphs of technical requirements (hereinafter TC), you can make reference within the proposal of the paragraph of technical requirements to any position or affixed view.
The textual information is the tolerances and dimensions, drawing zones, designations of types, cuts, sections, marking signs, marking of welded joints, position.
A change in the value of the textual information (for example, a part position number) will result in an automatic change in the TC text. If this position has been deleted, a warning will appear about the need to check a particular item of requirements.
The relationship between the TC and the designation works in the opposite direction. Changing the order of the points of the requirements (their numbering) is automatically reflected in the designation on the drawing (for example, a marking mark, the designation of which contains the item number from the technical requirements).

Editor



Main menu

- File
  - *Clear* - clear TC.
  - *Open file* - the command opens the TC from a previously saved file.
  - *Save file* - command saves TC to * .rtf file.
  - *Close without save* - the command closes the TC editing dialog box without saving.
- Edit
  - *Undo* - command cancels last action.
  - *Redo* - the command repeats the previously canceled action.
  - *Insert title* - command inserts a title in front of TC items.
  - *Delete title* - command removes TC title.

Toolbar

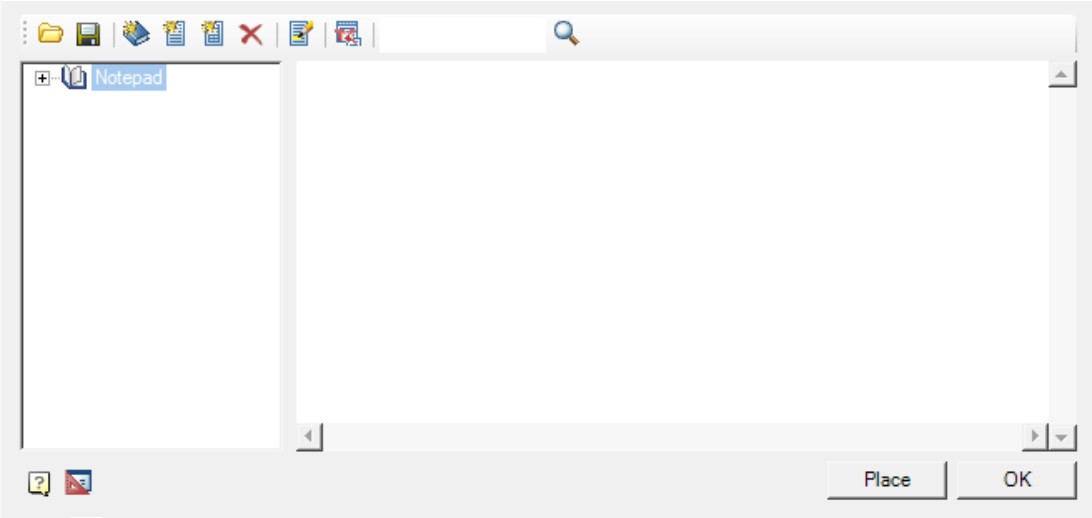
- *Open file* - the command opens the TC from a previously saved file.
- *Save file* - command saves TC to * .rtf file.

Important!	The special characters used in the editor are lost when they are transferred to an external file.

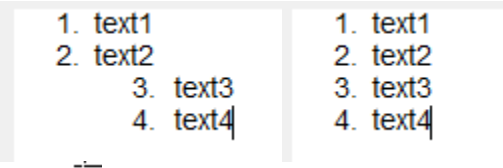
- *Fill in MSWord* - the command opens a new document in MSWord and inserts the text TC.

Important!	The special characters used in the editor are lost when they are transferred to an external file.

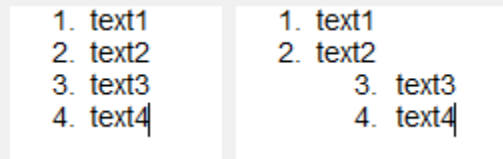
- *Notebook* - command opens interface notebook.



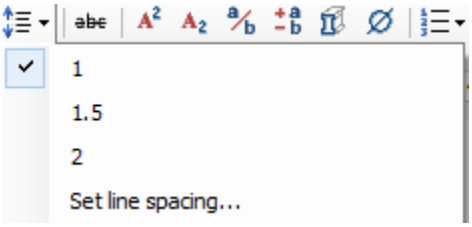
- *Undo* - command cancels last action.
- *Redo* - the command repeats the previously canceled action.
- *Add paragraph (CTRL+ENTER)* - command adds a new TC item.
- *Delete paragraph (CTRL+DEL)* - command deletes TC item.
- *Replace paragraph up* - the command moves the TC item at which the cursor is located, higher up in the list.
- *Replace paragraph down* - the command moves the TC item at which the cursor is located, down the list.
- *Decrease Indent* - the command reduces the indent of selected items.



- *Increase indent* - command increases the indent of selected items.



- *Line Spacing* - the command opens the list for setting the line spacing. Adjust line spacing of selected items.



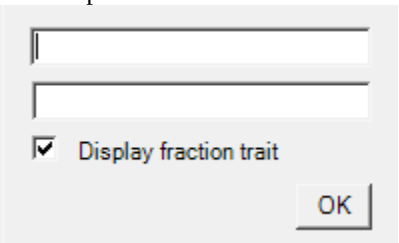
  - *1* - set 1 leading (unit for measuring line spacing). Line spacing is added after the TC item.
  - *1.5* - set 1.5 leading. Line spacing is added after the TC item.
  - *2* - set 2 leading. Line spacing is added after the TC item.

- *Set line spacing...* - the command allows you to specify your own line spacing. Line spacing is added after the TC item.

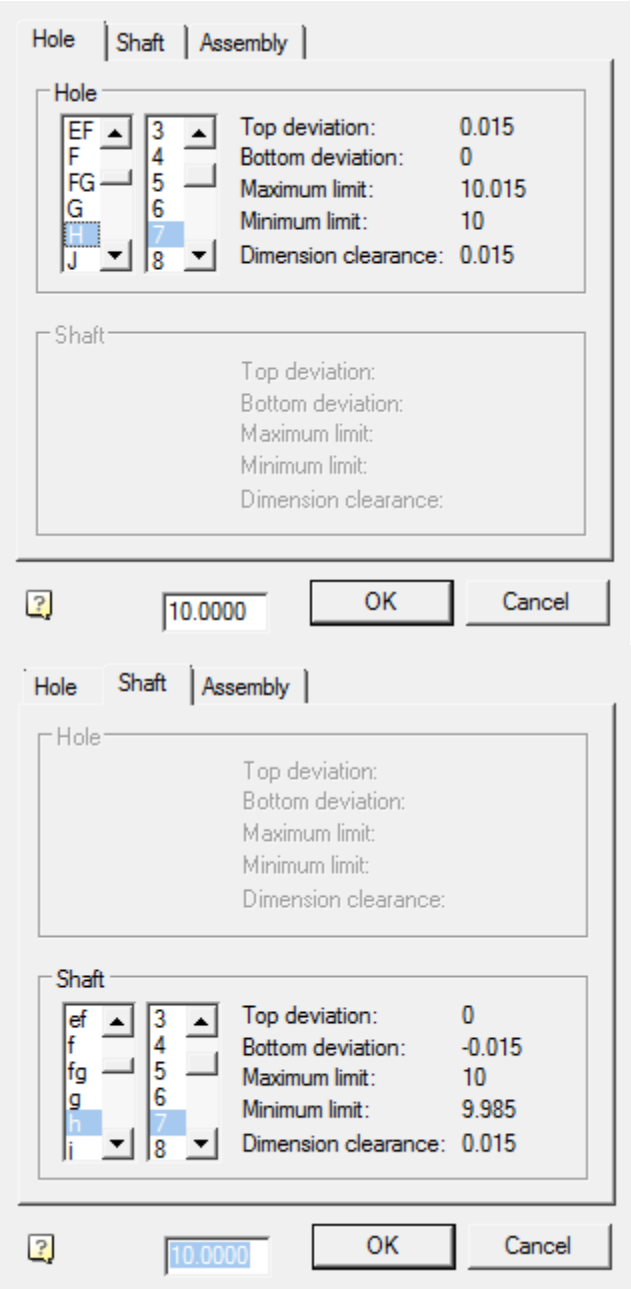
- *Superscript* - translates the selected text into superscript.

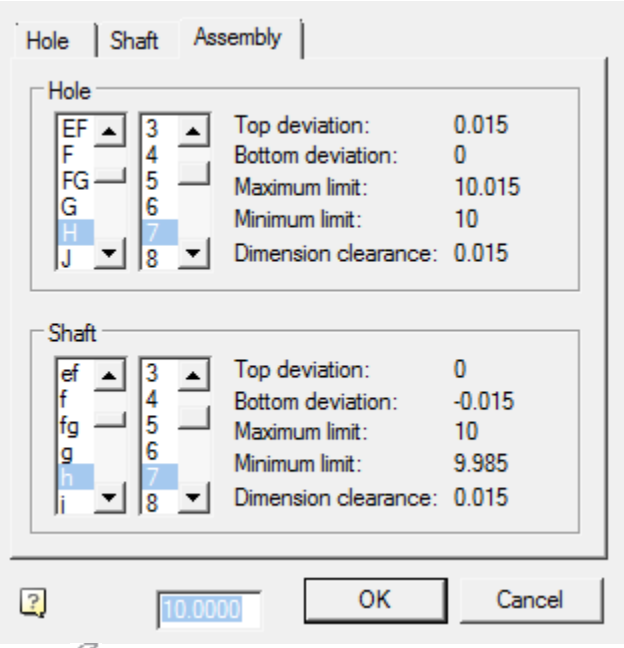
1. text$^{11}_{22}$

- *Subscript* - translates the selected portion of the text into a subscript.
- *Insert fraction* - the command calls the "Fraction" dialog box in which the numerator and denominator are specified. The fraction is inserted into the text TC.

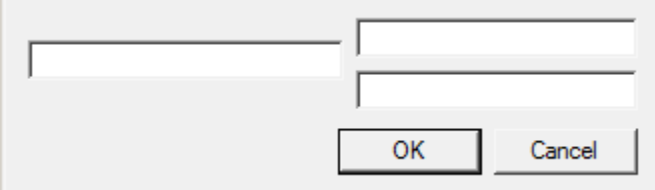


- *Insert tolerance* - provides the ability to insert a tolerance. When called, the "Tolerances" dialog box opens.

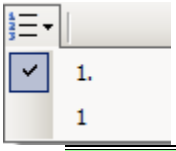

Hole
Shaft
Assembly
Hole
EF
F
FG
G
H
J
3
4
5
6
7
8
Top deviation: 0.015
Bottom deviation: 0
Maximum limit: 10.015
Minimum limit: 10
Dimension clearance: 0.015
Shaft
Top deviation:
Bottom deviation:
Maximum limit:
Minimum limit:
Dimension clearance:
10.0000
OK
Cancel
Hole
Shaft
Assembly
Hole
Top deviation:
Bottom deviation:
Maximum limit:
Minimum limit:
Dimension clearance:
Shaft
ef
f
fg
g
h
j
3
4
5
6
7
8
Top deviation: 0
Bottom deviation: -0.015
Maximum limit: 10
Minimum limit: 9.985
Dimension clearance: 0.015
10.0000
OK
Cancel



- *Insert material* - provides the ability to insert material. When called, opens the "Material" dialog.



- *Insert spec symbol* - allows you to insert special characters in the text TC.
- *Numeration* - the command allows you to select the design option for TC items.



Note:	To create a hierarchical list it is necessary to press the "Tab" key on an empty item. A new child level has been created. To return to the previous level, press the "Enter" key on an empty item.

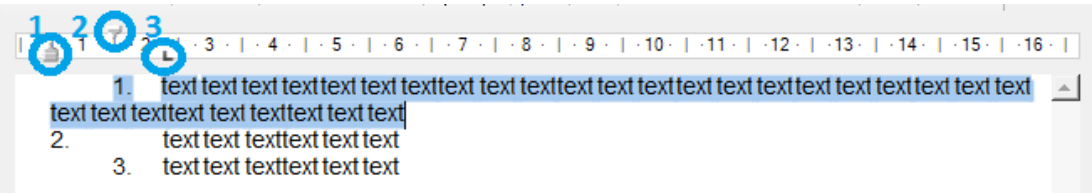
- *Insert position* - the command associates text fragments in the TC with a graphical representation of the positions in the drawing.

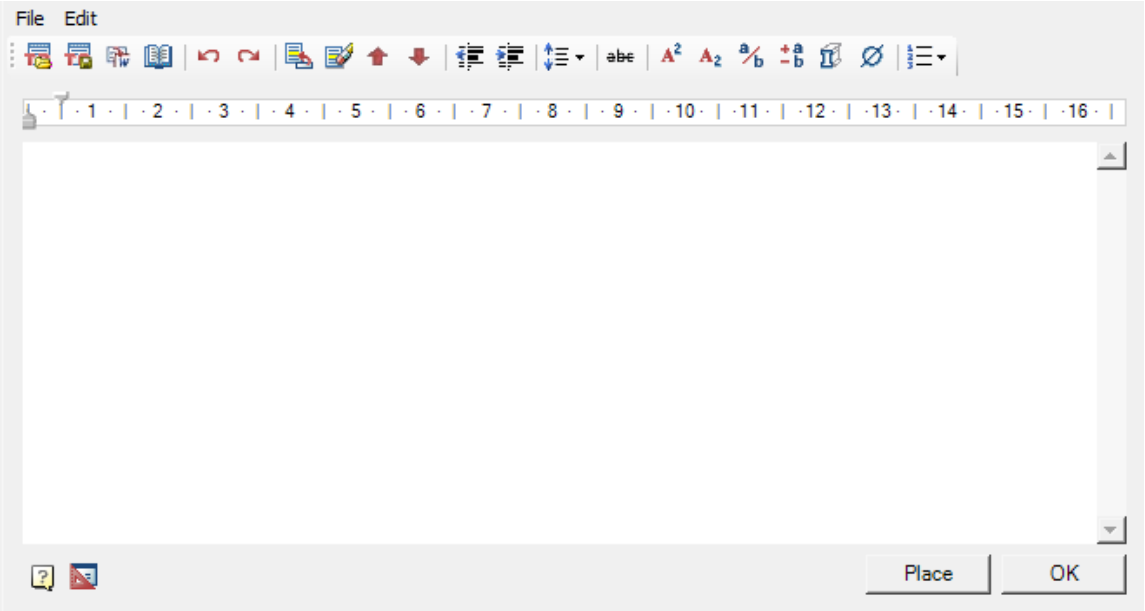
To insert a link in the TC editor, place the cursor on the insertion point of the associated symbol and click on the "Insert position" button, select a position in the drawing.
Related information is displayed in blue in the TC editor. When you click on a link, the application will display the associated graphical representation.

- *Insert view/cuts* - the command associates text fragments in TC with a graphical representation of views, sections and sections in the drawing.

To insert a link in the TC editor, place the cursor on the insertion point of the associated symbol and click on the "Insert view / cuts" button, select the view, section or section.
Related information is displayed in blue in the TC editor. When you click on a link, the application will display the associated graphical representation.

- Ruler. The ruler is used to position the lines relative to the entire TC document. Sliders are used to position the lines.


1
2
3
1. text text text



Main menu

- File
  - *Clear* - clear TP.
  - *Open file* - the command opens the TP from a previously saved file.
  - *Save file* - command saves TP to * .rtf file.
  - *Close without save* - the command closes the TP editing dialog box without saving.
- Edit
  - *Undo* - command cancels last action.
  - *Redo* - the command repeats the previously canceled action.
  - *Insert title* - command inserts a title in front of TP items.
  - *Delete title* - command removes TP title.

Toolbar

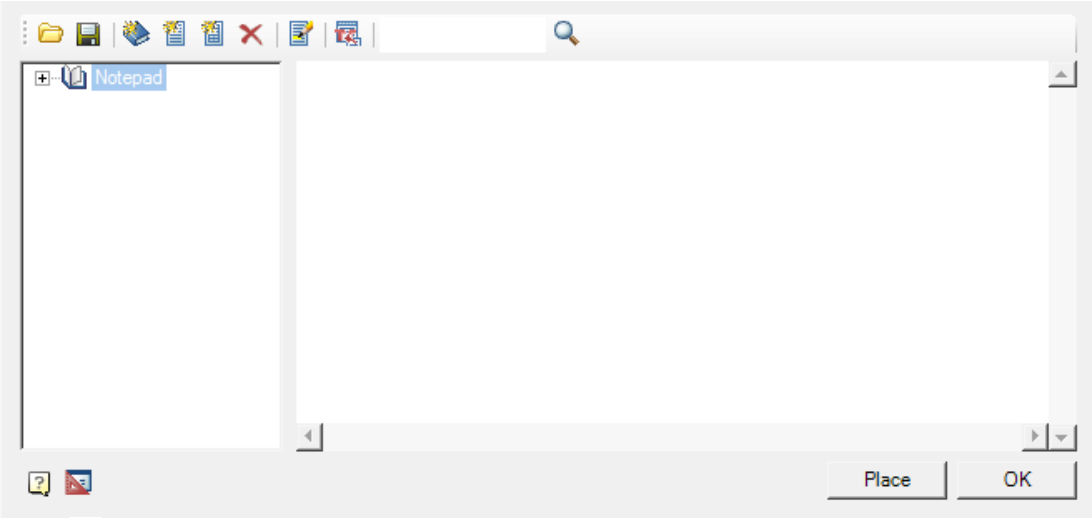
- *Open file* - the command opens the TP from a previously saved file.
- *Save file* - command saves TP to * .rtf file.

Important!	The special characters used in the editor are lost when they are transferred to an external file.

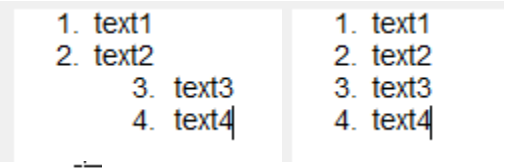
- *Fill in MSWord* - the command opens a new document in MSWord and inserts the text TP.

Important!	The special characters used in the editor are lost when they are transferred to an external file.

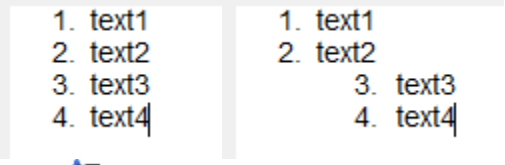
- *Notebook* - command opens interface notebook.



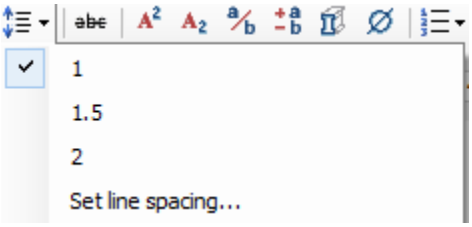
- *Undo* - command cancels last action.
- *Redo* - the command repeats the previously canceled action.
- *Add paragraph (CTRL+ENTER)* - command adds a new TP item.
- *Delete paragraph (CTRL+DEL)* - command deletes TP item.
- *Replace paragraph up* - the command moves the TP item at which the cursor is located, higher up in the list.
- *Replace paragraph down* - the command moves the TP item at which the cursor is located, down the list.
- *Decrease Indent* - the command reduces the indent of selected items.



- *Increase indent* - command increases the indent of selected items.



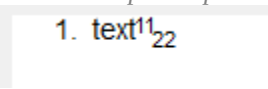
- *Line Spacing* - the command opens the list for setting the line spacing. Adjust line spacing of selected items.



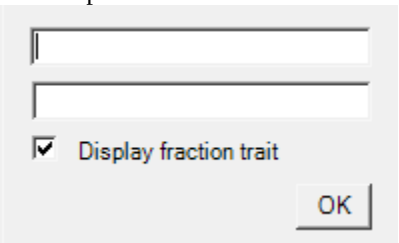
  - *1* - set 1 leading (unit for measuring line spacing). Line spacing is added after the TP item.
  - *1.5* - set 1.5 leading. Line spacing is added after the TP item.
  - *2* - set 2 leading. Line spacing is added after the TP item.

- *Set line spacing...* - the command allows you to specify your own line spacing. Line spacing is added after the TP item.

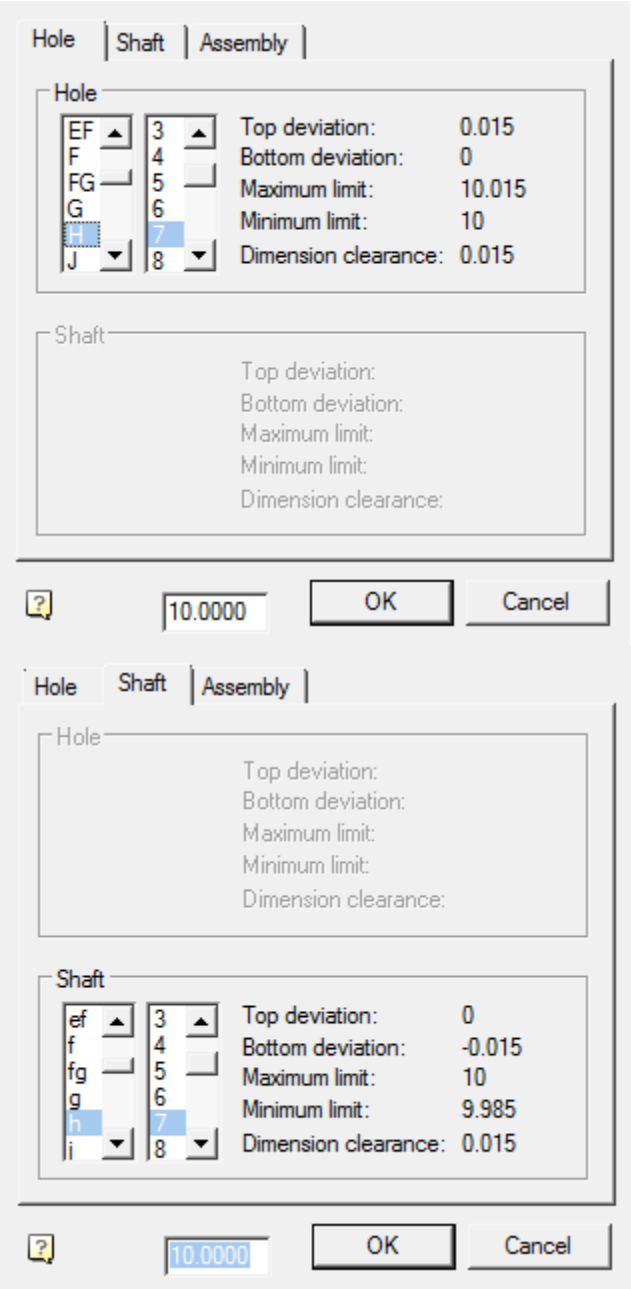
- *Superscript* - translates the selected text into superscript.

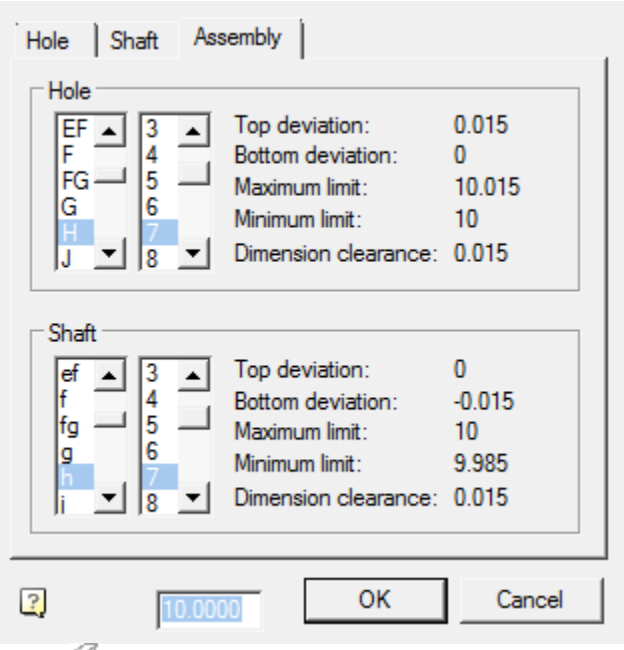


- *Subscript* - translates the selected portion of the text into a subscript.
- *Insert fraction* - the command calls the "Fraction" dialog box in which the numerator and denominator are specified. The fraction is inserted into the text TP.

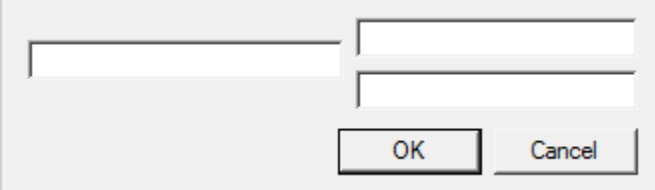


- *Insert tolerance* - provides the ability to insert a tolerance. When called, the "Tolerances" dialog box opens.

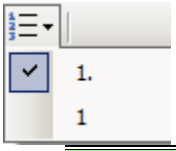

Hole
Shaft
Assembly
Hole
EF
F
FG
G
H
J
3
4
5
6
7
8
Top deviation: 0.015
Bottom deviation: 0
Maximum limit: 10.015
Minimum limit: 10
Dimension clearance: 0.015
Shaft
Top deviation:
Bottom deviation:
Maximum limit:
Minimum limit:
Dimension clearance:
10.0000
OK
Cancel
Hole
Shaft
Assembly
Hole
Top deviation:
Bottom deviation:
Maximum limit:
Minimum limit:
Dimension clearance:
Shaft
ef
f
fg
g
h
j
3
4
5
6
7
8
Top deviation: 0
Bottom deviation: -0.015
Maximum limit: 10
Minimum limit: 9.985
Dimension clearance: 0.015
10.0000
OK
Cancel



- *Insert material* - provides the ability to insert material. When called, opens the "Material" dialog.

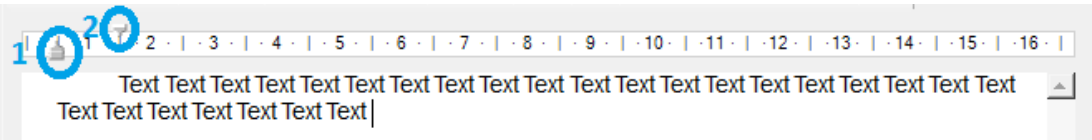


- *Insert spec symbol* - allows you to insert special characters in the text TP.
- *Numeration* - the command allows you to select the design option for TP items.



Note:	To create a hierarchical list it is necessary to press the "Tab" key on an empty item. A new child level has been created. To return to the previous level, press the "Enter" key on an empty item.

- Ruler. The ruler is used to position the lines relative to the entire TP document. Sliders are used to position the lines.



1. The bottom slider adjusts the indentation of all lines of the paragraph except the first line.
2. The top slider adjusts the indentation of the first line of the paragraph.

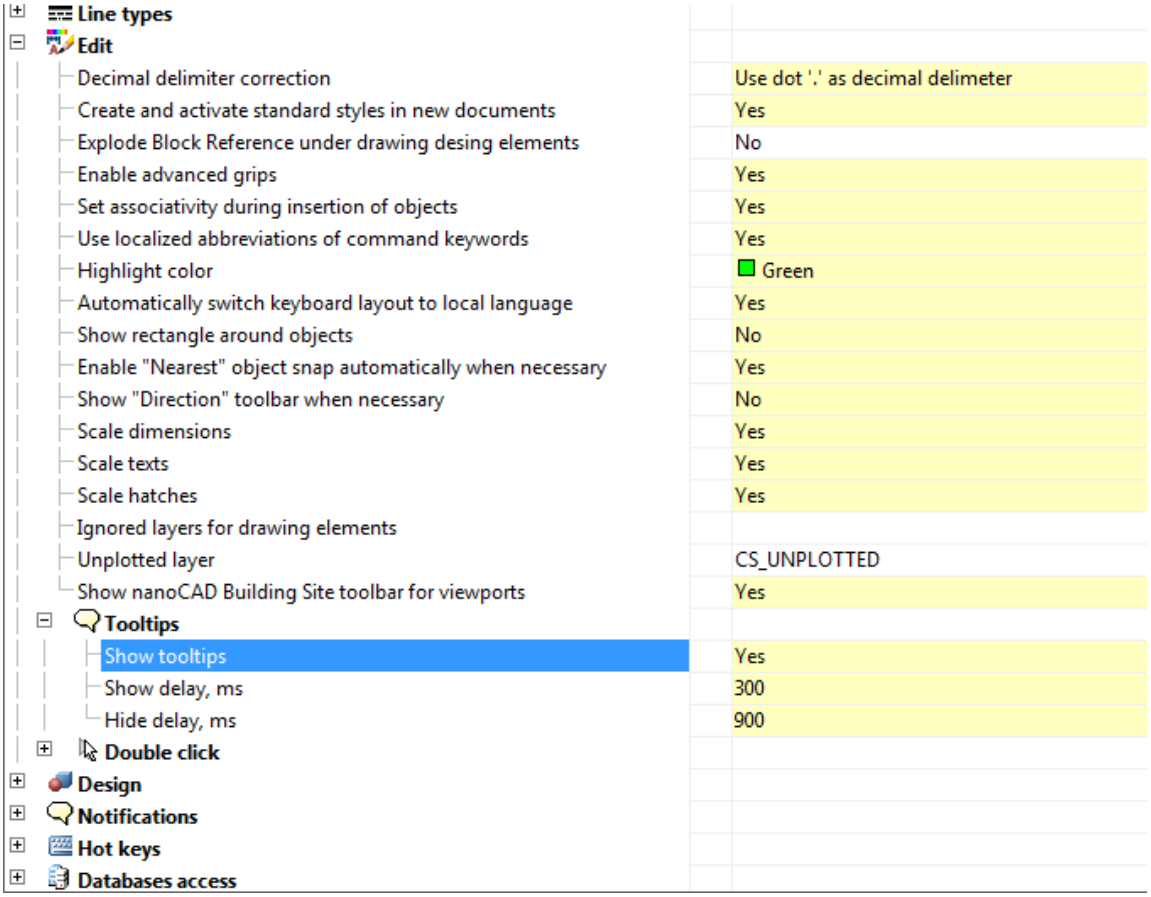
Context menu

The context menu uses standard commands for text fields.

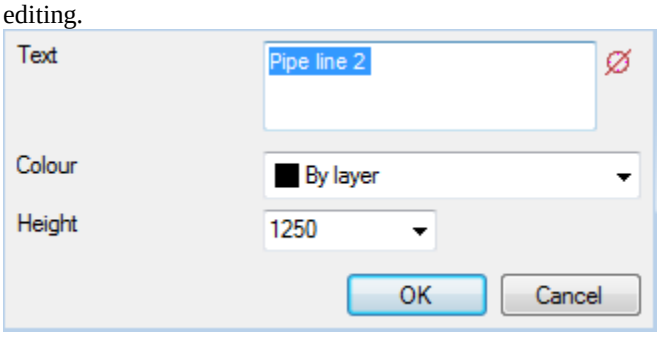
Setting text options

To set the TP text parameters (font height and slope, indent from the title block, eTP.) open the dialog box *"Settings"*.

Text editing



If it's set multiline text editing with tools nanoCAD Mechanica 20.1, then clicks on the test will be called dialogue editing.



In this dialog may ask, besides the text, its color, and the Height, and add special symbols.

Utilities

Creating a hatch

Main menu: ***Mechanical - Utilities -*** ***Create hatch***.

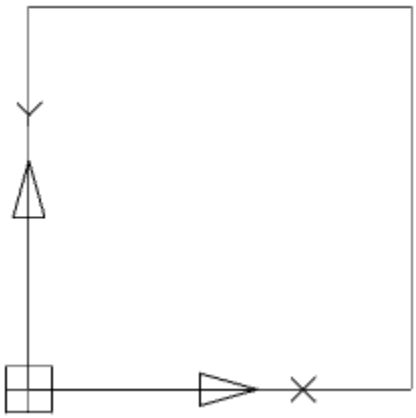
Toolbar: ***Utilities -*** ***Create hatch***.

Command line: ***MCHPATTERN***.

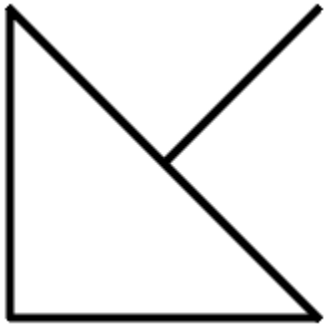
In nanoCAD Mechanica 20.1 it is possible to create hatches according to a given pattern - a sketch using the *"Create hatch"* tool. The hatch pattern created with it is written as an independent * .pat file and can be used in other projects or by other users.

Procedure

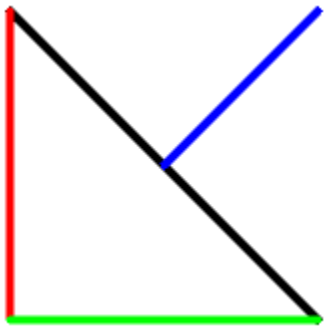
1.Create a border for the hatch sketch. Construct a 300x300mm rectangle and place its bottom-left corner at the origin of coordinate system 0,0,0. The sketch area must be at the origin of the coordinate system. Overall dimensions of the sketch area should not be more than 300x300mm.



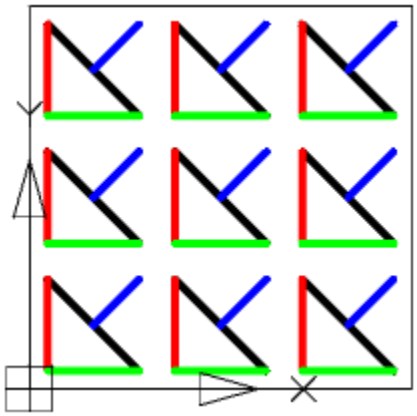
2. Sketch one element of the future hatch. The sketch needs to be done within the created rectangle. The sketch should only be done with lines. It is not allowed to sketch with arcs, circles. It is recommended to approximate curved objects in polylines without arc segments. Sketch polylines must be exploded into line segments.



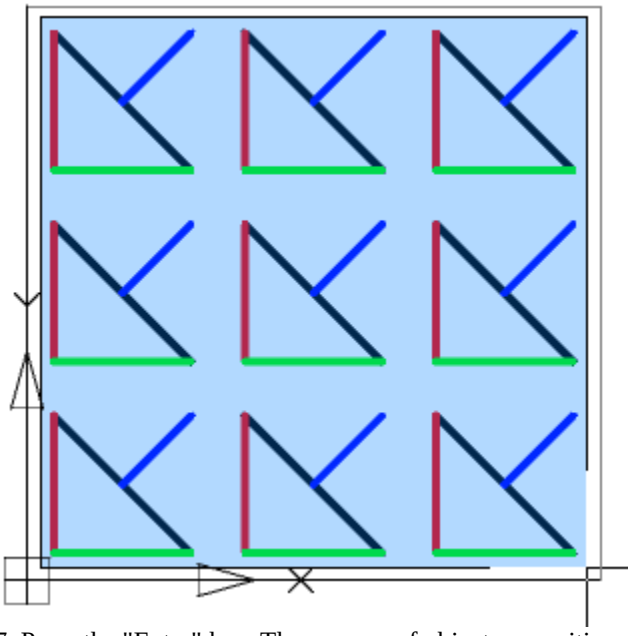
3. Assign a different color to each section of the sketch. The sketch should be done with segments of different colors. The hatch tool recognizes the periodicity in the sketch by the color of the lines.



4. Create an array of sketches within a rectangle. The sketch must follow an explicit periodicity. The sketch must include at least three repeating fragments of the future hatching.



5. Call the command *"Create hatch"*.
6. Create an array of sketches within a rectangle. The sketch must follow an explicit periodicity. The sketch must include at least three repeating fragments of the future hatching.



7. Press the "Enter" key. The process of object recognition and analysis will begin. During recognition, gaps in sketch lines are automatically determined, if in the selected pattern the line break is outside the selection boundary, the line will be defined in the hatch as continuous. If the sketch does not meet the conditions, a message box will appear, otherwise the file save dialog.
8. Correct the errors indicated in the message. Repeat step 5 - step 7.
9. In the file save dialog specify the path to the hatch file.

Important!	You cannot use the names "Acad.pat", "Acadiso.pat" as the name of the sample file. It is recommended to save in C:\ProgramData\Nanosoft\nanoCAD Mechanica 20.1\SHX

10. The custom hatch will be created. The hatch created with the *"Create hatch"* tool fully accepts all the properties of the nanoCAD hatch: associativity, the ability to change the scale, the angle of inclination, overlay with a closed tolerance.

Important!	A drawing file * .dwg that used a unique hatch must be transferred to another user along with the * .pat file of the hatch pattern. For a successful translation, use the (eTransmit) "Generate Set" command.

Objects numeration

Main menu: ***Mechanical - Utilities - Objects numeration***.

Ribbon: ***Mechanical - Utilities - Objects numeration***.

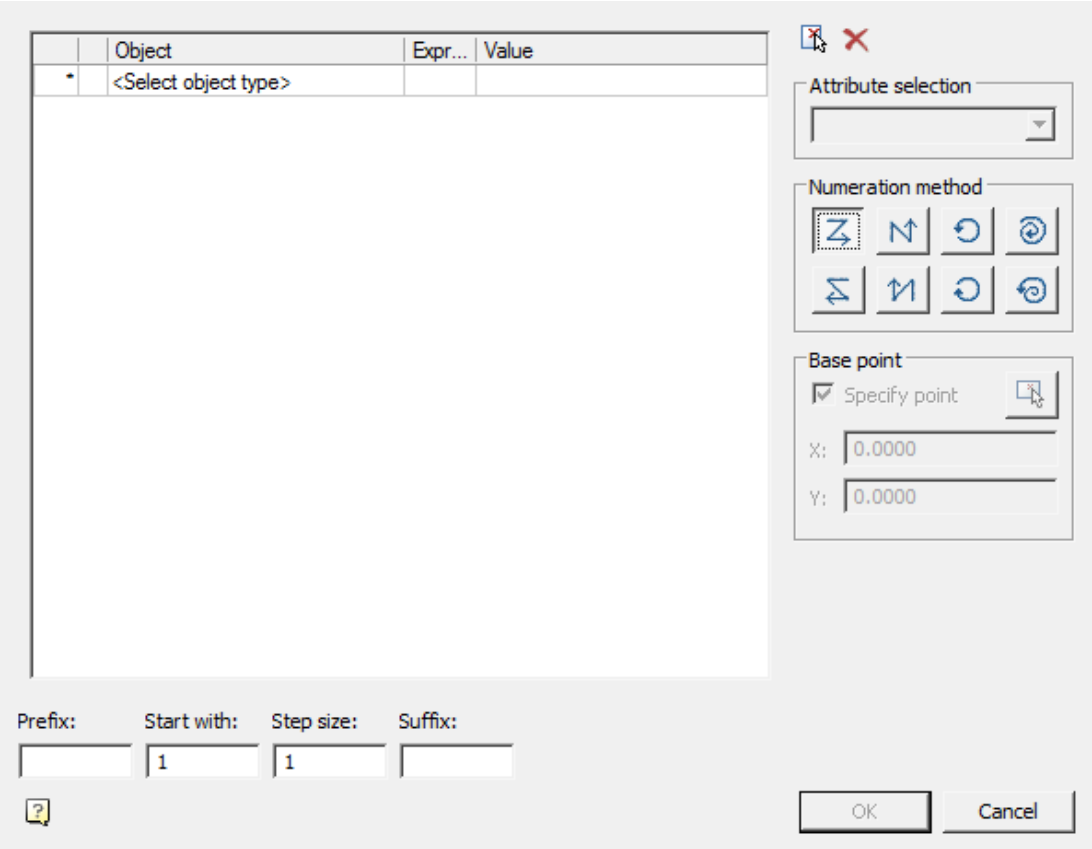
Toolbar: ***Objects numeration (on toolbar*** "MCS Utilities"***)***.

Command line: ***MCNUMOBJ***.

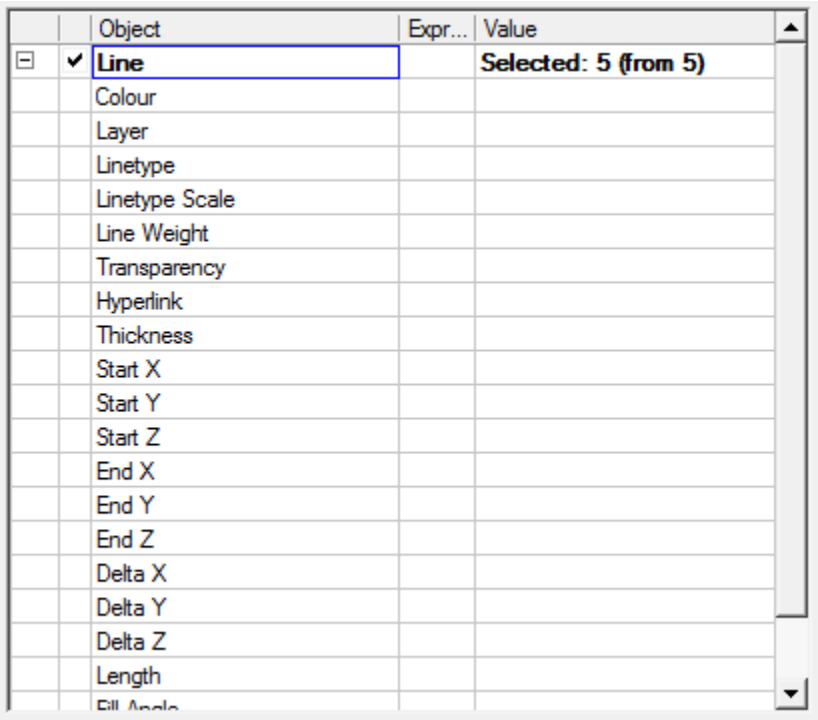
The command finds all specified objects with attributes and numbers it according to a given algorithm.

Procedure

1. Call the team. The "Object Numbering" dialog opens.



2. Select objects. Select an object using the quick selection. All objects that have text editable attributes (block, database object, universal marker, etc.) can be selected.

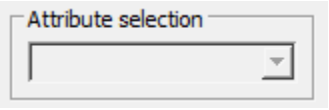


After selecting an object, a list of attributes common to the selected objects and available for editing is formed.
Additional list management commands:

Set selection - the command allows you to specify objects in the drawing.

Reset conditions - command clears the list of conditions.

3. Select the attribute in which the result of numbering will be recorded.



4. Select the numbering method:

- from left to rigth, from top to bottom. When choosing this method, the command compares the coordinates of points X and Y, respectively, starts numbering from the highest and leftmost objects to the lowest and right relative to the base point.

- from bottom to top, from left to right. When choosing this method, the command compares the coordinates of points X and Y, respectively, starts numbering from the lowest and leftmost of the entire object to the highest and right relative to the base point.

- from rigth to left, from top to bottom. When choosing this method, the team compares the coordinates of points X and Y, respectively, starts numbering from the topmost and to the right of the entire object to the lowest and leftmost relative to the base point.

- from bottom to top, from right to left. When choosing this method, the command compares the coordinates of points X and Y, respectively, starts numbering from the lowest and most right of the entire object to the highest and left relative to the base point.

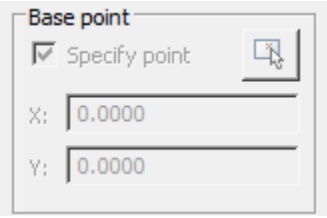
- radial anticlockwise. When choosing this method, the command compares the coordinates of points in the polar coordinate system, the distance from the center (specified by the user base point) and the angle between the zero direction (positive X-axis direction). Numbering is counterclockwise. When this method is selected, the selection of the base point becomes available.

- radial clockwise. When choosing this method, the command compares the coordinates of points in the polar coordinate system, the distance from the center (specified by the user base point) and the angle between the zero direction (positive X-axis direction). Priority in comparison has an angle. Numbering is clockwise. When this method is selected, the selection of the base point becomes available.

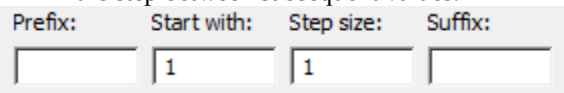
- spiral anticlockwise from center. When choosing this method, the command compares the coordinates of points in the polar coordinate system, the distance from the center (specified by the user base point) and the angle between the zero direction (positive X-axis direction). Priority when comparing has distance from the center. The numbering is counterclockwise from the center. When this method is selected, the selection of the base point becomes available.

- spiral clockwise to center. When choosing this method, the command compares the coordinates of points in the polar coordinate system, the distance from the center (specified by the user base point) and the angle between the zero direction (positive X-axis direction). Priority when comparing has distance from the center. Numbering is clockwise to the center. When this method is selected, the selection of the base point becomes available.

5. Indicate a base point if necessary. The base point is indicated either by entering the X and Y coordinates, or by pointing to the screen. When cocking the "Specify point" checkbox, the point is indicated after pressing the "OK" button.



6. Customize the brand. The formation of the brand comes from three parts. The prefix and suffix is specified in the editable field. And the brand value itself is calculated based on the initial value entered by the user and the step between subsequent values.



7. Click "OK."
8. If the "Specify point" checkbox has been set, specify the base point in the drawing.
9. Numbering will be made.

Measurements

Main menu: ***Mechanical - Utilities -*** ***Measure***.

Ribbon: ***Mechanical - Design -*** ***Measure***.

Toolbar: ***Measure (on toolbar*** "Utilities"***)***.

Command line: ***MCMEASURE***.

Measurements or tool with Mark drawing is designed to extract various properties of objects in the drawing. After selecting the command a dialog box *value picker* opens.

Measure distance. Measures the distance between two points, radius or diameter. Switching the measurement method is via the context menu.

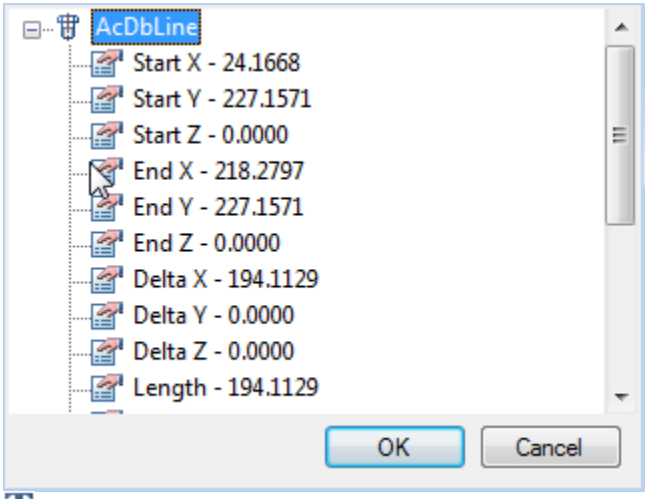
Measure angle. Measures the angle.

Measure Perimeter. Measure the perimeter of the closed area, the length of a closed polyline or circle.

Measure area. Measures the area of the closed area. Select an area in two ways: 1 - Click inside the closed loop, 2 - Click on the closed polyline or circle.

Complex area. Measures the area of several loops. Selecting the area is carried out: 1 - Click to specify the first closed loop, 2 - Click to specify the second closed loop 3 - Press ***Enter*** to complete the command.

Take from property. Allows you to get the value of the parameter selected in the drawing object nanoCAD Mechanica20.1.



Take from text. Returns the value of the selected single-line or multi-line text.

Calculate. Opens calculator.

Take from notes. Opens the notebook.

The measured values are displayed in the dialog box.

If universal measuring time was part of a command (for example, the dialog box *nanoCAD Mechanica20.1 text* context menu command), the measured value is transmitted to the command being executed.

Fragment Copy

Main menu: ***Mechanical - Utilities -*** ***Fragment copy***.

Ribbon: ***Mechanical - Utilities -*** ***Fragment copy***.

Toolbar: ***Fragment copy (on toolbar*** "Utilities"***)***.

Command line: ***MCCOPY***.

The command is used to create remote views of the drawing.

1. Select the drawing *center point for the detail view.*
2. Select the area of the drawing. In the context menu, you can choose a certain kind of marquee - circle or rounded rectangle. Secure selection click.
3. Specify the drawing *center remote fragment.* available in the context menu command "0-cut" to remove the original objects from the drawing.

Plate Array

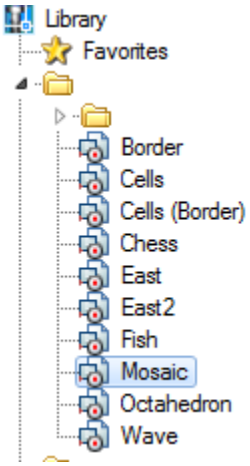
Main menu: ***Mechanical - Utilities -*** ***Plate array***.

Ribbon: ***Mechanical - Utilities -*** ***Plate array***.

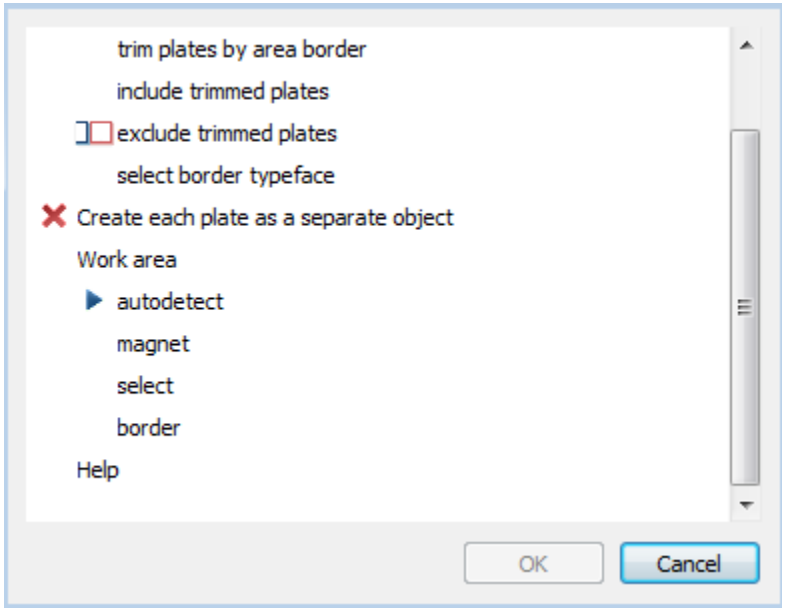
Toolbar: ***Plate array (on toolbar*** "Utilities"***)***.

Command line: ***MCPLTARRAY***.
Command to tile layouts for a given circuit.

1. Choose from a base tile pattern and place it on the drawing.

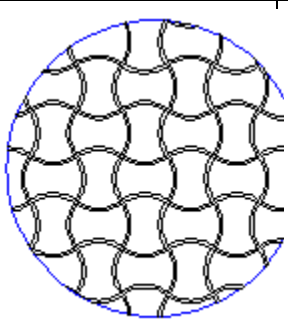
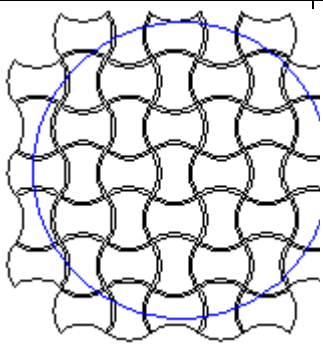
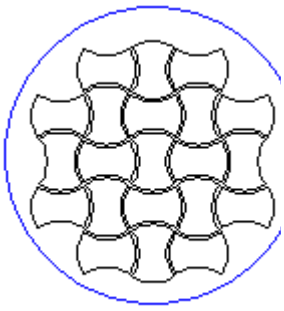
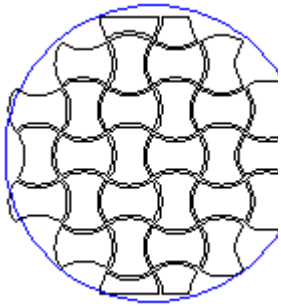


2. Press the ***plate array***.
3. In the opened window, click the *plate shema* and specify the *token group*.



4. Select *Trim mode extreme tiles* .
   - Trim plates by area border - tiles clipped to the boundary of the layout. Circuit tile becomes open. This tile is taken into account in property "Cut". This mode is not available if you enable the check box "Make each tile a separate."
   - Include trimmed plates - extreme tiles stacked atop circuit layout.
   - Exclude trimmed plates - tiles, overlapping circuit layouts do not fit.

- Sect border type face- if tiles have execution end elements will be selected most podhodyaeschee. When the layout of the area selected by the execution system names: Whole, Left, Right, Top, Bottom. When the layout along the border all versions checked sequentially, regardless of names.

Trim plates by area border	Include trimmed plates	Exclude trimmed plates	Sect border type face
			

5. *Create each plate as a separate object*.

If the box is checked, the result will be a group of independent layout objects - copies of the original objects. This mode does not support cutting elements on the boundaries of the area, but allows more flexibility to edit the result. When the check box is created off a single object that includes all geometry. In the properties of this object will be information on the number, weight and costs of each type of tiles, as well as the total weight (based on the whole tiles) and cost. You can delete individual tiles by clicking on them with the right. In this case the total amount will be adjusted, weight and cost.

6. Determine the *work area* tiles.
   - auto detect - layout area defined contour, which is located inside the tile pattern. In the inner islands stacked tiles will not.
   - magnet - loop selection tool "magnet".
   - select - area layout is selected in the drawing indicating one or more loops.
   - border - tiles splits along the perimeter or primitives nanoCAD. Cropping mode in this case will simulate laying curbs.
7. To complete the command, press *OK*.

Note:	Creating a template tile layouts can be done using Object Wizard.

Creating an array of objects

Main menu: ***Mechanical - Utilities -*** ***Array***.

Ribbon: ***Mechanical - Utilities -*** ***Array***.

Toolbar: ***Array (on toolbar*** "Utilities"***)***.

Command line: ***MCARRAY***.

Replication called ordered distribution of copies of objects by step and quantity. Duplication of graphics in the nanoCAD Mechanica20.1 is performed using a dialog box arrays.

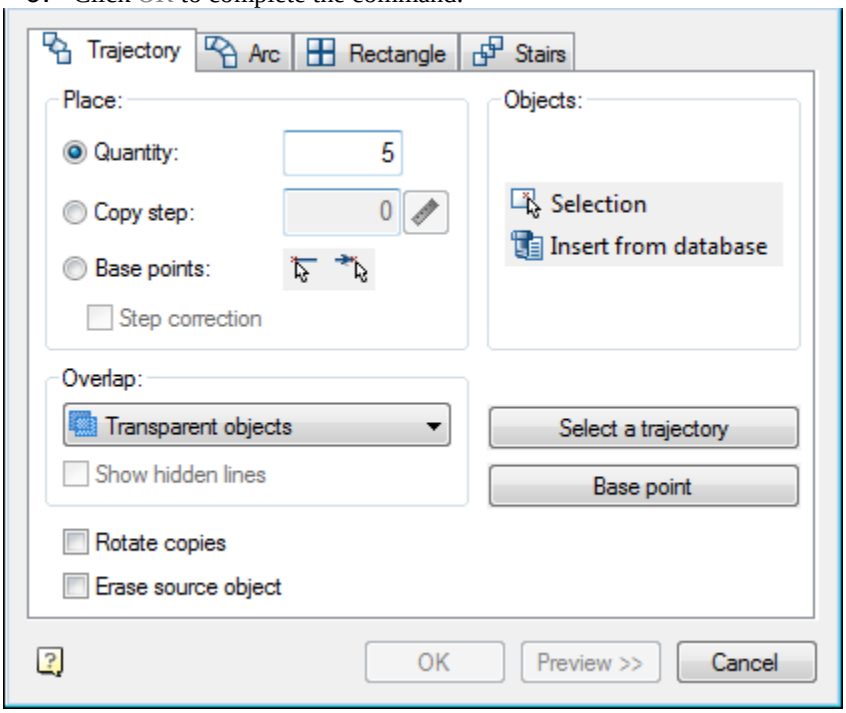
Four tabs in the Arrays dialog box define four (4) different object replications.

- Arrays along an arbitrary path
- Arrays for arc trajectory
- Rectangular array
- An array of step

For the distribution of objects within a specified area used [for an array field].

Arrays along an arbitrary path(Trajectory)

1. Tab *Path* enter the governing parameters.
2. Follow *The selection object* (object selection).
3. Specify *the base point*.
4. Select *trajectory.*
5. Follow *View* .
6. Click *OK* to complete the command.



At the top of the window is a group of controls *placement*, which allows you to select:

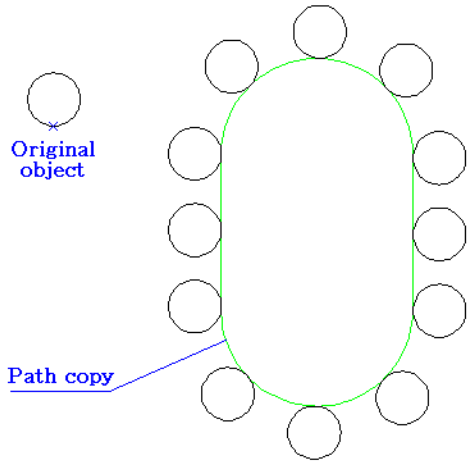
- Quantity;
- Copy step;
- Base point;
- Step correction;

In the drop-down list make a choice of Overlap:

- Transparent objects;
- Next under preceding;
- Next Over preceding;
- Interleave.

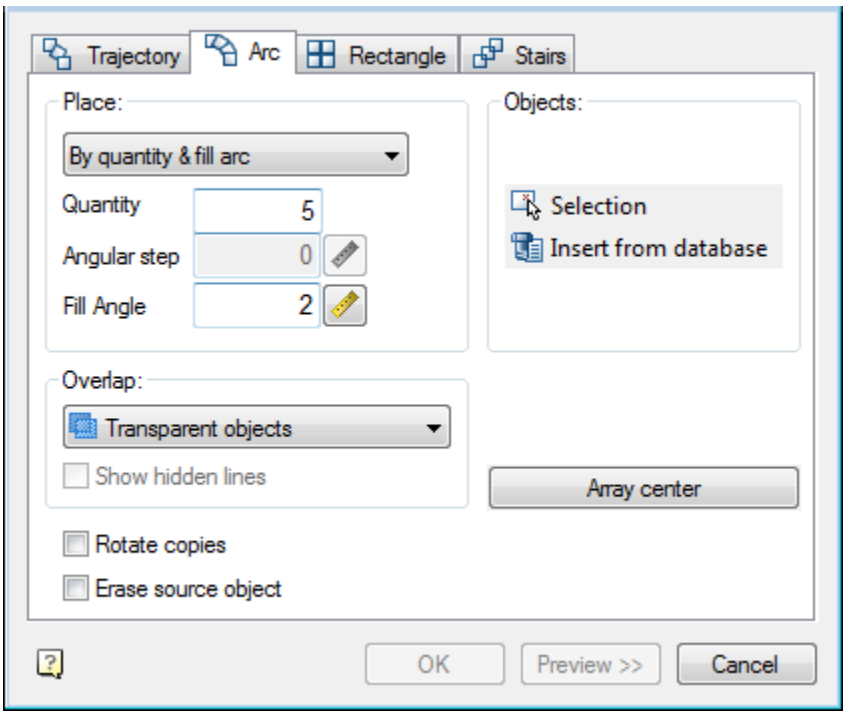
Button *pick from drawing* step allows you to select copies of the objects in the drawing.
Switch *Show hidden line* controls the display of hidden lines in the drawing. If the switch is on the hidden lines are displayed in the drawing.
Switch *Rotate copies* controls the orientation of the copied object. If the switch is on when copying objects are rotated so that the vector Y was directed perpendicularly to the path (as shown in the figure).
Button Selection *Selection* to select the replication object in the drawing.

Button Insert from database *from the base* to select the object from the database replication of standard elements.
Button *Base point* lets you choose the base point of duplicated object (normally it should be expressed in the center or corner of the object).
Button *select a trajectory* to select a graphic object, along which you will be replicating. As the trajectory can be used lines, polylines, arcs, circles.



Arrays for arc trajectory

1. Tab *Arc* enter the governing parameters.
2. Follow *The selection object* (object selection).
3. Select *Center array*.
4. Follow *Preview* .
5. Click *OK* to complete the command.



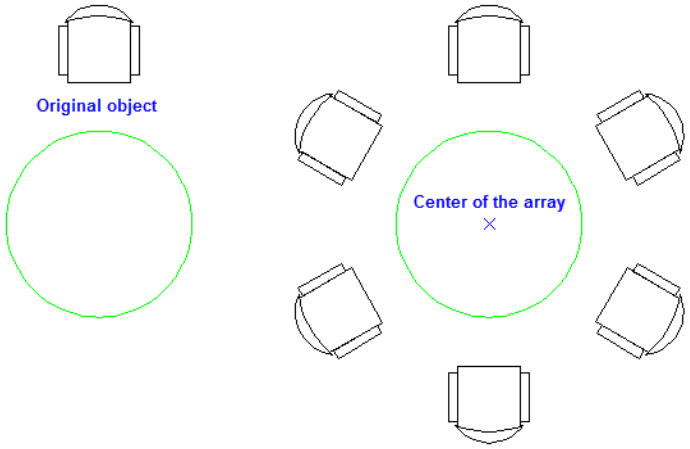
At the top of the window is a control group *accommodation*, which lets you choose placing of replicated objects:

- By number and step;
- By number and angle;
- By step and corner;

In the input field *Number* specify the number of replicated objects.
In the entry field *Step Angle* enter the angle between the replicable elements.
In the input field *angle* specify the angle required.
Button *with Mark drawing* step allows you to select the corner or angle with the drawing.
In the drop-down list make a choice of Overlap:

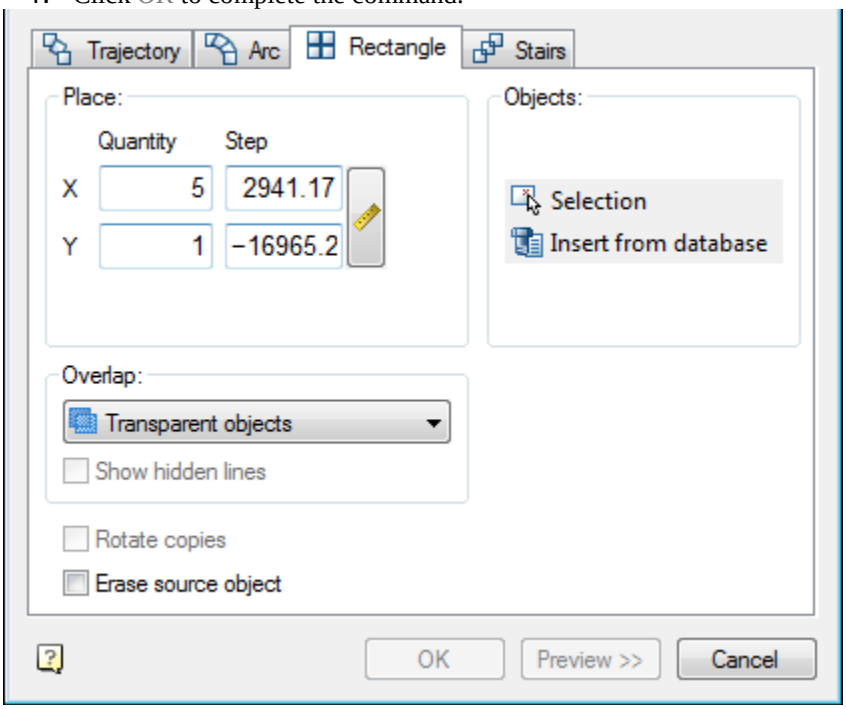
- Transparent objects;
- Next under preceding;
- Next over preceding;
- Interleave.

Switch *Show hidden line* controls the display of hidden lines in the drawing. If the switch is on the hidden lines are displayed in the drawing.
Button Selection *Selection* to select the replication object in the drawing.
Button Insert from database *insert from database* used for selecting a center point which the objects will be replicated.
Button *array center* used for selecting a center point which the objects will be replicated.
Switch *Rotate copies* controls the orientation of the copied object. If the switch is on the turn when copying objects are shown in the figure.



Rectangular array

1. Command *Rectangle* enter key parameters.
2. Follow *The selection object* (object selection).
3. Follow *Preview* .
4. Click *OK* to complete the command.

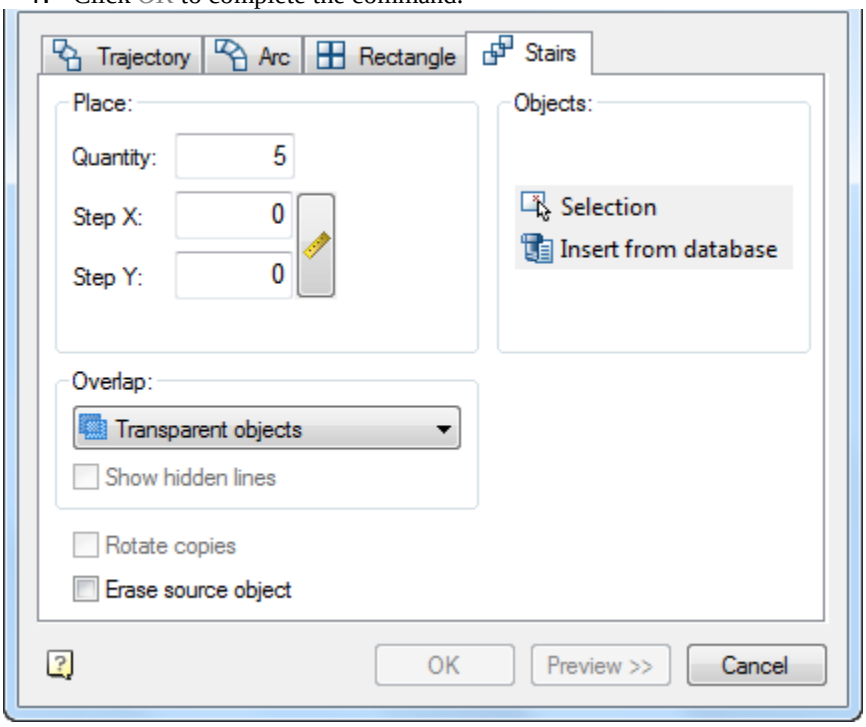


Enter the number of objects in the array horizontally and vertically, to specify the amount of horizontal and vertical steps between objects.

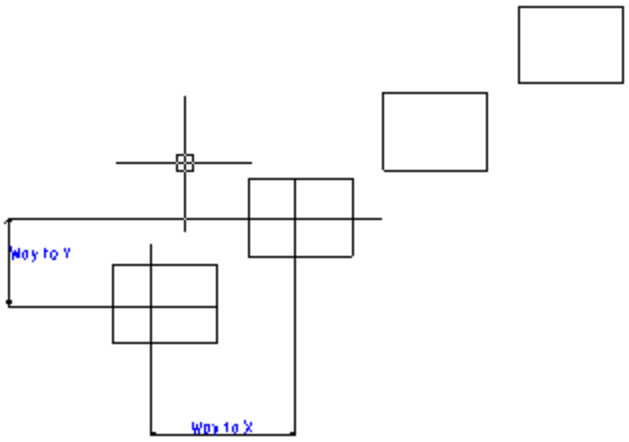
Steps of an array

1. Tab *steps* enter the governing parameters.
2. Follow *The selection object* (object selection).

3. Follow > *View* .
4. Click *OK* to complete the command.



Enter the number of objects in the array, specify the amount of horizontal and vertical step between objects.



Area array

Main menu: ***Mechanical - Utilities -*** ***Area array***.

Ribbon: ***Mechanical - Utilities -*** ***Area array***.

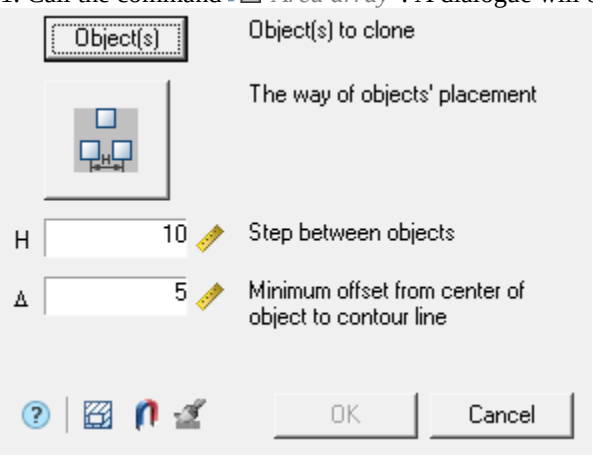
Toolbar: ***Utilities -*** ***Area array***.

Command line: ***MCARDARRAY***.

The tool is designed to replicate objects within a given area.

Procedure

1. Call the command *"Area array"*. A dialogue will open *"Area array"*.



2. Click the *"Object(s)"* button and select the original objects to copy in the drawing. Press the *"Enter"* key to complete the selection.
3. Select the method of placing the copied objects using the graphic switch:

- Rhombus.

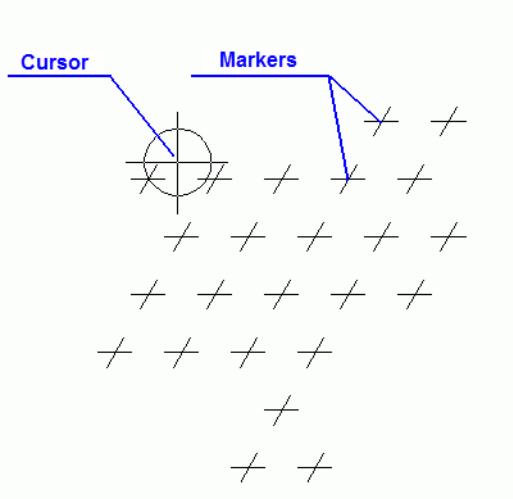
- Square.

- Triangle.

4. Specify the spacing (H) between adjacent copies. The "Pick from drawing" button allows you to select a step from the drawing.
5. Specify the minimum indent (Δ) from the edge of the area. The "Pick from drawing" button allows you to select an indent from the drawing.
6. Set the area of the array in one of three ways:

- *"Select an area by the point"*. The construction area is formed from a set of closed areas. The command allows you to add closed areas to the set by clicking the LMB inside a closed contour in the drawing. Clicking again inside the contour removes the selected area from the set. To complete the selection, press the *"Enter"* key.
- *"Select contour with the magnet"*. The contour of the array area is set using the utility "Magnet".
- *"Paint area"*. The command allows you to specify an arbitrary placement of copied objects, providing a set step between adjacent copies. This command is available only after selecting the source objects to copy.

The construction of the area is carried out by moving the mouse cursor over the drawing (construction must be started from the location of the original objects). As you move the cursor, markers will appear on the screen to indicate where the copied objects are inserted.



To remove insertion markers, press and hold the *"Shift"* key. When you hover over a marker, the marker will be deleted.
To temporarily disable the placement of markers, press and hold the *"Ctrl"* key.
Finish drawing the area by clicking LMB. An array by area will be built.
7. Click the *"OK"* button. An array by area will be built.

Centerlines

Main menu: ***Mechanical - Utilities -*** ***Centerlines***.

Ribbon: ***Mechanical - Utilities -*** ***Centerlines***.

Toolbar: ***Utilities -*** ***Centerlines***.

Command line: ***MCAXIS***.

The command is designed to draw centerlines.

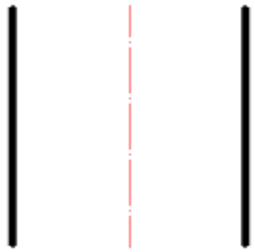
Procedure

The construction method depends on the selected objects:

- *Parallel lines*. The centerline will be drawn in the middle between these segments.

1. Call the command *"Centerlines"*.
2. Select the first line.
3. Select the second line.
4. Specify manually the first point of the centerline or press the *"Enter"* key. When you press the *"Enter"* key, the centerline is drawn automatically with a protrusion specified in the settings (by default 2mm).
5. Specify manually the second point of the centerline or press the *"Enter"* key. When you press the *"Enter"* key, the centerline will stop drawing, the second point will be in the calculated center between the lines.

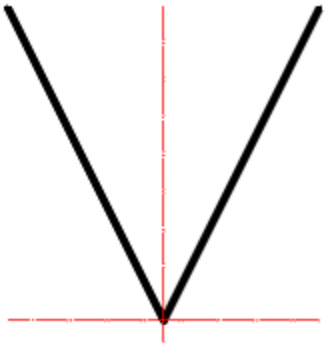
In the case of manual selection of the start / end point of the centerline, the amount of the protrusion is controlled by the user.



- *Non-parallel line segments*. The centerline will be drawn along the bisector of the angle between the line segments.

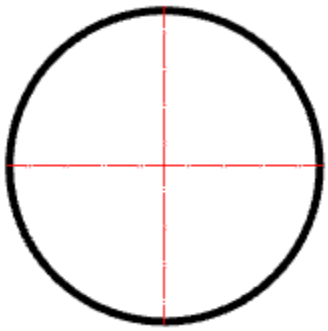
1. Call the command *"Centerlines"*.
2. Select the first line.
3. Select the second line.
4. Select the direction of the centerline. Using the *"Rotate"* context menu command, you change the direction of the centerline drawing (available only for intersecting line segments).
5. Specify manually the first point of the centerline or press the *"Enter"* key. When you press the *"Enter"* key, the centerline is drawn automatically with a protrusion specified in the settings (by default 2mm).
6. Specify manually the second point of the centerline or press the *"Enter"* key. When you press the *"Enter"* key, the centerline will stop drawing, the second point will be in the calculated center between the lines.

In the case of manual selection of the start / end point of the centerline, the amount of the protrusion is controlled by the user.



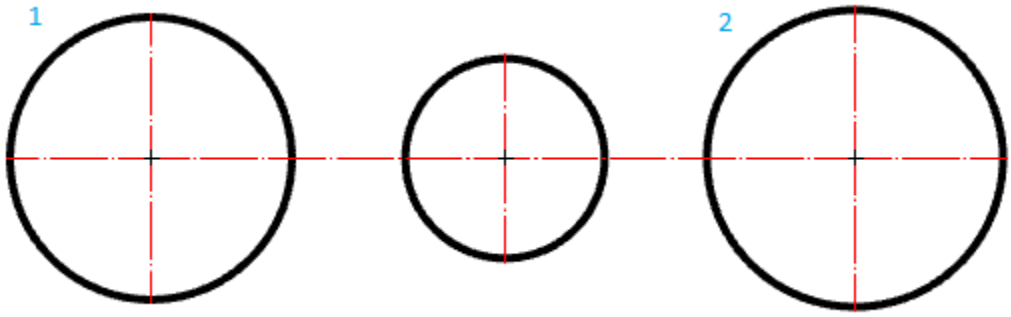
- *One circle*. Draws centerline centerlines on the selected circle.

1. Call the command *"Centerlines"*.
2. Pick a circle.
3. Press the *"Enter"* key.



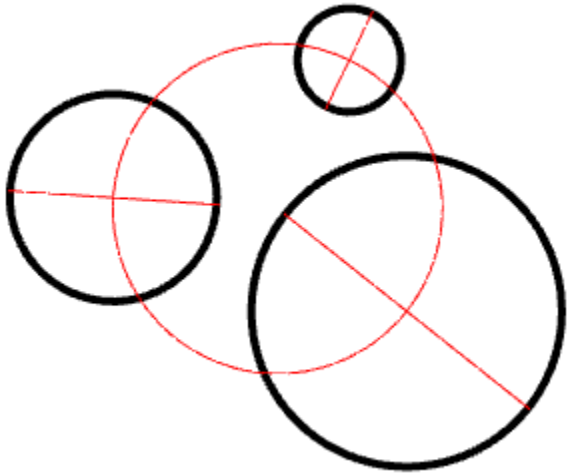
- *Two or more circles*. Draws centerline centerlines on selected circles with one common centerline.

1. Call the command *"Centerlines"*.
2. Select the first circle.
3. Select the second circle.
4. Press the *"Enter"* key. If on the line between the centers of the two selected circles there are also centers of other circles, then for such circles, axial ones will also be built.



- *Three mismatched circles*. Creates a circle of centers passing through the centers of the selected circles. Center axes are automatically added to all circles whose centers coincide with the constructed center circle.

1. Call the command *"Centerlines"*.
2. Select the first circle.
3. Select the second circle.
4. Select the third circle.



Fillet

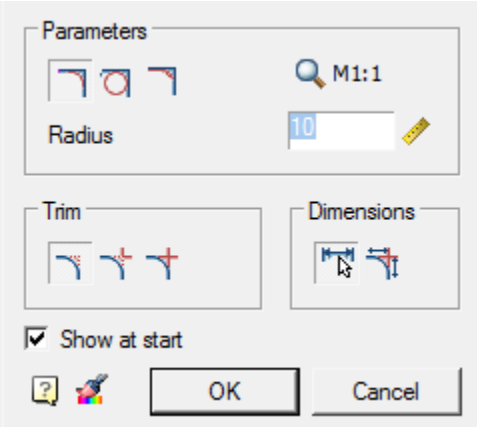
Main menu: ***Mechanical - Utilities -*** ***Fillet***.

Ribbon: ***Mechanical - Utilities -*** ***Fillet***.

Toolbar: ***Fillet (on toolbar*** "Utilities"***)***.

Command line: ***MCFILLET***.

Command Options



Pick from drawing

Mode *full trim contour lines*

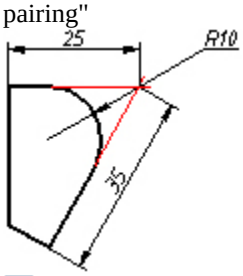
Mode *partial trim lines to their intersection*

Mode *without cutting*

Copying properties

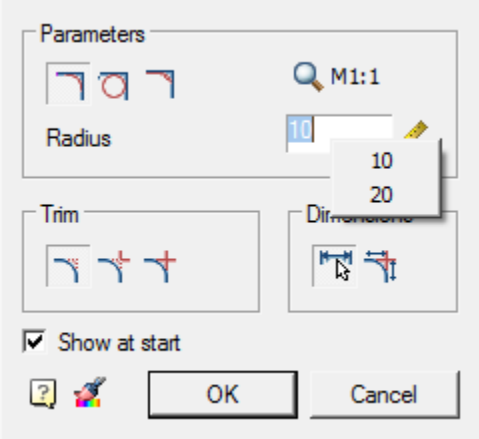
Measure fillet. Tabulated fillet radius and the number

Extra measurement. Stamped dimensions of the original geometry. Active only when the button "dimensioned pairing"



Insert circle as fillet.

Double click the left mouse button or click the right mouse button in the input fields of values, a context menu will open with a list of the last entry:

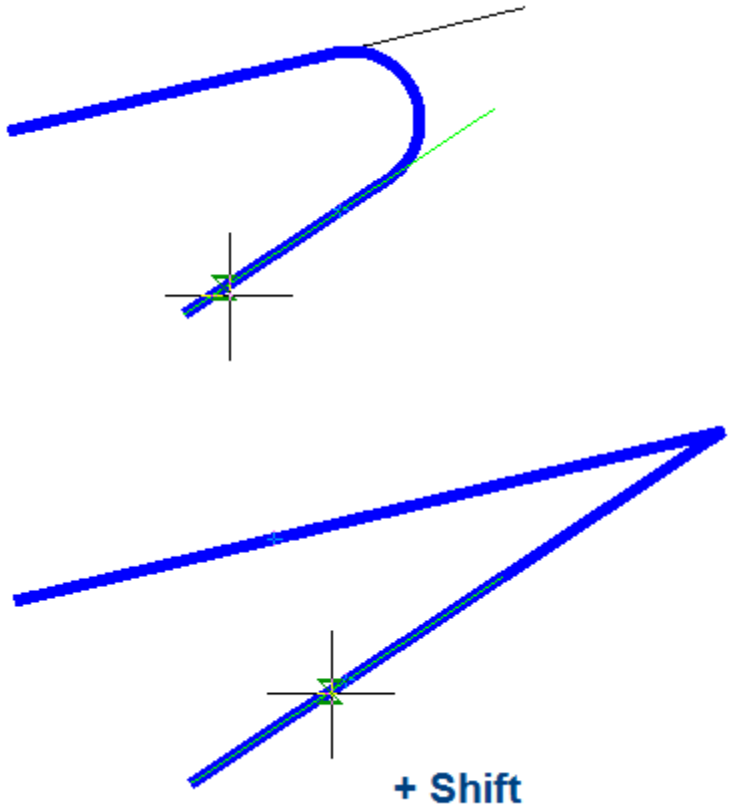


When a command from the context menu, the following options::

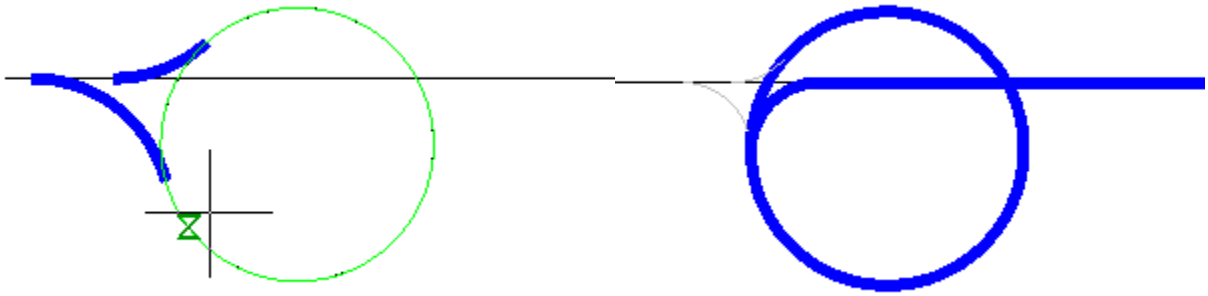
properties - Open the dialogue to change the chamfer .
new - Completion of a group of facets and the beginning of another. This command is used when you need to, for example, several facets of the same size on the same site, and then with the same dimensions - on the other:
Chamfer - Switch to build Circle. After you select the shortcut menu command opens this dialog Pairing parameter setting interface.
polyline - Switching to chamfering along the selected polyline. This option is available if you have selected the type of symmetric Circle: one length or two identical length. In operation, the cut-off modes are ignored and dimensioning.

Features of the Command

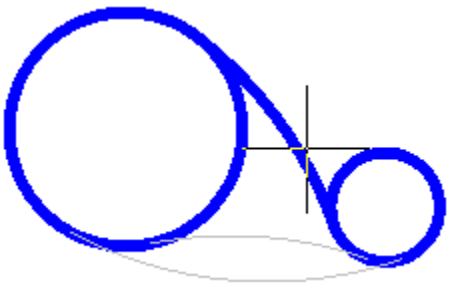
Command works in 3D. To execute the command source primitives must lie in one plane
If you select the second object, press and hold SHIFT, then the angle will be formed (at the intersection of the closure and isolation).



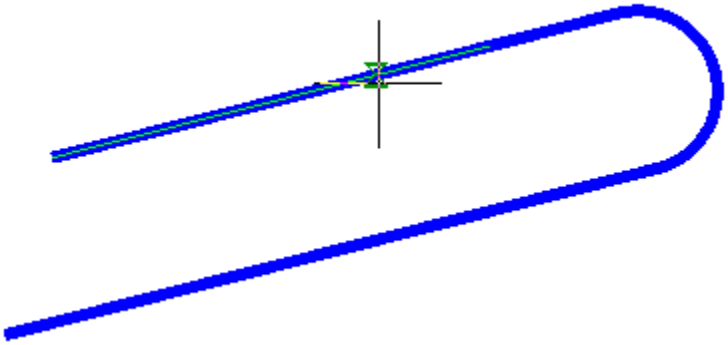
For a pair of ARC-ARC additionally available pairing, combining internal and external touch.



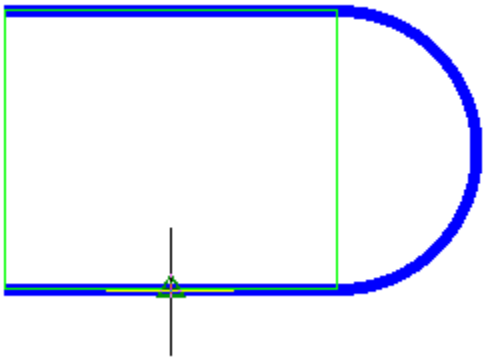
For a pair of ARC-ARC additionally available pairing, combining internal and external touch.



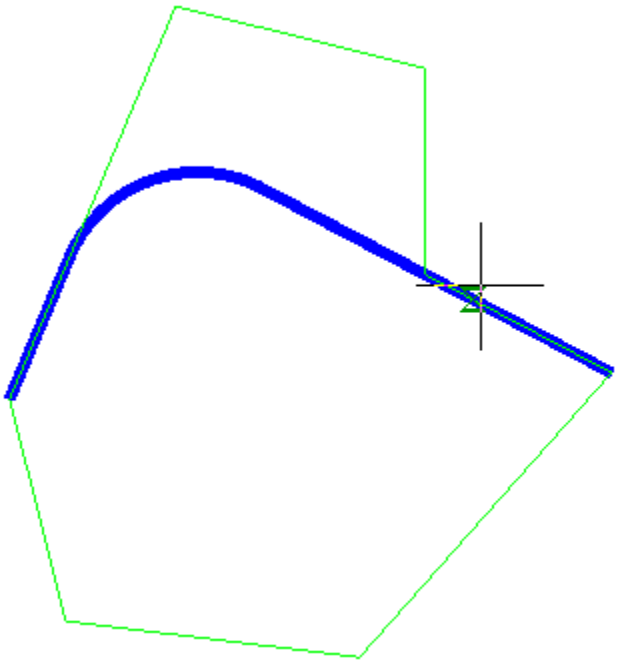
Availability of different types of interfaces depends on the fillet radius. To the outside and combining the radii need greater magnitude than the inner.
When two parallel lines will be constructed pairing radius equal to half the distance between them (regardless of the specified length). Automatically aligns the sides to the longest segment. When two parallel lines will be constructed pairing radius equal to half the distance between them (regardless of the specified length). Automatically aligns the sides to the longest segment.

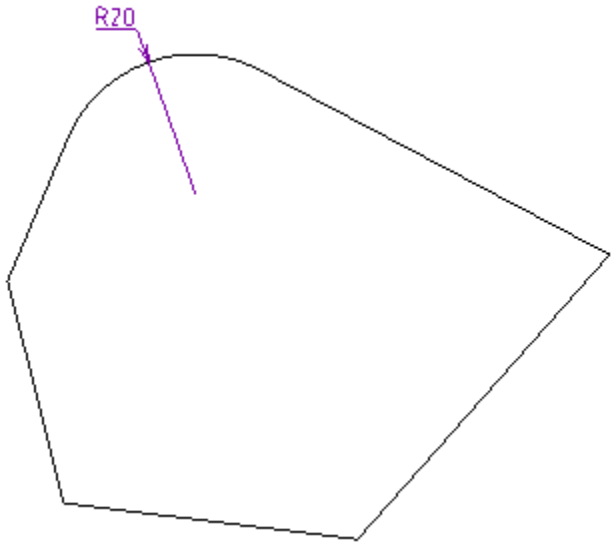


Pairing works parallel segments and regions of the same polyline.



Side interface depends on the position of the mouse cursor when the second object
When you insert a polyline retains its integrity.
If the team is formed of two polylines one, it will be object of a single polyline.
If you select one polyline segments separated by other sites, all the intermediate portions are removed.





Chamfer

Main menu: ***Mechanical - Utilities -*** ***Chamfer***.

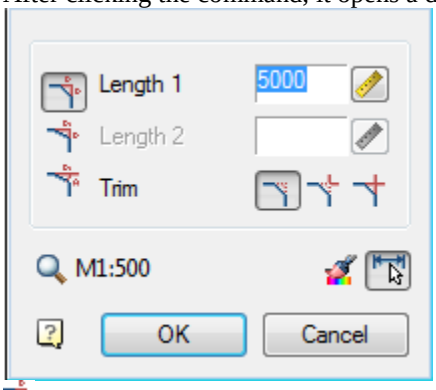
Ribbon: ***Mechanical - Utilities -*** ***Chamfer***.

Toolbar: ***Chamfer (on toolbar*** "Utilities"***)***.

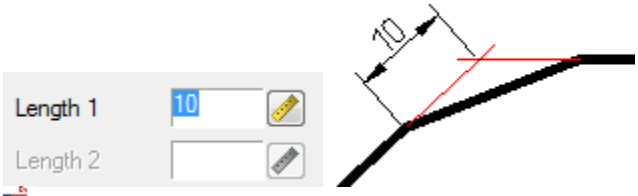
Command line: ***MCCHAMFER***.

Command Options

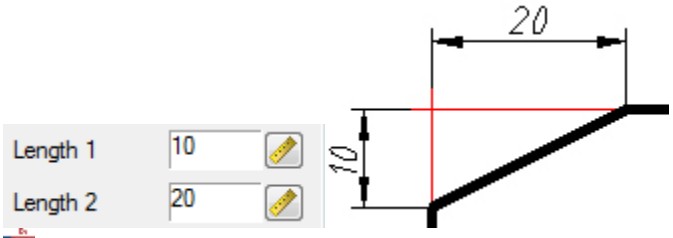
This command is used for automatic and semi-automatic placement of face on the details with the various versions and with the possibility of automatic dimensioning.
After clicking the command, it opens a dialog box settings Circle:



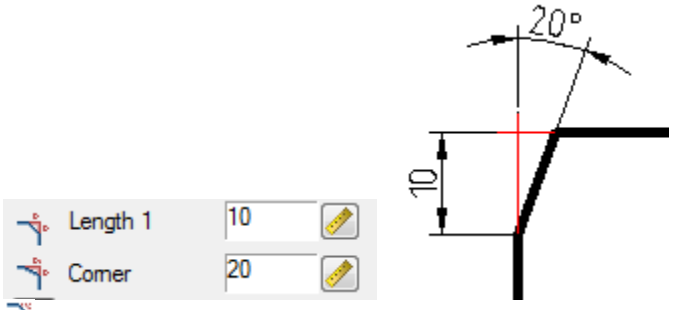
Mode *chamfering parts with two different sizes*



Mode chamfering parts with *two different sizes*



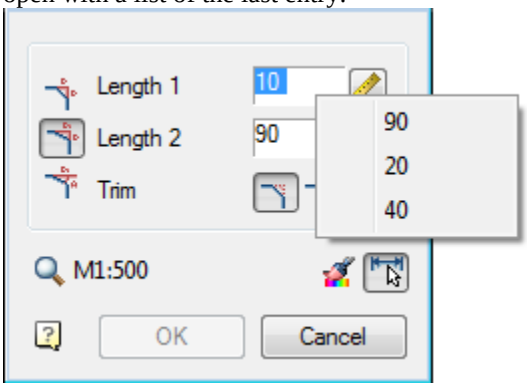
Mode *Chamfer length and angle* in this mode instead of the *length 2* displays the setting *angle.*



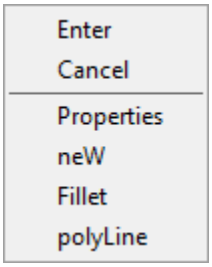
Mode *full cutting contour lines*

Mode *partial cutting lines to their intersection*

Mode *without cutting*
Measure Chamfer
Copy properties surrounding face
Pick from drawing
Double click the left mouse button or click the right mouse button in the input fields of values, context menu will open with a list of the last entry:



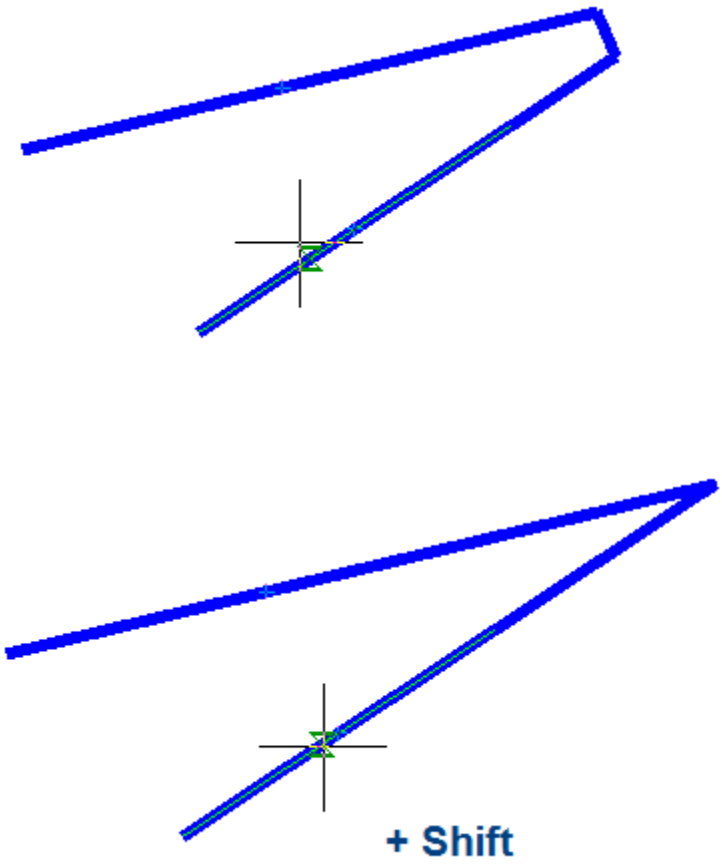
When a command from the context menu, the following options:



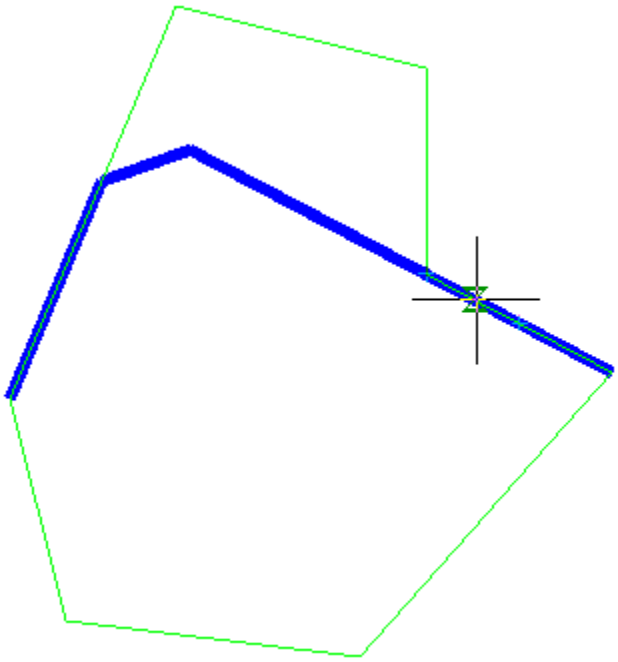
Propertise - Opening the dialogue to change the chamfer chamfer.
New - Completion of a group of facets and the beginning of another. This command is used when you need to, for example, several facets of the same size on the same site, and then with the same dimensions - on the other:
Fillet - Switch to build pairings. After you select the shortcut menu command opens this dialog ***Pairing*** parameter setting interface.
Polyline - Switching to chamfering along the selected polyline. This option is available if you have selected the type of symmetric Circle: one length or two identical length. In operation, the cut-off modes are ignored and dimensioning.

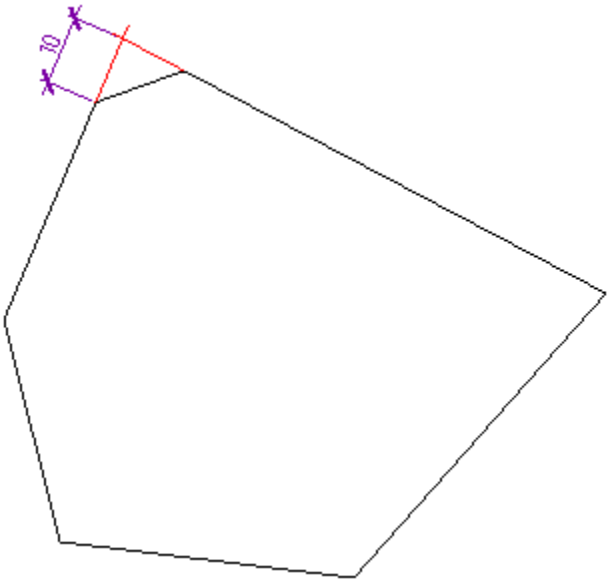
Features of the command

The command works in 3D. To execute the command source primitives must lie in one plane.
If you select the second object, press and hold SHIFT, then the angle will be formed (at the intersection of the closure and isolation).



When you insert a polyline retains its integrity.
If the team is formed of two polylines one, it will be object to a single polyline.
If you select one polyline segments separated by other sites, all the intermediate portions are removed.





Breaks

- Main menu: ***Mechanical - Utilities - Breaks -*** ***Break lines***.
- Ribbon: ***Mechanical - Utilities -*** ***Break lines***.
- Toolbar: ***Break lines (*** "MCSBreaks"***)***.

Command line: ***MCBREAK***.

Procedure

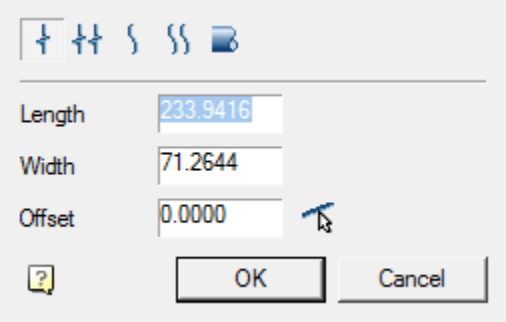
1. Select the first point to insert the break line.
2. In the context menu, select the type of break (break) (Linear, Two-lenear, Curved, tWo-curved, tuBular).
3. Pick the second break point.
4. In the context menu, select the type of cursor movement control (FreeOffset, OrthoOffset).
5. Specify the width of the cliff (break). A cliff (break) will be built.

Note:	The whole clipped geometry between the break lines is not actually deleted and can be restored either by removing the break line or editing its contour with the grips nodal points.

Note:	All objects nanoCAD Mechanica 20.1 have a so-called Z-Order. This value sets the level of the Z-coordinate in relation to other object-dependent details nanoCAD Mechanica 20.1. Editing the value of the order, you can control the mutual overlap of objects nanoCAD Mechanica 20.1.

Dialogue

The edit dialog is opened from the tooltip when you hover on a break (break) or when you double-click on a break (break).



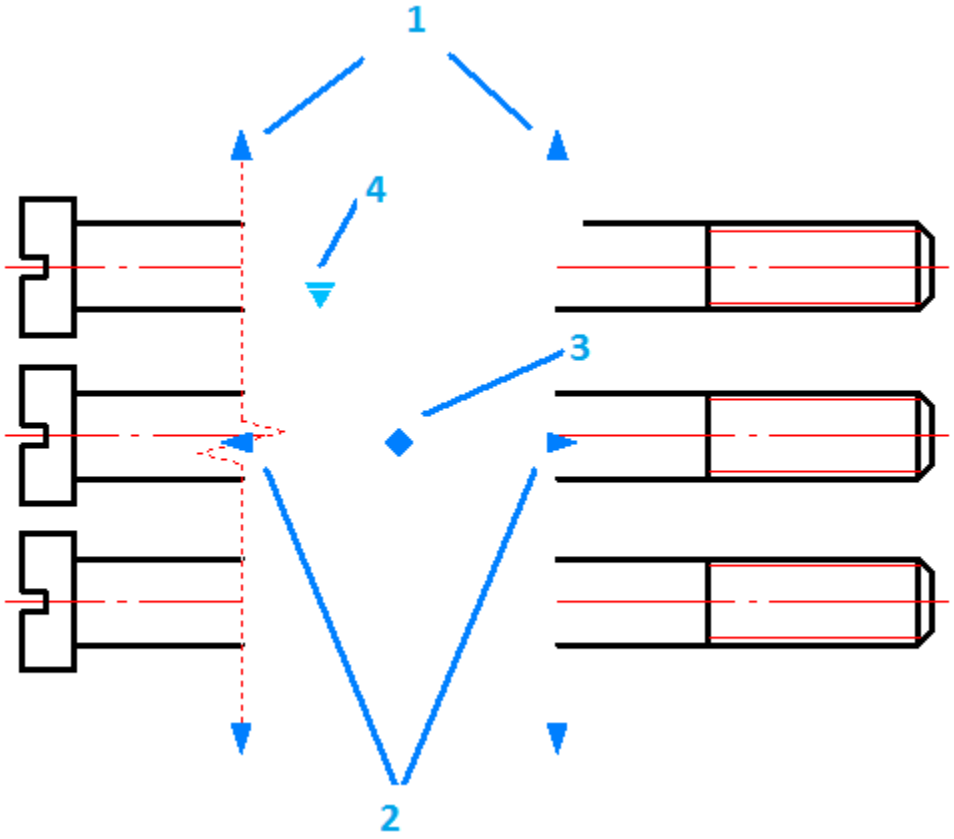
Editing dialog includes:

- Graphic break type switch;
- Field *"Length"*;
- Field *"Width"*;
- Field *"Offset"*. You can get the value using the offset command *"Parallel to specified line"*.

Grips

1. Grips length setting
2. Grips break width settings
3. Break displacement grip
4. Drop-down type grip
    - Linear
    - Two-linear
    - Curve

- Two-curve
- Tubular



Linear break

Main menu: ***Mechanical - Utilities - Breaks -*** ***Linear break***.

Ribbon: ***Mechanical - Utilities -*** ***Linear break***.

Toolbar: ***Linear break (*** "MCSBreaks"***)***.

Command line: ***MCBREAK_LINEAR***.

Two-linear break

Main menu: ***Mechanical - Utilities - Breaks -*** ***Two-linear break***.

Ribbon: ***Mechanical - Utilities -*** ***Two-linear break***.

Toolbar: ***Two-linear break (*** "MCSBreaks"***)***.

Command line: ***MCBREAK_TWOLINEAR***.

Curved break

Main menu: ***Mechanical - Utilities - Breaks - Curved break***.

Ribbon: ***Mechanical - Utilities - Curved break***.

Toolbar: ***Curved break (*** "MCSBreaks"***)***.

Command line: ***MCBREAK_CURVED***.

Two-curved break

Main menu: ***Mechanical - Utilities - Breaks - Two-curved break***.

Ribbon: ***Mechanical - Utilities - Two-curved break***.

Toolbar: ***Two-curved break (*** "MCSBreaks"***)***.

Command line: ***MCBREAK_TWOCURVED***.

Shaft break

Main menu: ***Mechanical - Utilities - Breaks - Shaft break***.

Ribbon: ***Mechanical - Utilities - Shaft break***.

Toolbar: ***Shaft break (*** "MCSBreaks"***)***.

Command line: ***MCBREAK_TUBULAR***.

Construction lines

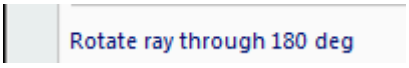
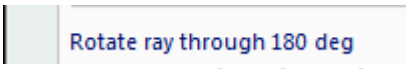
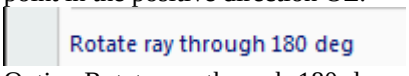
Main menu: ***Mechanical - Utilities - Construction lines***.

Ribbon: ***Mechanical - Utilities - Construction lines***.

Toolbar: "Construction lines".

Auxiliary lines are characterized by a layer in which they are placed by default (*MCSCLines*), color (red) and line type (thin solid), that are set in the settings. Additional settings - prostanovki rays or lines. Settings only apply to newly inserted objects. When you switch the existing objects remain as it is.
The table below describes which objects need to select and how formed auxiliary line in terms nanoCAD.
Some commands allow you to specify a sequence of auxiliary lines in the format XX|XX|XX, where XX - the amount of displacement of the beam (direct) in degrees or drawing units from the base position or previous beam, rendered in this command.

Command	Name	The algorithm works command		Objects of choice
		When drawing rays (*_ray*)	When drawing lines (*_xline*)	
Mcconsthor	Create a horizontal	Command is invoked *_ray*, in which the user is given the insertion point. The	Command is invoked *_xline*, in	Insertion point

	auxiliary line	second point is determined by the program to define the orientation of the beam is strictly horizontal. Point is determined by the positive direction of the X axis.  Option *R-Turn-ray-to-180-degrees* allows you to change the direction of the beam on the opposite	which the user is given the insertion point. The second point is determined by the program to define the orientation of the line is strictly horizontal.	
Mcconstver	Creating vertical reference line	Command is invoked *_ray*, in which the user is given the insertion point. The second point is determined by the program to define the orientation of the beam is strictly vertical. Point is determined by the positive direction of the axis Y.  Option *Rotate-ray-through-180-degrees* allows you to change the direction of the beam on the reverse.	Command is invoked *_xline*, in which the user is given the insertion point. The second point is determined by the program to define the orientation of the line is strictly vertical.	Insertion point
Mcconstz	Create an auxiliary line in the direction Z	Command is invoked *_ray*, in which the user is given the insertion point. Constructed beam emerging from this point in the positive direction OZ.  Option Rotate-ray-through-180-degrees allows you to change the direction of the beam on the reverse.		Insertion point
Mcconstcrs	Creating auxiliary lines intersecting	Command is invoked *_ray*, in which the user is given the insertion point. Built at the intersection of the rays of the insertion point in the positive direction of the axes OX and OY. Options allow you to select the quadrant bounded by the rays.	Command is invoked *_xline*, in which the user is given the insertion point. Constructed lines intersect at the insertion point.	Insertion point
Mcconsthb	Create an auxiliary line by two points or corner	Command is invoked *_ray*,in which the user is given the insertion point. Then to complete the command inserted rays at a predetermined angle with respect to OX (angle determined by the second point specified by the user) or a number of angles for group insert format XX | XX | XX.	Command is invoked *_xline*, in which the user is given the insertion point. Then to complete the command inserted straight with a predetermined angle with respect to OX (angle	Insertion point, a point which determines the angle (angle) or entering a number. The command completed after entering the series or after the forced shutdown

			determined by the second point specified by the user) or a number of angles for group insert format XX | XX | XX.	command (Esc or right button).
Mcconsthw	Create an auxiliary line at an angle to the segment	Command is invoked *_ray*, in which the user is given the insertion point. Specifies the position of the base angle (line). Then to complete the command inserted rays at a predetermined angle relative to the base angle or angles for a number of bulk-insert format XX | XX | XX. The angle is determined by the user of said second point.	Command is invoked *_xline*, in which the user is given the insertion point. Specifies the position of the base angle (line). Then to complete the command inserted straight with a predetermined angle relative to the base angle or angles for a number of bulk-insert format XX | XX | XX. The angle is determined by the user of said second point.	Insertion point, the point defining the base angle (or segment), which determines the point angle (angle) or entering a number. The command completed after entering the series or after the forced shutdown command (Esc or right button).
Mcconstpar	Creating an auxiliary line parallel at a predetermined distance	Command is invoked*_ray*, in which the user selects a segment, polyline, line or ray. Further, given the parallel position of the object as a point through which it must pass (in the case of a polyline - equidistant) or number of copies of an object in the format XX | XX | XX.	Command is invoked *_xline*, in which the user selects a segment, polyline, line or ray. Further, given the parallel position of the object as a point through which it must pass (in the case of a polyline - equidistant) or number of copies of an object in the format XX | XX | XX.	Beam line, polyline or line. Reference point or enter a number. The command completed after entering the series or after the forced shutdown command (Esc or right button).
Mcconstpar2	Creation of a parallel auxiliary line at half the specified	Command is invoked *_ray*, in which the user selects a segment, polyline, line or ray. Further, given the parallel position of the object as a point through which it must pass (in the case of a polyline -	Command is invoked *_xline*, in which the user selects a segment, polyline, line or	Beam line, polyline or line. Reference point or enter a number. The

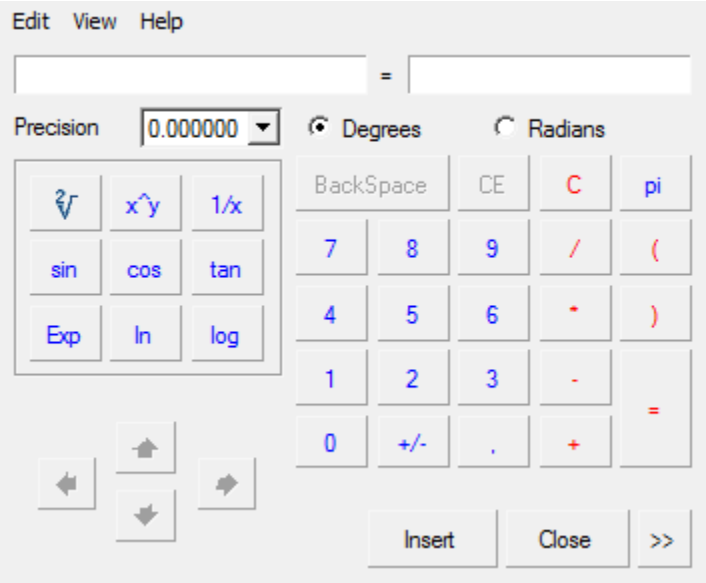
	distance	equidistant) or number of copies of an object in the format XX | XX | XX. The resulting objects are at half of the distance.	ray. Further, given the parallel position of the object as a point through which it must pass (in the case of a polyline - equidistant) or number of copies of an object in the format XX | XX | XX. The resulting objects are at half of the distance.	command completed after entering the series or after the forced shutdown command (Esc or right button).
Mcconstlot2	Establishment of a subsidiary of the perpendicular by two points	Command is *_ray*, in which the user is given the insertion point. Then, the user specifies the second point (a preview is rendered beam and segment coming from the insertion point, the second point is tied to the cursor) to specify the second point forms perpendicular to the beam. Ray is drawn perpendicular to the object.	Command is invoked *_xline*, that sets the insertion point. Then, the user specifies the second point (a preview is rendered straight and cut coming from the insertion point, the second point is tied to the cursor) to specify the second point forms perpendicular to the line. Direct drawn perpendicular to the object.	Insertion point, reference point on the perpendicular to the object renders.
Mcconstlot	Create an auxiliary line perpendicular to the segment	Command is invoked *_ray*, for which the user selected reference object and point to it. Ray is drawn perpendicular to this object at the selected point. Starting beam is at a selected point.	Command is invoked *_xline*, , in which the user selects the object and reference point on it. Direct drawn perpendicular to this object at the selected point. Middle line is the selected point.	Beam line, polyline or line. Point on the object.
Mcconsthm	Create an auxiliary line to the corner along the	Command is invoked _ray, for which the user selects two reference object. Ray is based on the bisector between the two reference objects.	Command is invoked _xline,in which the user selects two	Beam line, polyline or line.

	bisector		reference object. Direct built along the bisector between the two reference objects.	
Mcconsttan	Create two parallel auxiliary lines tangent to the circle	Selected circle or arc that it may determine. Constructed two rays tangent to the circle, it is necessary to ask a series of positioning angles or point specified segment emanating from the center of the circle and parallel rays (link).	Selected circle or arc that it may determine. Constructed two lines tangent to the circle, it is necessary to ask a series of positioning angles or point specified segment emanating from the center of the circle and parallel lines (link).	Circle or arc.
Mcconstcc	Creating auxiliary circle another circle concentric circle or arc is selected.	Automatically construct a circle concentric with the selected object and specify the diameter (or the given number of diameters). The command completed after entering the series or after the forced shutdown command (Esc or right button).		Circle or arc.
Mcconstc2	Creating auxiliary circle tangent to the segment	Selected reference line object, the user specifies a point lying on the circle defined by the radius of the circle (the system offers the minimum possible radius which is equal to half the distance along the normal from the selected point to the segment, and controls the maintenance of value (greater or equal)). We construct a circle at two points of tangency (one point is given conditional perpendicular) and radius..		Beam line, polyline or line. Point belonging to the circle.
Mcconstkr	Creating auxiliary circle tangent to two segments	Selected two reference linear object and specify the diameter. We construct a circle at two points of tangency and radius.		Beam line, polyline or line.
Mcconstkr	Creating auxiliary circle tangent to three segments	Selected two reference linear object. We construct a circle at three points of tangency.The radius is calculated automatically.		Beam line, polyline or line.
Mcconstcircli	Creating auxiliary square circumscribing circle	Selected circle or arc, automatically builds a square, a circle is described. The context menu provides switching between inscribed and described by the square.		Circle arc.
Mcconsttc	Create an auxiliary line	Selected consecutively two arcs or circles. Constructed beam tangent to the two	Selected consecutively two	Two arcs or circles

	tangent to two circles	circles, a definite choice (touch point - coming to a place of choice, the beginning of the beam - the first object).	arcs or circles. Constructed line tangent to the two circles, a definite choice (touch point - coming to a place of choice, the center line - the first object).	
Mcconst_circle	Creating auxiliary circle	This is the counterpart nanoCAD assigning properties auxiliary geometry.		
Mcconstxray	Creating a subsidiary of rays emanating from a single point	This is the counterpart nanoCAD assigning properties auxiliary geometry.		
Mcconstxline	Creating auxiliary lines passing through one point	This is the counterpart nanoCAD assigning properties auxiliary geometry.		
Mcconstlines	Auxiliary lines	This command displays the settings nanoCAD Mechanica 20.1 with a pointer to the section *Auxiliary lines.*		
Mcclineo	Enable / disable the auxiliary lines	This command automatically hides all auxiliary lines.		
Mcconsterase	Remove auxiliary lines	This command automatically removes all auxiliary lines.		

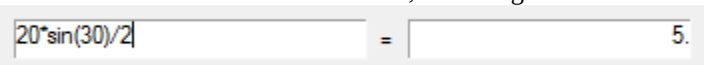
Calculator

The calculator is designed for complex mathematical calculations. The capabilities of the table and text editor are significantly expanded using the powerful functionality of the calculator.
The calculator is available in the table editor, text editor, when using the *"Inquiry"* utility in the *"Value picker"* dialog.
The *"Calculator"* dialog consists of a main menu and a set of grouped commands.



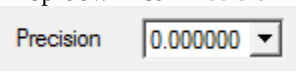
Main menu

- Edit
  - Copy Ctrl+C - the command copies data from the input field.
  - Paste Ctrl+V - the command pastes the copied data into the input field.
- View
  - NoteBook - command opens the notebook panel.
  - Degrees - the command switches the mode of calculating angles in degrees.
  - Radians - the command switches the mode of calculating angles in radians.
  - Expression - the command turns on the formula calculation mode. The input field is divided into two: on the left is the formula, on the right is the result.

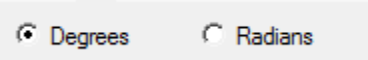


Commands

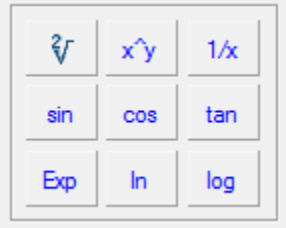
Drop-down list *"Precision"* - sets the rounding digit of the calculation result.



Switch *"Degrees - Radians"* - switches the angle calculation mode.



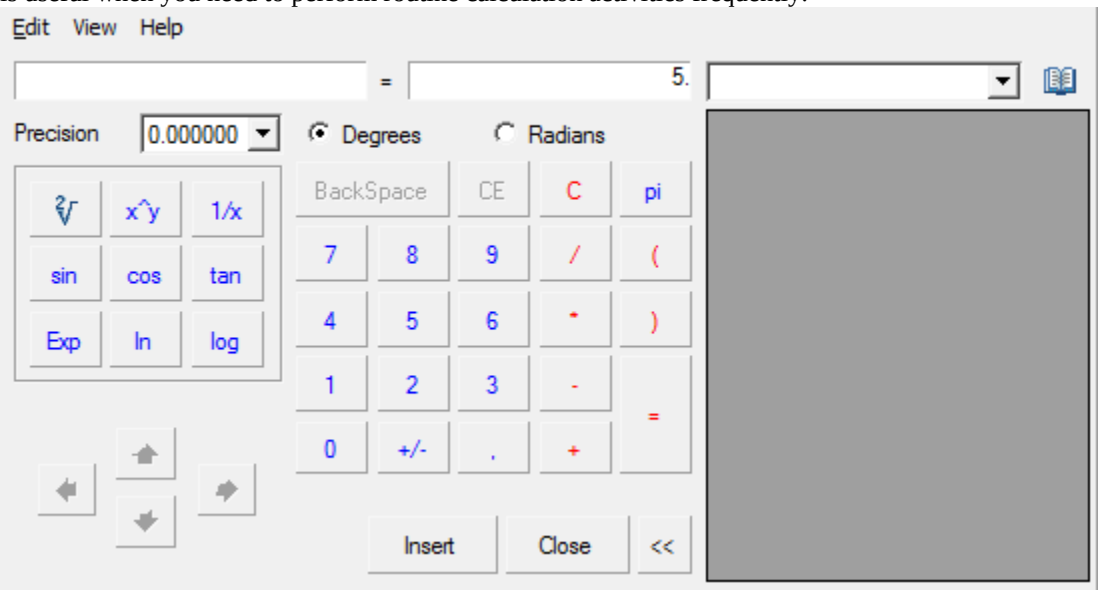
Group of buttons *"Mathematical operations"*.



The main group of calculator buttons. The composition of the buttons is similar to all standard calculators.

Buttons for navigating table cells. Available when editing a table cell. It also shows the name of the currently edited cell.

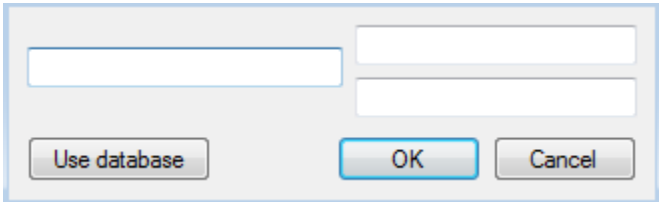
Button to open the "Notebook" sidebar. It is possible to connect the spreadsheet editor of the "Notebook" tool. The additional sidebar lists tables from the notebook in a drop-down list. After selecting a table, its composition will be displayed. Double-clicking on a value from the table adds this value to the input field. This tool is useful when you need to perform routine calculation activities frequently.



The *"Insert"* button is designed to insert the result of calculations from the calculator into an editable cell, a text entry field or the *"Value picker"* dialog.
The *"Close"* button closes the calculator dialog without transferring the value.

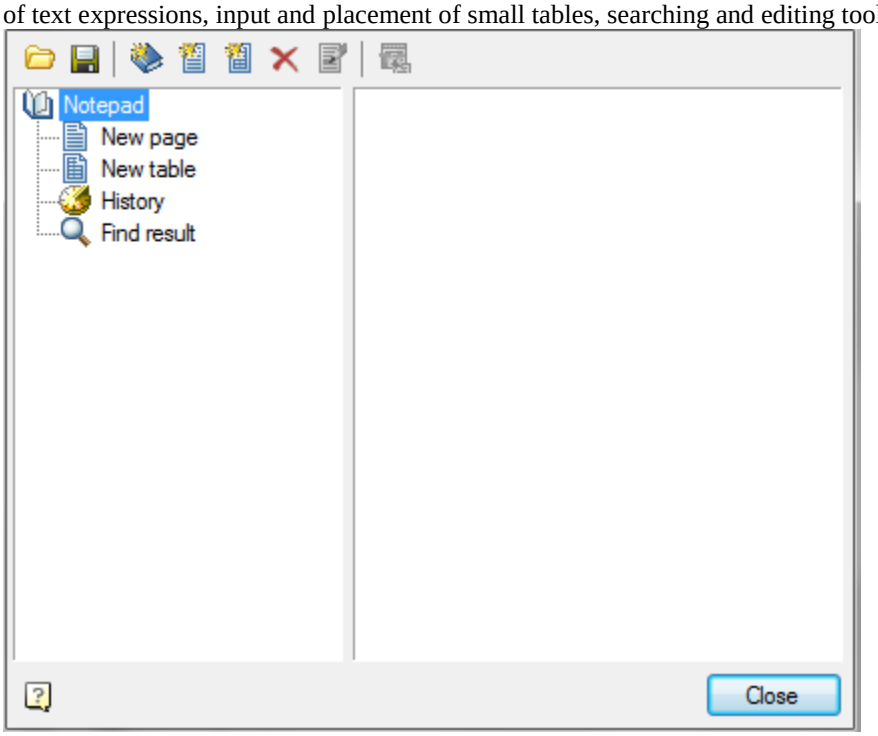
Insert material designation to the table

Button *material designation.*
This tool is designed to be inserted into the table, the technical requirements and specifications of the formatted string records on material and product mix. By pressing the button, a dialog box appears called material designation.



Notebook

Button *Notebook*
To enter recurring standard text expressions (abbreviations, symbols, etc.) used tool Notebook. This is a versatile tool for some editing tools nanoCAD Mechanica20.1, which is necessary to enter text. Notebook combines the functions of: storing a set of expressions and the structure of their placement in a separate file, input and placement of text expressions, input and placement of small tables, searching and editing tools introduced earlier records.



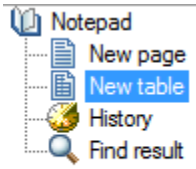
Dialog *Notebook* divided into two parts. On the left side there is a tree structure of the notebook where the user selects the ***sections*** and ***pages*** notebook. In the upper part of the window control buttons notebook.

Load from file. Opens the notebook file stored on disk.
Save. Saves the changes to the file in your notebook.

Important!	Recommended that you save a new notebook in a separate file in the first call to the tool Notebook using the button ***Save***.

To search for a text string in a notebook has a special search tool, Right-click on any section or page of the section and in the context menu, call the command ***Find***. Found links are placed in the section *results*
To accelerate the search procedure provided for viewing the latest expressions used in section *history*.
Management structure for the notebook with the buttons:
Add a partition

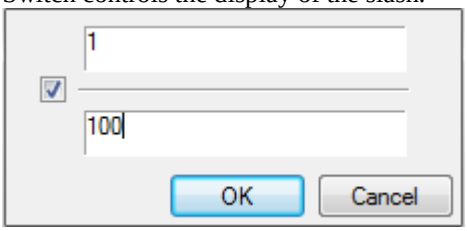
Add this page
Add a table
Delete tree item
In the notebook is allowed to create an unlimited number of sections and subsections, pages, and tables.



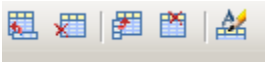
Moving sections, pages, and tables for structural wood can produce a drag and drop (drag and drop).
Rename the selected section or page using the command Rename the shortcut menu.
Select / Copy the selected text. Transmits the selected piece of text pages into editable table cell. If the tree structure of the selected page or table in your notebook, the contents displayed on the right side of the dialog. Highlight the desired entry cursor, and turn the Select / Copy the selected text.
Edit Page. This button is used to activate the edit pages and style notebook. It becomes active only if the tree structure is selected page element or table. In edit mode, the tree structure becomes inactive, and the right side activates the Edit toolbar.

Editing tools page

Clear. Clears the current page
Load from file. Allows you to import an existing text file *.RTF MS Word.
Save the file. Allows you to record the content into a file *.RTF for use in MS Word.
Add item.
Remove the item .
Move up.
Move down.
Superscript. Includes character mode superscript.
Subscript. Includes character mode subscript.
Insert fraction. dialog box set the numerator and denominator.
Switch controls the display of the slash.



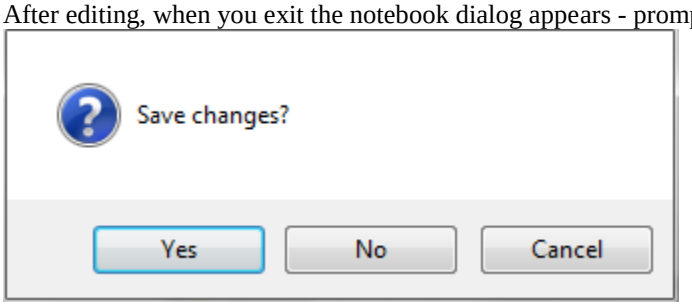
Insert tolerance. Opens ***Tolerances.***
Insert material. Opens material.
Calculator.



Tools for editing table

Insert Row. *Delete Row*. *Insert Column.* *Delete Column.* ***Rename column.*** Opens Properties column in which you specify the width and title of the selected column.

After editing, when you exit the notebook dialog appears - prompted to save the changes in a notebook.

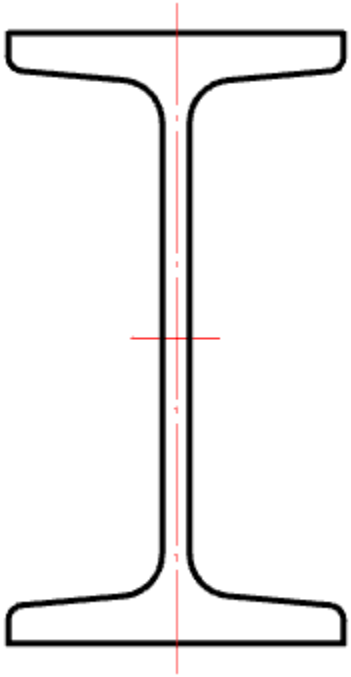


Magnet

Magnet used to draw complex paths based on existing graphics. The contour can be constructed by points or by sequential indication of adjacent segments or arcs.
Called by:

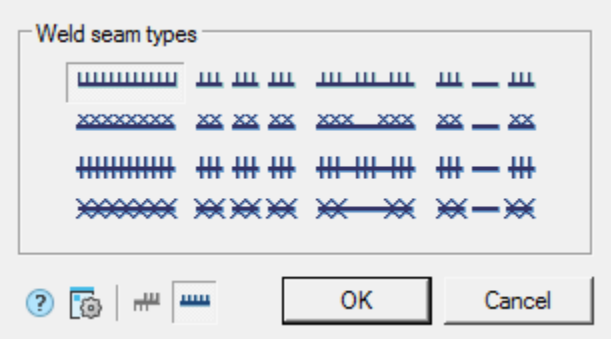
- from the context menu (*Weld seam*, *Simplified weld joint*)
- button *"Select contour with the magnet"* in the dialog (*Area array*).

An example of constructing a weld seam contour, based on an I-beam image inserted into the drawing

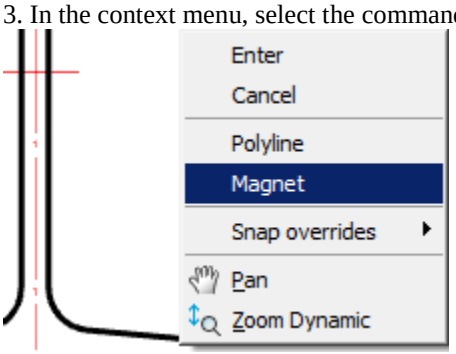


1. We call the command *"Weld seam"*. A dialogue will open *"Weld seam"*.

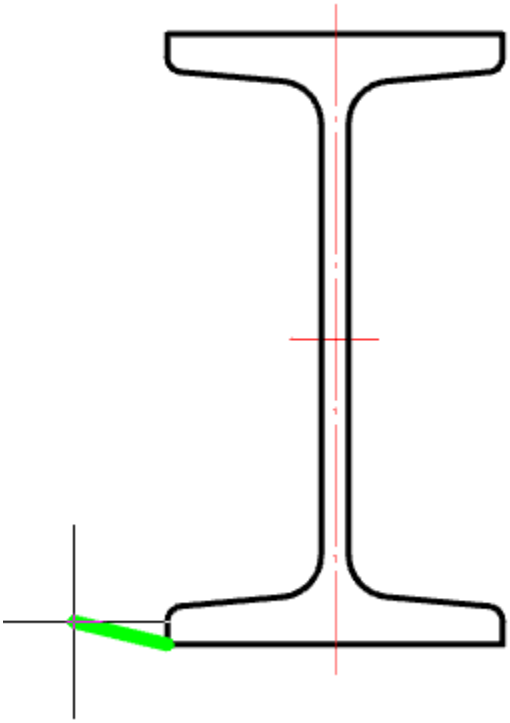
2. Select the type of seam and click *"OK"*.



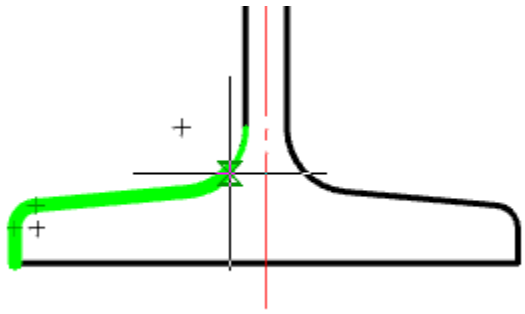
3. In the context menu, select the command *"Magnet"*.



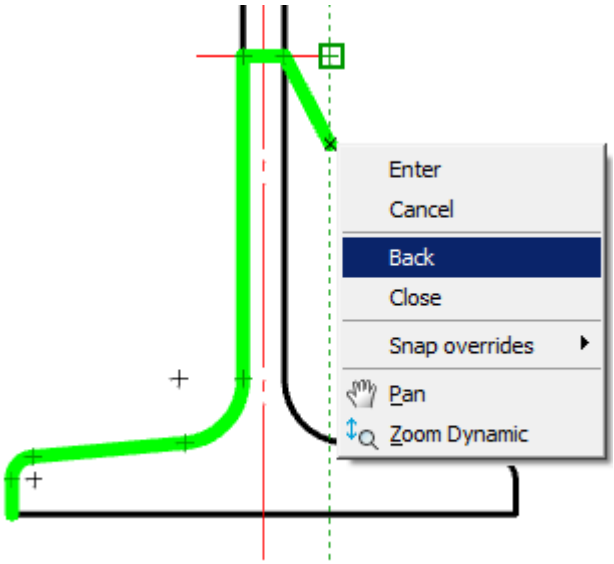
4. We indicate the starting point of the contour.



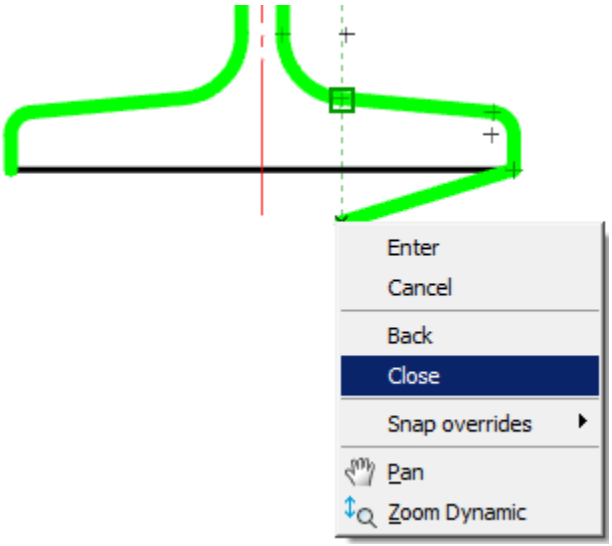
5. Sequentially move the cursor over the end points of the linear and arc sections of the I-beam contour.



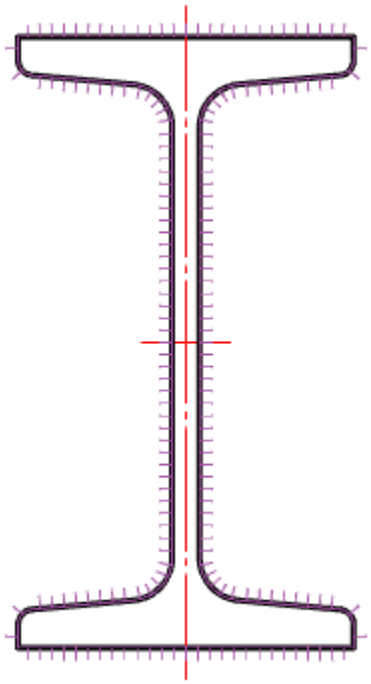
Deselection of the previous segment is made by re-pointing the cursor to the starting point of this segment or by the "Back" command from the context menu.



6. Use the "Close" command from the context menu to complete the construction of the closed contour.



7. The weld will be built along the specified contour.



Editing

Editing

Main menu: ***Mechanical - Edit -*** ***Edit...***.

Ribbon: ***Mechanical - Utilities -*** ***Edit...***.

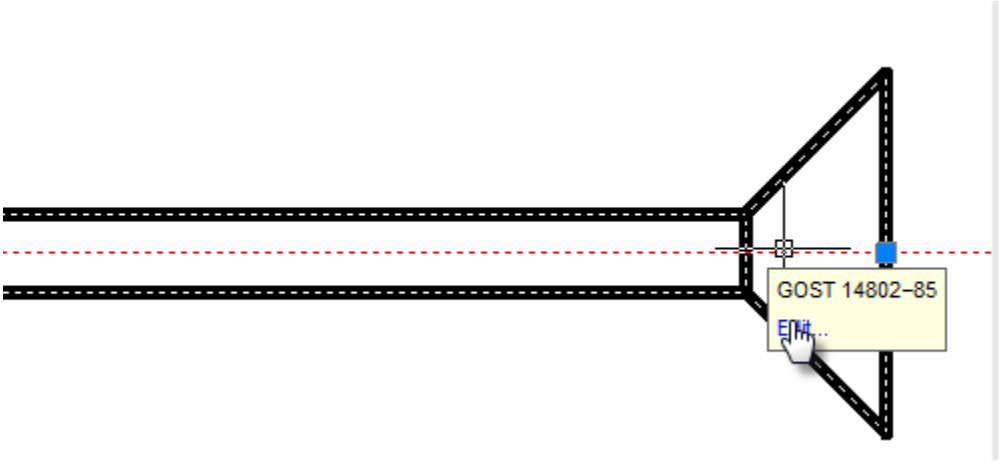
Toolbar: ***Edit... (*** "Edit"***)***.

Context menu: ***Edit (on select object)***.

Command line: ***MCEDIT, EDIT***.

Editing Building Ground20.1 - objects can be made in the following ways:

- right-clicking and selecting *Edit* Edit from the context menu;
- double-clicking the left mouse button on the nanoCAD Mechanica20.1 - object. This method is specified in the settings;
- with "grips";
- calling on the Edit tooltip that appears when you hover the cursor over an object:



- properties dialog box (properties);
- standard commands Erase (ERASE), Move (MOVE), Copy (COPY), Rotate (ROTATE), Stretch (STRETCH).

Edit in-place

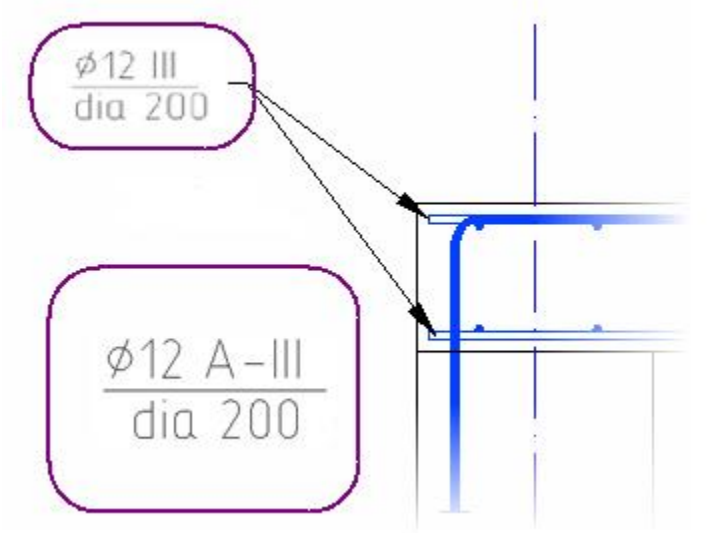
Main menu: ***Mechanical - Edit -*** ***In-place edit***.

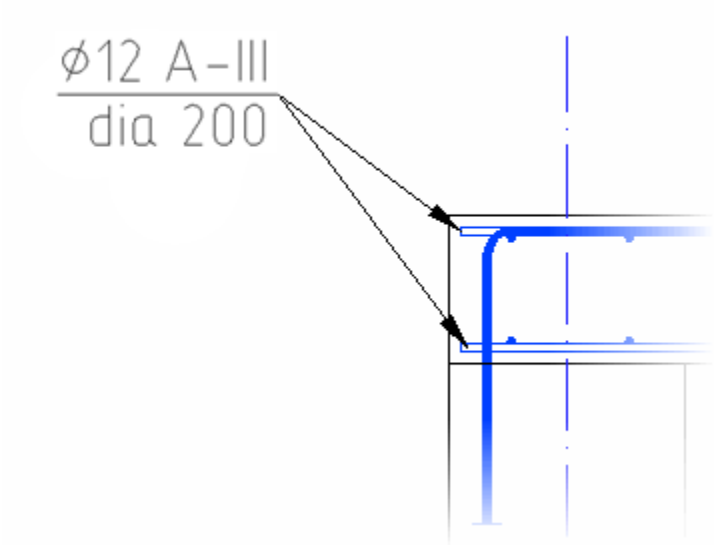
Ribbon: ***Mechanical - Utilities -*** ***In-place edit***.

Toolbar: ***In-place edit (*** "Edit"***)***.

Command line: ***MCIPEDIT, IPEDIT***.

Command allows you to edit the drawing objects Construction site containing text.
Editing can be done on site by clicking the left mouse button while pressing Ctrl.





Group Edit

Main menu: ***Mechanical - Edit -*** ***Group edit***.

Ribbon: ***Mechanical - Utilities -*** ***Group edit***.

Toolbar: ***Group edit (*** "Edit"***)***.

Command line: ***MCEDITGROUPS***.

The command allows you to edit bolted or riveted connections.

- Click on the button *Group edit*;
- Select any item in the drawing either a bolted or riveted joint;

In the window that opens, make the necessary changes.

Furthermore, editing group nanoCAD Mechanica20.1 - objects can be produced by double-clicking the left mouse button on the axis of the assembly.

Regenerate

Main menu: ***Mechanical - Edit -*** ***Regenerate***.

Ribbon: ***Mechanical - Utilities -*** ***Regenerate***.

Toolbar: ***Regenerate (*** "Edit"***)***.

Command line: ***MCREGEN, REGEN***.

- Updates the object's nanoCAD Mechanica
- Recalculates the table
- Updates overlapping objects
- Automatically updates the calculated values of the attributes

Explode all

Main menu: ***Mechanical - Edit -*** ***Explode all objects***.

Ribbon: ***Mechanical - Utilities -*** ***Explode all objects***.

Toolbar: *Explode all objects* ***(*** "Edit"***)***.

Command line: ***MCEXPLODEALL, EXPLODEPDEALL***.

Explodes all objects of nanoCAD Mechanica into drawing primitives.

Explode primitives

Main menu: ***Mechanical - Edit -*** ***Explode All Primitives***.

Ribbon: ***Mechanical - Utilities -*** ***Explode All Primitives***.

Toolbar: Edit - ***Explode All Primitives***.

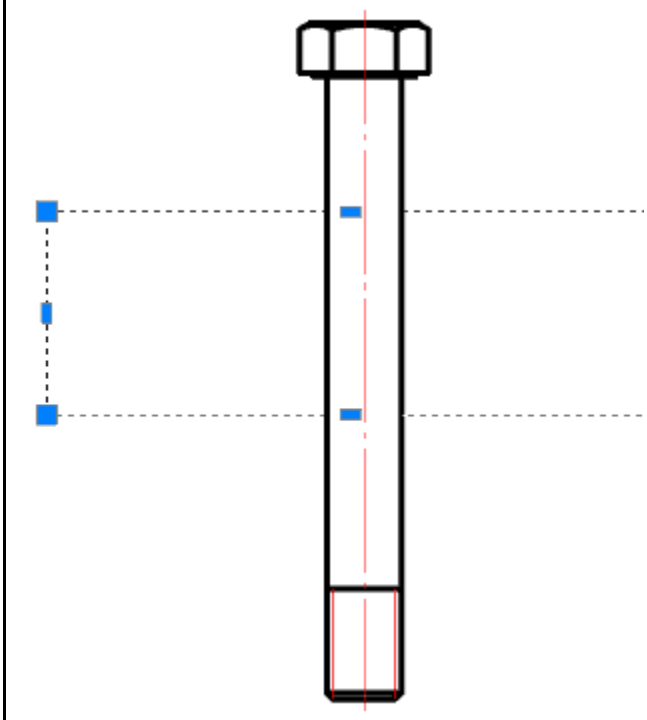
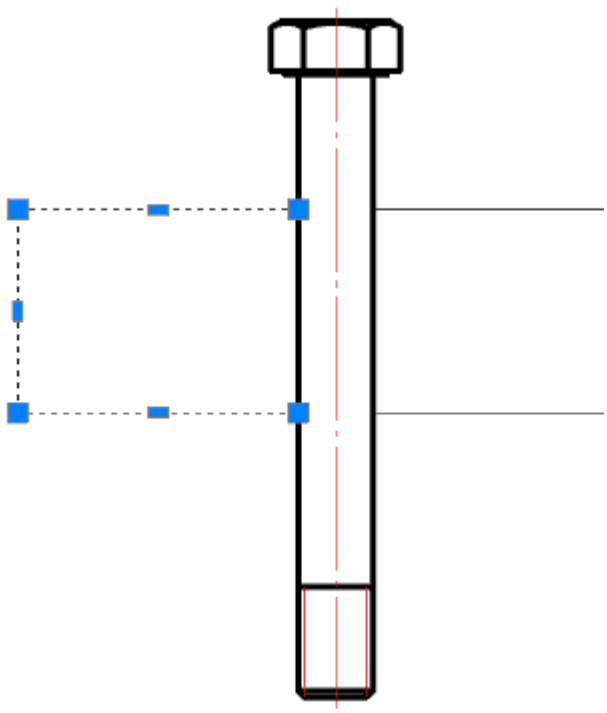
Command line: ***MCEXPLODEPSEUDO, EXPLODEPSEUDO***.

Explodes Primitives and Pseudo-Primitives that are generated when nanoCAD graphics are overlapped by objects nanoCAD Mechanica.

Procedure

1. Call the command.
2. Confirm the layout of the primitives.
3. The primitives will be broken.

Before	After

Before	After
	

Export drawing file

Main menu: ***Mechanical - Edit -*** ***Export to file***.

Ribbon: ***Mechanical - Utilities -*** ***Export to file***.

Toolbar: ***Export to file (*** "Edit"***)***.

Command line: ***MCEXPORTTOFILE***.

Saves selected objects in the drawing file.

1. Click *Export to file*
2. Select the drawing objects
3. In the dialog box, *type* the required settings drawing
4. Click *OK*
5. In the dialog box, specify a name for the drawing, and then click *Save*

Parameters Redefinitions

Main menu: ***Mechanical - Edit -*** ***Parameters redefinition***.

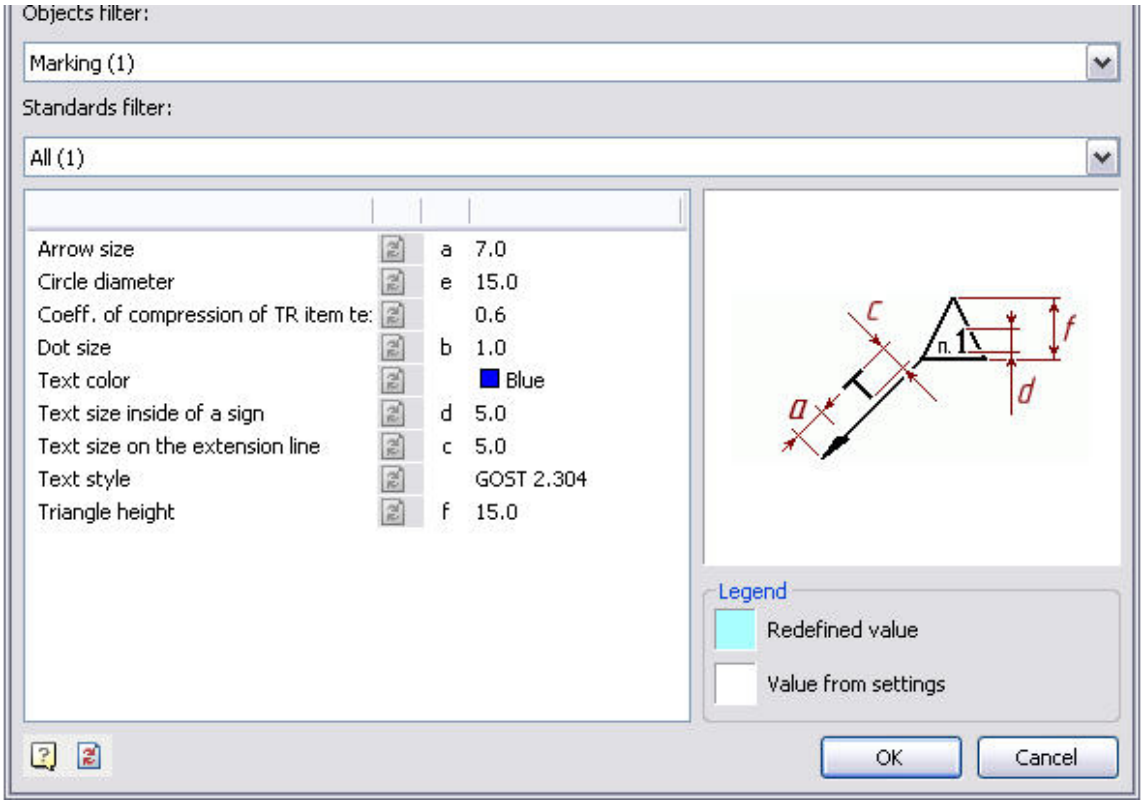
Ribbon: ***Mechanical - Utilities -*** ***Parameters redefinition***.

Toolbar: ***Parameters redefinition (*** "Edit"***)***.

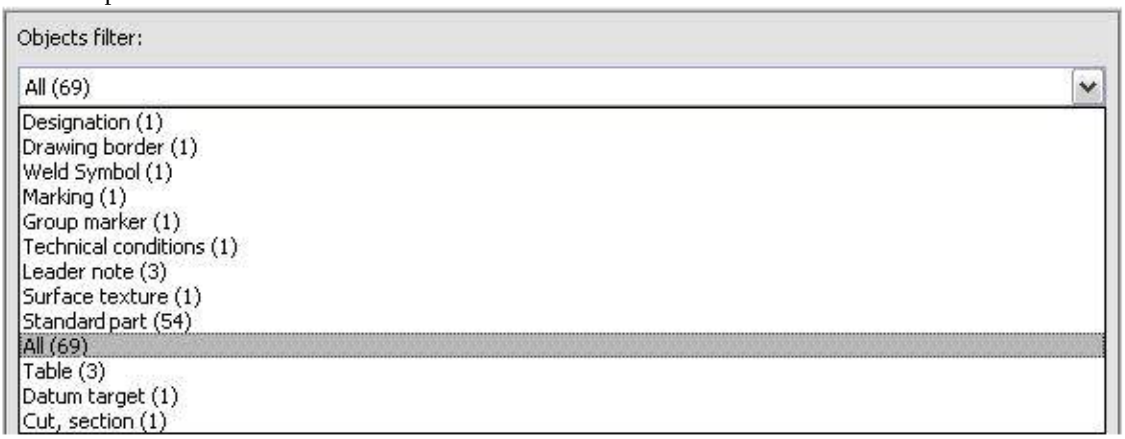
Command line: ***MCPR, PR***.

The basic rules

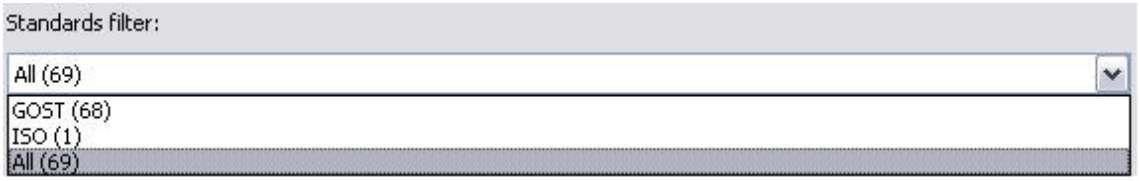
This command is used to define the variations of the wiseMechanical design objects settings in the current drawing.nanoCAD Mechanica20.1 The redefinition is controlled by the parameters set in the setting of design elements.



The objects are grouped by types in the dialog. The objects parameters are grouped by name.
To set the parameters or Redefinition removal filter lists are:



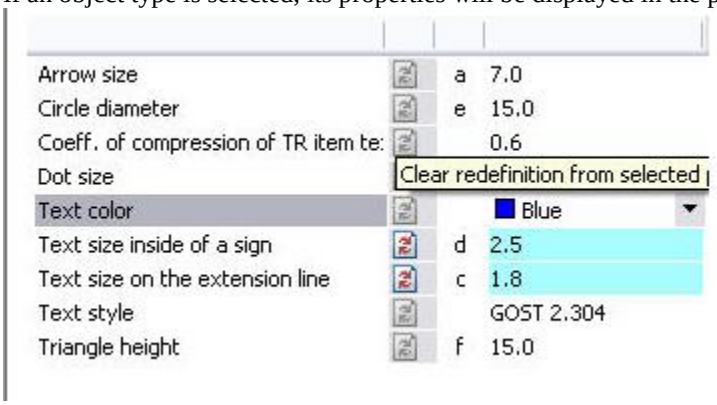
If any type is selected in the parameters table , the parameters list concerning the given type is displayed.
The standard filter list allows specifying different settings for single objects created by a different design standard.



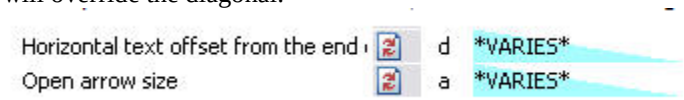
If you select "All" standards - the changes will be applied to all objects if you select only one of the standards (eg GOST), the changes will only be used for objects belonging to the chosen standard.

Command Options

If an object type is selected, its properties will be displayed in the parameters table.



Redefined parameters are shown in blue. To remove the overriding need to press the opposite setting.
In addition, you can manually specify a new value for the selected parameter. If the new value is the same default, the blue light will not be displayed. If the selected object group only some entries have the override, the backlight will override the diagonal.



To remove the override all objects need to click on the bottom left of the dialog.

Action sequence

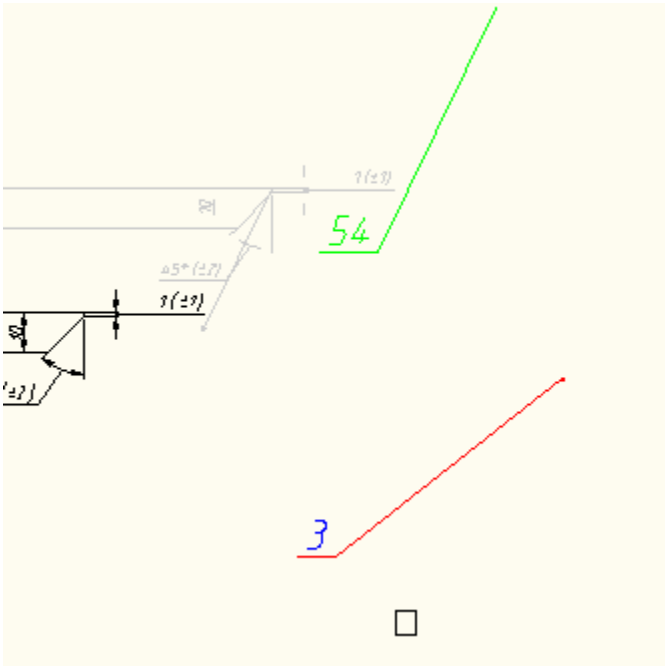
To change the settings of one or more design elements, you must:

- Select objects
- Choose the parameters redefinition command from the content menu.

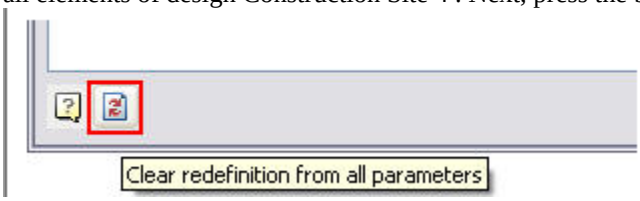
Next, in the dialog box to specify the new settings data display objects. Changes apply only to the selected objects.
Change the parameters of objects can also be by a separate command ***Parameters redefinitions***.

Checking the settings of design elements

To verify the design elements specify the settings, you must run the ***Parameters redefinition***. It is proposed to select the objects that will be exposed to the command.



When you press the Enter will select all elements of design Construction Site 4. When you press the Enter will select all elements of design Construction Site 4 . Next, press the button ***Clear redefinition from all parameters***



The result is a document issued in accordance with the settings specified in the enterprise (in terms of elements nanoCAD Mechanica20.1).

Find and Replace

Main menu: ***Edit -*** ***Find and Replace...***.

Main menu: ***Mechanical - Edit -*** ***Find and Replace...***.

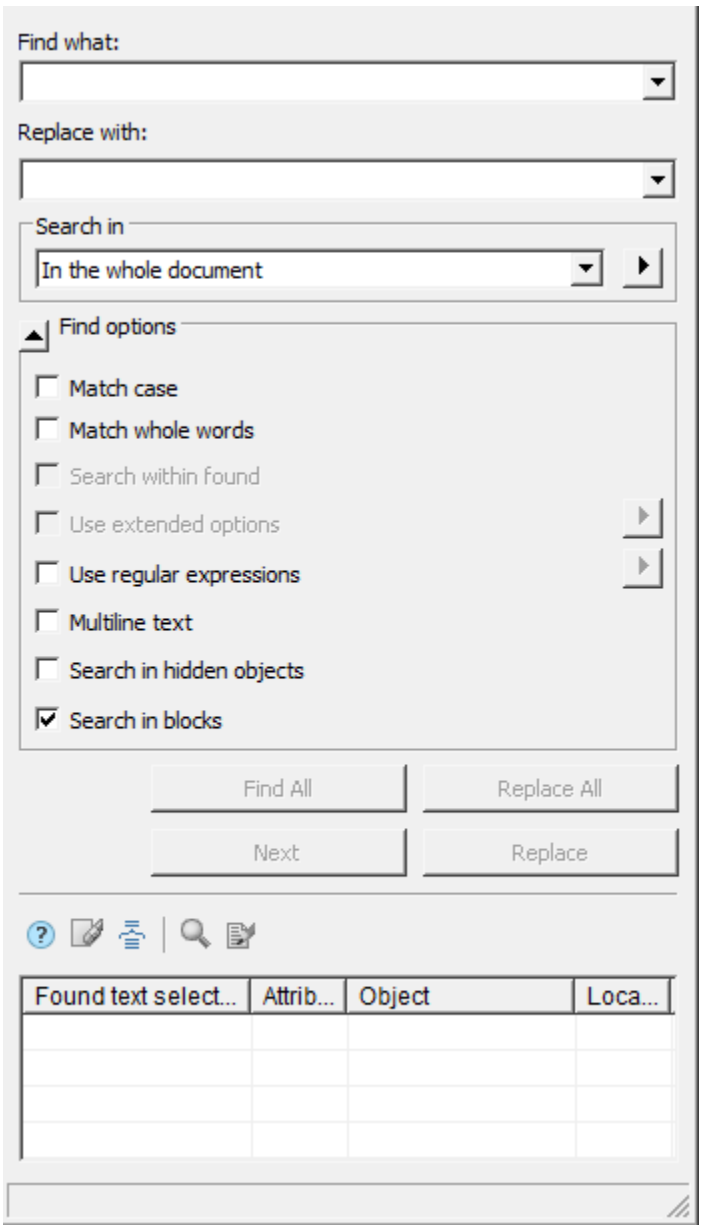
Ribbon: ***Mechanical - Utilities -*** ***Find and Replace...***.

Toolbar: ***Edit -*** ***Find and Replace...***.

Command line: ***MCFIND, FIND***.

Shortcut: ***Ctrl+F***.

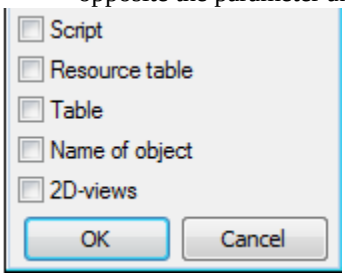
The *"Find and Replace"* tool is used to find and replace string values in nanoCAD Mechanica 20.1 objects in a drawing or in an object base.



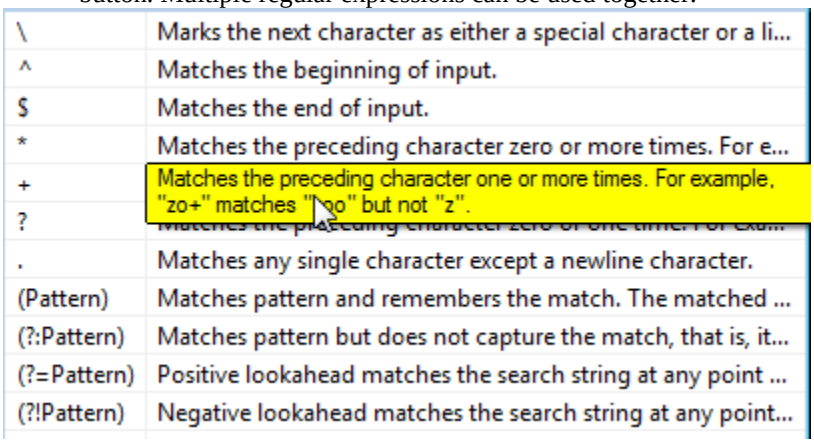
Search

1. Call the command. A dialogue will open *"Find and Replace"*.
2. In the *"Find what:"* field, enter the search phrase.
3. Select *"Search in"* from the list, or by using the additional button that defines the search area. The button action depends on the context of the *"Find and Replace"* command call. If the command is called from the object manager, the *"Quick selection"* window opens.
4. Select search parameters :
   - *Match case*. If the flag is set, the search is case-sensitive.
   - *Match whole words*. If the flag is set, the search is performed for individual words, that is, combinations of characters separated by punctuation marks or spaces.

- *Search within found*. The switch is available after the string has been searched and limits the search range.
- *Use extended options*. The switch is available when searching in the object database. Click the button opposite the parameter and in the window that appears, select the advanced search areas.



- *Use regular expressions*. Enables search mode with additional selection conditions. Click the button opposite the parameter and select the required expression from the list that appears. When you hover over a list item, a tooltip appears with a detailed explanation. The choice of expression is made with the left mouse button. Multiple regular expressions can be used together.



- *Multiline text*. Turns on the search mode for multi-line text.
- *Search in hidden objects*. Turns on the mode when both visible and hidden objects are searched.
- *Search in blocks*. Turns on the mode when the search is carried out inside the blocks.

5. Click the *"Find all"* button, the search process will start.
6. The search result will be displayed in the table.

Search result

The search result is displayed in the table. Data is sorted by mouse click in the column header. The *"Next"* button allows you to move the active row in the table.

Found text chunk	Attribute	Object
electrode, additional wire	String 1	Leader note
1. electrode, additional wire	Text1	Technical conditions

The table has additional commands:

- *Clear all* - clears the search result list.
- *Group results by objects* - displays results for objects.
- *Show object* - shows the object corresponding to the selected row in the drawing. The object is shown even if it is on an inactive sheet.
- *Edit object* - opens a dialog box for the object corresponding to the selected row.

Replace

1. Search for objects to replace.
2. In the *"Replace with"* input field, enter the text string to replace the found lines with.
3. Click the *"Replace"* or *"Replace All"* button:
   - The *"Replace All"* button replaces all found fragments with a new value.
   - The *"Replace"* button starts the process of successive replacement of the found fragments. To replace all found fragments, press the *"Replace"* button each time, while the active line will move through the table below. The *"Next"* button allows you to move the active row through the table, skipping the next found fragment in the list without replacing it with a new value.

Quick selection

Main menu: ***Mechanical - Edit -*** ***Quick selection***.

Ribbon: ***Mechanical - Utilities -*** ***Quick selection***.

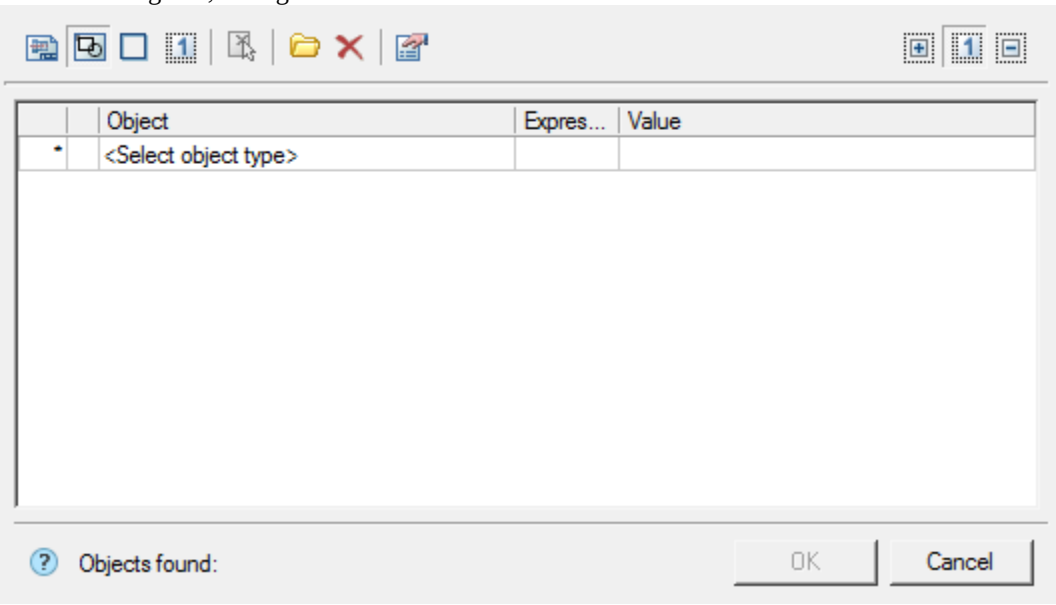
Toolbar: ***Edit -*** ***Quick selection***.

Command line: ***MCQS, QS, QSELECT***.

The *"Quick selection"* tool is used to select the nanoCAD Mechanica 20.1 objects inserted into the drawing according to certain conditions.

Procedure

1. Call the command.
2. In the dialog box, configure the search conditions.



3. Confirm the setting with the *"OK"* button. The objects in the drawing are selected according to the search conditions.

Dialog

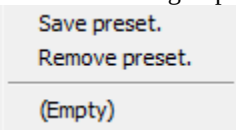
The dialog box consists of a toolbar and a list of search conditions. At the bottom left of the dialog, the number of objects found according to the specified search conditions is displayed.

Toolbar

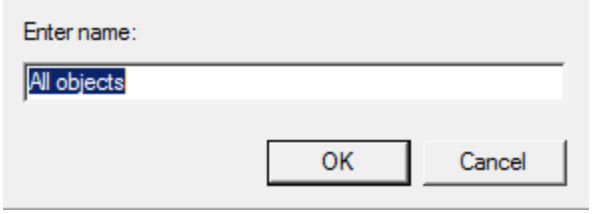
- *Selecting an object search area*. The choice of the search area is carried out using the modes:
  - *In the whole document* - selection of objects is carried out in the column of the list of conditions *"Object"*, the list of objects is taken from the entire drawing;
  - *In the current space* - selection of objects is carried out in the column of the list of conditions *"Object"*, the list of objects is taken from the current sheet;
  - *In bounding box* - objects are selected by specifying a rectangular area in the drawing, to redefine the area, use the command *Set selection*;
  - *In current selection* - objects are selected by specifying a set of objects in the drawing, to redefine a set of objects, use the command *Set selection*;
- *Selecting a method for adding found objects*. The choice of the method of adding is carried out using the modes:
  - *Set result to new selection* - objects matching the search conditions will be selected in the drawing, and all other objects will be deselected;
  - *Add result to selection* - the found objects will be added to the objects already selected in the drawing;
  - *Subtruct result from selection* - found objects will be deselected if they were selected.
- *QS Params*. Opens a dropdown menu:

Include Frozen and Invisible layers - the switch determines whether to search for objects on frozen and hidden layers. Does not search by default.

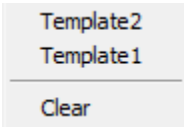
- *Set selection*. The command allows you to override a set of objects or a rectangular area. Available for *"In bounding box"* or *"In current selection"* modes.
- *Presets*. Opens a dropdown menu. Using the set of tools included in this command, you can select objects according to previously established and saved conditions:



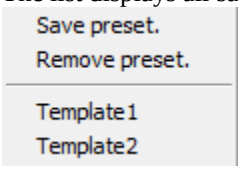
  - *Save preset*
    Opens a dialog box in which the name of the template is entered, after which the template is added to the list of templates.



  - *Remove preset*
    Opens a dialog box in which a template is selected to be removed from the list, or the entire list is cleared.



- *List preset*
  The list displays all saved templates. If the list is empty, "(Empty)" is displayed.

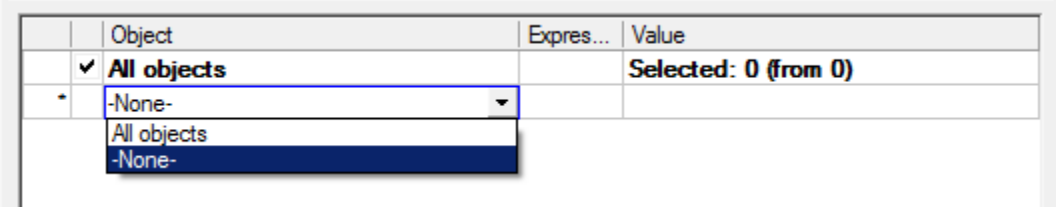


- *Reset conditions*. The command clears the list of conditions.

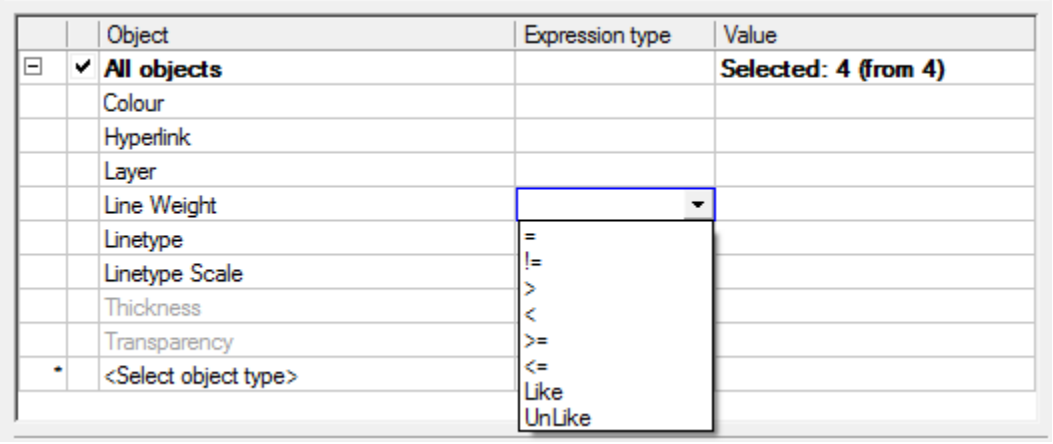
List of conditions

The list of conditions consists of three main columns:

- *"Object"* - displays objects and their parameters. If you selected the search area *"In the whole document"* or *"In the current space"* you must add objects yourself. In the *"Object"* column, select the desired type of object from the drop-down list of the cell. To deselect or remove an object from the table, select the *"-None-"* value.



- *"Expression type"* - a comparison operator is assigned to object parameters. Each type of object has its own parameters for which conditions are set.



Possible operators:

- = - equally;
- *!=* - not equal;
- <, > - less - more;
- >=, <= - no less, no more;
- *Like* - substring search. According to this condition, all objects are selected for which the parameter value contains the substring indicated in the *"Value"* column.
- *Unlike* - substring search. According to this condition, all objects are selected for which the parameter value does NOT contain the substring indicated in the *"Value"* column.

- *"Value"* - indicates the value by which the selection will be made. The value can be text, numeric, or selected from the drop-down list.

Viewport context toolbar

Context menu viewport appears when you right-click in the space of *Layout*.

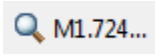
Activate the form. Opens in model space viewport.

Update dimensions. Sets the scale size of the form in accordance with the layout space.

Cut viewport. Overrides viewscreen on existing polyline or build a new one when you select the context menu option *Polygon*.

Fix view. Commits scale and position relative to the model space viewport.

Fit all. fits into the viewport model space objects.



Select scale. Opens the context menu to select the viewport scale.

MechWizard

MechWizard

MechWizard implements the following features:

- Creating a custom library of parts.
- Description of rules of conduct parts in the assembly.
- Assignment of output parameters to specify details.

MechWizard supports two types of iParts: static parts and parts with internal intelligence.
Static parts are based on values in the table. When they are inserted the following sequence of actions:

1. Select the item.
2. Indicate the denomination of the value table.
3. Apply the spare parts drawing.

The dynamic parts (with internal intelligence) in addition to the geometrical parameters, include pre-assembly constraints. Working with such objects looks:

1. Select an object-dependent part.
2. Bring the item to the mating piece.

Winning such an approach: Automatically detects the denomination detail, the insertion point, the direction of drawing, assembly constraints.

ScriptMaster

Main menu: ***Mechanical - Standard parts - MechWizard -*** ***ScriptMaster***.

Ribbon: ***MechWizard - MechWizard -*** ***ScriptMaster***.

Toolbar: ***MechWizard -*** ***ScriptMaster***.

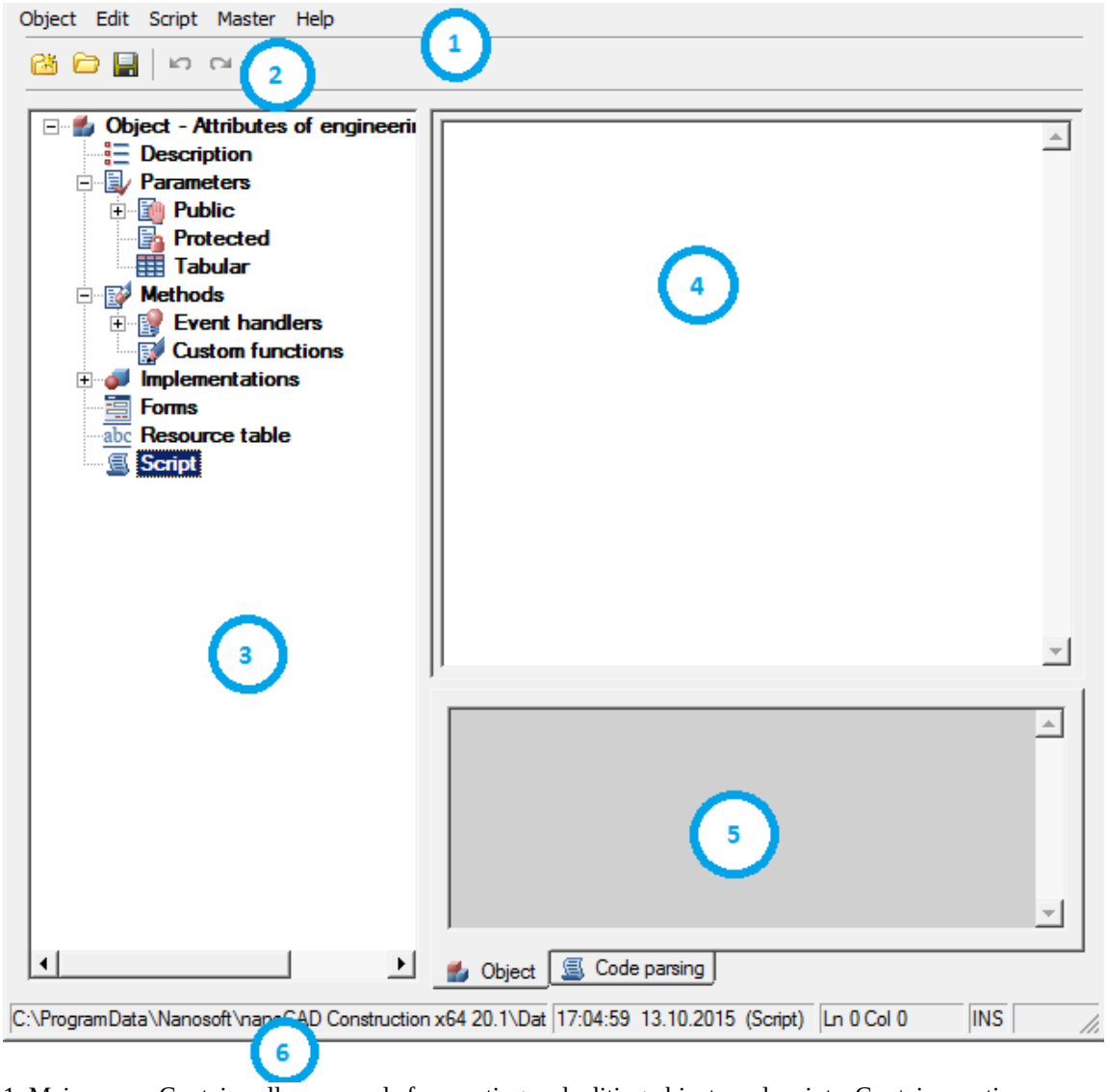
Library: ***Context menu command*** *"Open with ScriptMaster"* ***on any element library***.

Command line: ***MCSMASTER***.

The command opens *"ScriptMaster"* for creating and editing a database object.

Dialog

The title of the current element is indicated in the headers of the *"ScriptMaster"* dialog. An "*" signifies that unsaved changes were made to the item.



1. Main menu. Contains all commands for creating and editing objects and scripts. Contains sections:
   - Object
   - Edit
   - Script
   - Master
2. Toolbar. Contains basic commands for creating and editing objects.
   - *"New object"*. The command opens a dialog for creating a new base element.
   - *"Open"*. The command is designed to select another element from the base.
   - *"Save"*. The command saves changes to the current object.
   - *"Undo"*. The command cancels the previous action.
   - *"Redo"*. The command redo the last undone action.
3. Object tree. In the object tree, you can navigate through the object properties and manage them through the context menu.
4. Workspace. The script and table, as well as other properties of the object are edited in the work area.

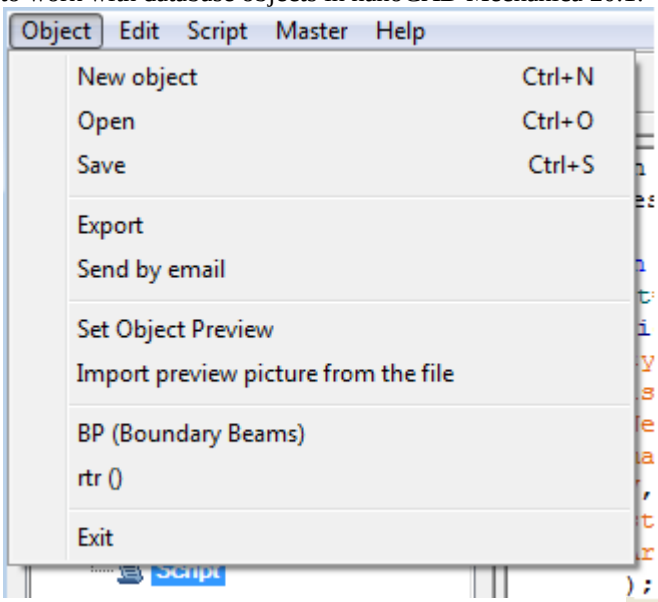
5. Notification area. The results of the script check and search / replacement are displayed in the notification area.
6. Status bar. The status bar displays the current state of work with the object:

- Setting the path to the current database;
- Current cursor position;
- Edit state - insert or replace;
- Script sign for reading or editing.

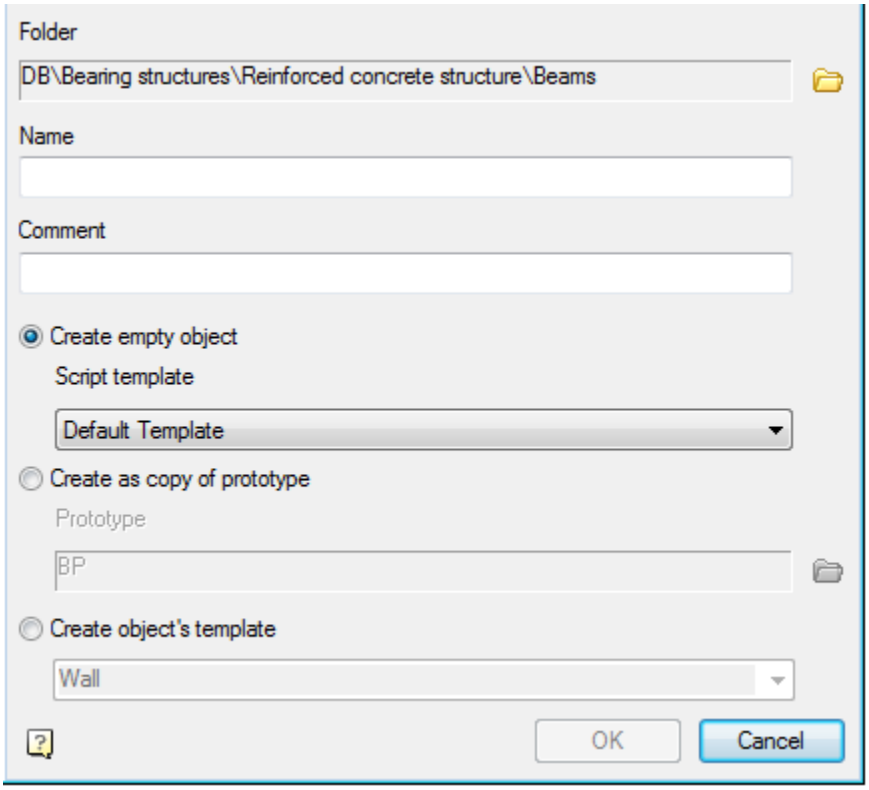
The menu bar

Object

Menu item *object* allows it to work with database objects in nanoCAD Mechanica 20.1:

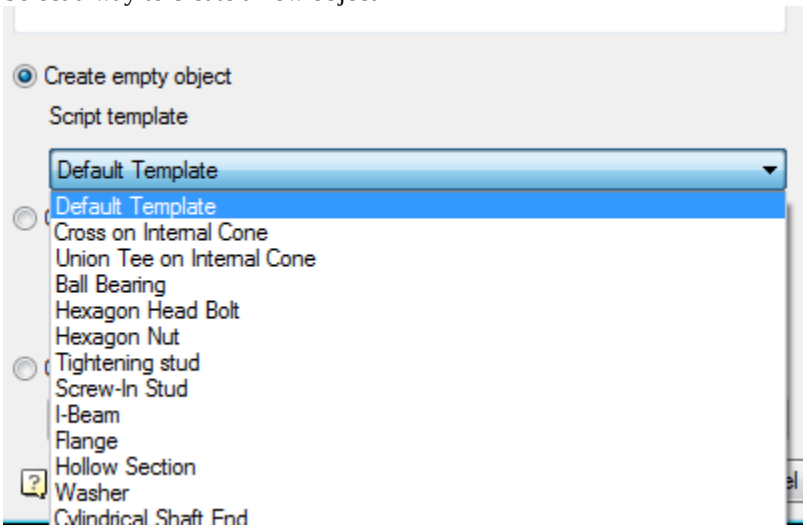


Object - creates a new element base (By pressing keys *Ctrl+N* or clicking on the icon *Create* on the toolbar). After issuing this command, a dialog box appears to *New object*



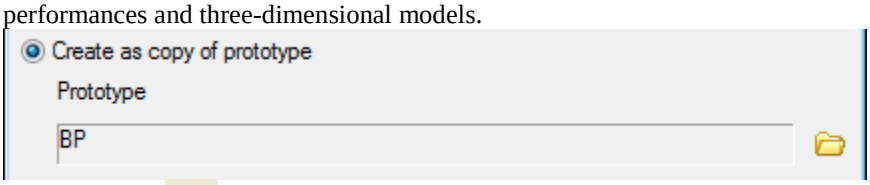
To create a new object, you must specify:
Folder in which to save the new object
Name (will be displayed in the browser)
Comment (will be displayed in the tooltip)

Select a way to create a new object



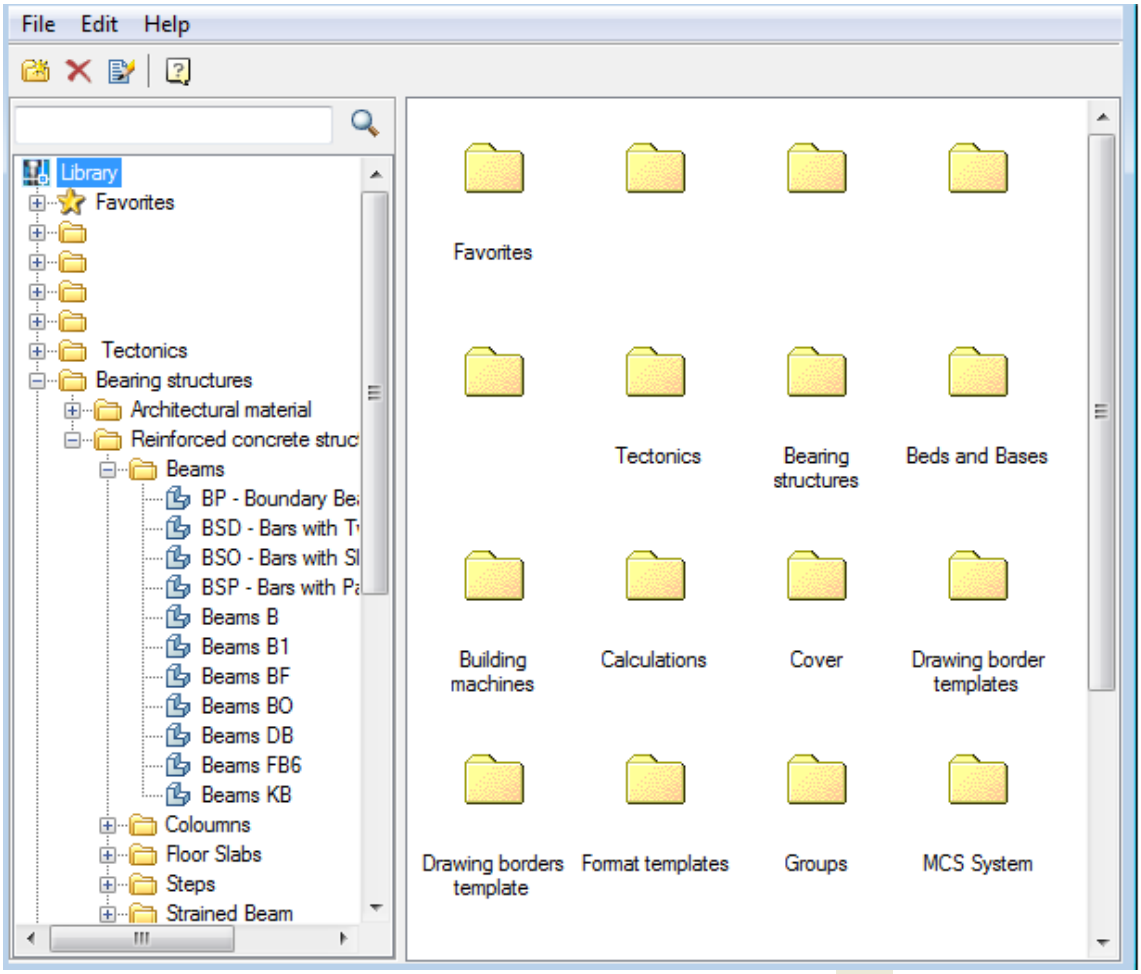
Create an empty object - creates a table and designs without having one script selected from the template.
Default template - empty script.

Create as copy of prototype - creates an object by copying the entire prototype object (along with the script, tables, performances and three-dimensional models.



Use the button to select an object to create a prototype object.

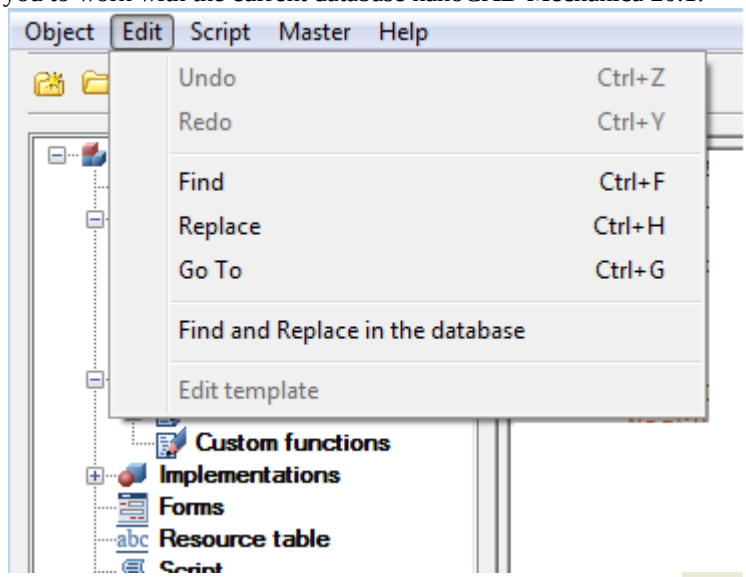
Open - Open an existing part (keyboard shortcut *Ctrl+O* or icon *Open* on the toolbar).
After selecting the command, a dialog box appears *Select the details* allowing you to select or find the item in the current database in Construction Site 4.



Save - save changes to the current member base (keyboard shortcut *Ctrl+S* or icon *Save* on the toolbar).
Set object preview - allows you to set as a preview for the object in the browser database objects nanoCAD.
Exit - close Master objects
In paragraph *object* available list of recently used items.

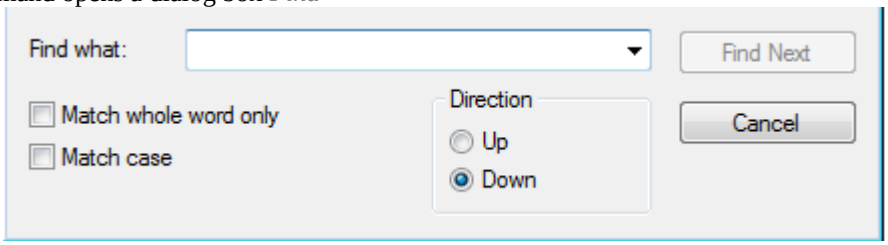
Editing

Menu item *Edit* allows you to work with the current database nanoCAD Mechanica 20.1:

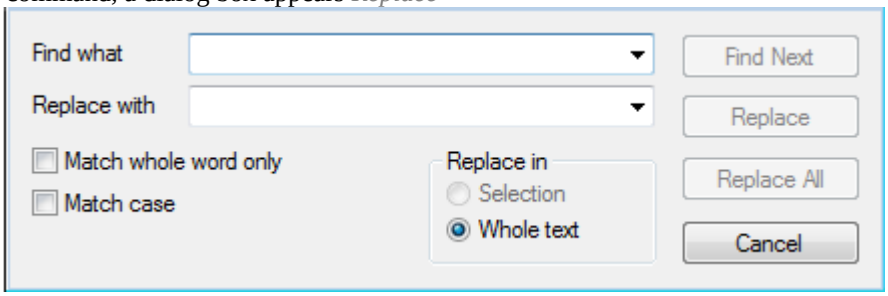


Undo - cancel the last action (promptness pressing keys *Ctrl+Z* or clicking on the icon *Undo* on the toolbar).

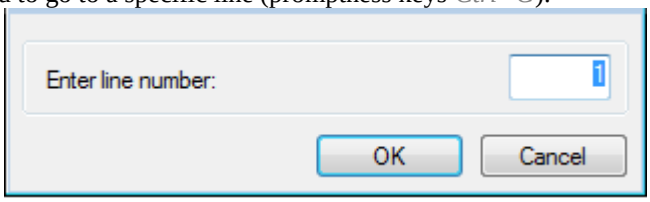
Redo - undo the last action (promptness pressing keys *Ctrl+Y* or clicking on the icon *Repeat* on the toolbar).
Find - search for a given string on the script of the current element (promptness keys *Ctrl+F*).
Calling this command opens a dialog box *Find*



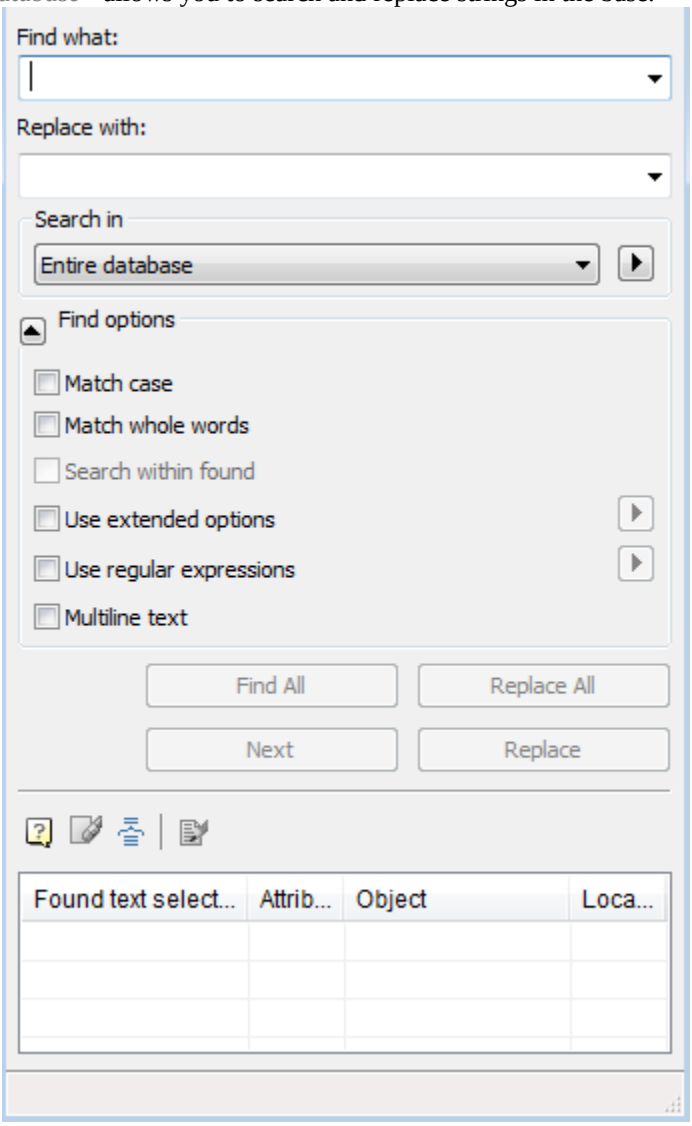
Looking provided an opportunity to take into account only whole words and case search string using by appropriate switches.
Also have the opportunity to select the search direction.
Go to the next matched string by pressing *find Next.*
After you click *Cancel* , you return to the previous menu.
Speas dropdown input field allows you to select the last used values.
Replace - search for a given string with a new one.
After calling this command, a dialog box appears *Replace*



When replacing provides an opportunity to consider only whole words and case search string using by appropriate switches. It is also possible to carry out replacement in the selected area or throughout the text.
Daylight without replacing the matched string to the next by pressing *find Next.*
When you click *Replace* matched string is replaced by a new one at the current position.
After you click *Replace All* all found strings will be replaced.
Button to return to the previous menu, press *Cancel*.
Go to - command allows you to go to a specific line (promptness keys *Ctrl+G*).



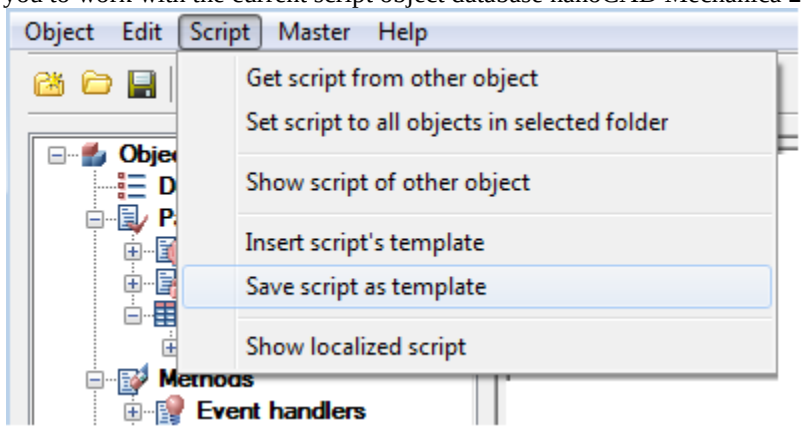
Find and replace on the database - allows you to search and replace strings in the base.



There are opportunities to search the entire database or in a folder, the search is case sensitive, whole words, as well as multi-line test, the search for the script, table, table name resources and facilities (*Use advanced options*)

Script

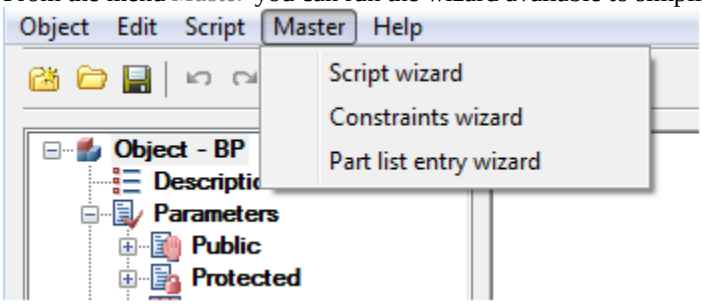
Menu *script* allows you to work with the current script object database nanoCAD Mechanica 20.1:



Get script from other object - command allows you to copy the script from another object.
Set script to all objects in selected folder - script command sets the current object to all objects in the selected folder database elements nanoCAD Mechanica 20.1.
Show script of other object - you can view the script of another object without changing it.
Insert script template - command allows you to take in the script as the current object previously saved template.
Save script as template - allows you to save the current script object as a template for future use.
Show localized script - command allows you to view localized script based on the data of the resource table.

Editing

From the menu *Master* you can run the wizard available to simplify the creation of objects:



- Script Wizard
- Constrains Wizard
- Part list entry Wizard

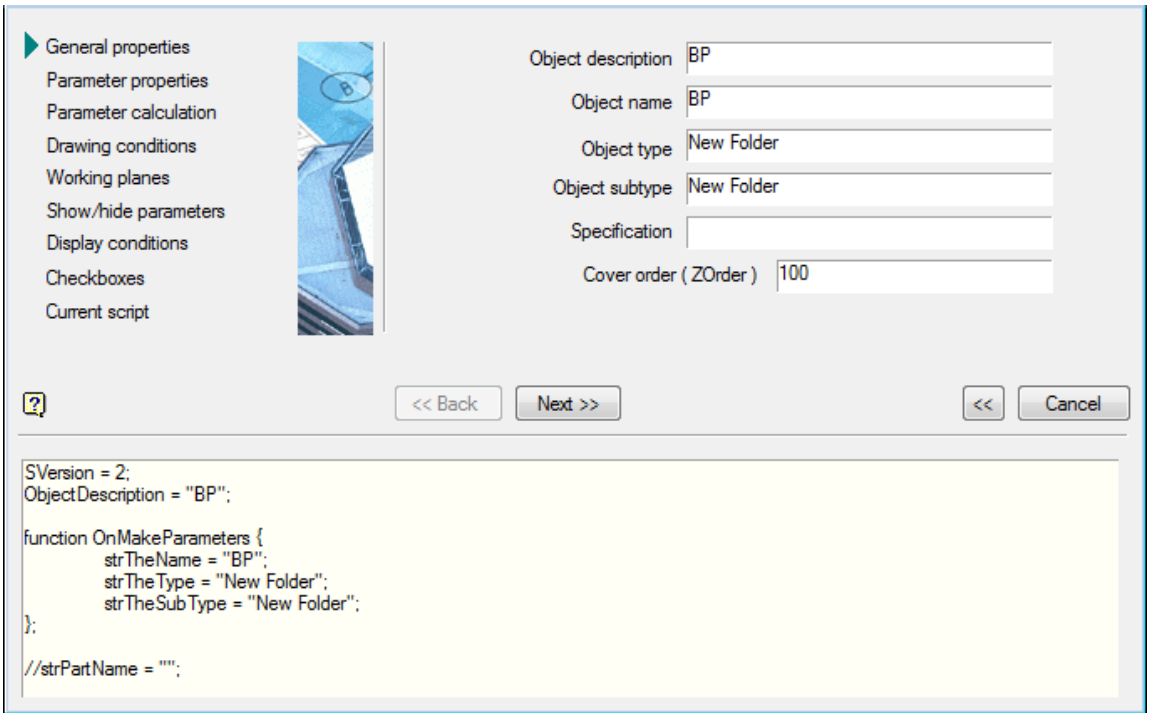
Script Master

In general, writing the script for the majority of standard parts is not required. Just open the script details, like on purpose, and copy it. However, it must be remembered that the parameters in the model (figure) must strictly match the prototype.
nanoCAD Mechanica 20.1 is a convenient mechanism for self-compilation step algorithms that control the behavior of the items in the time of insertion and editing object - *Master browser*.
Call *Master browser* via the menu item *Master / Script wizard*

- General properties
- Parameter Properties
- Calculation parameter
- Drawing Conditions
- working planes
- show / Hide parameters
- Display conditions
- Check boxes
- Current script

At any step of the script wizard may move buttons *<< Back* and *Next >>* .
Button *Cancel* exits the wizard.

Basic properties



Object description

ObjectDescription (Property Description). This parameter is optional, it describes the common name of the object. For example, nut, bolt, etc.

Object Name

strTheType (type). This parameter is required, it is advisable to fill it in Latin characters. Serves to identify the object as belonging to a group. For example, StdArborParts - standard parts shaft, StdJointParts - fastenings.

Object type

strTheName (Name of facility). This parameter is required, it is advisable to fill it in Latin characters. Defines elements within the group, given strTheType. For example, a screw bolt - Screw, for nuts - Nut, for washers - Washer etc.

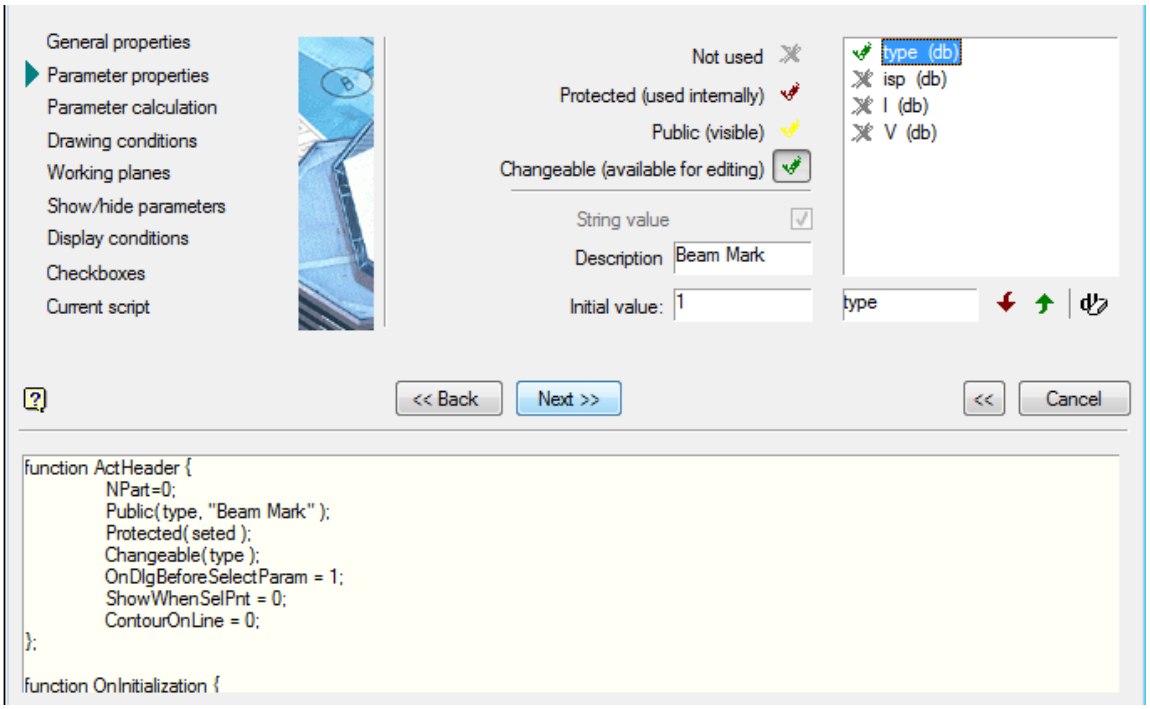
Object subtype

strTheSubType (subtype). Additional parameter, it is desirable to fill it in Latin characters. Defines a specific subtype of GOST. For example, if the name of the object - Nut (Nut), then simple nuts subtype is Simple, from crown - Castle etc.
These parameters set by the user, based on its project object interaction, and can be arbitrary. Hard list is not, and they can be changed at any time. They will only be used in scripts to automatically establish dependencies (details - in the description of scripts).

Specification

Description line items for transmission specification.

Parameter Properties



Not used

All parameters specified in the table, are automatically available for operations over them from the program during the insert or edit the object. If a variable is used only in the process of creating a graphic image, then use it in the script is not necessary.

Protected (used internally)

If necessary, display the parameter dialog during insertion.

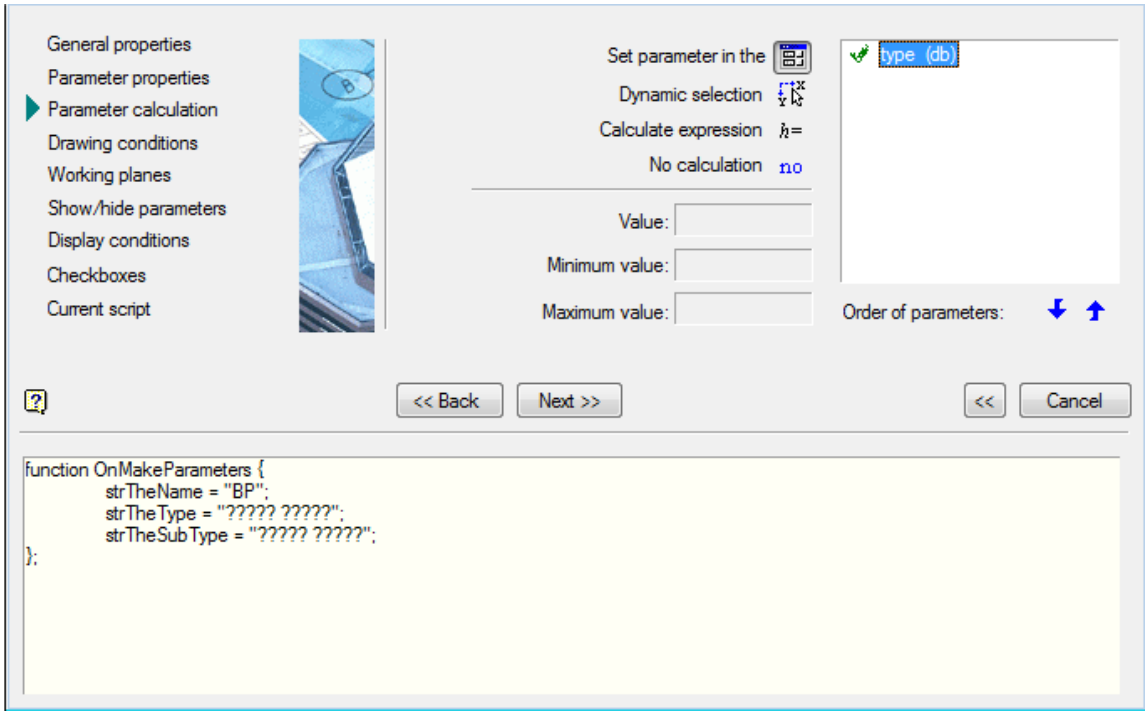
Public (visible)

Used to transfer parameters during insertion and editing of other objects.

Changeable (available for editing)

Available to change other objects in the editing process. Variable of the property, will pick up from the table and the necessary parameters on the basis of the obtained values to reconstruct the details of the model.

Calculation of parameters



Set parameter in the dialogue

Variable for which this flag is set to be available for editing of dialogue insert details.

Dynamic selection

Processed when the position of the cursor relative to the insertion point of the object according to the formulas in the designated parameter value in the range of "minimum value" - "Maximum".

Calculate after dialouge is shown using dynamic selection by expression

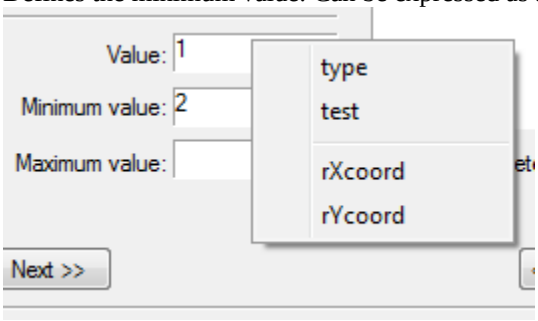
Calculated after the call dialogue and visual selection as expression in the Parameter Value.

Dont calculate parameter

Defines the minimum value. Can be expressed as a function of other parameters. Determination of the value of the parameter as a function of the cursor or other parameters. Available functions may be called from the context menu or manually entered into the appropriate input box.

Minimum value

Defines the minimum value. Can be expressed as a function of other parameters.



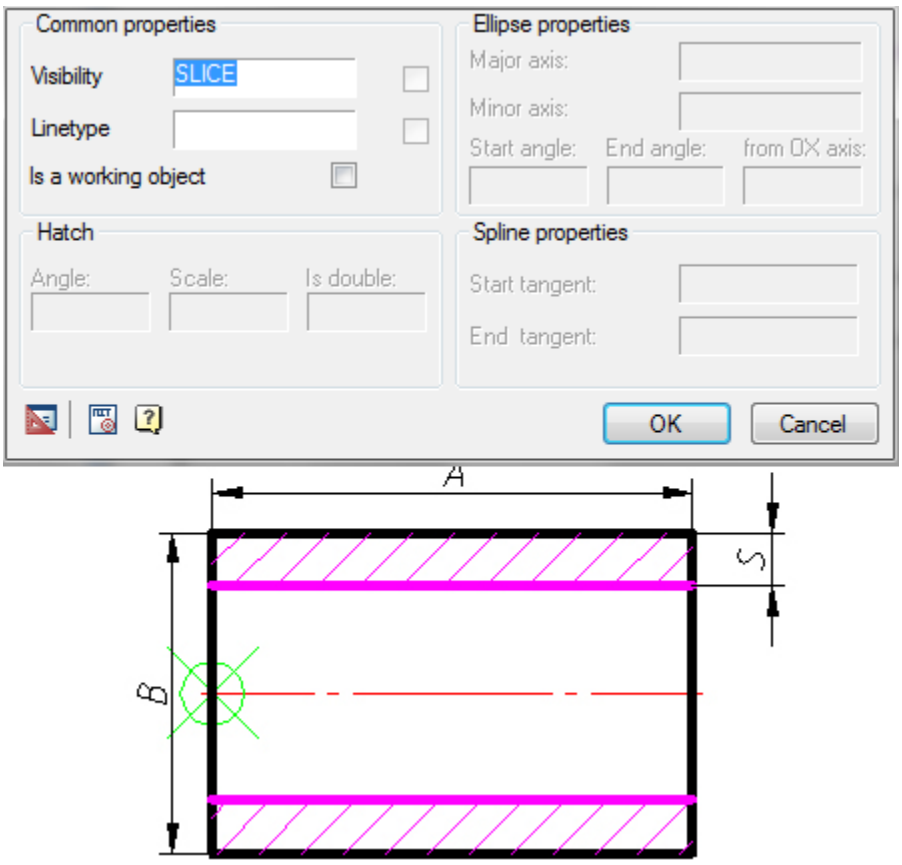
Context menu is available in the input field.

Maximum

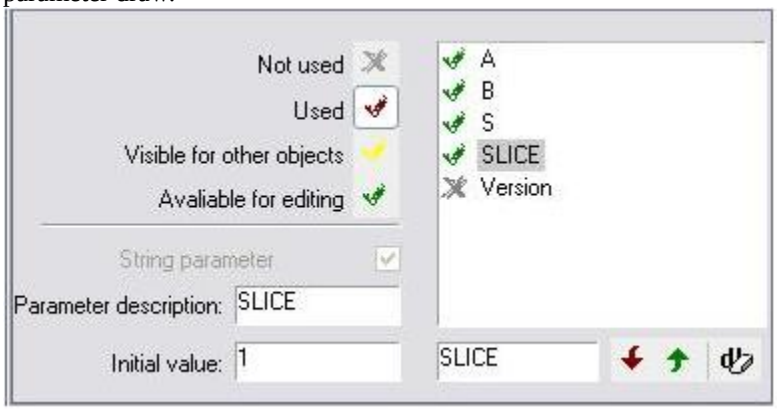
Defines the maximum value. Can be expressed as a function of other parameters. Context menu is available in the input field.

Conditions rendering

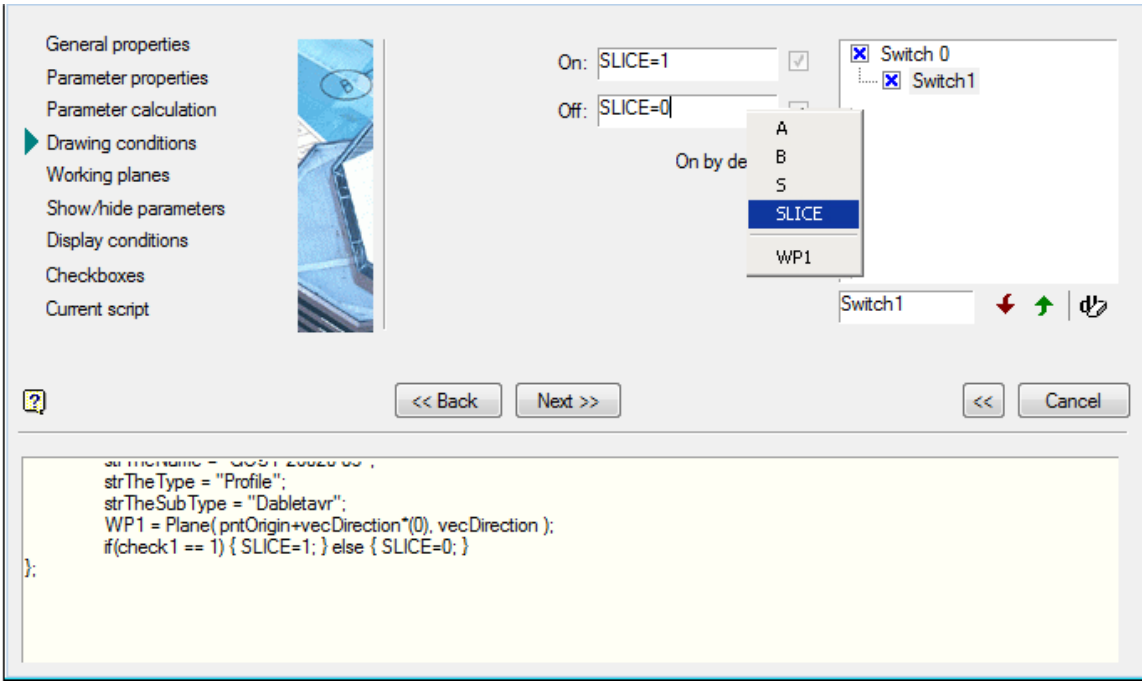
Add a line in the drawing model, rendering which we will operate from the dialog during the insertion of parts (for example, they will be highlighted *Magenta*). Pressing the button will open a dialog box *object properties* and assign as a condition of their appearance in the drawing text parameter. In this case ***SLICE***.



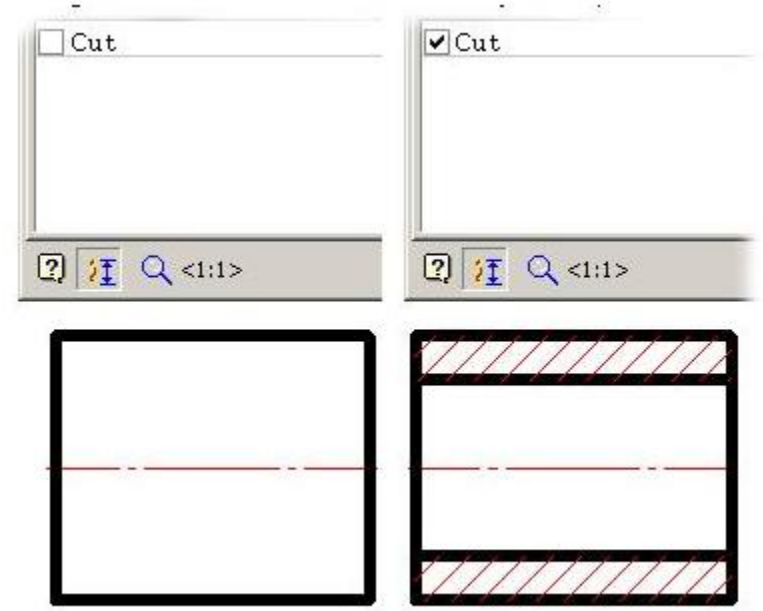
Size "S", necessary for proper recognition, automatically appears in the parameter list, but you must enter a parameter draw.



Fragment dialog box ***Properties settings***

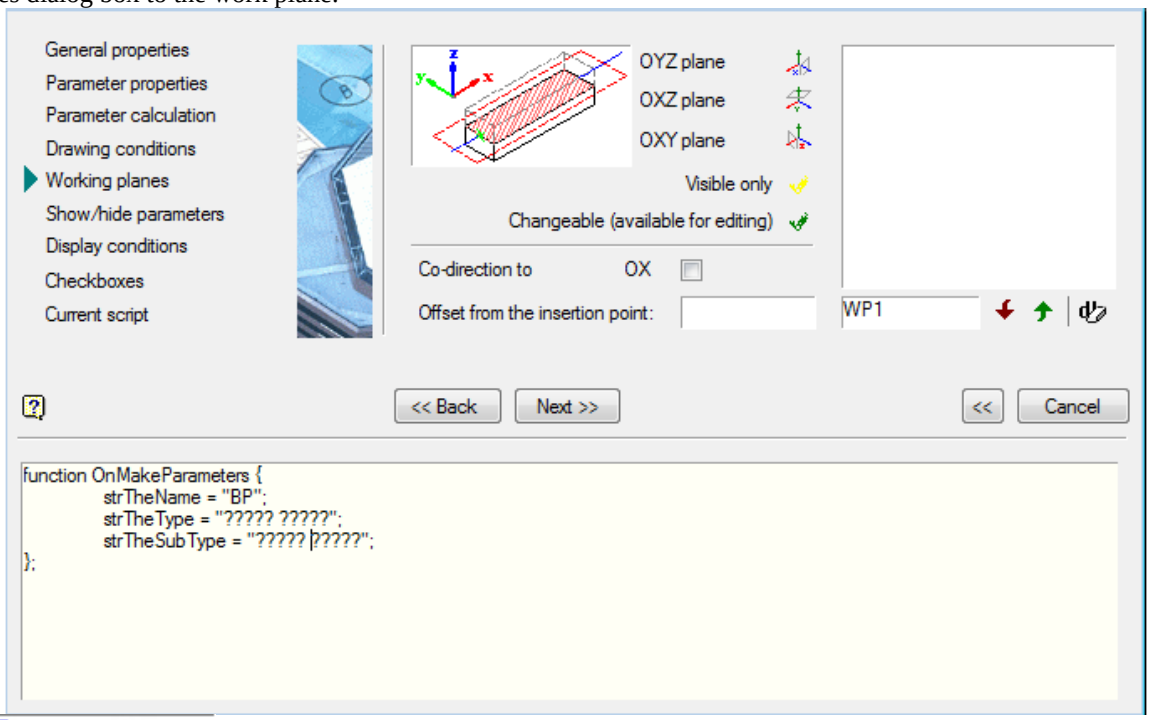


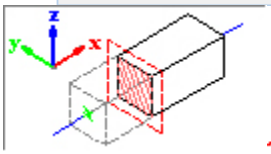
In the name field switch is set *"Cut"*. The parameter rendering paste from the context menu variable *"SLICE"*. After the completion of the script retain the item and exit the editor. Of the database tree select our item and insert it into the model. In the editing dialog parts we have a switch incision through which we can control the rendering of relevant lines.



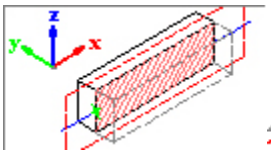
Working planes

Workplanes determine the position of the items in the time of insertion and editing. Determination of working planes dialog box to the work plane.

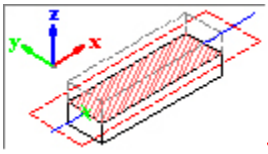




Creating a plane parallel to the plane OYZ



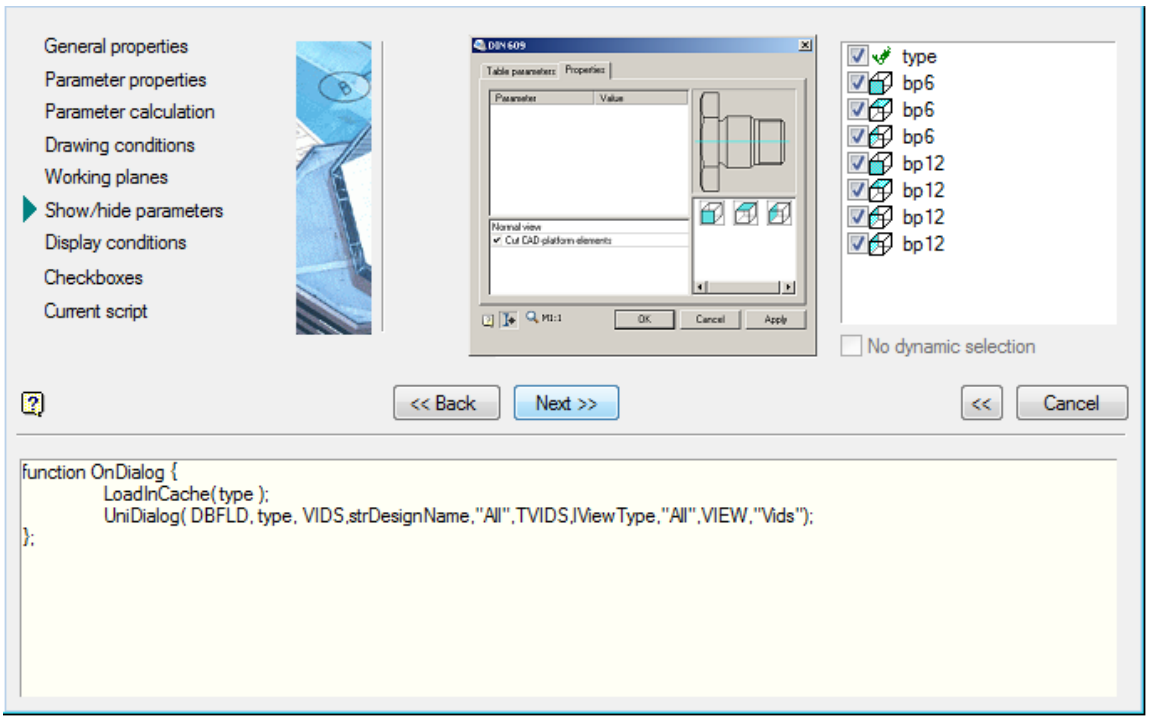
Creating a plane parallel to the plane OXZ



Creating a plane parallel to the plane OXY

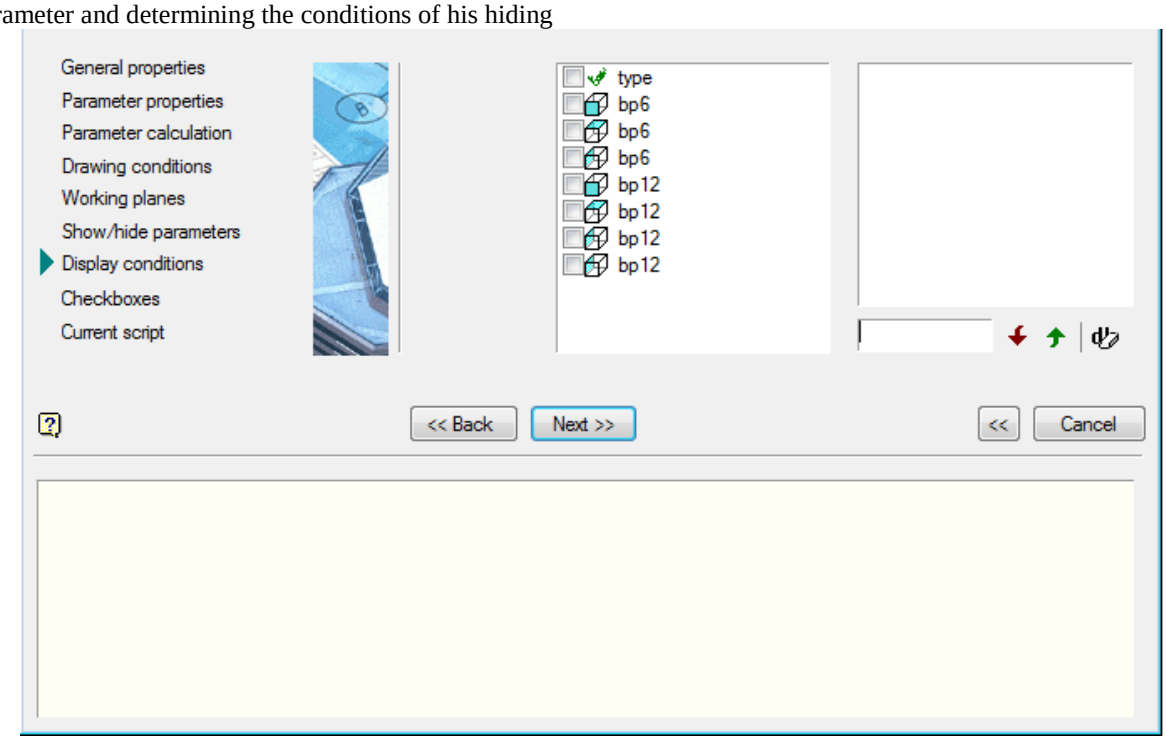
Show / Hide parameters

In this step, you must identify the available options in the menu items. The selection is made by setting a checkmark APPLICABLE parameter



Display conditions

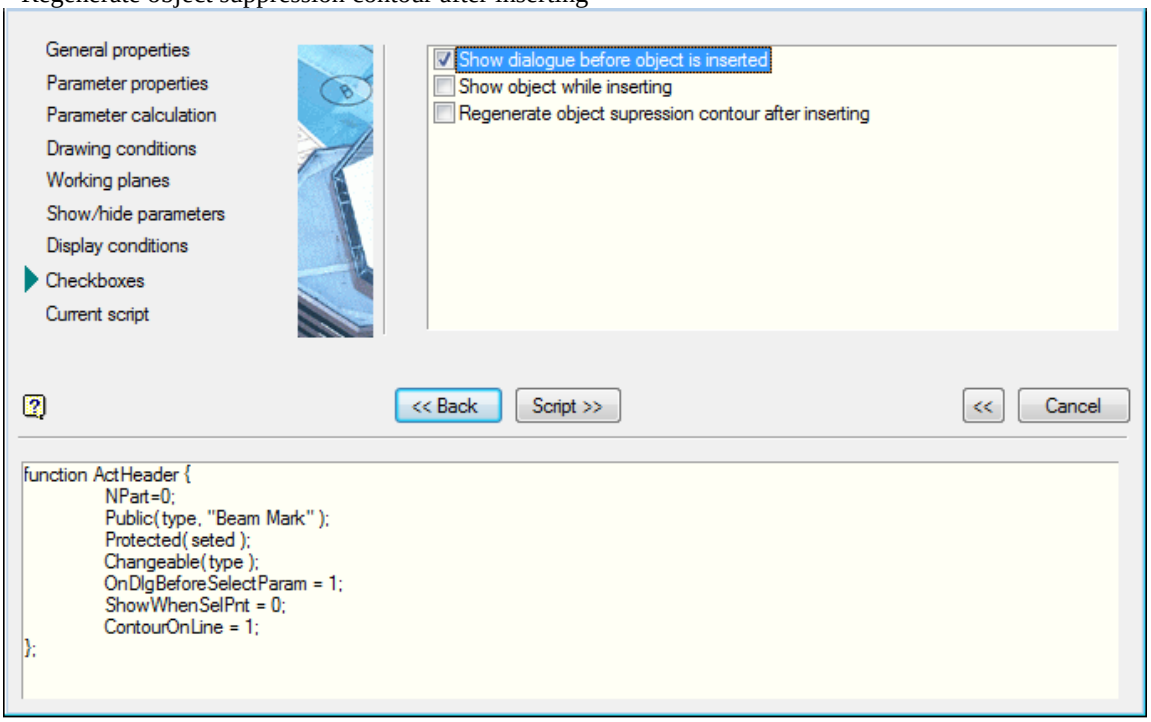
In this step, display conditions defined species . The selection is made by setting a checkmark APPLICABLE parameter and determining the conditions of his hiding



Check boxes

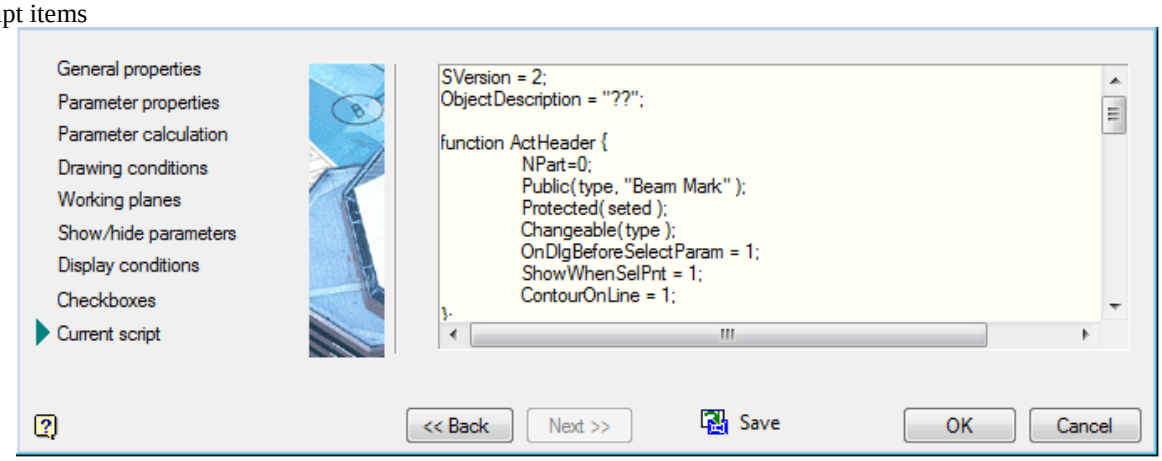
In this step, there are the following settings:

- Show dialogue before object is inserted
- Show object while inserting
- Regenerate object suppression contour after inserting



Current script

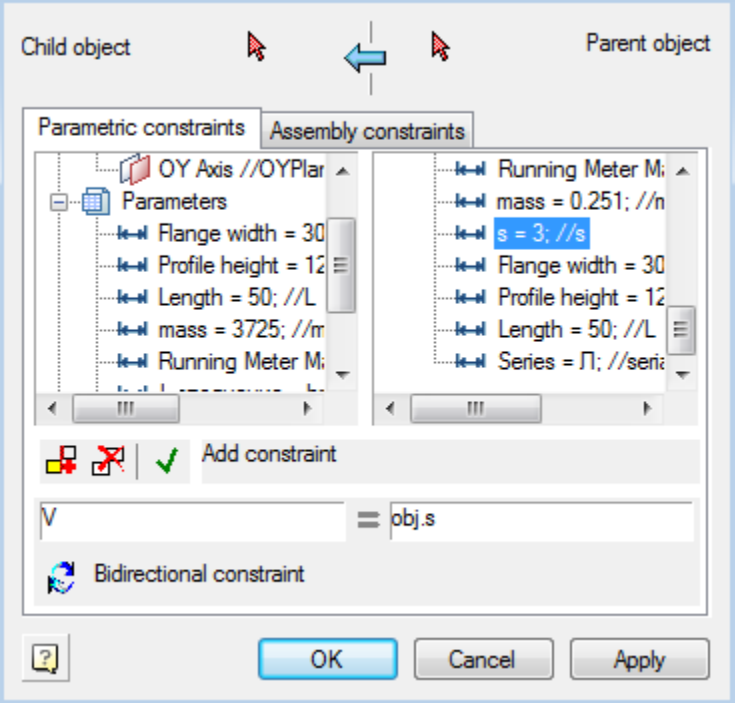
In this step, you can view and edit the resulting script. Clicking on the button *Save* and OK to apply the current script items



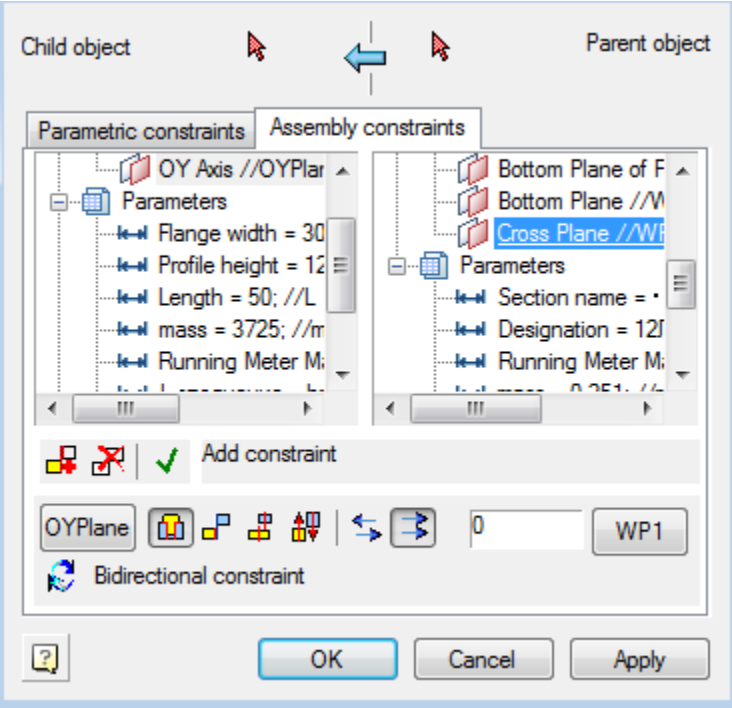
This is the final step.

Master dependencies

Creating dependencies for interaction with other items going online.
Call *Master dependencies* via menu *Master / Master Dependency* .
Further, following the screen prompts on the command line, place the drawing model details and select the items with which it must interact.
After placing the parts in the drawing, a dialog box *Managing Dependencies*. Functionally, the dialog box is divided into two parts bookmarks. Bookmark this *Parametric dependence* and Assembly constraints.



Definition of parametric relationships occurs directly in the dialog box. It's enough to double-click on the option to specify the details of the child (it will automatically appear in the left-hand side) and bring it into line with the parameter of the parent object. In the right-hand side can be used mathematical expressions.



Types of assembly and parametric relationships are defined in the dialog box. Parameters can serve as table settings objects and definable expression.

Axial and surface alignment

Surface alignment

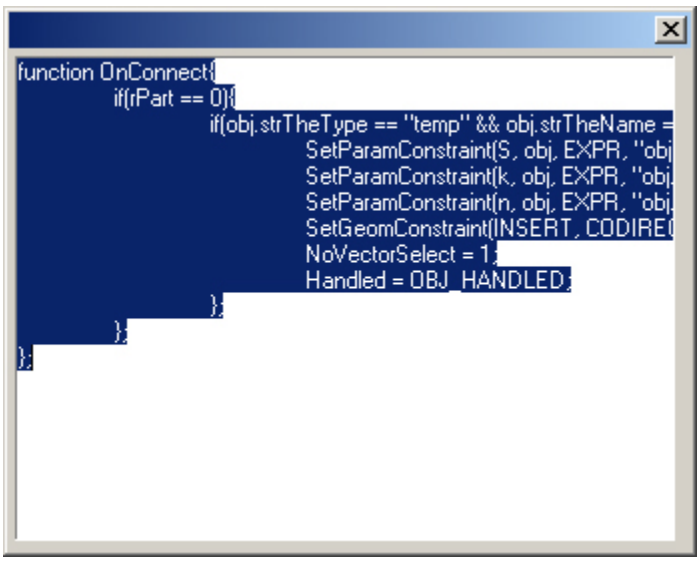
Axial alignment

Direction

Sets the direction for the dependencies (Counter directional / Co-directional)

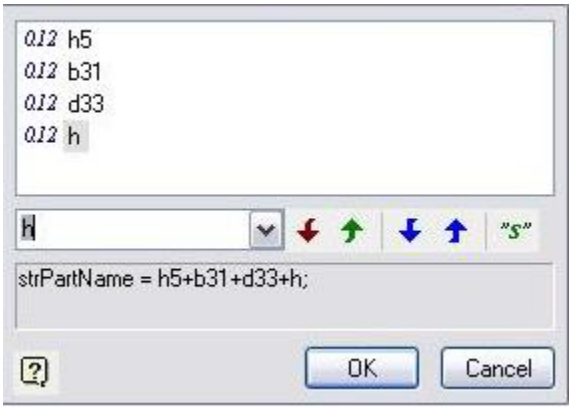
Once all the dependencies are installed, you can click OK, successively pass through the dialog boxes and set the options you want.

Eventually, an automatically generated string specifications necessary to automatically detect dependencies.

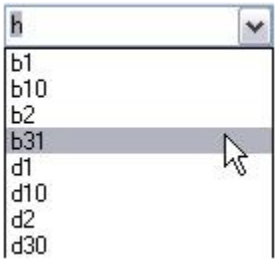


Master of string of specification

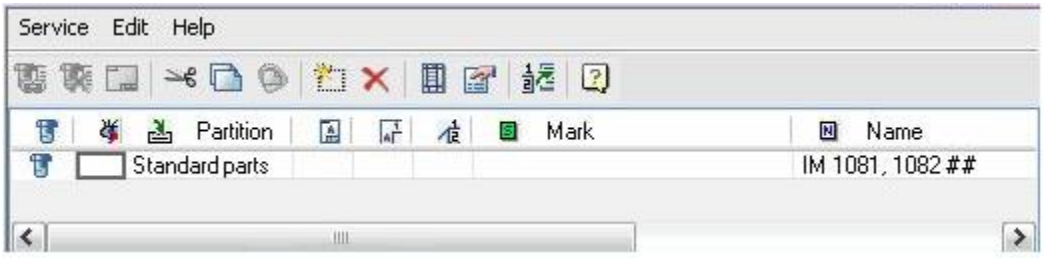
Formation data for transmission to the specification can be done in automatic and semi-automatic modes. Some information can be inserted on the basis of the table of parameters associated with the geometry of the figure, the part can be entered manually in the editing dialog standard parts.
Call *Master BOM line* via menu *Master / Master BOM line* .



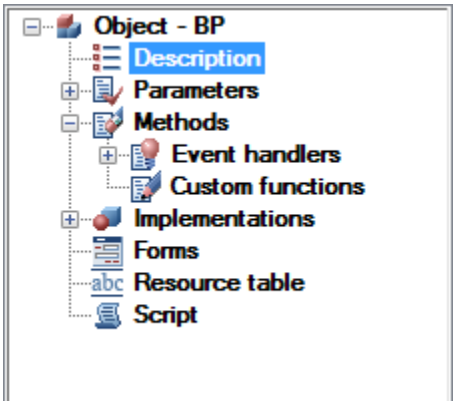
In the drop-down list are tabulated parameters involved in drawing details.



For a complete description of information on standard parts must be able to adjust or supplement information. For example, cover screws or the class thread. In this case, just add a "#".
The editor position will be able to insert text between the two lattices, and then save.



Object tree



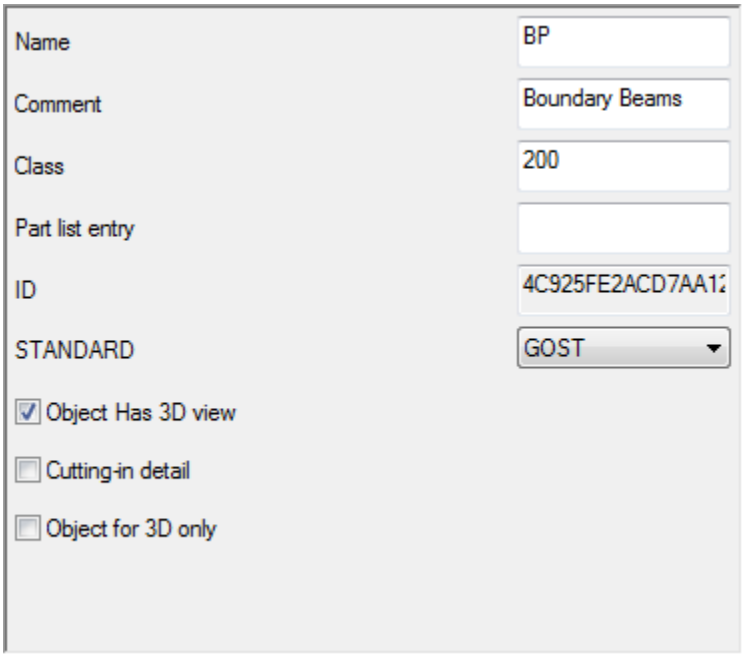
Tree object contains the following sections:

- Description ;
- Parameters;
- Methods;
- Function;
- Forms;
- Resources table;
- Script .

If the open file has been modified, its wood properties can be irrelevant. To actualize the properties tree to choose the context menu to update the object tree.

Description

When you click on the section *Description* of the workspace displays the following properties.



Property name - the name of the object to be displayed in the browser database.
Comment is displayed in the tooltip to the object.
Standard product allows you to set the standard (GOST, DIN, ISO, etc.).
Part list entry, is usually not be filled in the properties, and the variable defined in a script.
You can also put a tick:
Object have 3D view.
Cutting in detail
Object for 3D only.
In addition, the properties of the unique identifier for the object.

Parameters

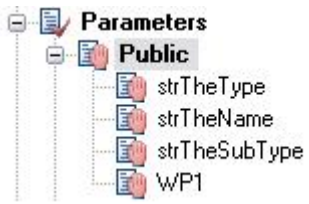
Section *Parameters* contains three subsections:

- public,
- protected,
- tabular.

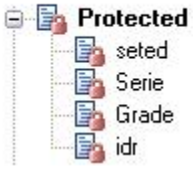
The public section *Upcoming* shows a table of open (public) parameters along with their comments.

	Name	Comment
1	strTheType	Object's type
2	strTheName	Object's name
3	strTheSubType	Object's subtype
4	WP1	Work Plane 1

Furthermore, the parameters are displayed in a tree branch.

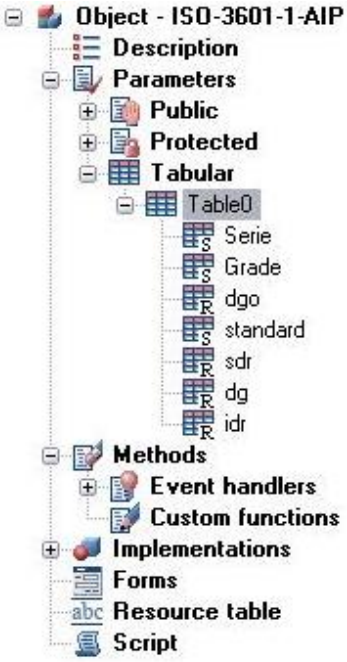


Section *Protected* contains a list of protected (protected) parameters, which are also displayed in a table and drop-down list of branches in the tree object.



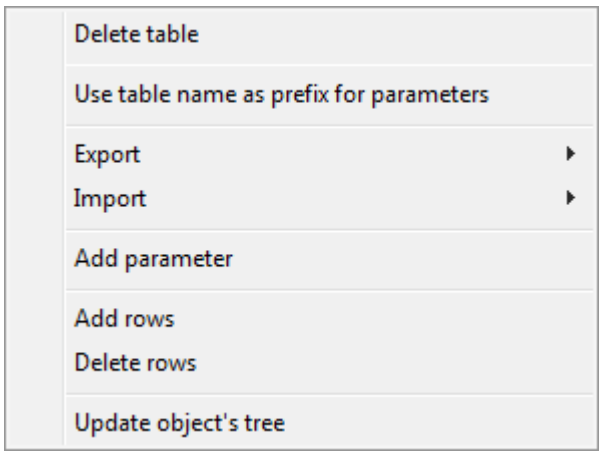
	Name
1	seted
2	Serie
3	Grade
4	idr

Section *Table* includes a list of tables detail

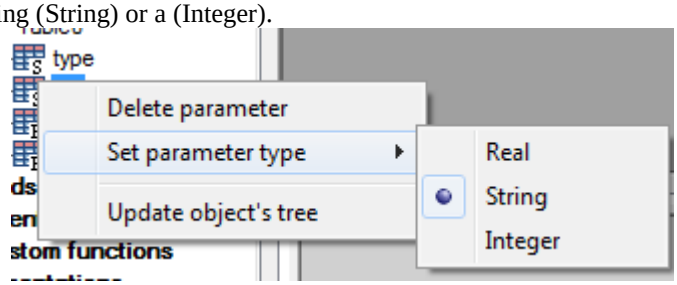


	ID	Serie	Grade
Type		String	String
Comm...	Identif...	Series	Class
1	1	G	N
2	2	A	N
3	3	G	N
4	4	A	N
5	5	G	N
6	6	A	N
7	7	G	N
8	8	A	N
9	9	G	N
10	10	A	N
11	11	G	N
12	12	A	N
13	13	G	N
14	14	A	N

Table and column of the table in the current workspace synchronized depending on the position of the cursor in the object tree. Tables in the object may be a few, while the identification is made on behalf of the table. Popup menu when the cursor position on the table includes the following tools:



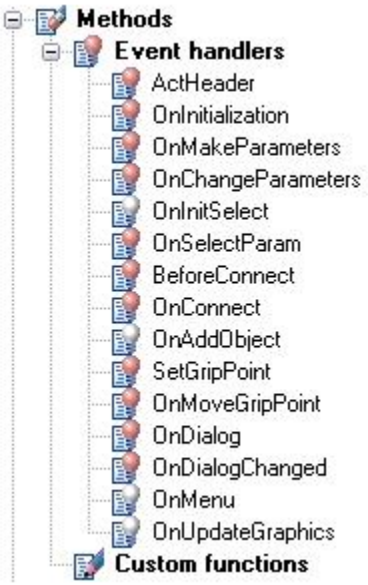
When you select *table name as a prefix to the parameters* of an appeal to the names of variables are prefixed with the table name. For example,Table0.dr.
You can add / delete rows from a table and add a parameter to a table.
There is also exported to Excel, text format and import from them.
Whenever the context menu on a separate parameter tables are available options remove a single parameter and set the type (Valid (Real), String (String) or a (Integer).



To rename a table parameter, select it and click on his name left mouse button, or press F2. If the parameter name has changed, it will automatically run search and replace in the script to a new name parameter correctly processed by the script.

Methods

Section *Methods* includes event handlers and custom functions.



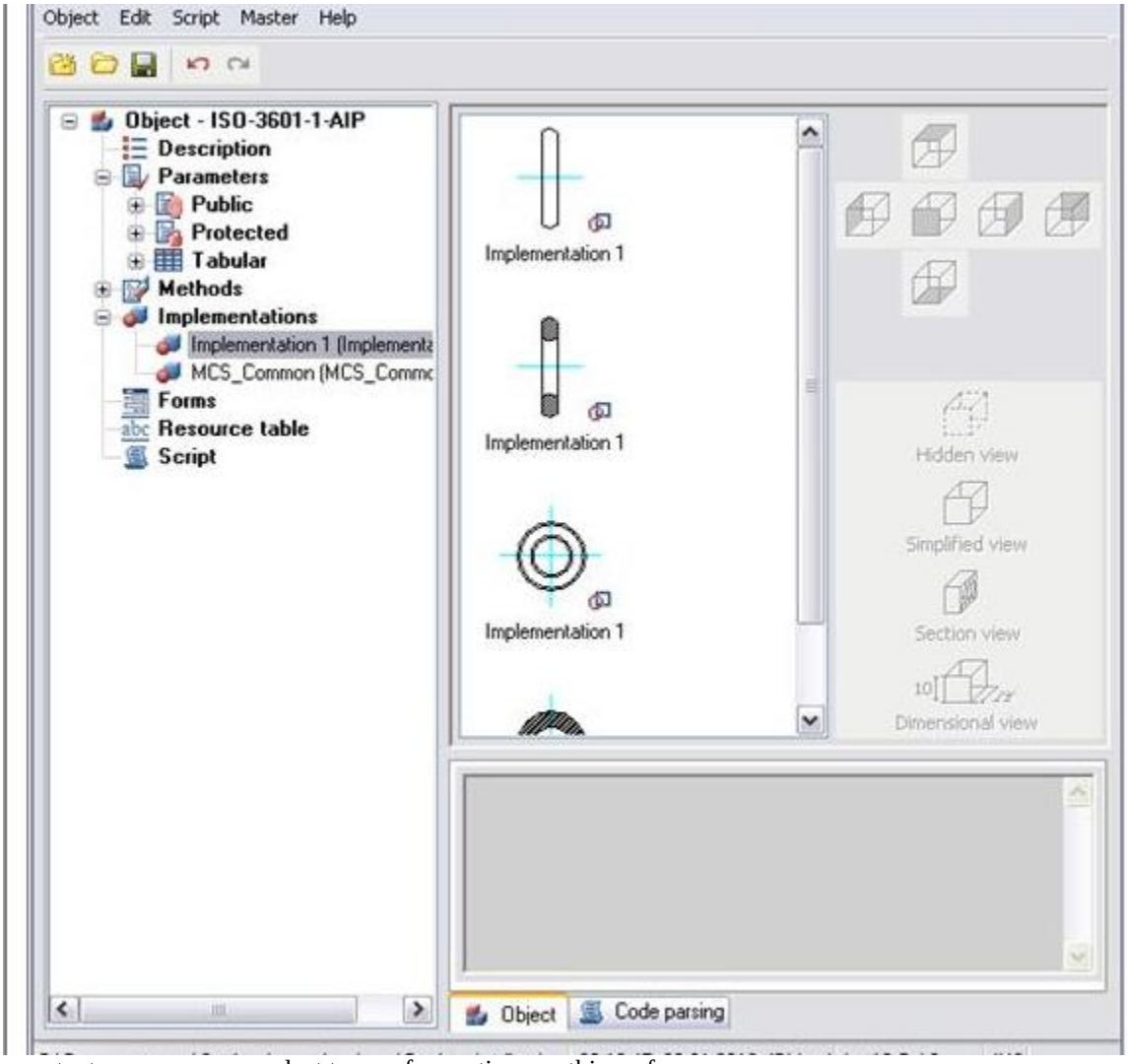
When selecting a specific function in the script is its description (synchronized current line).
Red light icon in the function indicates the presence of a function in the script is present, white - about lack.
When you double clicking on the corresponding function in the script created by its pattern.

Implementation

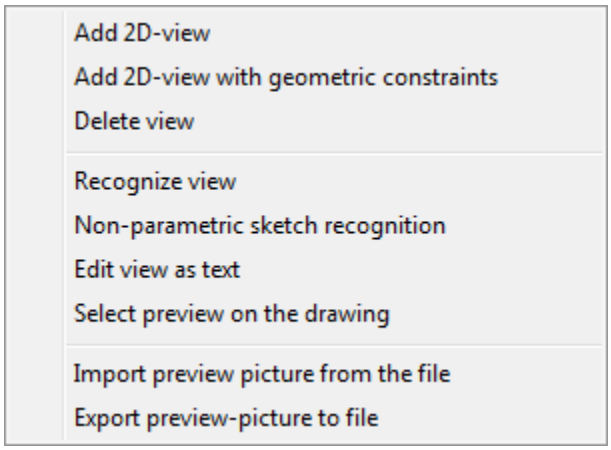
Section *Versions* list includes 2D and 3D versions
When selecting the root partition *versions* are all the performance, and the choice of a particular (for example, *Version 1*) - displays the filtered views of this execution. To create a two-dimensional view of a new version should be specified in the context menu *Add performance - automatically create 2D views.*
To create this kind of performance you need to select the required performance and the shortcut menu - Add 2D view.

- *2D view* - parametric graphic nanoCAD
- *3D view* of the Construction Site *20.1* - script to create a solid model (opening section of the shaft, etc.)

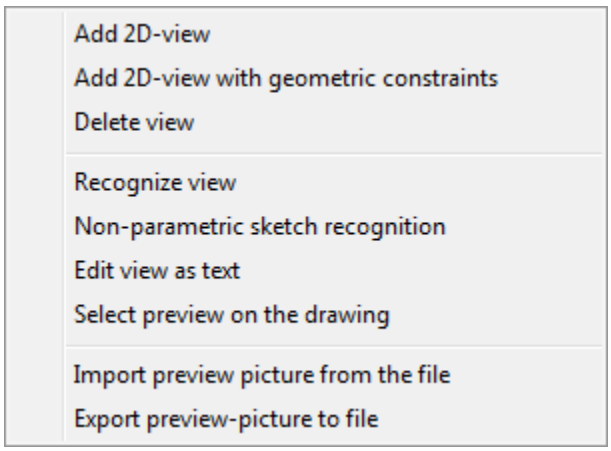
Deleted when you delete all kinds of performance of this version.
To set the parameters of the form, select it, and the right side of the workspace to set view orientation (*Front, Left, Top ,* etc.) and the type of form (*cut, cut C, C size* etc.).



In the context menu, you can select types of operations on this performance.
These options add \ remove kinds of current version:



The following context menu options are designed to work with the selected view of the current execution.



Recognize view - parametric recognition graphics nanoCAD.
Nonparametric recognize view - simple nonparametric reference charts, graphics taken from the object as it is.
Edit view as text - description of species opens in a text editor.
Set preview from objects - objects selection nanoCAD to create a preview of this species.
Import preview picture from file - creating thumbs on the basis of the previously saved file.
Export preview picture to file - Saves the preview in a file.

Form editor

Form Editor

Database objects nanoCAD Mechanica20.1 supports the use of custom dialogs for inserting and editing. Creating such dialogues (forms) is made in Form Editor, which can be accessed from the dialog boxes *Create Group*, *Create marker* and *Object Wizard* .
Button Open the ***form editor***.

Menu

File

- ***Test*** . Opens the form for viewing and testing.
- ***Output***. Closes the form editor.

Correction

- ***Cancel***. Cancels the previous execution of operations.
- ***Cut, copy, paste*** . Command working with the clipboard.
- ***Select all***. Selects all the controls on the panel shape.
- ***Remove***. Deletes the selected controls from the form.
- ***Group***. Group the selected controls.
- ***Ungroup***. Cancels the grouping selected controls.

View

- ***By ... edge***. Alignment commands selected controls along one edge.
- ***Align dimensions***. Aligns the selected controls dimensions.
- ***Above and below***. Reverses the order of the tab selected control by moving it up a level or down

- ***Forward, backward*** . Changes the tab order of the selected control by moving it to the first or the last level.
- ***Tab order***. Includes display serial numbers tab controls.
  - To set the tab order to be pressed sequentially controls on the left mouse button. Tab order will be set according to the sequence control option.
- ***Properties***. Enables and disables the additional window Properties.
- ***Variables***. Enables and disables the additional window variables.
- ***Grid***. Turns on and off the grid and snap to grid.
- ***Indents*** . Enables or disables the frame shape

Inset

There are commands insert controls

- ***Input field .***.
- ***Checkbox .***.
- ***Switch***.
- ***List***.
- ***List from the database***.
- ***Combo box***.
- ***Combo box from the database*** .
- ***Text***.
- ***Group***.
- ***Figure***.
- ***Panel***.
- ***Button***.
- ***Table with filtering***.
- ***Table***.
- ***Types and performance***.
- ***Viewing***.

Forms panel

On the forms produced occupancy controls.
In the context menu of the forms are available editing commands: *cut, copy, paste, group, ungroup*.
To add a control to the form by one of the following ways:

- Select the desired control from the menu *Insert* or toolbar. Specify the insertion point and set the dimensions of the control panel on the form.

After adding the control you want to bind it to a field in the *Properties*.

- In the open window, *Variables* , select the variable for which you want to add a control. Left-click on the variable in the list, and holding down the mouse button, move the cursor to the panel shape. Release the mouse button. Panel forms added ***group controls*** - *text* and *input field* .

If you insert a control from the window *variables* hold down Control, then it is possible to select the type of control.
Selection of the control panel forms can be accomplished in several ways:

- ***Single click of*** the left mouse button. Selected one control.
- ***Single click of*** the left mouse button while pressing Control . If the control is not selected, it is added to the current set. If a control is selected, it is removed from the current set.

- Choosing *a crossing* . Click the left mouse button in the space bar form. Holding the button down and dragging a selection box set. Selected controls that are fully or partially captured by the selection frame.

Editing the size of the control is done by using the " grips ". When the grid is resized with a certain step.

Changing the position of the selected controls performed with the left mouse button on one of the selected items. When the grid movement occurs incrementally.

Align controls

In the menu *View* available commands align controls on ***edge*** and ***size***.

Controls aligned to the left, and then by size .

Grouping controls

Working with multiple controls may need to be fixing their relative position. If grouping is fixed relative position of the controls, and when you move one of the other grouped elements also move.
Select panel forms controls and issue the Group. Selected controls will be grouped.
Removing Controls made the command Delete.

Test the form

Before completing the form creation is recommended to execute the test to view the form.

Variables window

In the ***Variables*** are parameters of the object (*marker , database object , group*).
Icon ✔ marked parameters that are associated with controls editable form

Properties window

In the Properties lists the properties of the selected controls on the form. At the bottom there is a line prompt with a brief description of the selected property

Form properties

General	
Help URL	
Caption	
Editor	
Top margin	8
Height	400
Left margin	8
Bottom margin	8
Margin	☑
Right margins	8
Grid	☑
Grid step	8
Width	400

Note:	In the "Help" parameter, the path to the user help is entered. User help will be invoked if the focus is on form control. If the focus is on the "Cancel" or "OK" buttons, the help of standard elements will open.

Input field. The input field is used to enter numeric or symbolic expressions for further processing.
Properties command:

View	
Visible	☑
Alignment	On the edge
Multiline	☐
Flat	☐
Tip	
Linking	(None)
Enable	☑
Text	None
Formatting	☐
Basic	
Name	editbox1
Class	After entering
Variable	
Type	Line
Position	
Upper limit	56
Height	56
Left boundary	176
Lower limit	112
Right boundary	232
Width	56
Measurements	
Meter	☑
Mode	Auto
Regime change	☐
Developments	
Activation	

Deactivation	
To change	

Checkbox. Allows you to monitor on/off.
Properties command:

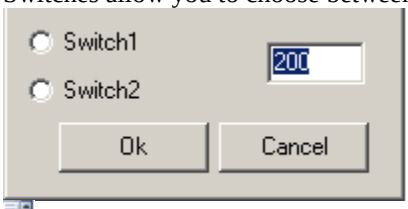
View	
Visible	☑
Tip	
Linking	(None)
Allowed	☑
Text	Flag
Basic	
Enabled	☐
Name	checkbox
Class	Flag
Variable	
Position	
Upper limit	8
Height	16
Left boundary	8
Lower limit	24
Right boundary	72
Width	64
Developments	
by clicking	

Switch .Switches to monitor the status on/off. Several individual switches interconnected to form a "group switch" to track more states. Example of use of the group switch.
Description:
Several switches are associated with the same variable script. Each of them set any value. Variable is set to the included switch. Value is of type string. If set to "-1" value is assigned automatically starting with 1, in the tab order for TAB. In this case, the value type is numeric. Switches associated with a single variable group are considered. In the group may be included in only one switch. If a variable is assigned, grouped switches located inside one of the control. Usually this group or panel.
Properties command:

View	
Visible	☑

Tip	
Linking	(None)
enabled	☑
Text	Switch
Basic	
Included	☐
Value	-1
Name	radiobutton
Class	Switch
Variable	
Position	
Upper limit	24
Height	24
left boundary	8
Lower limit	48
Right boundary	112
Width	104
Developments	
by pressing	

Example of using.
Switches allow you to choose between discrete values, the result is displayed in the input field



List. Used to select one value from a specified list.
Properties command:

View	
Visible	☑
Tip	
Linking	(None)
Enable	☑
Text	-1
Basic	

Data	"Element 1"
Name	Listbox
Class	List
Variable	
Current	-1
Position	
Upper limit	48
Height	40
Left boundary	8
Lower limit	88
Right boundary	104
Width	96
Developments	
Activation	
Deactivation	
To change	

List from the database.. Used to select one value from a pre-specified list in the database.
Properties command:

View	
Visible	☑
Tip	
Linking	(None)
Enable	☑
Text	
Basic	
Data	
Name	Listbox1
Class	List of base
Variable	
Current	
Position	
Upper limit	96
Height	60

Left boundary	8
Lower limit	156
Right boundary	104
Width	96
Developments	
Activation	
Deactivation	
To change	

Combo box. Designed to select from the specified list or enter an arbitrary value.
Properties command

View	
Visible	☑
Tip	
Linking	(None)
enabled	☑
Text	Combo box
Basic	
Data	
Name	combobox
Class	combobox
Variable	
Editable	☑
Position	
upper limit	168
Height	76
left boundary	8
Lower limit	244
Right boundary	104
Width	96
Developments	
activation	
deactivation	
to change	

Combo box from the database. Designed to select from the specified list or enter an arbitrary value.
Properties command:

View	
Visible	☑
Tip	
Linking	(None)
Enabled	☑
Text	
Basic	
Data	
Name	combobox1
Class	combobox
Variable	
Editable	☑
Position	
Upper limit	208
Height	64
Left boundary	8
Lower limit	272
Right boundary	104
Width	96
Developments	
Activation	
Deactivation	
To change	

Text . Static (non-editable) text formatting support. Specify the formatting is applied to the entire text.
Properties command:

View	
Visible	☑
Headset	(Info)
Tip	
Linking	(None)
Transparent	☑

Text	Text
Background	
Basic	
Name	static
Class	Text
Label	
Variable	
Position	
Upper limit	280
Height	32
Left boundary	8
Lower limit	312
Right boundary	104
Width	96

Group. Decorative element, designed to visually group several controls on the form.
Properties command:

View	
Visible	☑
Tip	
Linking	(None)
Enabled	☑
Text	
Basic	
Name	groupbox1
Class	Group
Variable	
Position	
Upper limit	320
Height	38
Left boundary	8
Lower limit	358
Right boundary	104
Width	96

Developments	
Activation	
Deactivation	
To change	

Figure. Insert a simple bitmap to the form.
Properties command.:

Basic	
Name	Image
Class	Drawing
Variable	
View	
Visible	☑
Data	(Info)
Tip	
Linking	(None)
Transparent	☐
Proportionally	☑
Stretch	☑
Position	
Upper limit	360
Height	57
Lleft boundary	8
Lower limit	417
Right boundary	105
Width	97

Panel. Designed for grouping and align multiple controls on the form.
Properties command:

Basic	
Name	panel
Class	Panel
View	
Linking	(None)
Position	

Upper limit	16
Height	49
Left boundary	112
Lower limit	65
Right boundary	185
Width	73

Button. Control "button." Response by pressing the left mouse button.
Properties command:

View	
Visible	☑
Tooltip	
Linking	(None)
Allowed	☑
Text	Button
Main	
Icon	-1
Name	button
Class	Button
Variable	
Position	
Upper Bound	72
Height	32
Left bound	112
Lower limit	104
Right border	184
Width	72
Developments	
by clicking	

Table with filtering. Table variables to restrict the number of columns, and automatic filtering of the list columns.
Properties command:

Basic	
Name	Control
Class	Table with filtering

Variable	
View	
Visible	☑
Tip	
Linking	(None)
Enable	☑
Position	
Upper limit	112
Height	32
Left boundary	112
Lower limit	144
Right boundary	184
Width	72

Table . Table variables with the ability to view and select a row.
Properties command:

Basic	
Name	Control1
Class	Table with filtering
Variable	
View	
Visible	☑
Tip	
Linking	(None)
Enable	☑
Position	
Upper boundary	152
Height	40
Left boundary	112
Lower limit	192
Right boundary	184
Width	72

Views and performance. Table for selection and execution of standard projection of the object from the database.
Properties command:

View	
Visible	☑
Tip	
Linking	(None)
Enable	☑
Basic	
Name	Control2
Class	Views and performance
Execution	
Show performance	☑
Types	
	☑
	☑
	☐
	☑
	☐
	☐
Display Types	☑
Shows the type form	☐
Position	
Upper limit	200
Height	32
Left boundary	112
Lower limit	232
Right boundary	184
Width	72

Preview. Preview box of the selected object from the database.

Properties command:

View	
Visible	☑
Tip	
Linking	(None)

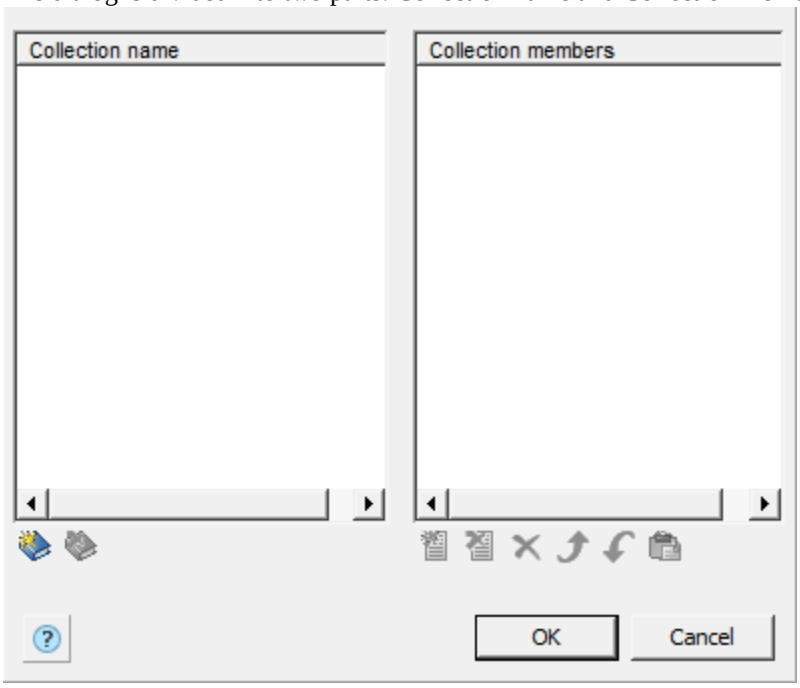
Enabled	☑
Text	PreViewControl
Basic	
Name	control3
Class	Review
Position	
Upper limit	240
Height	40
Left boundary	112
Lower limit	280
Right boundary	184
Width	72

Collection Editor

Serves for quick creation of lists and their subsequent use in form controls and custom sorting in tables.

Dialog

The dialog is divided into two parts: Collection name and Collection members.



Collection management tools

Add new collection.

Delete collection.

Collection members management tools

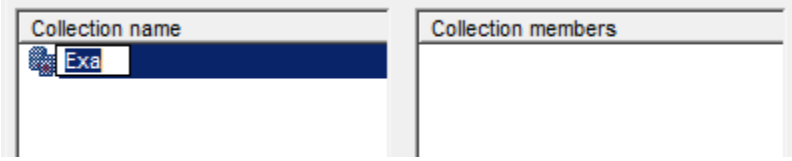
Add new collection member.

Delete collection member.
Delete all collection members.
Move collection member upper.
Move collection member down.
Paste from clipboard.

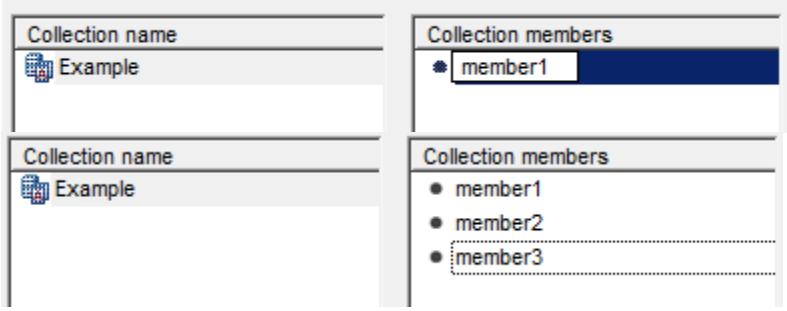
Create and assign a collection while editing a form

In order to create a collection you need to:

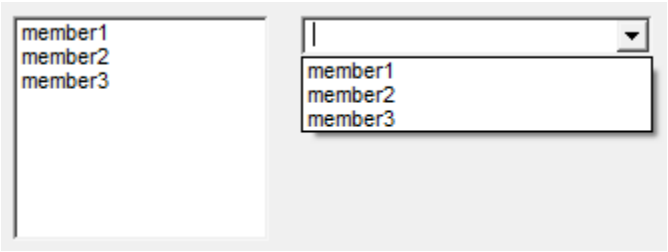
1. Click the *"Data"* option in the DB-Listbox or DB-combobox. A dialog box will appear.
2. In the list of collections, call the *"Add new collection"* command and enter a name for the collection.



3. Set the composition of the collection.



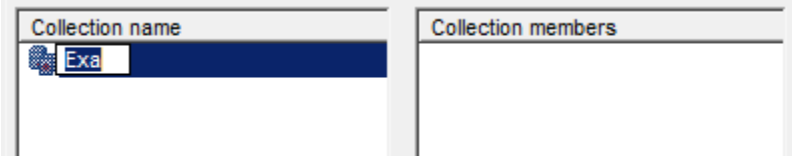
4. Select a collection and click *"OK"* in the Collection Editor dialog box.
5. he created collection will look like this:



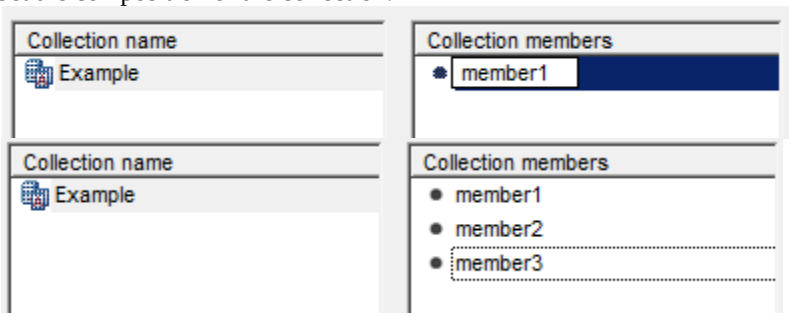
Creating custom filtering on a table

In order to create a collection you need to:

1. Go to Edit Table Dialog *"Main Menu - Rows - Custom Sort..."*.
2. In the list of collections, run the *"Add new collection"* command and enter a name for the collection.



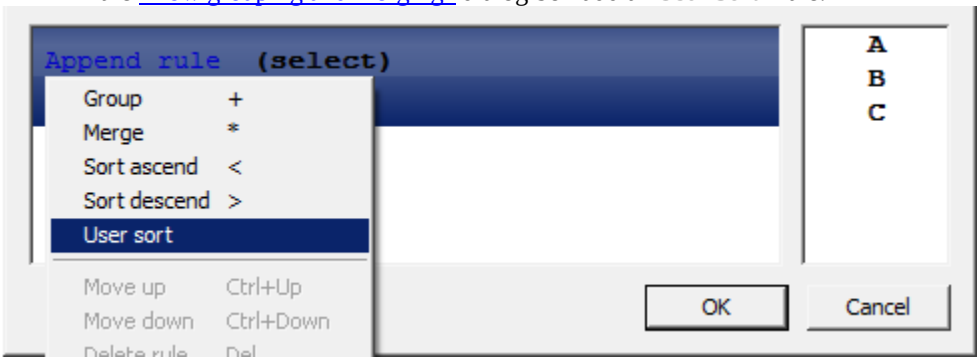
3. Set the composition of the collection.



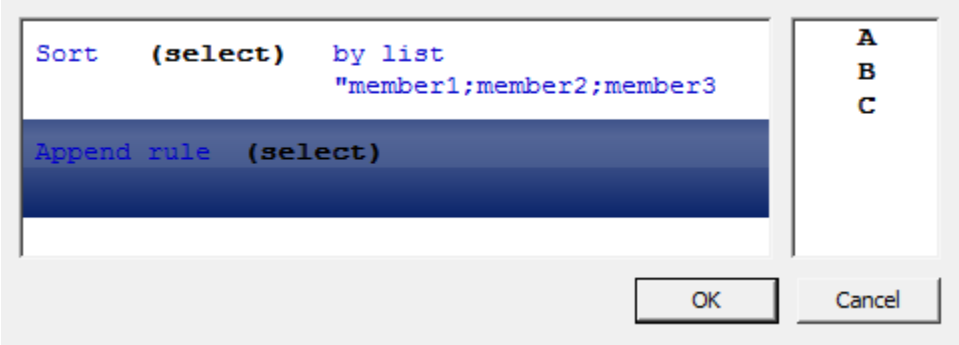
4. Click *"OK"* in the Collection Editor window.

Purpose of custom filtering in a table

1. In the "Row grouping and merging" dialog box add a *"User Sort"* rule.



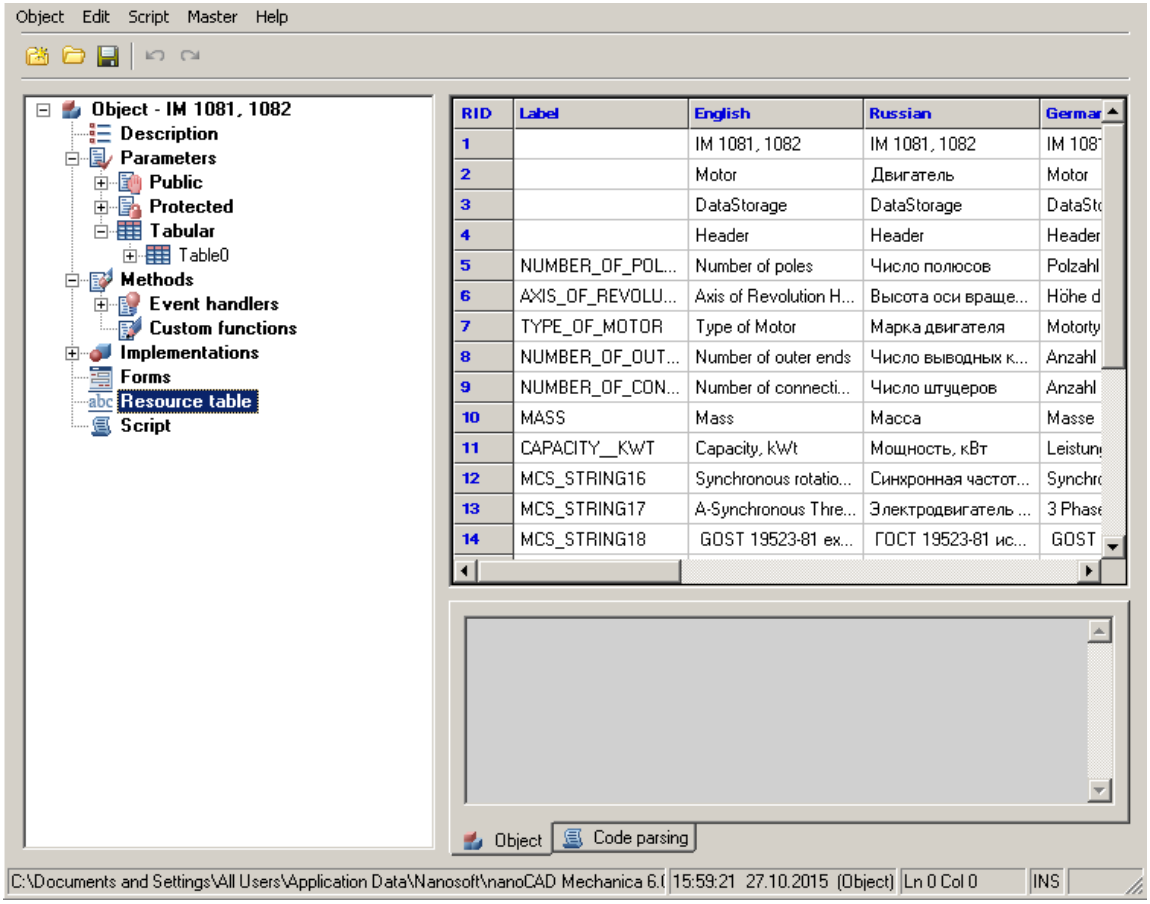
2. Select a collection and click *"OK"*.
3. The custom sort created will look like this:



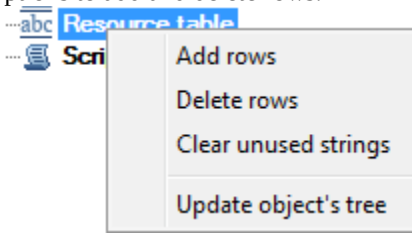
Important!	Collections are saved to the database, they can be used in other objects nanoCAD Mechanica 20.1. Forms and tables have separate stores. Collections created by editors or administrators are available to everyone and cannot be edited by the user (the collection editing buttons will be grayed out). User-created collections are only accessible to the user.

Table resources

Section *Table of resources* designed to create a multilingual resource strings details.



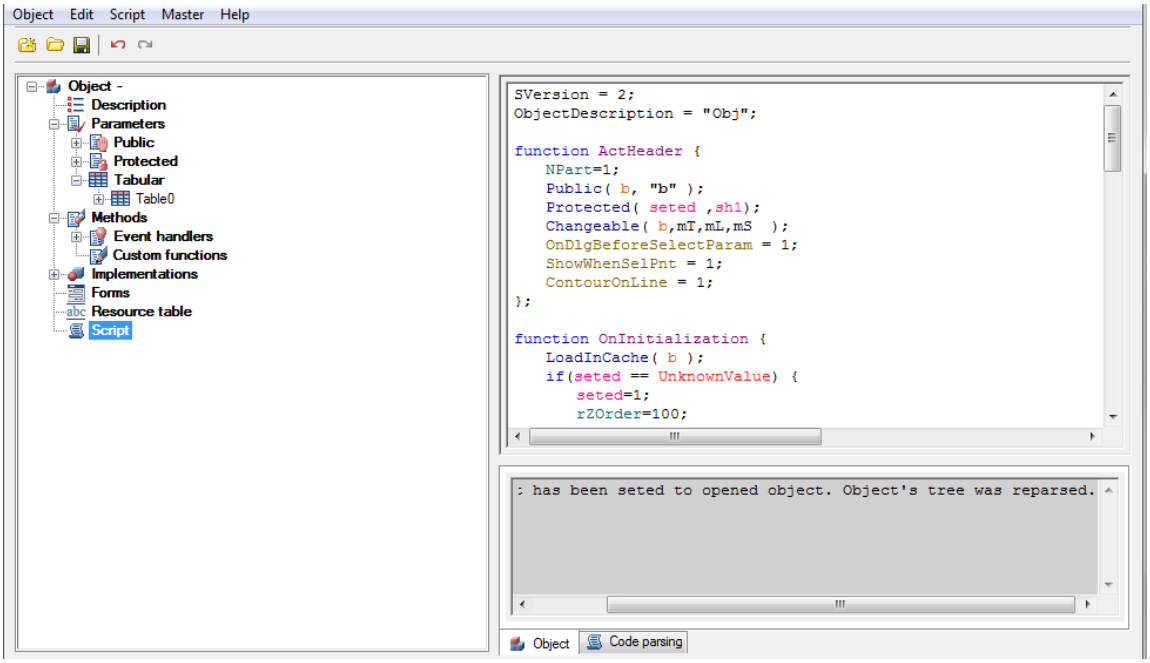
In the context menu section, there are options to add and delete rows.



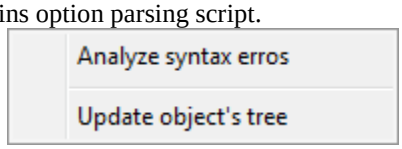
Each resource can have a name (*Label*), by which it will be referenced from the script (by writing the *@Resource Name*). The property is selected, the row that corresponds to the name of the resource (or its ID) and the column corresponding to the application language. Thus, we can ensure the correct operation of applications of different locations on the same database. Of course, the current local line must be completed, otherwise for non-Russian locale is selected resource corresponding to the English language.

Script

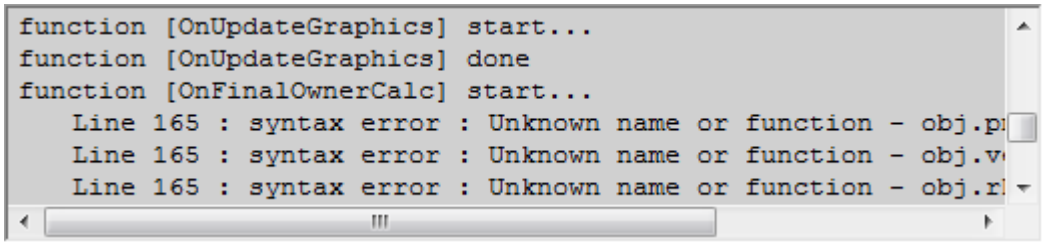
When switching to the section *script* in the workspace displays the script object.



In the context menu of the *script* contains option parsing script.



While parsing the functions performed consistently script object and passed to the notification system error messages. By double-clicking on a line in the notification area on the transition line of the script that contains the error.



Symmetry axis

Main menu: ***Mechanical - Standard parts - MechWizard -*** ***Symmetry axis***.

Ribbon: ***MechWizard -*** ***Symmetry axis***.

Toolbar: ***Symmetry axis (*** "MechWizard"***)***.

Command line: ***MCWIZSYM***.

Select the segment to set it as the axis of symmetry. Drawing can contain multiple axes of symmetry. Symmetrical layout sketch relating to a particular axis is determined automatically by the recognition of a parametric model.

Set Parameter

Main menu: ***Mechanical - Standard parts - MechWizard - Set Parameter***.

Ribbon: ***MechWizard - Set Parameter***.

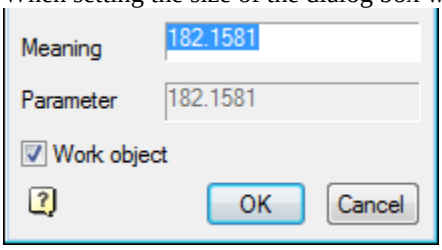
Toolbar: ***Set Parameter (*** "MechWizard"***)***.

Command line: ***MCWIZPARAM***.

Button ***Set parameter*** is used for setting targets sketch. Press the button and select the object (sketch element). This will bring up a dialog box, the form of which depends on which object was selected.

Setting size

When setting the size of the dialog box will look like:



Enter a *parameter* expression to control the size. The parameter can be set by a variable name or expression that is allowed to use the number and names of the variables related to mathematical and logical functions.

Important!	*Object parameter names should not coincide with the names of the teams nanoCAD (for example, can not be assigned to the parameter name HATCH or LINE)*

Expressions can be used the following functions:

- *arithmetic operators* +, -, *, /
- *trigonometric functions* ***sin(), cos(), tg()*** (the function argument is given in degrees)
- *inverse trigonometric functions* ***asin(), acos(), atg()***
- *taking the square root function* ***sqrt()***
- *exponentiation function*^ (recording format: x ^ y - x raising y)
- *absolute value* ***abs()***
- *rounded to an integer value* ***int().***

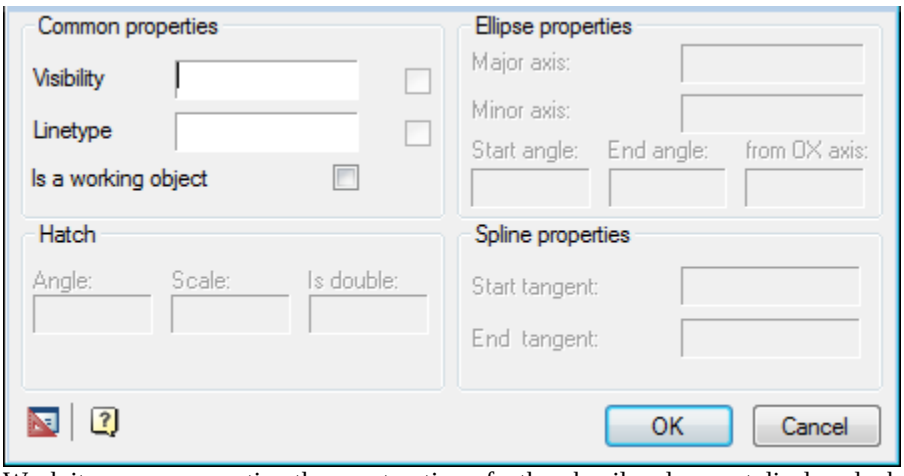
Logical operations:

- ***Min(x,y)*** - returns the minimum value of the argument;
- ***Max(x,y)*** - returns the greatest value.
- ***Iff (boolean, x, y)*** - returns the value of the argument x if the logical expression is true, or the value of y if the logical expression is false. Example of using the function, the result of which is similar to the function Min (x, y): Iff(x<y, x, y).

Valid logical conditions: >, <, == (equal), ! = (Not equal), > = (not less than), <= (not more).
Function within an expression can be nested.

Setting the graphic element

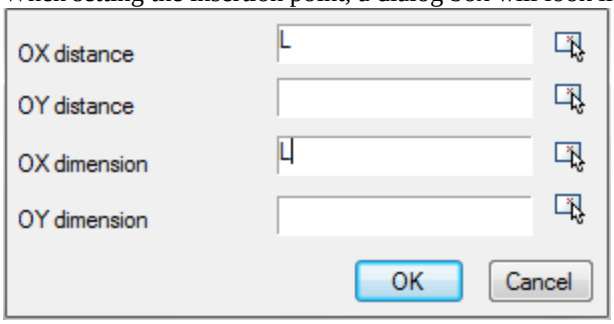
When you set a graphic element sketch (segment, circle arc, circle, spline or shading) dialog box will look like:



Work items are supporting the construction of a thumbnail and are not displayed when you insert a model from the database. Performance object can be any graphic sketch element: line, arc of a circle, etc.

Setting the insertion point

When setting the insertion point, a dialog box will look like:



Distance along the axes OX and OY determines the position of the handle in the object. Distance can be set expression or numeric value.
Parameter on the x-axis and OX and OY is the name of the script variable whose value will change when stretching the handle.
Button *specify parameter* lets take a parameter or drawing away.

Enter additional values

In the sketch may include additional expressions for calculation of parameters. These expressions should be written a single line text entry format each line of text as follows:
X = EXPR
where X - the variable name;
EXPR - an expression that contains the names of variables, mathematical or logical functions.
Examples of additional rows of values:

- a = b*2 - assigning the result of an arithmetic operation;
- alpha = asin(b/c) - the use of trigonometric functions;
- k = iff(a>b, 1, 0) - assign a value to the condition;
- d = min(m1, 100) - the assignment of the smallest of the two values;
- g10 = sin(alpha*2) + sqrt(b^2 - c^3) / min(sin(a), cos(b)) - nested functions.

Sequence computing additional rows values - from left to right and top to bottom. Additional values values recognizing parametric form must be selected along with other sketch entities.

Insert point

Main menu: *Mechanical - Standard parts - MechWizard - Insert point*.

Ribbon: *MechWizard - Insert point*.

Toolbar: *Insert point (* "MechWizard"*)*.

Command line: *MCWIZINS*.

The insertion point should always be shown on the sketch of the parametric model. Click to choose its position on the sketch.

If you try to check the model or recognize the form of a sketch in which the insertion point is not specified on the command line, you are prompted to "Select insert point". Then you need to specify the sketch position of the insertion point.

In the case of several insertion points in the sketch, recognition will be considered only of the one that has been specified later, and all others will be ignored.

Suppression contour

Main menu: *Mechanical - Standard parts - MechWizard - Supression contour*.

Ribbon: *MechWizard - MechWizard - Supression contour*.

Toolbar: *Suppression contour*.

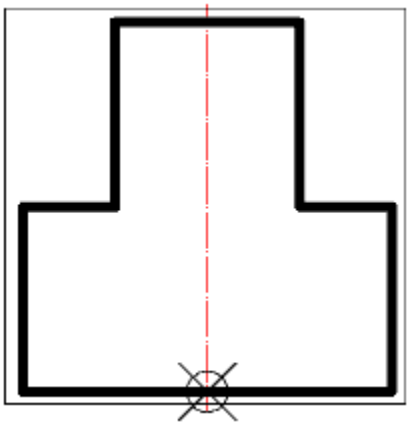
Command line: *MCWIZCONTOUR*.

Important! The command prepares the sketch for recognizing the execution type of the base object.

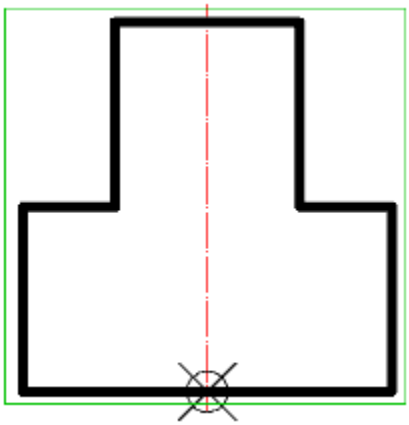
A suppression contour limits the area that the object overlaps with other graphics in the drawing. If the suppression contour is not specified, it is created automatically when the model is recognized by the external outlines of the part, and service objects are not taken into account.

Procedure

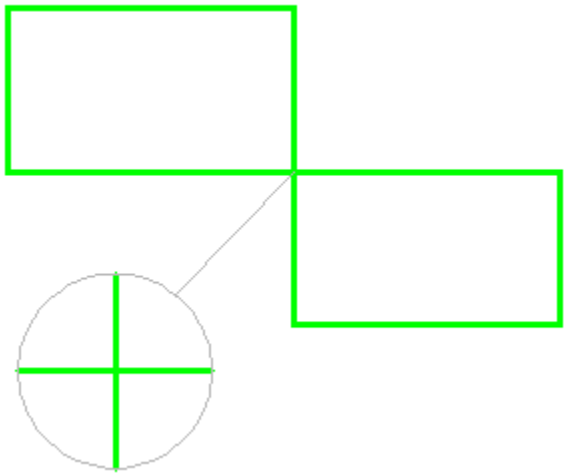
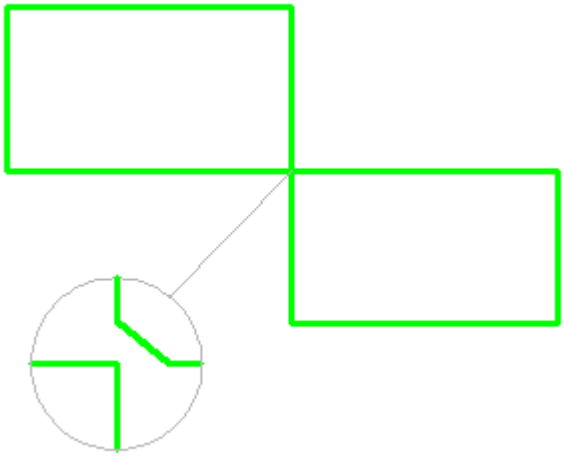
1. Draw a closed polyline on the sketch that matches the shape and size of the required suppression contour (in the figure, a rectangle around the sketch).



2. Call the command *"Supression contour"*.
3. Specify the constructed polyline. The contour is highlighted in green in the sketch.



A prerequisite for the contour is the absence of self-intersections. To avoid self-intersections for a compound contour, create the contour manually, leaving a minimum "isthmus" between the areas:

Wrong	Right
	

If the suppression contour cannot be created automatically (the outer boundaries of the object do not form a closed contour), then when the object is inserted into the drawing, a corresponding warning message will be displayed. However, errors in the suppression contour do not affect the applicability of the recognized object, and can only appear in the wrong overlap of other objects in the drawing.

Hatch contour

Main menu: ***Mechanical - Standard parts - MechWizard -*** ***Hatch contour***.

Ribbon: ***MechWizard - MechWizard -*** ***Hatch contour***.

Toolbar: ***MechWizard -*** ***Hatch contour***.

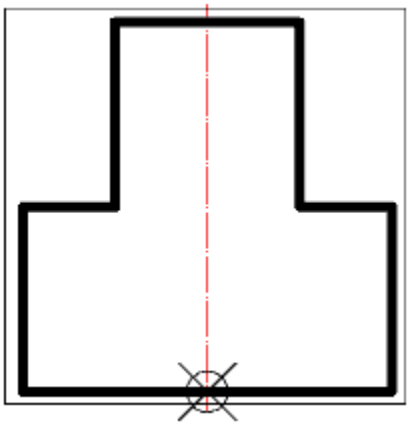
Command line: ***MCWIZSECTCONTOUR***.

Important! The command prepares the sketch for recognizing the execution type of the base object.

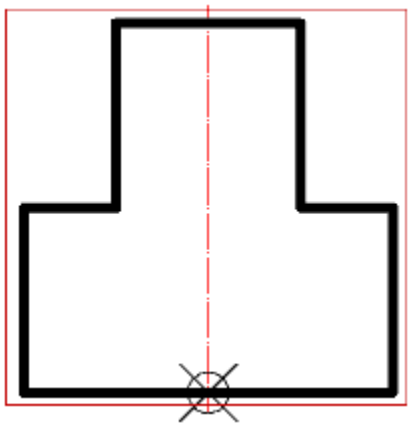
A hatch contour defines the shaded area of an object in the drawing.

Procedure

1. Draw a closed polyline in the sketch that matches the shape and size of the required hatch contour (in the figure, a rectangle around the sketch).

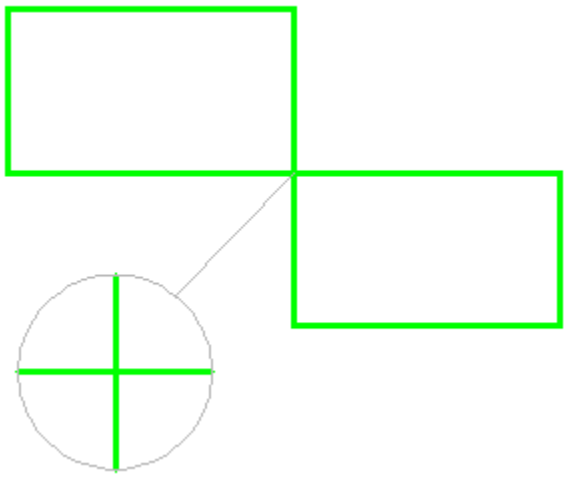
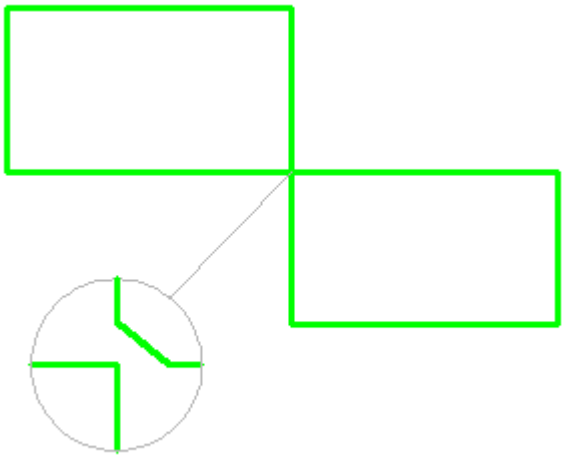


2. Call the command *"Hatch contour"*.
3. Select the constructed polyline. The outline will be highlighted in red in the sketch.



A prerequisite for the contour is the absence of self-intersections. To avoid self-intersections on a compound contour, create the contour manually, leaving a minimum "isthmus" between the areas.

Wrong	Right

Wrong	Right
	

Array

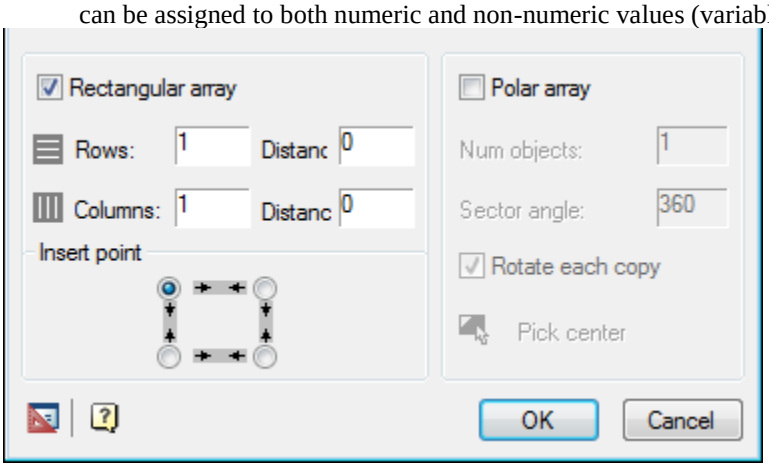
Main menu: *Mechanical - Standard parts - MechWizard - Parametric model*.

Ribbon: *MechWizard - Parametric model*.

Toolbar: *Parametric model (* "MechWizard"*)*.

Command line: *MCWIZARR*.

1. Create primitives of nanoCAD block and select the required dimensions in the sketch to determine its size and location.
2. Press the *array* and selected block in the drawing.
3. In the dialog box, select an array type and parameters that control the construction of the array. Parameters can be assigned to both numeric and non-numeric values (variable names and expressions).



Every single unit with a unique name that is inserted into a sketch can be used only in a single array.

Test Model

Main menu: *Mechanical - Standard parts - MechWizard - Test model*.

Ribbon: *MechWizard - Test model*.

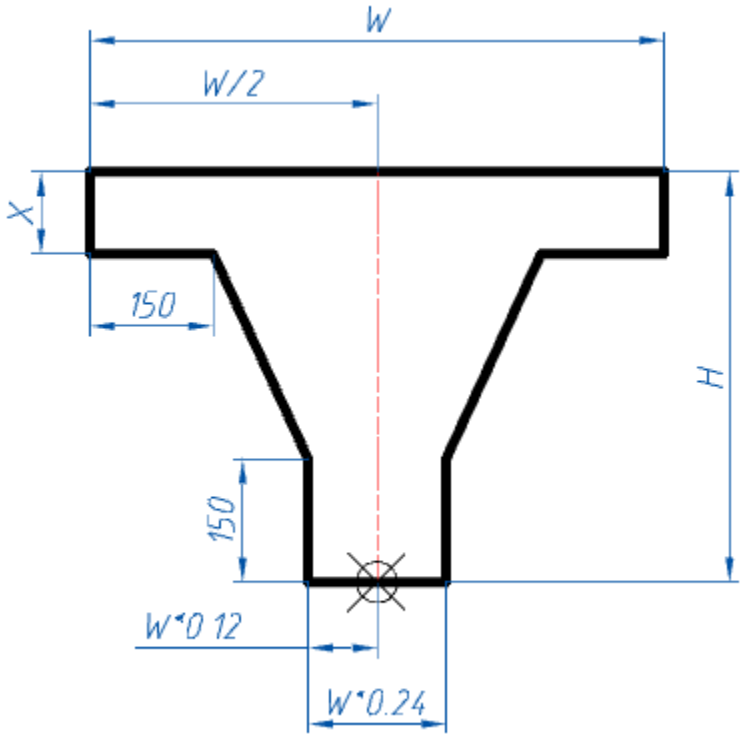
Toolbar: *Test model (* "MechWizard"*)*.

Command line: *MCWIZ*.

Checks sketch of a parametric model.
Highlight crossing window thumbnail and press Enter. If the sketch contains no errors, a message appears
If the sketch is not large enough, a message appears warning of an error.

Create sketch

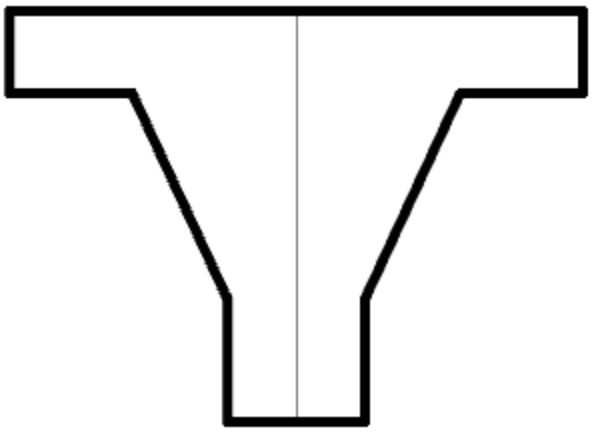
The sketch is the original model of the parametric view of the object.



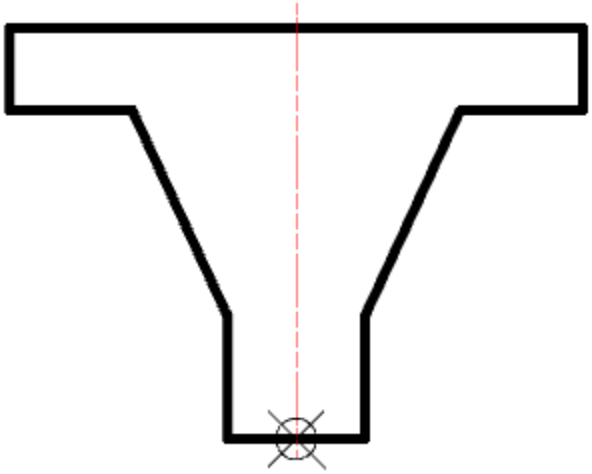
When creating parametric objects, the commands of the *"MechWizard"* toolbar are used.

Procedure

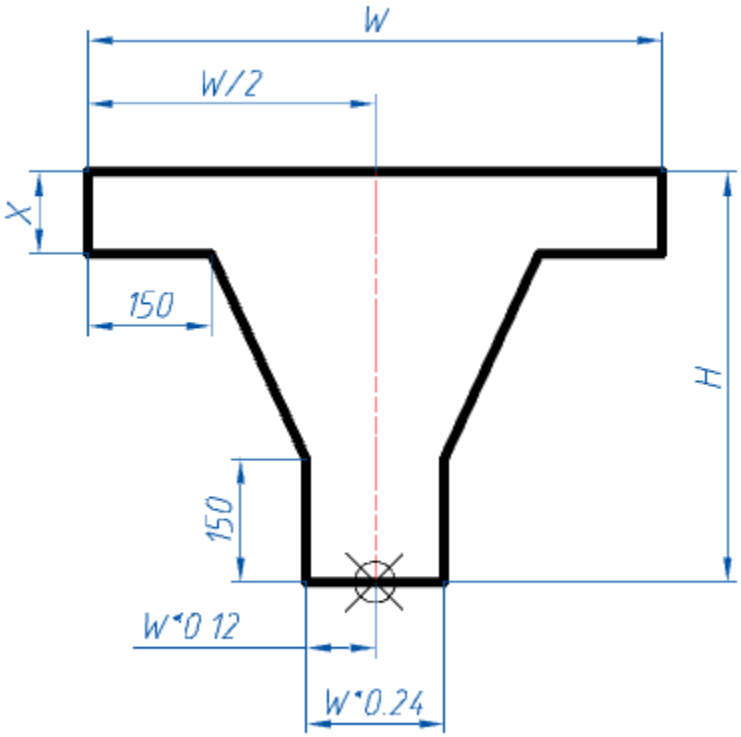
1. Prepare a sketch. For correct recognition, all graphic elements of the sketch (including hatching) must be drawn with the appropriate linetype. Similar types (solid thick and thin, dash-dotted thin and thick) should be distinguished by the color set in the nanoCAD Mechanicasettings.



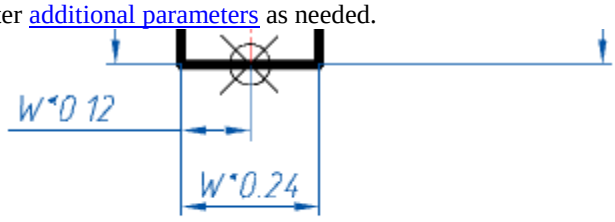
2. Set the insertion point, optionally specify the axis of symmetry, sketch element options, suppression contour, hatch contour and paramentric array.



3. Dimension and assign variables and expressions to them.

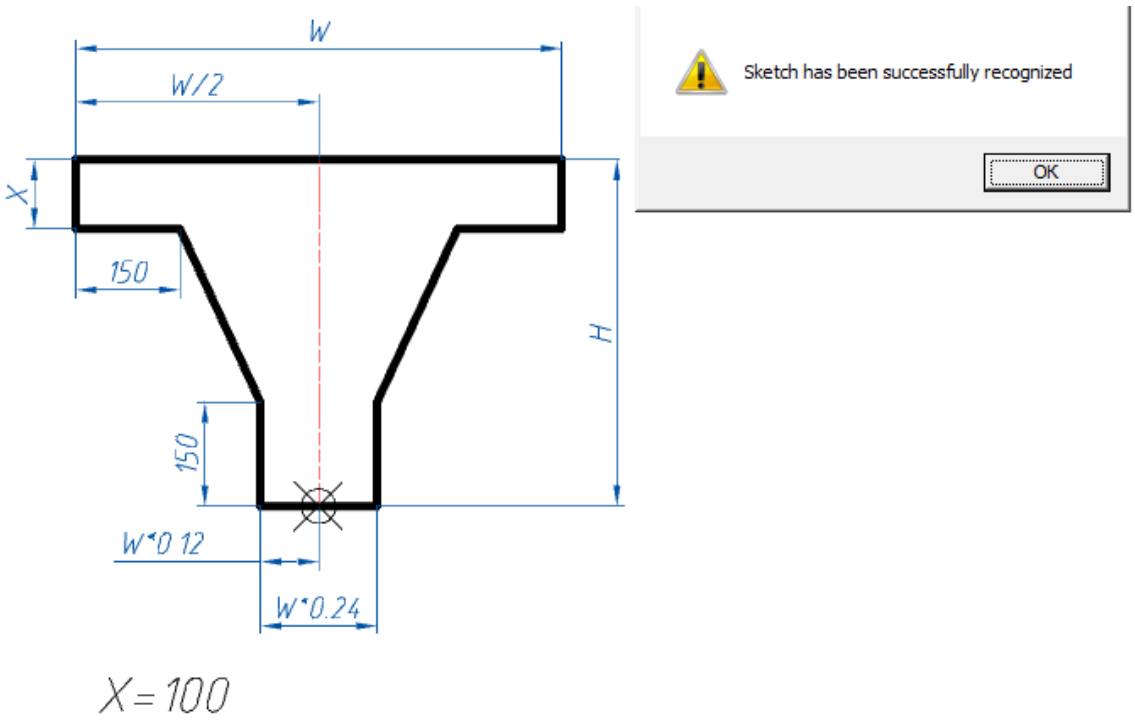


4. Enter additional parameters as needed.



X = 100

5. Check the sketch and make sure there are no errors.

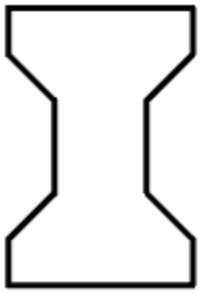


Examples

Creating a template layout tiles

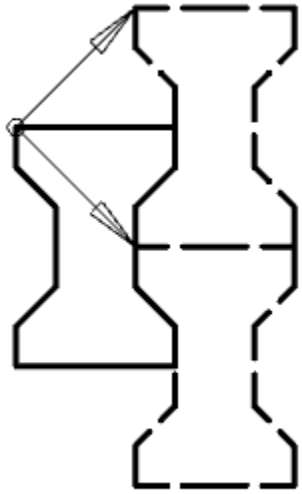
Example of creating a template layout tiles.

1. Create a sketch of the object. If the size of the tiles will not change, then the sketch is not required to specify the size.
   Terms and method of creating a sketch of the parametric object described in section Rules for creating sketches .



2. Create an object and save it to the database. In a script, you must set the variable "type" (strTheType = "Plate")

3. Position the ***Bone*** in the drawing, insert the base of the two vectors and create a group. (see. Create user group). These vectors determine the pace and direction of replication of the original group. The basic point of the vector must coincide with the insertion point of the object.



If only one vector will create one line layout in any mode selection area.

Developer's Guide Database

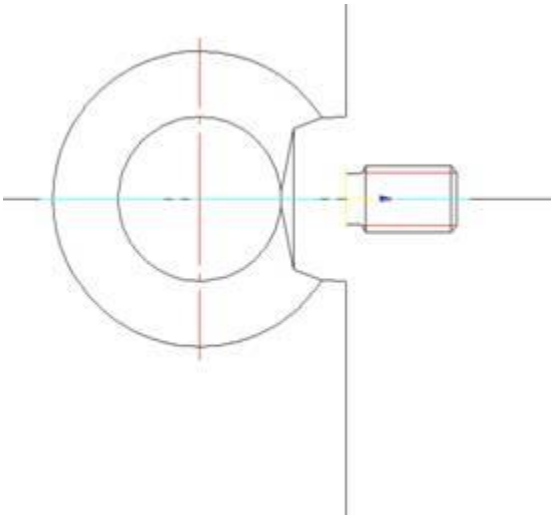
Terms

Functions of the standard objects

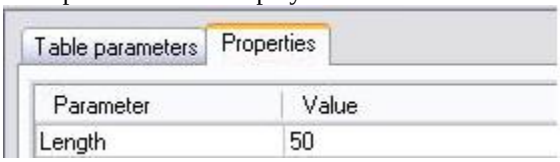
The database of the standard details was created to help a constructor to create graphics. Use the details from the database to create graphics. The standard details behave intelligently in the drawing so that the number of additional actions in the development environment are decreased.

What can the standard details do

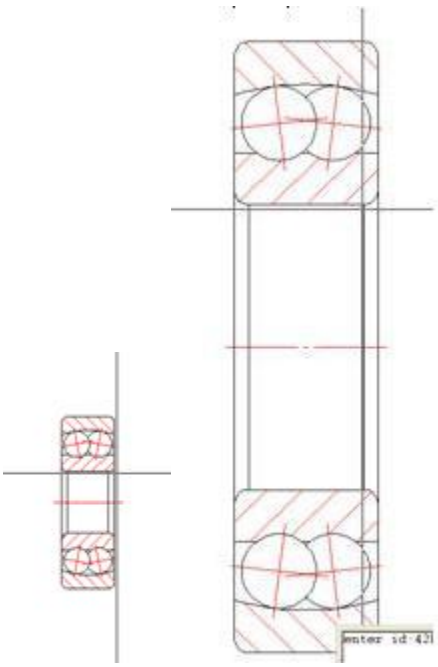
Insertion from the database. The database is organized as a hierarchical structure of objects having similar functions. (For example, the details of arbors, pipelines etc. are grouped). There are also different types of objects: the groups and the markers of the objects. You can search for the required standard in the database. When inserting a detail, specify the insertion point and the direction of the detail's local X axis (position and orientation of the detail). The details can be fully displayed or displayed as a representation of the base plane. The position of the detail is specified by a click, the orientation by a double-click.



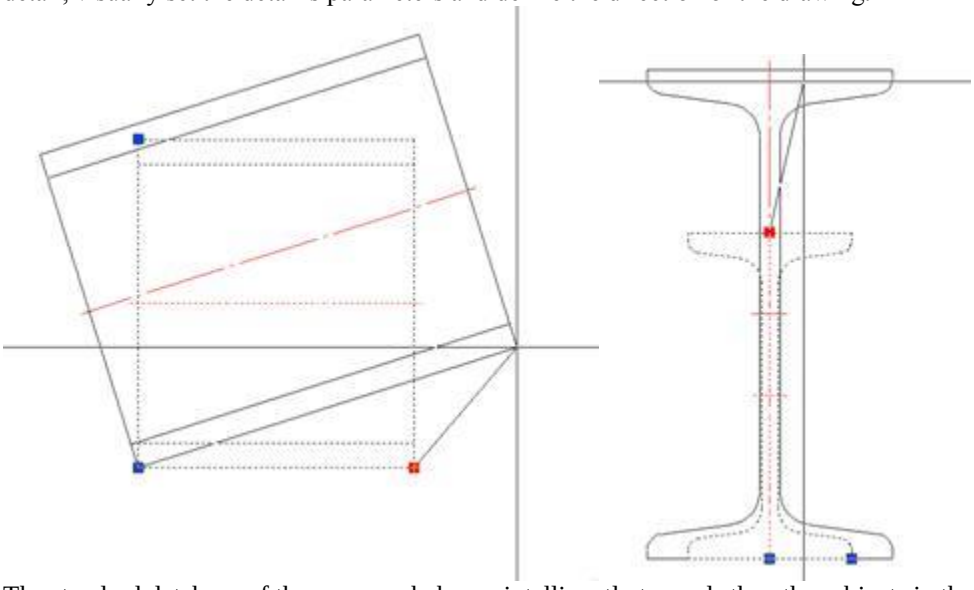
If GOST of the detail contains the table of the parameters, it is recommended to limit the dimension values for the elements of the construction. One detail can have several tables. (For example, if the parameters of one element of the construction do not depend on the parameters of the other elements).
If the parameters of the object's graphic are not specified by GOST, the user specifies their values during insertion. Such parameters are displayed in the second tab of the insertion dialog. Such property is the length of the I-Profile.



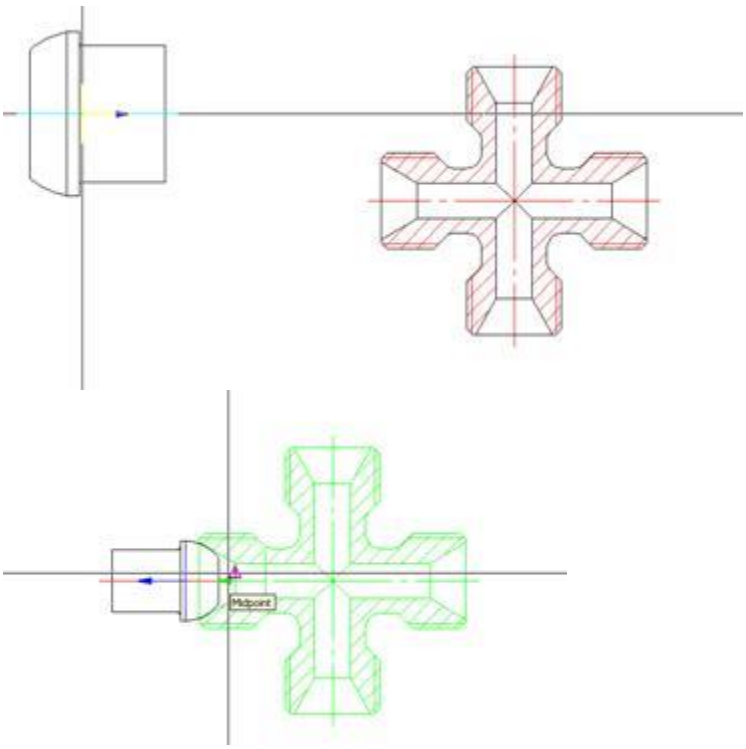
Sometimes it is convenient to specify the parameters in the drawing according to the parameters of other details. In this case, dynamical selection of the detail's parameters is used. This mode is selected with the *Selecting parameters dynamically* button. The parameters are selected towards the insertion point, after the insertion point is specified. Dynamical selection can be executed from the table or by using other parameters. The values, selected dynamically, sometimes set additional constraints (the maximum and minimum values) to provide correct drawing.



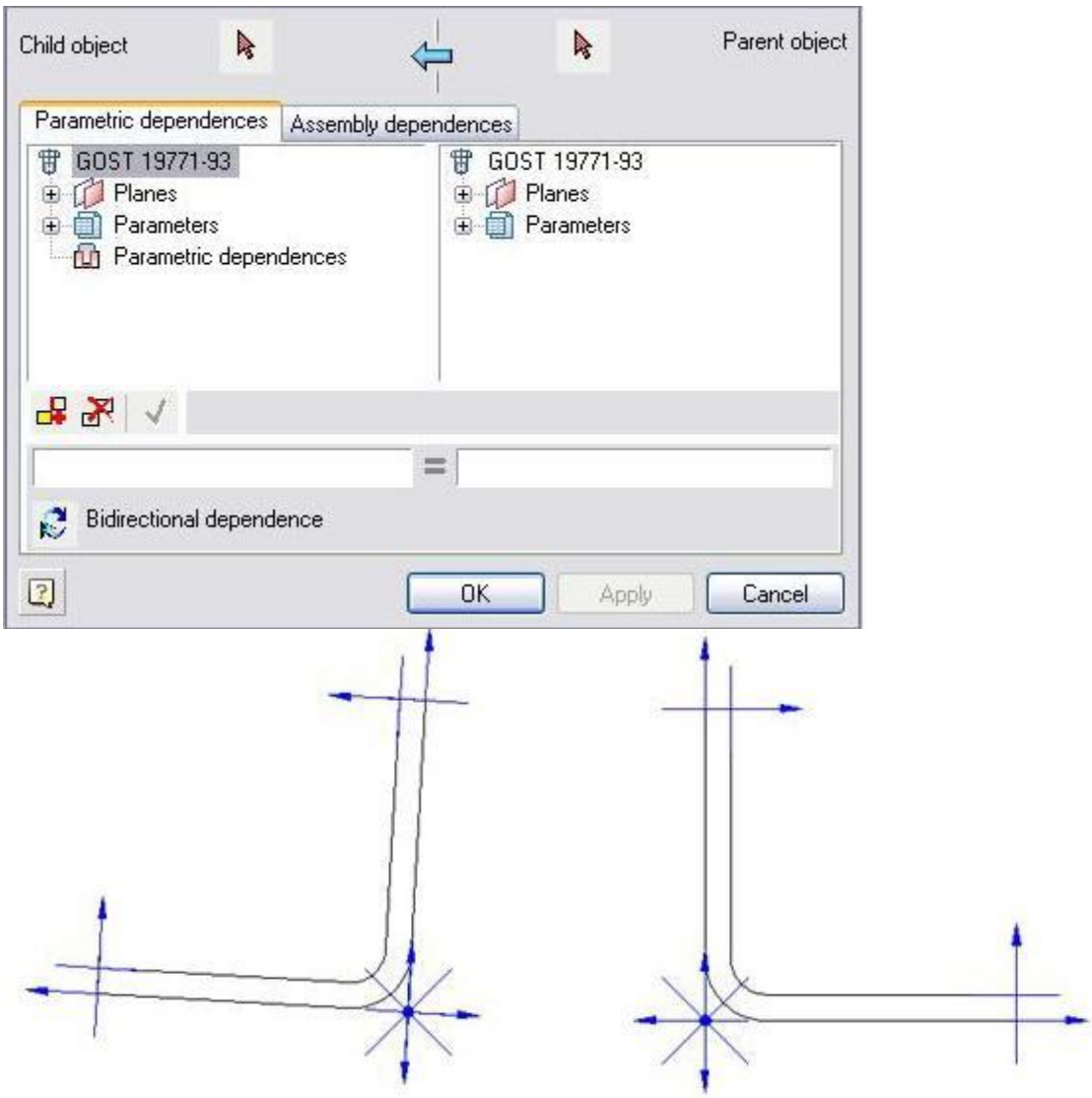
The grips are very useful tools to edit objects, and if the object has grips, they add additional functionality to the object. You can move, rotate and mirror the object, using the grips; you can also change the standard size of the detail, visually set the detail's parameters and define the direction of the drawing.



The standard database of the program behaves intelligently towards the other objects in the drawing. For example, when the pipe fitting with the internal cone is inserted, the nipple, inserted after it, will be automatically connected with the fitting.



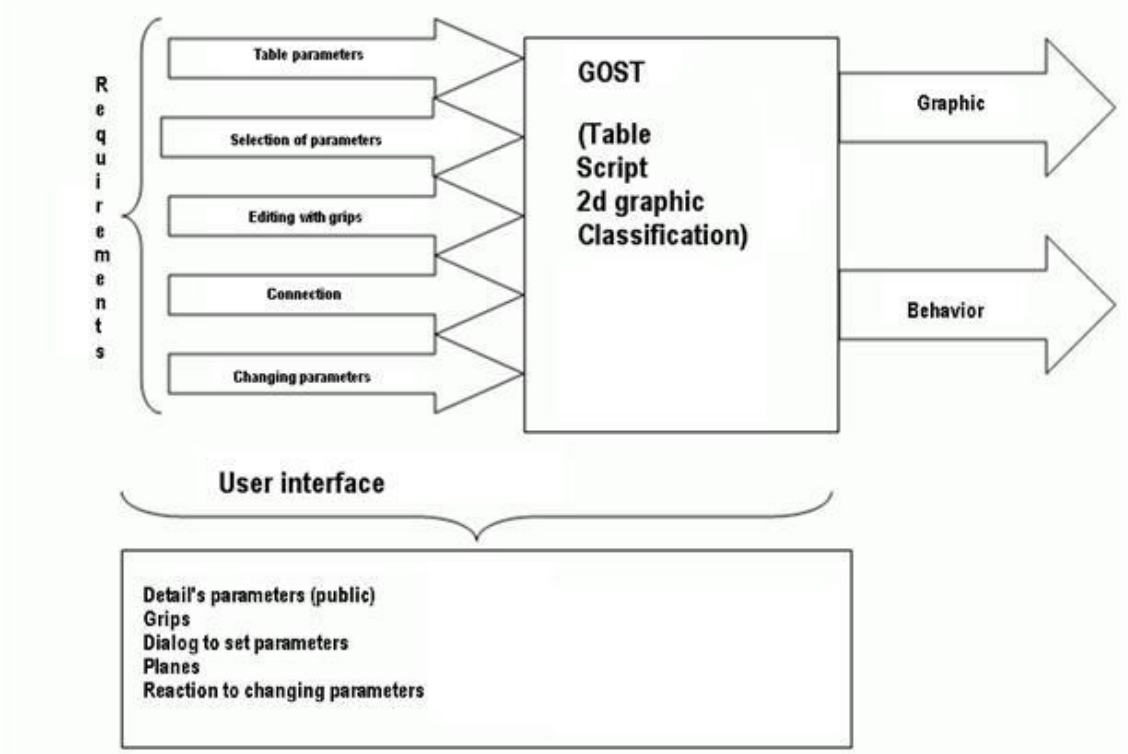
It is achieved by specifying parametric and geometric dependences between the details. The dependences for the details should be set according to the standards.
The user can set the dependences for the objects already placed in the drawing (for example, if the automatic connection is not specified). These dependences are custom and are stored in the drawing.



The standard databases can also have additional functions that are used in the other tools.

Abstract model of the detail

A detail should be a black box for the user.



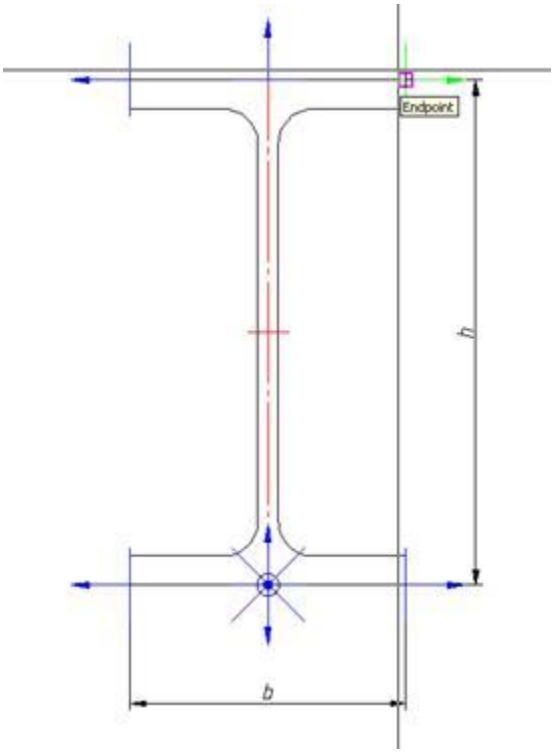
To specify a nominal size and graphic of the detail, use its parameters. It is necessary to specify these parameters, and not all parameters should be available for the user, but they should be known for graphic recognition. There are the public parameters, which the user can see, and the protected parameters, which define graphic, but the user cannot see them. These parameters can have values, defined by GOST, or the user can set their values.

When editing the detail with grips, you should define their number and the parameters they specify; define grips to specify the insertion point and the detail's orientation.

The view of the insertion dialog (or its data form) is also added to the user interface of the detail. Its view and behaviour is the black box of the detail, the form and the dialog should be convenient for the user to specify the nominal size and the parameters of the detail.

The geometric dependences are set by the detail's planes, which are a vector and a point together. The number and the position of these planes should be defined to set the dependences. The planes can be changed if they specify the detail's position according to the dependence with another detail, and unchanged (if another detail is linked to this plane); they do not change the insertion point of the detail according to the dependences.

The planes are defined according to the local coordinate system. The insertion point is its origin (pntOrigin). The direction vector (vecDirection) and vector in the drawing's plane (vecPlane), orthogonal to it, are X and Y axes. The mathematically highlighted plane will be in the b/2 horizontal and in the h vertical distance from the insertion point. The Vector of the plane will be collinear to the vecDirection direction vector. It is enough to specify the new position of the detail if the plane is changed according to the dependence.



When changing a detail's parameters (when there are parametric dependences or when specifying the values of the parameter in the dialog), it is necessary to define new values for the parameters, select new values from the table, change the planes etc.
To define discrete values of the parameters, select them from the table. The table is organized in such a way that one row corresponds with one nominal size and the columns of the table contain the detail's parameters. The parameters can have String, Integer and Real types. The parameters have a comment, which is used in the insertion dialog. Columns and rows have unique identifiers, saved during the import and export processes of the table to provide the correspondence of the nominal sizes and already existing details in the drawing (selected rows and parameters in the table when editing the object).

Object's graphic

To display the object in the drawing, the orthogonal views are used. The views have different types: general, views with cuts and simple views with dimensions. The views with dimensions are system and hidden. Their graphic is realized when the dimensional view is inserted from the context menu.
The local coordinate system in 2D views is set that the X-axis is directed to the right and the Y-axis is directed up in the frontal view. The direction of the coordinate axes is defined by the projective dependence in the other views.
The parametric recognition defines the parameters and the relative position (horizontality, verticality, perpendicularity and parallelism) of the graphic primitives and their interaction with each other (intersection points, tangents etc.). It is necessary to specify the insertion point of the detail (origin of the local coordinate system). It is necessary to describe a view as a text (graphic parameterization).
Recommendations for creating 2D views.

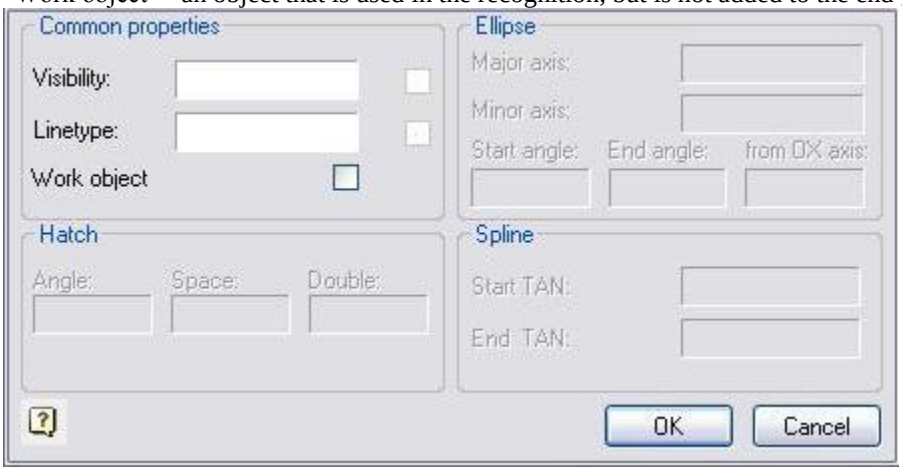
- Use the object snap for endpoints and intersections.
- Use the polar tracking and ortho modes.
- Arc segments and tangent arcs should be dimensioned.
- Use the helping geometry for recognition (circles in the centres of the tangent arcs).
- Dimension should be clearly set.

- Sometimes to create a difficult graphic, the view is separated into parts, and null distance is set between the parts.

The purpose of the recognition is to define the position of all primitives towards the insertion point. The parameters and dimensions are set from it.
The requirements to graphic are specified in the particular recommendations.
To make a 2D graphic parametrically controlled, set the detail's geometric parameters as dimensions instead of the nominal sizes. Some parameters are related to graphics.
Common properties: "Visibility" - is a logic expression or variable, defining whether or not to draw the primitive.
Linetype:

- 0 - main.
- 2 - thin.
- 4 - dashed.

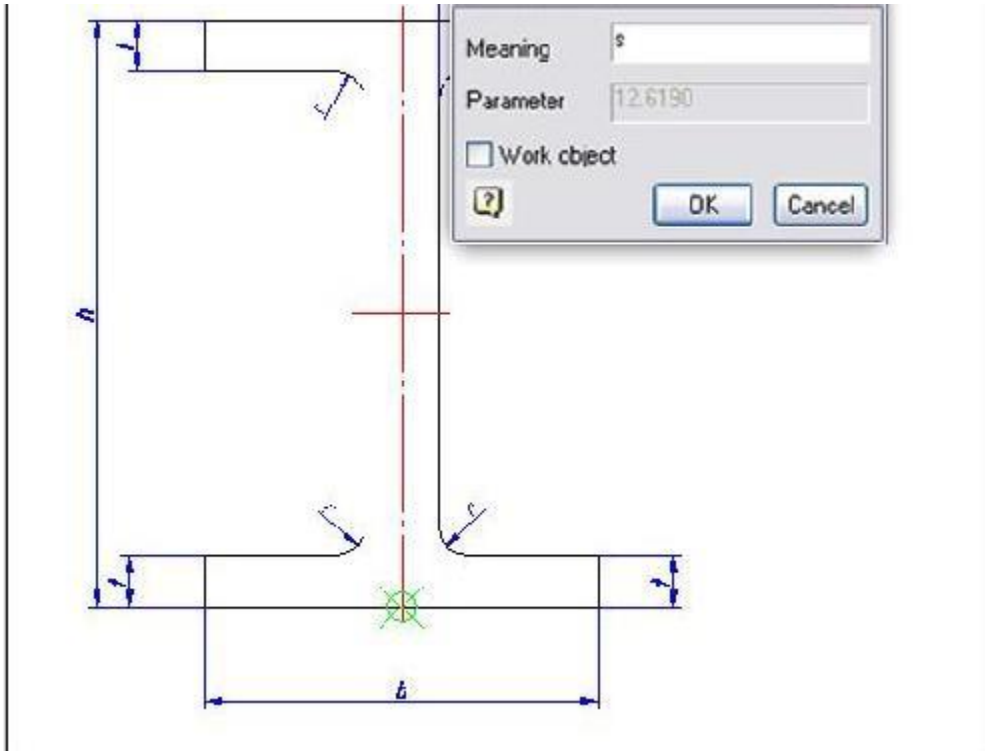
“Work object” - an object that is used in the recognition, but is not added to the end graphic. It is a helping drawing.



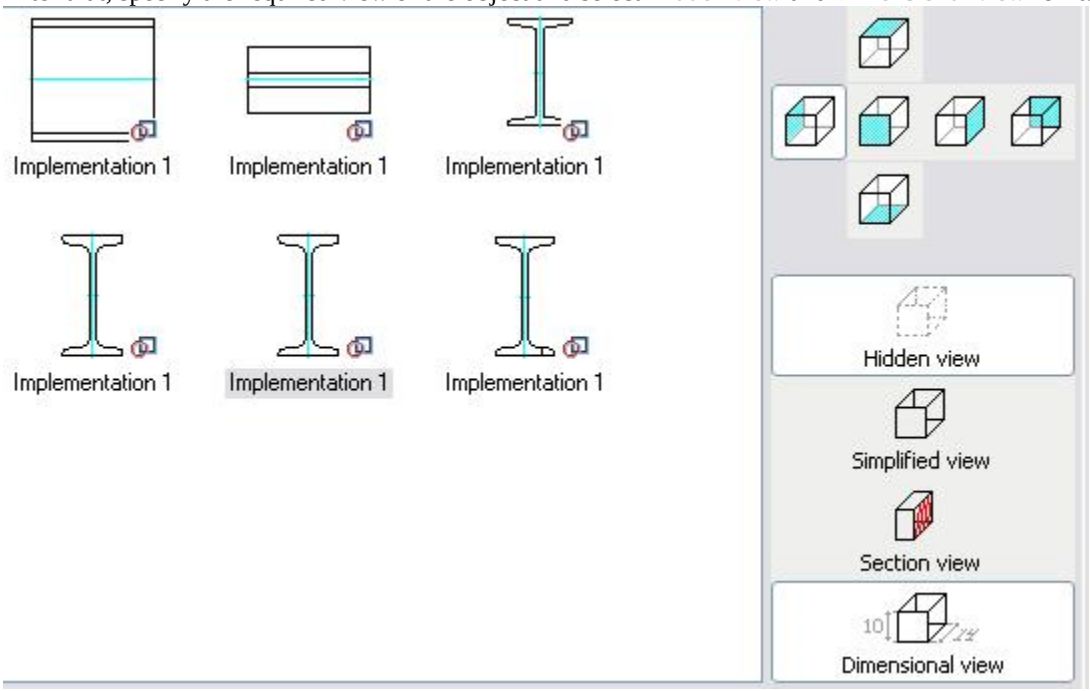
Parameters of hatch - the angle of slope, hatch space and double hatch feature (logical condition)
Parameters of ellipse - the major axis of the ellipse and the minor axis of the ellipse. The start angle and end angle of the ellipse - if the elliptic arc is drawn. The sloping angle to the OX axis is an angle, on which the local coordinate system of the ellipse is relatively rotated to the coordinate system of the object’s view.
Parameters of spline - the start and end tangent angle. All points of the spline should be dimensioned.
If the ***hatch contour*** does not follow the required contour, a closed polyline is created with the parameter - hatch contour.
Sometimes it is necessary to set a ***contour of suppression*** for the detail (if the automatically created contour does not follow the requirements of the detail). In this case, the parameter of the suppression contour is also set for the closed polyline.

Creating the dimensional views of the standard details

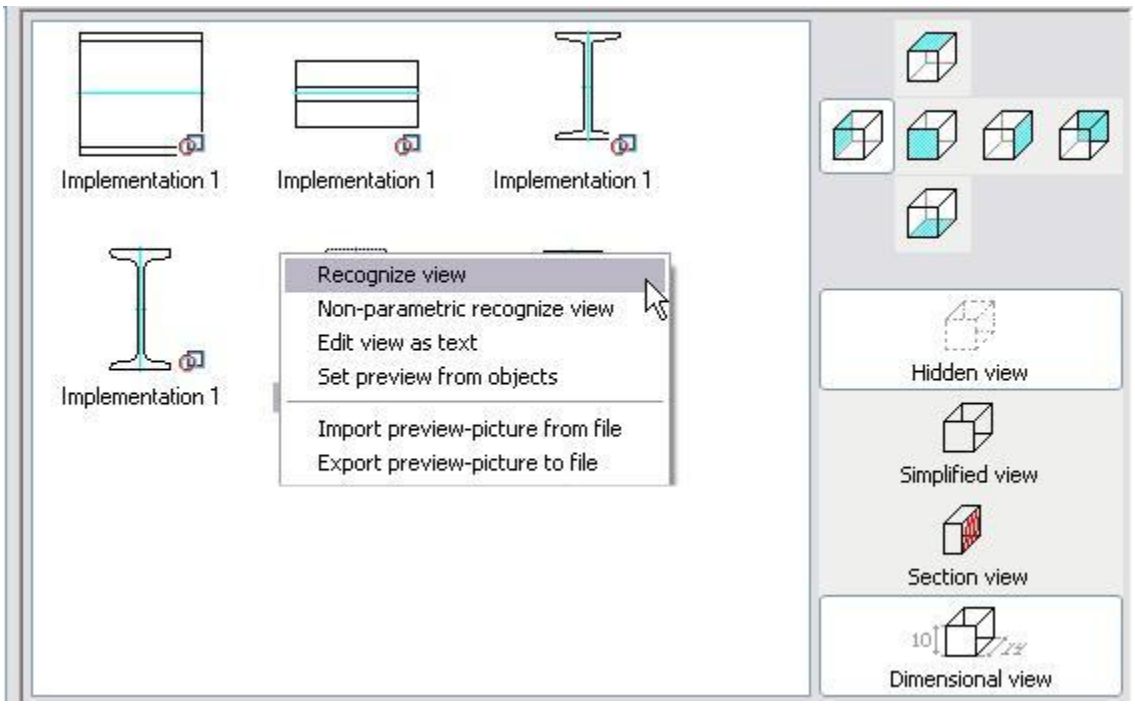
The same graphic is used to create the dimensional views of the standard details, as is used to recognize the main view.
Add only the required elements to the graphic of the dimensional view.
To recognize the dimensions as constructive objects, switch off the *Work object* checkbox in the ***Parameter*** dialog.



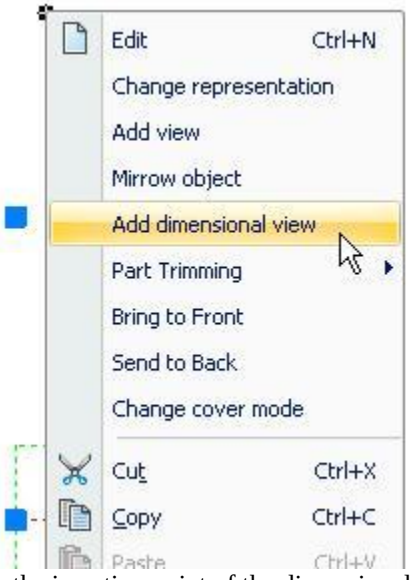
After that, specify the required view of the object and select *Hidden view* and *Dimensional view* for it.



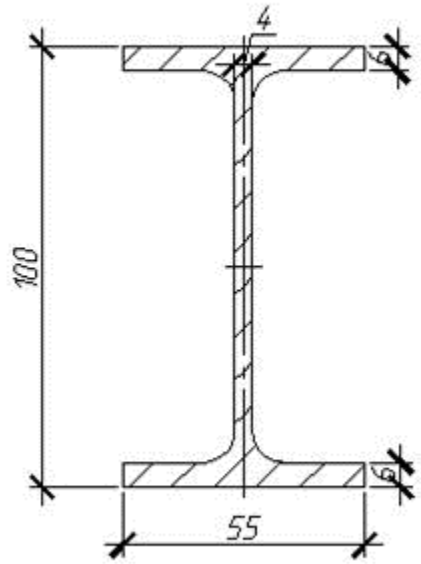
Select the *Recognize view* from the context menu.



Specify the graphic in the drawing and confirm the selection. Save the object.
To test the dimensional view, insert the object with the created direction of view. From the context menu of this view, select the *Add dimensional view*.



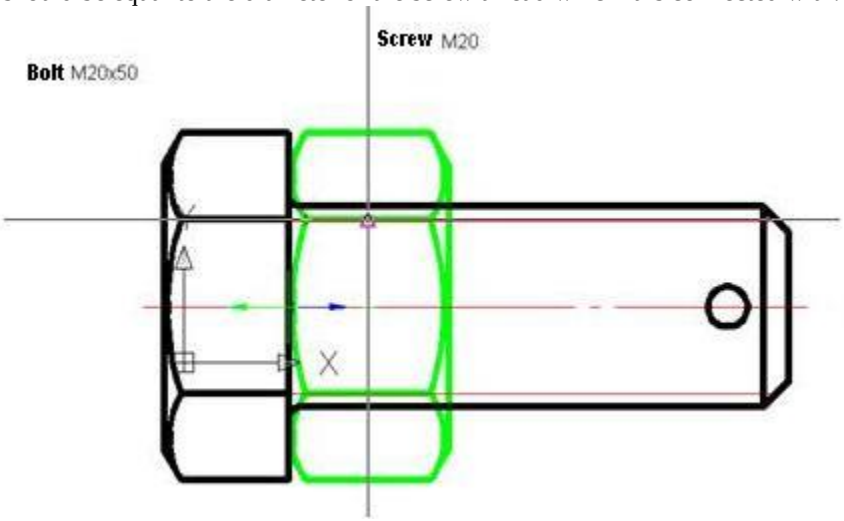
Specify the insertion point of the dimensional view.



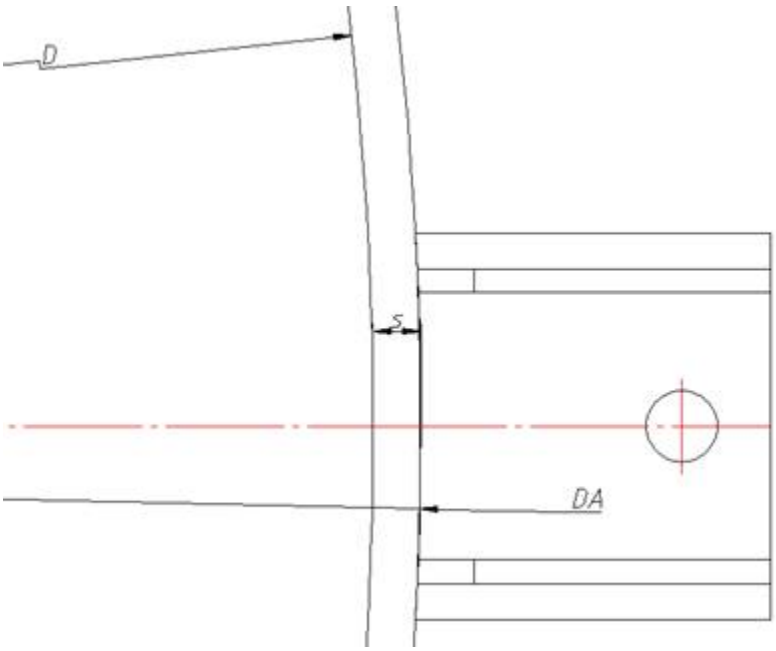
Note:	Dimensional views are inserted with the current dimension style and symbol scale.

Details in the database. Dependences

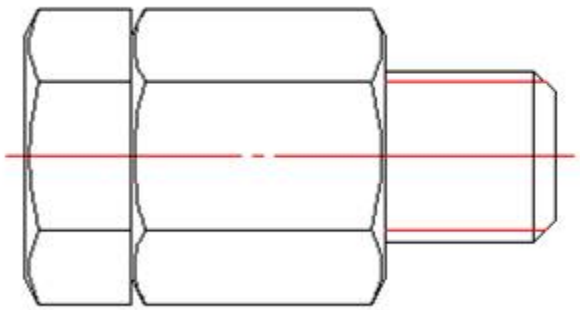
The details can be connected with each other. How does it happen By setting the parametric and geometric dependences. The parametric dependence - an expression meaning that one parameter of one object equals one parameter or the arithmetic expression of the parameters of another object. For example: the diameter of the bolt thread should be equal to the diameter of the screw thread which it is connected with.



The thread diameter parameter (dr) of the screw is equal to the thread diameter parameter (dr) of the bolt (dr=dr).



When setting the support for the shell, the dependence of the parameters' arithmetic expression is set: the diameter of the support is equal to the sum of two shell widths and the inner diameter of the shell. DA=D+2*s.
When setting the dependences there are a child and parent objects, the dependences can be one-directed and two-directed. The one-directed dependences affect only the child object (change of the parent object affects the child object, but change of the child object does not affect the parent object), the two-directed dependences affect two objects, change of one object affects the other object.
The geometric dependence between the objects is realized by movement and reorientation of the objects. The new position of the object is calculated according to the new position of the object's planes.
The views of geometric dependences:
INSERT - the combination of the insertion points and planes' vectors. The function has a parameter - offset - the distance between planes' points of the detail along the OX axis.



MATE - the collinearity of the direction vectors if the abscissae of the details are relatively equal. There is a parameter - distance along the OX axis.
AXIS - the combination of the direction vectors with the unfixed coordinates along the OX axis. The parameter of the dependence - distance along OY axis.
The combination along direction (angular dependence) - the collinearity of the direction vectors with the unfixed relative coordinates.

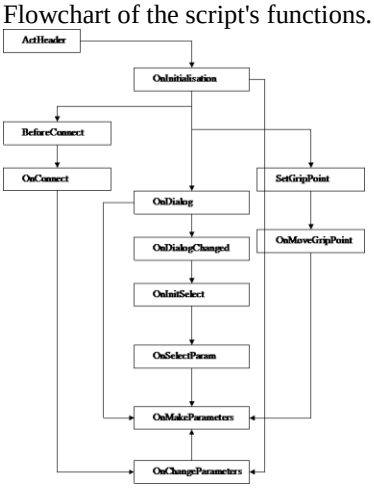
The geometric dependences can be codirectional and opposite directional according to the orientation of the local vector of the OX axis.
There is no need to connect all objects with all objects and that is why it is necessary to set the dependences automatically. The classification of the objects is set to define what details should connect with each other. The classification is made according to the number class (used in the program tools) and the public parameters of the object. These are strTheName, strTheType and strTheSubType variables. To connect a bolt and a screw there is a check of these variables and, if they coincide, the dependence is set. Fasteners are connected only with fasteners, not with pipe fittings, but both types have the thread diameter parameter. Every object has a unique identifier, used for the insertion from the browser.

Suppression contour and display order

Graphics, like the standard details from the database, have a suppression contour - a closed area, hiding the graphical primitives and other details. (For example, the contour of the screw overlaps a line of the bolt.).
Display order is set by the rZOrder variable. Details with an rZOrder value greater than the rZOrder value of other details will overlap them. This property works with the intervals divisible by 2000. The object from the database can have a suppression contour formed with its outer boundary, a specified suppression contour or have no contour at all.

Eventive Object's model

Flowchart of the script's functions.

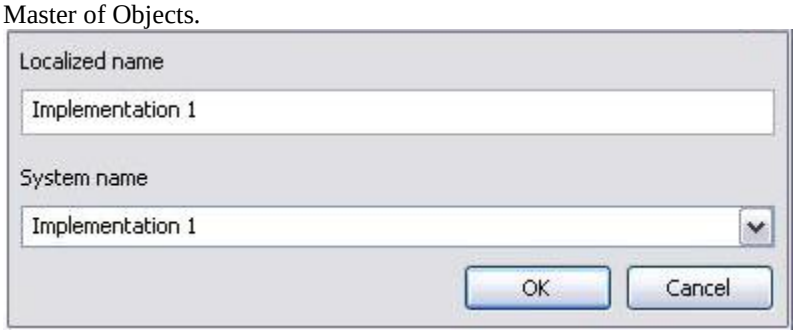


ActHeader - the description of the object's parameters. It includes the description of the public and protected parameters, specification of the reserved variables and description of the subobjects.
During every activation of the object, the OnInitialization function is executed. Using set variable the single setting of the object's parameters needs to be checked.
The OnDialog function is activated after specifying the insertion point and direction from the context menu during the dynamical selection of the parameters.
The OnDialogChanged allows you to add interactivity in the insertion dialog of the object. It is executed every time the parameters are changed in the dialog. In the function's body, it is possible to hide the controls to provide the correct selection of the object's parameters.
The OnInitSelect is executed before the dynamical selection of the parameters. There is a tooltip string (for example, the current value of the length).
The OnSelectParam is executed during the dynamical selection of the parameters. It defines what and how parameters should be selected. It usually contains checks of request number for the selection of the parameters and checks of the current view.

The OnMakeParameters function is final. All final calculations of the parameters are executed and the classifiers and planes are specified. As this function is executed after all object events, the constraints for the parameters and the relative selection of implementations are added.
The SetGripPoint function sets grips for the object. Their number and position are set there.
The OnMoveGripPoint function receives the number of the edited the NMovingGrip grip, the relative position of the grips is processed and the object's parameters are changed according to it.
The BeforeConnect is executed before the dependences are set. The previous dependences are reset and a tooltip string is specified when selecting the object to connect.
The OnConnect function is executed when one object is connected to another object. It contains a check of the object's classifiers and operators for setting the dependences according to this check.
As the result of the OnConnect is a new value for the connected object, this is received as a new object. The OnChangeParameters function is executed when the object's parameters are changed according to another object's parameters using the parameters toolbar. This function determines the object's reaction to changing its parameters.

Localization mechanism

The following is a mechanism of localization:
The object's implementations are activated by their system names, which can be changed in the context menu of the Master of Objects.



Activation of the implementation in the script:

```
if (strDesignName == "Implementation 1") {
  strPartName = @BOLT_M+dr+step+" # x"+L+@___GOST_7798_70;
};
```

If the use of the string values in the local coding is required in the script, change them with the resources.
The resource is a reference to the table row containing the list of the string values according to the current application language.

RID	Label	English	Russian	German	Polish	Czech
8	ROD_LENGTH	Rod length	Длина стержня	Länge	Długość trzpienia	Délka dříku
9	THREAD_DIAMET...	Thread diameter	Диаметр резьбы	Gewindedurchmesser	Średnica gwintu	Průměr závi
10	WORK_PLANE_1	Work Plane 1	Рабочая плоскость 1	Arbeitsebene 1	Płaszczyzna konstru...	Pracovní ro
11	SMALL_THREAD_...	Small thread step	Мелкий шаг резьбы	Feingewinde	Mały skok gwintu	Krátký závit
12	BOLT_OK	Bolt OK	Болт Ok	Bolzen OK	Śruba OK	Šroub OK
13	BOLT_HEAD_WID...	Bolt Head Width	Размер под ключ	Schlüsselweite	Szerokość łba śruby...	Šířka hlavy
14	THREAD_LENGTH	Thread Length	Длина резьбы	Gewindelänge	Segment gwintowany	Délka závit
15	MASS	Mass	Масса	Masse	Masa	Hmotnost
16	SELECT_LENGTH	Select length	Выберите длину	Länge auswählen	Wybierz długość	Vyberte dél
17	BOLT_M	Bolt M	Болт M	Bolzen M	Śruba M	Šroub M
18	___GOST_7798_70	# GOST 7798-70	# ГОСТ 7798-70	# GOST 7798-70	# GOST 7798-70	# GOST 7
19	BOLT_3_M	Bolt 3 M	Болт 3 M	Bolzen 3 M	Śruba 3 M	Šroub 3 M
20	BOLT_4_M	Bolt 4 M	Болт 4 M	Bolzen 4 M	Śruba 4 M	Šroub 4 M
21		Thread diameter	Диаметр резьбы	Gewindedurchmesser	Średnica gwintu	Průměr závi
22		Detail length	Длина детали	Detaillänge	Długość	Délka
23		Thread Length	Длина резьбы	Gewindelänge	Segment gwintowany	Délka závit

To use the local resource in the table, call its name (Label) with "@" prefix.

```
Public(
  L, @ROD_LENGTH,
  dr, @THREAD_DIAMETER,
  WP1, @WORK_PLANE_1,
  swTr, @SMALL_THREAD_STEP,
  rScrewOk, @BOLT_OK,
  s, @BOLT_HEAD_WIDTH,
  b, @THREAD_LENGTH,
  massa, @MASS
);
```

When the script is executed in the specified location, the required values will be selected instead of the @<resource name> references:

```
Public(
  L, "ROD_LENGTH",
  dr, "THREAD_DIAMETER",
  WP1, "WORK_PLANE_1",
  swTr, "SMALL_THREAD_STEP",
  rScrewOk, "BOLT_OK",
  s, "BOLT_HEAD_WIDTH",
  b, "THREAD_LENGTH",
  massa, "MASS"
);
```

Requirements of the objects

Requirements of the standard details

The required functionality means that the objects can be inserted from the database, their parameters can be specified, changed, the details can be connected with each other etc.

The details should *satisfy the standards* (GOST, ISO, etc.) and engineering practice. It is required to follow not only the table parameters but also ESKD, Mechanical etc. requirements. The details should not break the rules of design and product documentation.
Standardization. To make the creation and classification process of the standard details easier, use similar graphics for the details that have similar functions.
As the database is used in multilingual applications, it can be *localized*; the names should be correctly displayed in English or in Russian.
The situation, where several people have developed ONE GOST, is very common. The standardization helps to transfer the product from the developers to the users. Everybody should understand how the scripts are created.

Requirements of the objects. List of the requirements. Graphic and script creation of the objects from the database using MechWizard

The requirements should be applied for the newly created and edited objects from the database.

Object's properties

The name of the object should contain the name and number of its standard (for example, "GOST 7798-70", "ISO 657/16").
The comments attached to the name should contain the object's name according to the standard (for example, "Hot-Rolled I-Shape with Sloped Flange").
The specification string should be specified by the strPartName variable.
All the comment fields of the table parameters from the DBFLD and DBINF sections of the UniDialog function should be filled in with the descriptions. All the comment fields of the table should be filled in.
It is recommended to group the table fields so that they have the main parameters grouped to the left.
By default, the name of an implementation is "Implementation1" (without space), if another name is not specified by the standard.
If the Russian standard is used, GOST is set in the properties (if the detail was made by OST, TU or ATK).

Parameters

When creating the parameter's name you should take into account:

1. Similar parameters of the objects from the database (for example, dr - thread diameter, p - thread pitch, L- rod length).
2. Names of parameters in the standard (for example, D1 - flange outer diameter, Dy - drift nominal diameter, b - thread length. The function of the parameter should be specified by the comment in the table or by description of the public parameter.
3. Rules of the variable's name for C language.

Rules to define the variables:

- Identifier is a sequence of symbols, which are variable's, function's or operator's names.
- Identifier can consist of Latin letters, numbers and underscore character.
- Identifier should be started from a letter or underscore character.
- Identifier, consisting of the same sequences of symbols but in a different case (upper/lower case), are different identifiers.
- Maximum length of the identifier is limited to 50 symbols.

Recommendations for variables names:

- Use prefixes for variable types (r - real, i - integer, str - string, b - logic, vec - vector, pnt - point)
- Use descriptive variables' names (for example, rLength instead of L, bSection instead of sh)
- Use different cases of letters in the long names of the variables (for example, rWP2offset, bUseFlatSeal)

The parameters should be described as public if:
1. It is needed to specify a connect for the object
2. It is needed to specify a nominal size for the object (the parameters are important)

3. It is needed to extract the parameter for further use (by universal marker or to add to the table fields and etc.)
In any other cases the parameter should not be added to the public section.
You can specify only values of the strTheName, strTheType and strTheSubType variables in English. The values of these parameters should be specified in a way to provide the maximum standardization of the connect procedures.
All parameters of the Public type should have descriptions. This also applies to the descriptions of the working planes. The descriptions of the parameters should be started from the object noun (for example, "Bolt head width").
It is not recommended to describe the parameters with dimensions in the Public section. The descriptions of parameters in the table and in VFLD field can have dimensions. In this case, the dimension is specified in SI units after a comma (for example, "Outer diameter, mm").
All parameters names of the VFLD and BFLD sections should be filled in. It is not allowed to use the variable name as the description of the variable (e.g. L,"L").
If the preview is set with the dimensions of the detail, it is allowed to use the comments of any parameters in the VFLD L, "L, Length".
It is recommended to specify the description for all parameters used in the script (as comments).
If a detail's mass is specified in the standard, it should be added to the public parameters and in the UniDialog DBINF section.
The mass of the detail is specified as a "massa" parameter and in kilos per one detail.
If the mass of the implementations is specified in the table, these parameters have names: massa1, massa2 and etc.
To provide the correct localization of the scripts, all string values should be references to the resources, except for the implementation's name strings, strTheName, strTheType, strTheSubType classifiers, form names etc.
Resource names are specified in English upper case letters without spaces.
For example:

```
@DIAMETER_OF_THREAD
@WORKING_PLANE_1
```

The system names should contain only English letters.

Script

It is recommended that the comments are created in English
It is not recommended to place several operators in one line.
When creating the script, follow the order of the functions:

1. ActHeader()
2. OnInitialisation()
3. (Custom functions)
4. OnMakeParameters()
5. SetGripPoint()
6. OnMoveGripPoint()
7. OnDialog()
8. OnDialogChanged()
9. OnChangeParams()
10. OnInitSelect()
11. OnSelectParam()
12. OnMenu()
13. BeforeConnect()
14. OnConnect()

The operators, setting the strPartName and strPartDescription variables, should be placed at the end of the OnMakeParameters parameters.
The script version and the GOST description should be placed in the first lines of the script. General comments of the object can be placed after them. For example:

```
  SVersion = 2;
   ObjectDescription = "GOST 19425-74";
   // Slope of flange should be taken
   //between 4 and 10%.
   //In particular 8%
```

It is recommended to use the tabulation to specify an offset for the composite statement and is not recommended to use spaces.
Composite statements should satisfy the following requirements:

- Open bracket is on the same level as the statement
- Nested statements are placed in the same level with the 1 Tab offset.
- Close bracket is on the same level as the composite statement.

Example:

```
function OnConnect {
   if (rPart == 0) {
      if (obj.strTheSubType == "vaBottom") {
         setWorkId (0, obj.objectID);
         Handled = OBJ_HANDLED;
      };
      if (obj.strTheSubType == "vaShell") {
         setWorkId (1, obj.objectID);
         Handled = OBJ_HANDLED;
      };
   };
   …
};
```

The UniDialog function should satisfy the following requirements:

- The Function is formed as a composite statement
- Every keyword is started from a new line
- Every option of multi-version sections starts from a new line with the offset.

Example:

```
UniDialog(
  DBFLD, seria, h, b,
  DBINF, num, mas1,
  VFLD,
   L,"length of I-Profile, mm",
   strRefDataHeader,"Reference data:",
   A, "A, sm^2",
   Ix, "Ix, sm^4",
   ix, "ix, sm",
   Wx, "Wx, sm^3",
   Sx, "Sx, sm^3",
   Iy, "Iy, sm^4",
   Wy, "Wy, sm^3",
   iy, "iy, sm",
  BFLD, bHid, "Show hidden lines",
  TVIDS,lViewType,"AnyWBK",
  VIEW,"Vids"
);
```

It is recommended to break difficult arithmetic expressions over several lines, or use several additional variables.

It is recommended to use the descriptive comments to group lines of the code which have a close logic structure (for example, block of statements to specify working planes etc.).

Graphics

The drawing should provide recognition of the parametric 2D view of the object from the database with maximum legibility and contain all required views for the element (including the graphic for which the preview was created).
The expressions "single parametric element from the database", "detail" and "GOST" (in lower case so that the letters are not confused with the standards) are set as equivalent.
The drawing of the detail or details should be created in one file. The file name should contain all the numbers of the standards used to create the detail and, if needed, the description. (for example, "GOST 7798-70 Hexagon Head Bolts Product Grade A").

General requirements

By default the scale is set 1:1. The scale to draw a graphic is selected by the user, but the drawing should be legible.
It is recommended to draw a detail according to its values from the table.
All objects are drawn in the model space. Default settings are used to create elements.

Layers

By default the line width is set to default for all layers and objects.
Layers, used for the drawings:

Layers name	*Parameter - linetype of the object*	*Colour*	*Objects*
MAIN	0 or not specified	White	Objects, which should have this linetype Insertion point
THIN	2	Red	Objects, which should have this linetype Hatch
DASHED	4	Green	Objects, which should have this linetype
AXIS	Specified by the command	Red	Objects, which should have this linetype
WORKING	Checkbox "working object"	Yellow	Objects, which should have this linetype
DIMENSIONS	Not specified	Blue	Dimensions and text formulas Names of GOSTs, implementations, views and other comments

The "centre Line" parameter should be set for graphic objects after their replacement on the specified layer.

Placement of graphic

The separate details and the separate implementations of one GOST should be separated from the other GOSTs and implementations in the drawing (zonally or with the rectangular frames). If there is more than one GOST or implementation, the GOST number or implementation name should be written above of the group of implementation or GOST views.
The views should be in the projective connection in the drawing.

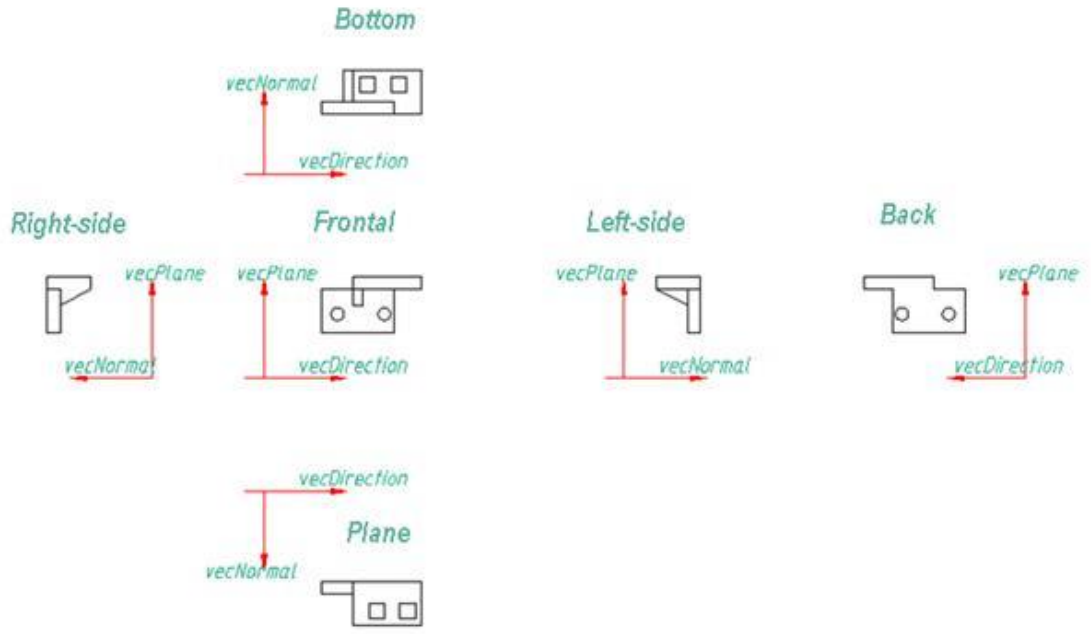


Figure 1. Placement Drawing Views.

The views, which are out of the projective connection, should have clear names to define the relation.
It is recommended to group the views with cuts, simplified views and views with dimensions near their projective views.
If one or several views belong to one GOST/implementation, they should have clear names to define the relation (for example: "Left-side view (all implementation)", "Frontal (GOST 7798,7805, 7806)"). The graphic of the details view should have a name.
The detail should be oriented that the detail's direction (vecDirection of script) should coincide with the OX axis (to the right for the frontal view).

Dimensions

To set the dimension, take into account:

1. Correct recognition;
2. Legibility of the drawings;
3. Unified system for design documentation requirements.

It is not permitted to place the dimensions inside the hatched area (the hatch is not allowed to flow over the dimension values).

Hatch

Hatch should be executed by the "User defined" type, slope angle 45 degrees. A hatch step should be specified parametrically according to the width of the narrowest hatched part (s):
For example:
1. For thin-wall objects: s/2;
2. For massive objects: s/8;
3. Intermediate step: s/4.
A hatch type can be specified in absolute units divisible by 10 (0.1;1;10;100)
You can ignore these rules if you need to hatch in both sides.

Parameters

Using the "Set parameter" tool, set the parameter for the dimensions. It is possible to switch off the "Work object" checkbox for the dimensional view only.
It is recommended to avoid text formulas in the drawing fields (place the calculations in the script).
Text formulas are allowed to be placed in the drawing, if you need to calculate the parameters of the implementation's view and it is defined that their use is not needed in the script.
Use of non-English letters is only allowed in the names
When specifying the parameter's name take into account:

1. Similar names existing in the database
2. Parameters names in the GOST
3. Meaningful parameters names, corresponding to the syntax of the C language.

If possible, avoid difficult arithmetic expressions set in the dimension parameters (it is recommended to use the Protected variables).

Other requirements

The views names should be specified with text of 10mm height. Text formulas should have the text height of 5 mm.
The views names are placed above the view, text formulas are placed below the view.
Specify meaningful names for the blocks in the array in English (for example, hole, chamfer etc.). The insertion point of the blocks, used for creating an axial array, should be placed in the centre of the axial array.
It is possible to add an image of the detail, copied from the GOST, into the drawing field, if it does not cause difficulties in legibility and overlap other objects.

Syntax scripting language

Script - is the code in a language similar to SI, which determines the nature of the object database software parameters, the plane impact on the response object. Automatic installation of dependencies, etc. A script comprises a series of blocks, which perform defined events from outside the object.

Identifiers

Identifier is a sequence of characters representing the name of a variable, function, or operator. Rules describe identifiers MechWizard similar to the rules in C:

- Identifier can consist of letters of the alphabet, numbers, and the underscore character.
- The identifier must begin with a letter or underscore.
- Identifiers consisting of identical sequences of letters, but in a different case (uppercase / lowercase) are different identifiers.
- The maximum identifier length is limited to 50 characters.

Recommendations to the name variable names are listed in the document.

Keywords

Keywords - are reserved identifiers that are endowed with a certain sense. They can be used only in accordance with the value of the known macro-language interpreter. Keywords cannot act as an identifier.
List of key words listed in the annex.

Data Types

In the macro language, there are five main types of data:

1. Valid floating-point number (double analogue in C). Range-1.7e308 .. 1.7e308.
2. Line.
3. Three-dimensional point (Point)

4. Three-dimensional vector (Vector)
5. Plane defined by the reference point and a normal vector (Plane).

The script made an agreement to the names of real type variables prefixed r, to add a prefix to a string variable str or s, to the point type variables prefixed pnt or p, the variables of vector type to add a prefix or vec v, and the plane denoted WP (WorkPlane).
Initialization of variables is carried out without prior specification as in BASIC. To declare a variable, you just need to assign it a value. If the variable is not described in the header function in the sections ActHeader Public or Protected, it will be considered local and will retain its value from the first moment until the end of the definition, in which it is described.

```
StrName = "ObjectA";
rPI = 3.1415926;
rA = 2.0;
```

To initialize the points and vectors used special constructor functions Point (), Point (x, y, z), and Vector (), Vector (x, y, z) respectively.

```
pnt2=Point();//handle length
pnt1= Point(1.0, 10.0, 10.0); //This is the point with specific coordinates
vec1 = Vector(); // vector of unit length, which coincides with the axis OX (1,0,0)
vec1 = Vector(1.0, 1.0, 0); //Vector specific coordinates
```

To separate the coordinate point or vector can be accessed using the qualifier:

```
pnt1:x = 12.5;//Removing the abscissa of the point
vec1:y = 25.6;//Removing the ordinates of the point
```

Plane used to initialize a constructor function Plane (pnt, vecNormal).

```
WP1 = Plane( pntBase, vecNormal );
//pnt- This is the basic point for the plane WP1
//vecNormal- This is the normal vector to the plane of WP1
```

Eg

```
WP1 = Plane( pntOrigin, vecDirection);
```

Indexing variables
If you declare a number of variables of any type with names that differ only by the number in the end, then these variables can be accessed by index, as an array.

```
rParam1 = 1.0;
rParam2 = 2.0;
rParam3 = 4.0;
...
rParam[2] = 45.7;
rParam[3] = 9.1223;
```

Comments

Comments in the script begins with the double slash line and ends at the end of the line.

```
//First set up a plane in counter-clockwise order
vecNormal=getLocalNormal(vecDirection,vecPlane);

WP1 = Plane( pntOrigin, vecDirection );
WP2 = Plane( pntOrigin-vecPlane*(0), -vecPlane );

//Right side
```

```
WP3 = Plane( pntOrigin+vecNormal*(b/2), vecNormal );
WP4 = Plane( pntOrigin+vecNormal*(b/2)+vecPlane*h, vecNormal );

WP5 = Plane( pntOrigin+vecPlane*(h), vecPlane );
```

Operators

All operators of the macro language can be divided into the following categories:

- arithmetic operators
- Boolean operators
- conditional operator
- loop operator
- other operators (empty statement, the assignment operator, the operator access to internal members)

All operators end empty statement - a semicolon ";".

Empty operator

Empty statement consists only of a semicolon. When this statement is executed, nothing happens. This statement is a delimiter for other operators, functions, and lines in the script.

Composite operator

A compound statement is a few operators and announcements enclosed in curly braces {}. Execution of the compound is in the sequential execution of its constituent operators. At the end of a compound statement the semicolon is placed.

The assignment operator

The assignment operator is denoted by the sign of equality.
expression ***A = B;*** means that variable ***A*** is assigned the value of expression ***B***, and the type of the variable ***A*** type of expression is established ***B***.

Operator access to internal members of the complex data types

To access the internal members of the complex data types, a special qualifier operator ":". For example, it can be used to access the individual coordinates variables of type Point and Vector.

Arithmetic and logical operators

Syntax of arithmetic and logical operators in macro language script syntax is similar to SI.

Arithmetic		***Puzzle***	
A + B	Addition	A == B	A is equal to B
A - B	Subtraction	A != B	A nonequilibrium In
A * B	Multiplication	A > B	A more in
A / D	Division	A < B	And in less than
		A && B	A and B
		A || B	A or B

For arguments of type Point subtraction result is a vector. A multiplication of a vector by a new vector will equal the scalar product of the vector by.

```
vecDirection = pntEnd - pntStart;
vecScaled = vecDirection * rScale;
```

The conditional operator

Format statement:

```
if ( <expression> )
    operator_IF1;
else
    operator_ELSE1;
    //<expression>- it is a logical condition to perform operator
```

As operatora_IF1 and / or may act _ELSE1 composite operator, ie group of statements enclosed in braces.
Execution if the operator starts with the evaluation of expression.
Further execution is as follows:

- if the expression is true (ie, different from 0), then the operator _IF1.
- if the expression is false (ie, equal to 0), then the operator _ELSE1
- if the expression is false and there is no operator _ELSE1, then the following if statement in

Block of statements after else may be missing along with the word else. In this case, the conditional statement is as follows:

```
if ( <expression> )
    operator_IF1;
```

As an expression can be any logical or arithmetic operator or a sequence of statements separated by parentheses.

```
if (A>1)
    operator_IF1;
if ((A>1)&&(B<5)&&(C==25))
    operator_IF1;
```

You can use nested conditional statements, ie conditional statement can be included in the design ***if*** the design or ***else*** another conditional statement.

```
if (A > 1){
    if (B < 5){
        if (C == 25)
            operator_IF1;
    }
};
```

Cycle Operator

Format statement:

```
while (expression)
    operator_BODY;
```

Requirements for the expression, such as in the if statement. _BODY Operator called the loop body.
Executing the cycle begins with the evaluation of the expression. Further execution is as follows:

1. If the expression is true (not 0), the loop body is executed.
2. If the expression is false, then the next while the operator.
3. The cycle is repeated from step 1.

Functions

Function - a set of operators and other function call. Each function has a name. Function declaration begins with the keyword function. The function body is enclosed in braces. In this version of the browser functions have no parameters.
Format function:

```
function Function_Name{
  operator_1;
  ...
  operator_N;
}
```

Allowed to call one of the other functions, but called from funcA functions must be declared above in the body of the script.
Example:

```
function user_Function_1{
  MessageBox("This is my first feature");
}
function OnInitialization{
  user_Function_1(); // The correct call. The called function is declared above!
  user_Function_2(); // Wrong call. The called function is declared below!
}
function user_Function_2{
  MessageBox("This is my second feature");
}
```

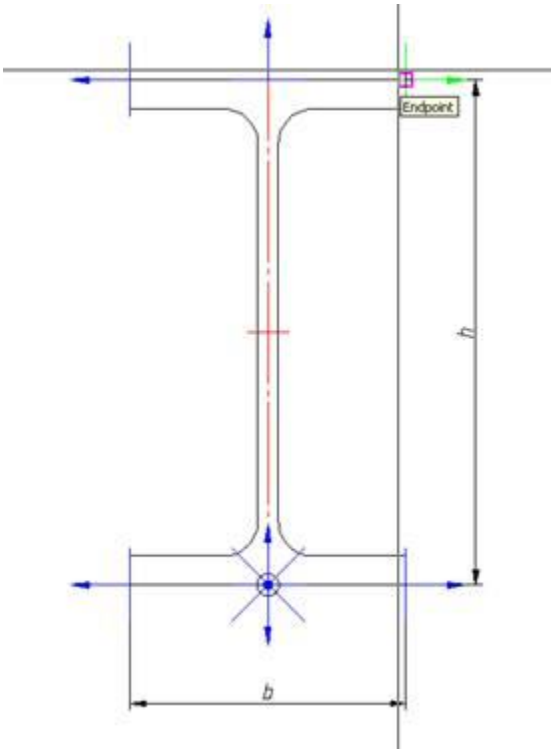
The interpreter has a number of built-in functions as listed in Appendix 2.

Specifying the object plane

Target abstract object geometry (points vectors planes) may be in coordinate form (almost not used) or relative to the local coordinate system. Position of the points defined by specifying operators addition point and plane. In the example, the position of the reference point is defined as the plane

```
  pntWP4=pntOrigin+vecDirection*(b/2)+vecPlane*h
```

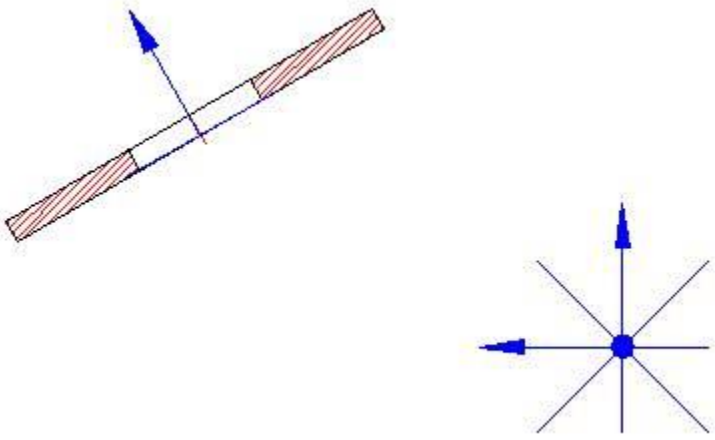
Normal vector of the plane collinear insert vecDirection



Consequently, she illuminated plane WP4 will be set as

```
WP4=Plane(pntWP4,vecDirection);
```

If the surface normal vector is not orthogonal coordinate system, it is often determined by the operator or RotateBy GetLocalNormal.



For example, for lining the conical bottom, surface normal vector is defined as the WP2

```
vecNormal=getLocalNormal(vecDirection,vecPlane);
 vecFitting = rotateBy(vecDirection,90-ang/2,vecNormal);
 //ang - is the angle of the bottom in terms of
```

Apart from the above identified, operators are used to find the vector normal plane and the base point:

```
   pntWP1 = Point(WP1);
    vecWP1 = Vector(WP1);
```

To find the distance between the point and the plane used the difference point and plane

```
   rDistance = WP1-pntOrigin;
```

Vector length (distance between points) define the operator vecLen:

```
 rLength = vecLen(pntWP1 - pntOrigin);
```

Orthogonal vector operator is getPerp:

```
vecPlane = getPerp(vecDirection);
```

Worksheet

To work with the table object used by two operators and LoadInCache SelectInCache.
The operator loads the LoadInCache specifies the parameters in the cache of the table. This operator in the new versions must be called only when the sample is forced with an expression to evaluate the parameters of the (..., "+", ...).
SelectInCache operator selects from the table according to the specified criteria (see syntax in the appendix)
Sampling by "kFirst" selects one first row. Sampling by "kFilter" selects a range of records from a table that match the specified filter.
Dining sample-specific parameters "~" - the approximate equality parameter selects the value in the record. "=" - An exact equality, "+" - the sample on the filter. Picks by filter oznachet that if the logical filter condition is satisfied, then the record is selected from the table (subjected to further checks on the conditions).
For example, the

```
SelectInCache("kFirst", "dr", "~", rdr, "L", "+", "L>=rMinLen&& L<=rMaxLen&& (L-b<=rTrLen)");
```

selects the first record of the table, where the combination of parameters dr, L, b such that dr≈rdr, rMinLen<=L<=rMaxLen, (L-b)<=rTrLen.
Function SelectInCache important order of the sampling conditions. This means that in the first place will be selected from the table entry corresponding to the first condition, then - of which selects records corresponding to the second condition, etc.
If there are multiple items in the tables, then the coincidence parameter names must enable the checkbox "Use the table name as a prefix to the parameters" in the context menu of the corresponding table.
In this case, an appeal to the parameter in the script will be as follows:

```
 <Name of the table>.<Parameter name>
```

For example:

```
If (Table0.dr==Table1.Thread1) {
  ...
 };
```

To provide a sample of the first table is not necessary to modify the operator SelectInCache.
For example, if you need to select from the table Table1 parameter dr, equal to 10, then the first condition in the operator to select from a table must follow the table index (starting from 0):

```
 SelectInCache(1, "kFirst", "dr", "~", 10);
```

For a table with imenenm Table0 table name can be omitted. Ie it is also true, and another expression:

```
SelectInCache ("kFirst", "dr", "~", 10, "L", "~", 50);
 SelectInCache ("0","kFirst", "dr", "~", 10, "L", "~", 50);
```

Dialog object. Making function UniDialog

View Insert Object dialog box is defined by specifying the key features UniDialog.
See the appropriate section of the application.
It should be easy to compare that
DBFLD - the defining table parameters.
DBINF - reference table settings.
VFLD - are arbitrary parameters and reference data.
VIDS - a list of versions of the object (possibly omitted if there is only one execution.
TVIDS - different views of the same design (possible keys - by type, All - those present, AnyWBK - opposing views generated mirror (for symmetrical objects).
VIEW - thumbs species.
Key OnDglBeforeSelParam in ActHeader determines whether the dialog is called before the dynamic parameter selection.

Connecting UserForm

If an object is given shape, then in order for it to be called, you need to function in place of the function OnDialog UniDialog cause concrete form.
For example, if the object is given form named frmMain, you should write the following:

```
functionOnDialog{
  ShowForm("frmMain");
 }
```

Similarly, the terms can cause several different forms of the object, referring to them by name using the operator ShowForm.

Set dependencies

Operators set up dependencies implement geometric and parametric dependence.
Parametric dependencies

```
SetParamConstraint(< param >,obj,EXPR,< expression to constrain>,bBidirect);
```

<param> - name of the parameter object to which the time dependence.
Obj - which object set.
EXPR - so far only the dependence on the expression
<expression to constrain> - arithmetic combination of parameters on which dependence is placed (must be in quotes).
bBidirect - bidirectional or unidirectional relationship.
Geometric constraints

```
SetGeomConstraint(rTYPE, rDIRECTION, obj, < WPname >, < WPtoConnect >,offset,bBidirect);
 SetGeomConstraint(ANGE,CODIRECT,< obj >,< wpSourcePlane >,< wpObjPlane >,< rAngle >,< bBiderect >);
```

Types of dependencies rTYPE = INSERT (inset)) MATE (alignment on the plane) AXIS (alignment axis) DIRECTION (aka ANGLE) (angular orientation).
Direction rDIRECTION = CODIRECT (co-directed) CONTRDIRECT (opposite directions).
<WPname> - The name of the plane, which are attached.
<WPtoConnect> - The name of the plane, joined.
Offset - numerical or parametric expression that defines:
INSERT - axial distance between the planes.
MATE - the normal distance between the planes.
AXIS - relative displacement in terms of the axes.
DIRECTION - angle relative orientation of the planes.
For dependencies DIRECTION (ANGLE) there are certain conditions:

Syntax:
SetGeomConstraint(ANGE,CODIRECT,< obj >,< wpSourcePlane >,< wpObjPlane >,< rAngle >,< bBiderect >);

ANGLE - keyword angular dependence.
CODIRECT - always for the angular dependence. It is believed due to the minimal angle.
<obj> - which object set.
<wpSourcePlane> - the plane of the source object to which dependence is placed.
<wpObjPlane> - the plane of the object to which you subscribe.
<rAngle> - numerical value of the angle in degrees, depending.
<bBidirect>=(TRUE,FALSE) - unidirectional or bidirectional relationship. Default FALSE.
Depending on the type ANGLE should be added after the dependency type INSERT, MATE, AXIS.
When the dependency type INSERT, MATE or AXIS rotation is relative to the axes of the planes dependence.
In the absence of dependency type INSERT, MATE AXIS or move it in the normal to the plane in which both vectors lie in the direction of the minimum angle of rotation.
If we consider the vectors are collinear, move it about the axis OX.
Example:
SetGeomConstraint(ANGLE,CODIRECT, obj,WPnormal,obj.WPnormal,45);
SetGeomConstraint(ANGLE,CODIRECT, obj,OYPlane,obj.WP2,15,TRUE);
Examples of different types of dependency can be found in the appendix.

Function ShowValue

Using the debugger windows can detect errors in the script and determine the values of parameters during the execution of the script. Syntax, see Appendix.

Object versioning support

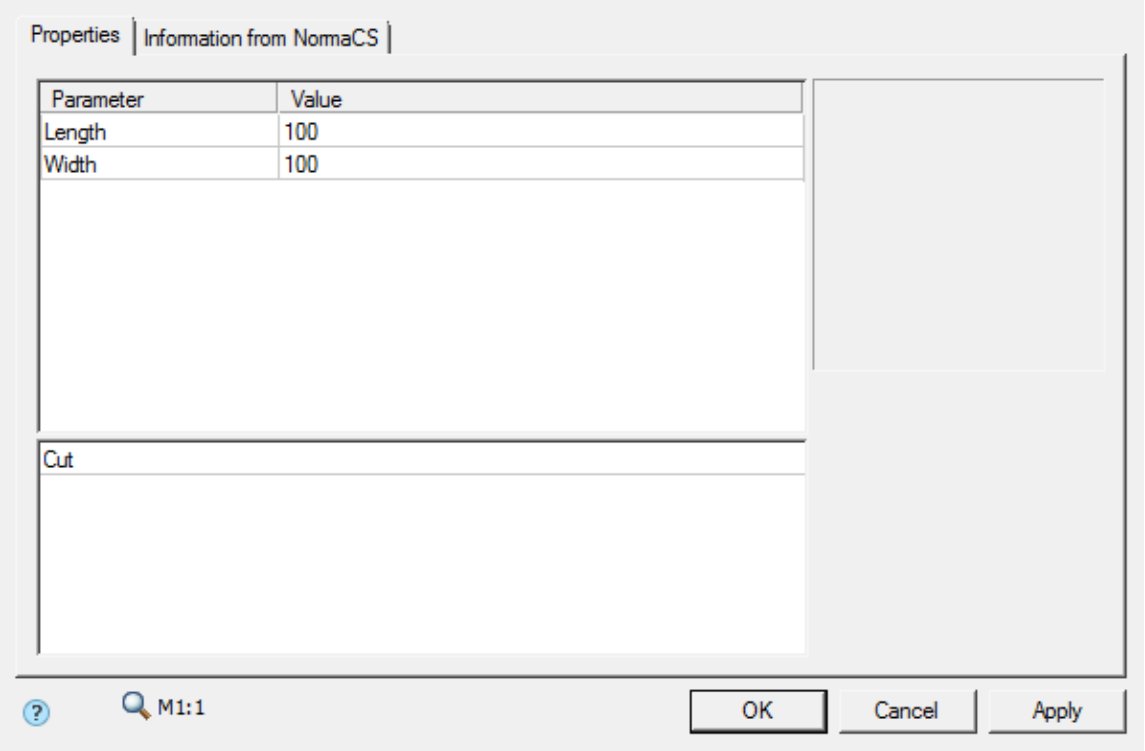
The object versioning is necessary in order for the new parameters in the updated object to be correctly displayed and used on the previously inserted objects.
The seted protected parameter is used to support the object versioning. The parameter is automatically generated when using the script wizard and can be renamed or added manually if necessary.
Consider the script for creating a rectangle (the script is not complete, only methods relevant for the example are shown):

```
.....    ...
function ActHeader {
.....    NPart=0;
.....    Public(a,"Length", b,"Width");
.....    Protected(seted);
.....    Changeable(a,b );
.....    OnDlgBeforeSelectParam = 0;
.....    ShowWhenSelPnt = 1;
.....    ContourOnLine = 0;
};
.....    ...
function OnInitialization {
.....    if(seted == UnknownValue) {
.....    .....    seted=1;
.....    .....    a = 100;
.....    .....    b = 100;
.....    };
};
.....    ...
```

```
function OnDialog {
.....    UniDialog( VFLD, a,"Length", b,"Width", TVIDS,lViewType,"All",VIEW,"Vids");
};
.....    ...
```

The ActHeader method specifies the seted protected parameter and the public length and width parameters.
In the OnInitialization method for an object that has no version, add the length and width parameters and set version 1.
In the OnDialog method, we call a dialog with the width and length fields.

Let's insert an object (hereinafter Object1). When placing an object on a drawing, a window with default parameters will open.

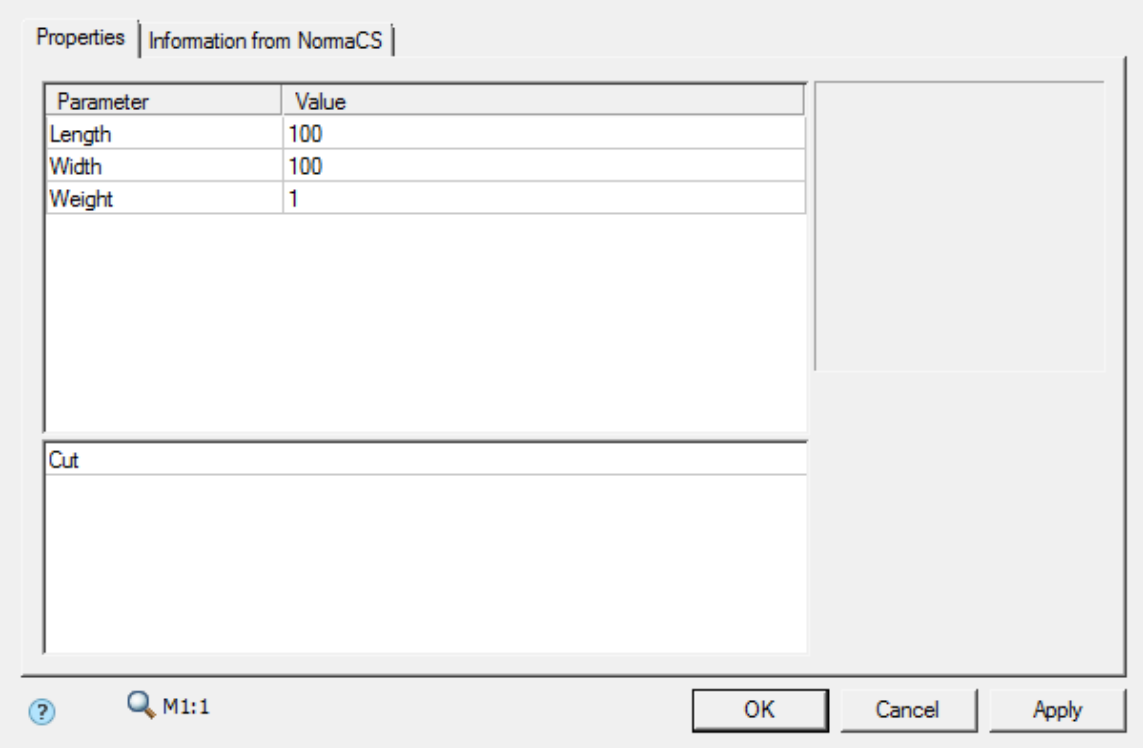


Suppose we need to add line thickness for the rectangle.
To do this, we add the "weight" parameter:

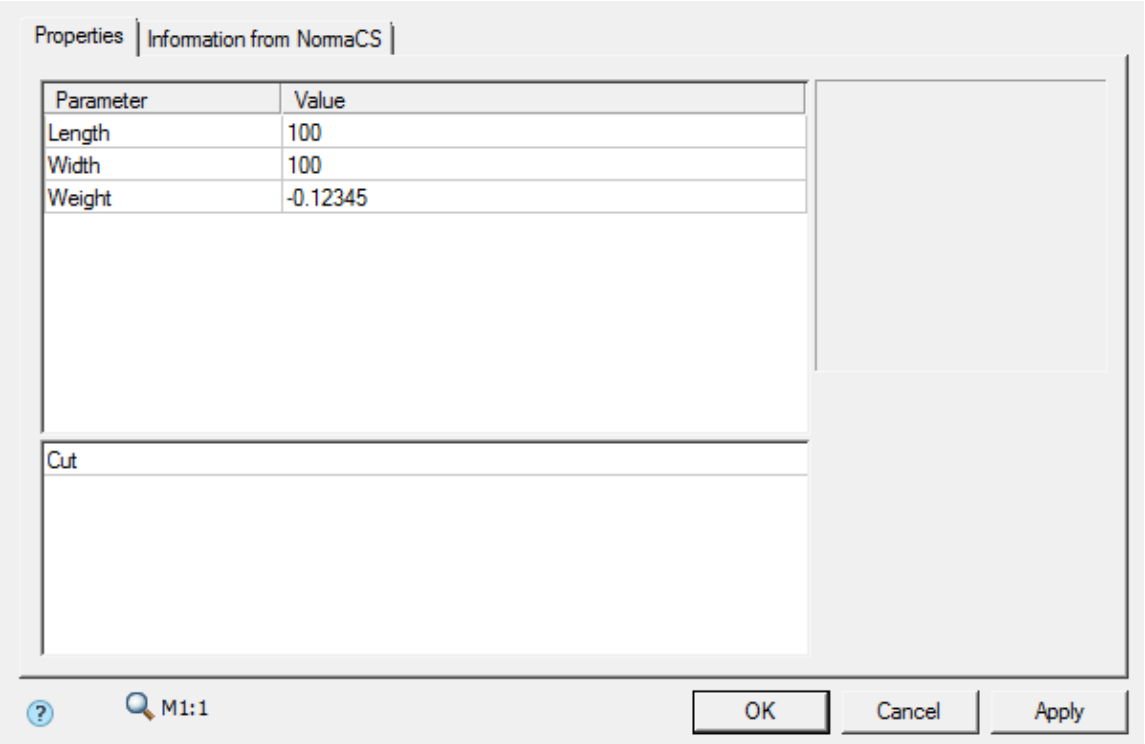
```
.....    ..........
Public(a,"Length", b,"Width", weight, "Weight");
.....    ...
function OnInitialization {
.....    if(seted == UnknownValue) {
.....    .....    seted=1;
.....    .....    rZOrder=100;
.....    .....    a = 100;
.....    .....    b = 100;
.....    .....    weight=1;
.....    };
};
```

```
.....     ...
function OnDialog {
.....     UniDialog( VFLD, a,"Length", b,"Width",weight,"Weight"  TVIDS,lViewType,"All",VIEW,"Vids");
};
.....     .....     .....     .....     .....
.....     .....     .....     .....     .....
```

Let's insert an object (hereinafter Object2). The dialog correctly displayed the line weight to us.



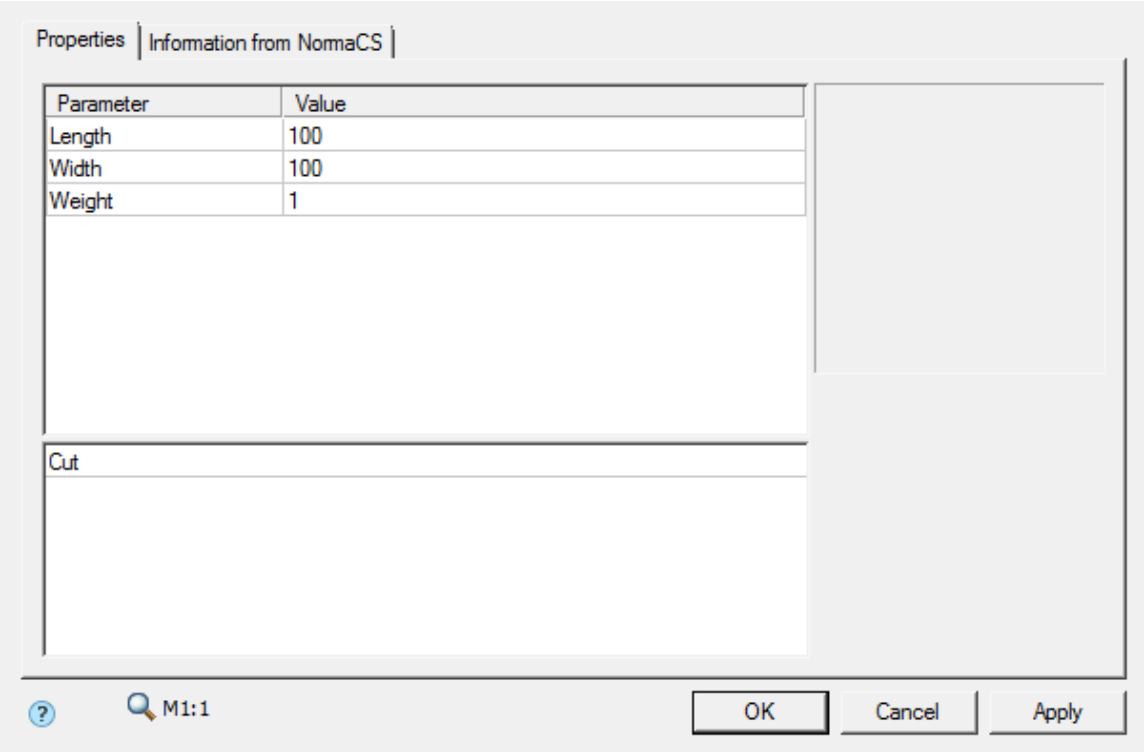
Let's open Object1 for editing and see that the "Weight" parameter is displayed incorrectly.



The point is that Object1 already has version (seted) 1 and the weight is not assigned.
Let's fix the OnInitialization method. If there is no version, we will increase it to 2, add a condition if we have version 1.

```
function OnInitialization {
.....    if(seted == UnknownValue) {
.....    .....    seted=2;
.....    .....    rZOrder=100;
.....    .....    a = 100;
.....    .....    b = 100;
.....    .....    weight=1;
.....    };
.....    if(seted == 1) {
.....    .....    seted=2;
.....    .....    weight=1;
.....    };
};
```

Now objects of older versions will open correctly.



Appendix

Appendix 1. A list of keywords and reserved variables

SVersion=2;	The first line of the script, which determines the version of the interpreter
strTheType	Object type
strTheName	Object Name
strTheSubType	Subtype object
pntOrigin	Insertion point
vecDirection	Vector insertion direction
VecPlane	Orthogonal vector to vecDirection
Scl	Object Zoom
rZOrder	Ranking (used for overlap)
strDesignName	Execution system name
rXcoord	Relative abscissa cursor with dynamic selection of the parameters
rYcoord	Relative ordinate cursor with dynamic selection of the parameters
VecCoord	Mostly available in OnSelectParam VecCoord: X - X coordinate of the relative VecCoord: Y - Y coordinate of the relative

pntGrip#	(# Number) point pens with the number #
rPart	Serial number of the request.
rKbd	Flag keyboard input. Equal to 1 if the value is entered from the keyboard.
strPartName	Name of detail in the specification (section name)
strPartDescription	Part description in the specification (section designation)
strPromt	Help line (in the dynamic selection or when selecting items for a connection)
NSelect	The number of requests when inserting objects.
NPart	Number of selectable parameters if 0 or undefined then simply called dialogue.
BreakCur	End the current dynamic parameter selection
BreakAll	Complete the entire cycle of selection of dynamic parameters
NoVectorSelect	Does not indicate the vector inserts (used at a connection)
Handled	Result processing facility
NGrip	Number of Handles
NMovingGrip	Phone editable pens
lSimpleView	Variable specifies the type of species, UniDialog created and installed automatically. If == 1 - a simplified view. If == 2 - sectional view
scaleDisable	If this parameter is set to 1 in UniDialog prohibits the changing Scale
ShowWhenSelPnt	If this box is checked, the object will be drawn at the time of insertion, if not specified, it is enabled by default
ContourOnLine	Generation circuit is not drawing from a script, and on the fly after drawing
IsAHole	The object is a hole (Embedded object)
OnDlgBeforeSelectParam	Call dialogue before calling OnSelectParam.
IsACalculator	Is set equal to 1 for the calculations
No3dViews	Disables automatic views
AutoShift	Automatic pseudo-pressing of the Shift button
UseOnlyZOrder	Used only for crossing rZOrder, do not use Z-coordinate
fUseCommonScale	Maintain external control scale
MINPOSSIBLEVALUE	The minimum acceptable value for the application
NotStdBody	If == 1, then the item is placed in the component catalog
SpecPartition	Unit specifications, which gets the item
UnknownVal=nknownValue=-0,12345	The value assigned to the default variable. If the variable is not defined, it is assigned a value.

Filter List function SelectInCache:

- "kAsk" - Ask the user.
- "kFirst" - Find the first value in the table that satisfies a specified condition.
- "kLast" - Find FOLLOWING value in the table that satisfies a specified condition.
- "kFilter" - all the values from the table that satisfy the specified condition.
- "kAskIfNeed" - Ask user if the selection was not successful.

Values that the variable can Handled:

- OBJ_UNHANDLED - Dependencies are not processed.
- OBJ_HANDLED - Dependencies processed
- OBJ_WAITMORE - Waiting for Input
- OBJ_WAITMORE_M
- OBJ_WARNING- Error installing dependencies. When returning this code, the parent object will seek to establish the same parameters as that of the child, and then update the child object is repeated.
- OBJ_ERROR

Constants for the variable type lViewType

- VFRONT - Front view
- VLEFT - Top view
- VRIGHT - Bottom View
- VTOP - Left View
- VBOTTOM - Right Side View
- VBACK - Rear View

Types depending

- INSERT - Insert
- MATE - Combination by plane
- AXIS - Combining axis
- DIRECTION - Combining the direction
- ANGLE - The angular dependence

Subtypes depending

- CODIRECT - co-directional dependence
- CONTRDIRECT - contra dependence

Recurrence dialog codes or function MessageBox:

- IDOK
- IDCANCEL
- IDYES
- IDNO

Codes for the function MessageBox:

- MB_YESNO
- MB_OKCANCEL
- MB_OK
- MB_ICONINFORMATION
- MB_ICONQUESTION
- MB_ICONERROR
- MB_ICONWARNING

Example:

```
MessageBox("Test");
if(IDYES == MessageBox("Yes or No", MB_YESNO, MB_ICONQUESTION)){
   ...
}
```

Appendix 2. Built-in functions

Mathematic functions to work with the surfaces

Function	Comment
sin(alfa) cos(alfa) tg(alfa)	Trigonometric functions. ***alfa*** - angle in degrees Returns a real number
asin(c) acos(c) atg(c)	Arc-trigonometric functions. ***c*** - number Returns an angle in degrees.
DegToRad(deg)	Function converts degrees to radians. ***deg*** - angle in degrees
RadToDeg(rad)	Function converts radians to degrees. ***rad*** - angle in radians
int(value)	Function rounds a real value to integer ***value*** - real number
abs(value)	Function gets the modulus of the number
sqrt(value)	Functions returns the square root of the number
min(a,b) max(a,b)	Function returns maximum and minimum numbers from ***a*** and ***b***
iff(expr, a, b)	Function is analogue of the ***if arithmetic operator***. Returns ***a***, if the result of the expression ***expr*** >= 1, ***b*** otherwise.
VecLen(vec)	Function returns the length of the ***vec*** vector.
VecUnit(vec)	Function gets the normal vector (of unit length) from the ***vec***.
VecCodirect(vec1,vec2)	Function checks if the ***vec1*** and the ***vec2*** are codirectional. Returns 1 if the vec1 and the vec2 are codirectional, 0 otherwise.
anglePi(vec1, vec2)	Function returns the angle between the ***vec1*** and ***vec2*** in the interval from0 to PI.
angleTwoPi(vec1,vec2)	Function returns the angle between the ***vec1*** and ***vec2*** in the interval from0 to 2*PI.
Point() Point(x, y, z) Point(plane)	Function returns a point. Can be used as a point constructor. If it is without arguments, it returns the P(0,0,0) point. If the argument is a plane, it returns a base point.
Vector() Vector(x, y, z) Vector(plane)	Function returns a vector. Can be used as a vector constructor. If it is without arguments, it returns the V(1,0,0) vector. If the argument is a plane, it returns a normal.
Plane(pntBase,vecNormal)	Function is a constructor of the plane. Returns the plane containing the ***pntBase*** point and having the ***vecNormal*** normal.
getPerp(vec)	Function returns a vector, orthogonal to the ***vec*** vector.
rotateBy(vec1,rAngle,vecNormal)	Function rotates the ***vec1*** vector by the ***rAngle*** angle (in degrees) in

	the plane, having the ***vecNormal*** normal. Returns the result of the operation.
getPerpendicular(pnt1,pnt2,pntFrom,pntBase)	Function returns the length of the perpendicular from the ***pntFrom*** point to the line, specified between the ***pnt1*** and the ***pnt2*** points. In the ***pntBase*** variable, the point of the perpendicular base on the specified line is returned.
getMiddle(pnt1, pnt2)	Function returns a point in the middle of the line, specified by the ***pnt1*** and ***pnt2*** points.
getLinesIntersect(pnt1,vec1,pnt2,vec2) getLinesIntersect(pnt11,pnt12,pnt21,pnt22)	Function returns a point of intersection of the lines, specified by the pairs of the points: [***pnt11, pnt12***] and [***pnt21, pnt22***] or by a point and a vector: [***pnt1, vec1***] and [***pnt2, vec2***]. if the lines do not intersect, the Z coordinate is -1 in the returned point.
getLocalNormal(vec1,vec2)	Function returns a vector product of the ***vec1*** vector and the ***vec2***.vector.
projectOnPlane(pnt,plane2,rDistance) projectOnPlane(plane1,plane2,rDistance)	Function returns a projection of the ***pnt*** point in the plane, placed in the ***rDistance*** distance from the ***plane2*** plane. If the plane is a first argument, the function returns the plane through the Point (***plane1***) point projection on the plane, placed in the ***rDistance*** from the ***plane2*** plane and having a normal vector, coinciding with the normal vector of the ***plane2*** plane.
Subtraction of the point from the plane	Example: d = WP - pnt; Operation returns the distance of ***pnt*** point from the ***WP*** plane. If point ***pnt*** is on side where normal of ***WP*** plane points, direction is positive, negative otherwise.
Formation of the string by the numbers	To the string variable you can add a number. The result is a string. Example: A = 1; B = 5; Str = "Type " + A + ". Modification " + B + "."; The result is a string ***"Type 1. Modification 5."***
FmtText(text, number, formated_text,...,)	Returns an rtf text, if among the parameters there is even one formatted, ordinary otherwise. Various numbers of the parameters can be a result of the function. Expressions can be used as parameters.
FmtSuper(Text); FmtSub(Text);	***FmtSuper()*** - returns an rtf text, formatted as the Superscript (upper index) ***FmtSub()*** - for the Subscript (lower index). ***Text*** can be a string or a number.
FmtDiv(Text1, Tex2, Splash)	***FmtDiv()*** - returns a fraction, if ***Splash != 0*** the fraction will have a divisor. Example of use: a = "super"; b= 1; c = 3; strPosition = FmtText("beginning of

	the text", "Upper index-", FmtSuper(a), "Lower index-", FmtSub("sub"), " Text", "Frection- ",FmtDiv(b, c, TRUE), "fraction without a divisor-",FmtDiv(b, c, FALSE));
FmtDigit(rNumber, rDelimeter)	Returns a string, formatted by a decimal symbol. ***rNumber*** - number or string, containing a number, if the ***rDelimeter == 0*** the divisor is a point, if the ***rDelimeter == 1*** the divisor is a comma, if the ***rDelimeter == 2*** the divisor is taken from the system local. ***Example:*** a = 1.5; Formatted = FmtDigit(a, 1) As a result the ***Formatted = "1,5"***

Functions to process surfaces

Function	Comment

GetNearestPlane(pnt)	Function returns the name of the nearest plane to the ***pnt*** point of the selected object.
	pnt - variable of the Point type.
	Example:
	function OnConnect{
	if(obj.strTheType == "StdJointParts"){
	// ...
	} else {
	strName = GetNearestPlane(pntOrigin);
	if(strName != UnknownValue){
	SetGeomConstraint(INSERT, CODIRECT, obj, WP1, strName, 0);
	NoVectorSelect = 1;
	Handled = OBJ_HANDLED;
	}
	}
	}

//variant 1
findNearest(
pnt,
Name1,
Name2,
... ,
NameN
)
//variant 2
findNearest(
pnt,
arrName,
arrIndexStart,
arrSize
)..... .....
	Returns an index of the nearest point or plane according to the parameters types, specified with the ***Name1***, ... ***NameN*** or ***arrName*** names.
	Function supports two types of call:
	1) the particular names of the public parameters are specified as arguments
	2) the array of the public parameters with the starting index for searching and calculating a number of the array elements, is specified as the arguments
	pnt - variable of the Point type.
	Name1, NameN, arrName - names of the public parameters of the selected object
	arrIndexStart - starting index for searching in the array
	arrSize - array size
	Example:

	//variant 1
	rNearest = findNearest(pntOrigin, obj.WP1, obj.WP11, obj.WP21);
	// rNearest takes 0, 1 or 2 values
	//variant 2
	rNearest = findNearest(pntOrigin, obj.WP, 1, 4);
	// rNearest takes 1, 2, 3 or 4 values..... ....
	....

//variant 1
nearestPlaneName(
pnt,
Name1,
Name2,
... ,
NameN
)
//variant 2
nearestPlaneName(
pnt,
arrName,
arrIndexStart,
arrSize
)..... .....
	Returns an index of the nearest point or plane according to the parameters types, specified with the ***Name1***, ... ***NameN*** or ***arrName*** names.
	Function supports two types of activation:
	1) the particular names of the public parameters are specified as arguments
	2) the array of the public parameters with the starting index for searching and calculating a number of the array elements, is specified as the arguments
	pnt - variable of the Point type.
	Name1, NameN, arrName - names of the public parameters of the selected object
	arrIndexStart - starting index for searching in the array
	arrSize - array size.
	Example:
	//variant 1
	strNearest = nearestPlaneName(pntOrigin, obj.WP1, obj.WP11, obj.WP21);
	// strNearest takes "obj.WP1", "obj.WP11" or "obj.WP21" values
	//variant 2
	strNearest = nearestPlaneName(pntOrigin, obj.WP, 1, 4);
	// strNearest takes "obj.WP1", "obj.WP2", "obj.WP3" or "obj.WP4" values

restoreBasis(vecOld, vecNormal, vecNew);	Function converts an old basis according to the new position of the vector. Following combinations are applied in the next order: (x, y, z, newX) (y, z, x, newY) (z, x, y, newZ) vRrev - previous vNext in the XYZ
MakeVectorsFromView(lViewType, vecDirection, vecPlane);	Sets the position of the vectors from the view type ***Example:*** MakeVectorsFromView(VTOP, vecDirection, vecPlane);
setBasePlaneForSelect(planeName, vecPlane = vecPlane);	Sets a plane coordinate system, where the SelectParam will work, i.e. the rXcoord, rYcoord will depend on this system. Applying. When selecting non-symmetrical details whose selection direction depends on the fixed plane and the distance is calculated from any plane; for example, the standard ends of the arbors, not from the pntOrigin. ***Example:*** setBasePlaneForSelect(WP1); Used for connecting the ends of the arbors.

Functions for working with tables

Loading a table from the database to Cache

```
LoadInCache(name1, name2, ... , nameN)
```

Loads the selected fields from the table.
Function has various numbers of arguments.
name1, ... , nameN - names of the table parameters.
Example:

```
LoadInCache(D);
LoadInCache(dr, dH);
```

Selection from the table

```
SelectInCache(strFilterKey, strParamName, strOperation, Value
 ... ,
 strFilterKeyN, strParamNameN, strOperationN, ValueN)
```

Executes selection from the object's table cache. Various numbers of arguments.
strFilterKey - key of filtration - string, taking the following values:
"kFirst" - select the first from the values, satisfying the filtration condition
"kLast" - select the last from the values, satisfying the filtration condition
"kFilter" - select the range of the values
"kAsk" - ask user (activates the dialog for manual selection of the value)
"kAskIfNeed" - ask user if automatic selection cannot be preformed
strParamName -name of the table parameter - string, containing the name of the table parameter, on which selection is made.
strOperation - operation of filtration - string that can take values :
"=" - equals
">=" - more or equals
"<=" - less or equals
"~" - approximately equals
">" - more
"<" - less
"+" - calculation of the expression
Example:

```
LoadInCache(dr, L);
SelectInCache("kFirst", "dr", "~", 14, "L", ">=", 100);
```

```
LoadInCache(dr, L, b);
SelectInCache("kFirst", "dr", "~", rdr, "L", "r;+", "L>=rMinLen && L<=rMaxLen && (L-b<=rTrLen)");
```

Dialogs of the database objects

UniDialog function

```
UniDialog (
   [DBFLD, D, Dn, ... ,]
   [DBINF, Number,Seria,...,]
   [VFLD, rA, "A", rB, "B", ... ,]
   [BFLD, bKey1, "Key1", bKey2, "Key2",... ,]
```

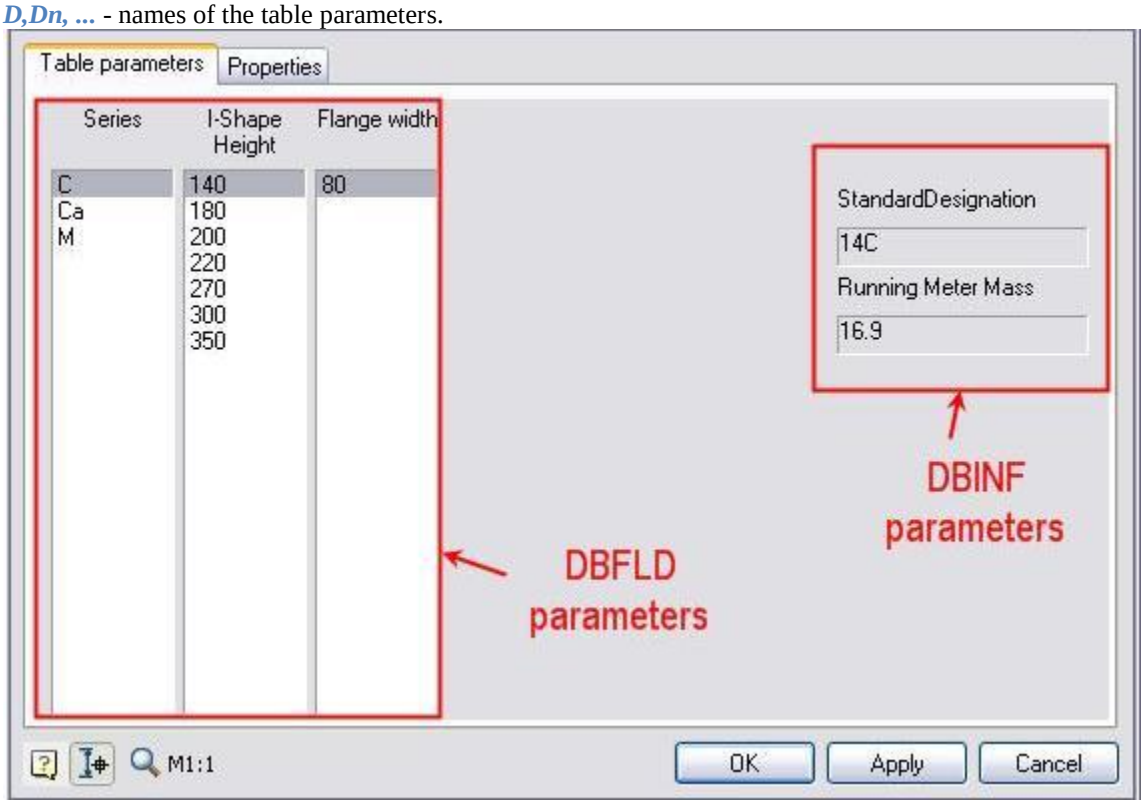
```
    [RADIO, rKey1, "Version1", "Version2", ... ,]
    [VIDS, strDesignName, "Design1","Design2", ... ,]
    [TVIDS, lVidType, {"F", "T", "R", "L", "All"},]
    [VIEW, {"Vids", "Hdr", "None"}]
)........    ........
........     ........
```

The function activates the dialog to edit the standard object from the database. The function returns the ***IDOK***, if the OK button was pressed, and the ***IDCANCEL***, if the Cancel button was pressed.

[ ] - for optional parameters

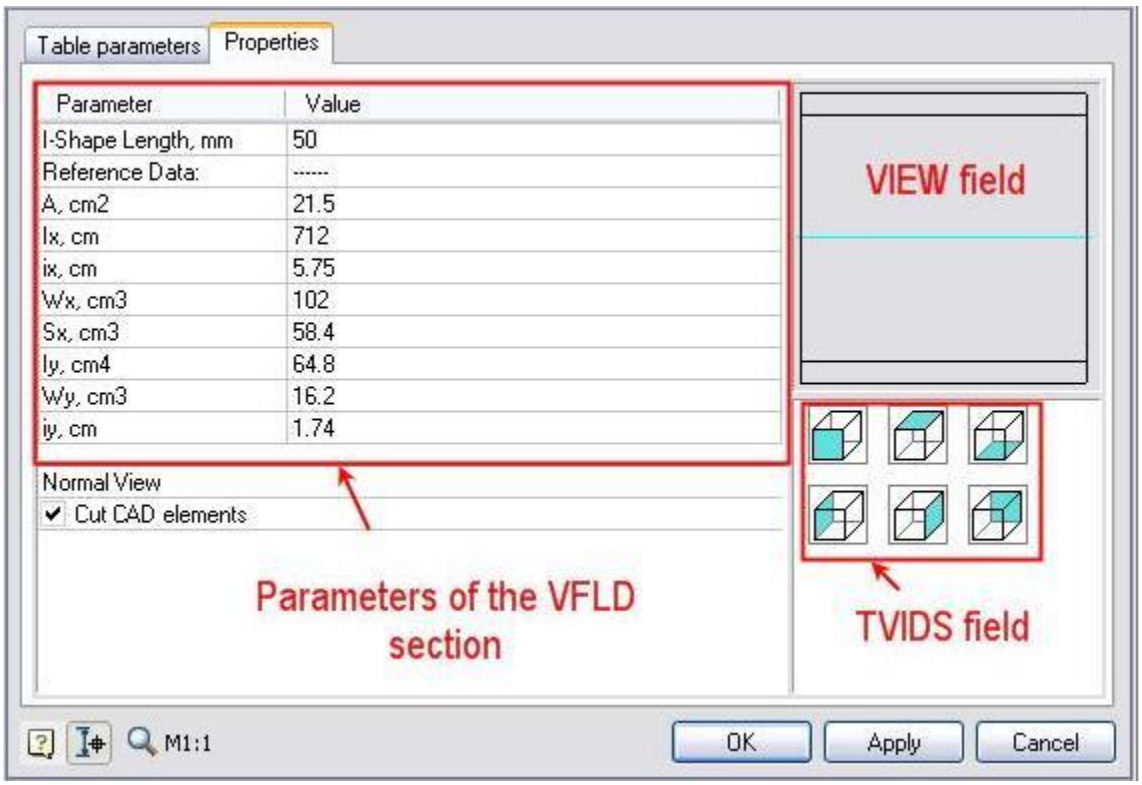
DBFLD - keyword, after which the table's parameters are listed.

D,Dn, ... - names of the table parameters.

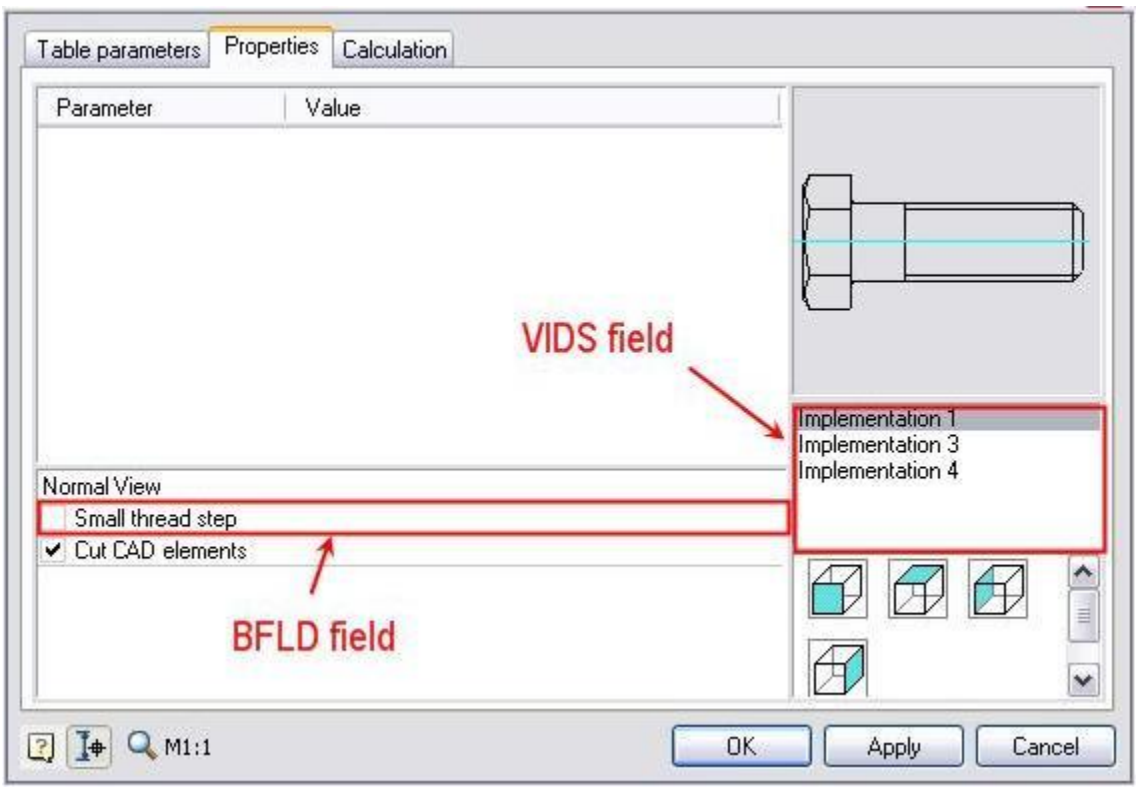


VFLD- keyword, after which the object's parameters are listed, announced in the ***Public*** or ***Protected*** sections.

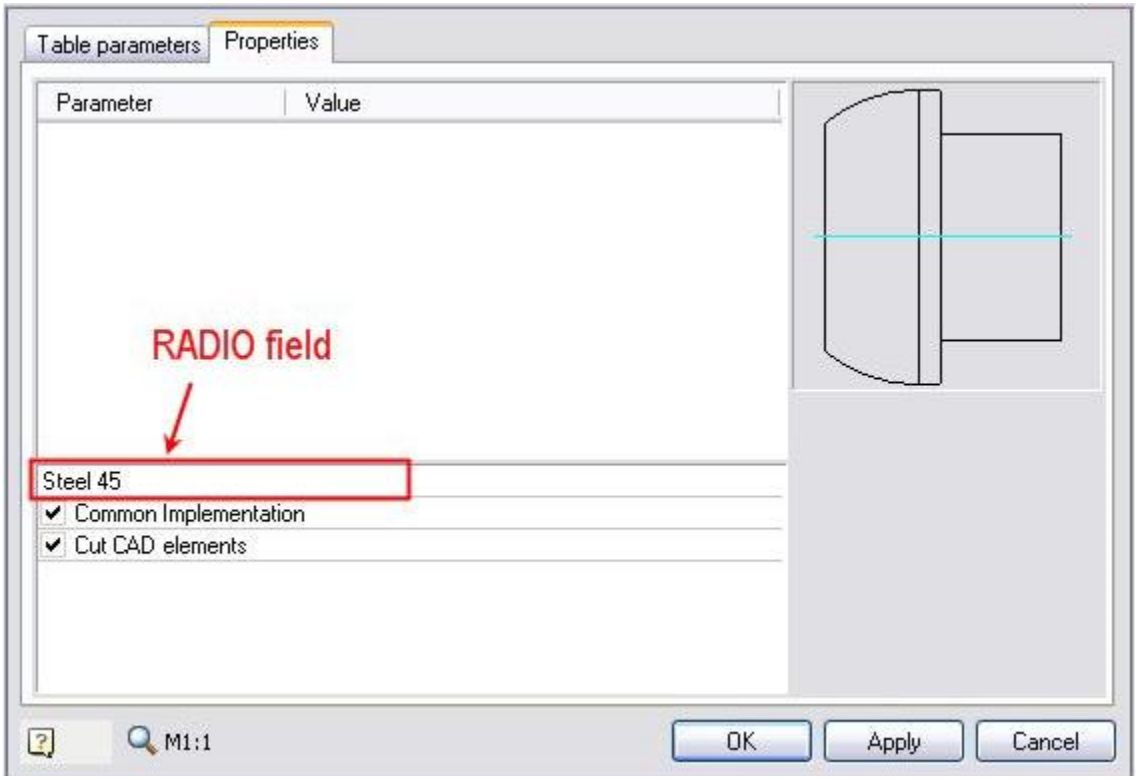
rA,rB, ... - names of parameters with the comments.



BFLD - keyword, after which the checkboxes are listed - variables, taking only 1 (CHKD) and 0 (UNCHKD).
bKey1, "Key1",bKey2, "Key2" - names of the variables with names of the checkboxes.
A block of the parameters, starting form the ***BFLD*** keyword, is a list of the dialog's components, which are standard ***CheckBox***(es) and has two modes: Checked and Unchecked. The parameter-variable, going after the BFLD, according to the checkbox mode takes 1 for Checked and 0 for Unchecked values. Such checkboxes are used for describing the properties of the standard objects in the application.



RADIO - keyboard, after which the group checkboxes are listed - variables, taking discrete integer values.
rKey1, "Variant 1", "Variant 2", ... - names of the variables of the group checkboxes with names of selection variants.



A block of the parameters, starting from the ***RADIO*** keywords, is a list of the dialog's components, which are standard drop-down lists with the permanent number of the unchanged strings. The parameter-variable, going after the RADIO, takes the index values of the selected string in the ComboBox. The top string has 0 index, the last is a subtraction of the number of strings and 1. Such checkboxes are used to select a material for the standard object in the application.

For the described example, the bCommon parameter takes 0 or 1 integer values, and the rd1 parameter takes 0, 1 or 2 values.

Parameters for the BFLD and RADIO can be described in the Public and Protected sections. Before activating the dialog, such parameters should be initialized (usually in the ***OnInitialization***).

VIDS- keyword, after which all an object's implementations are listed.

strDesignName, "Design1", "Design2" - variable's name, storing the name of the object's implementation with the available implementations' names.

TVIDS- keyword, after which the object's views are listed.

lViewType- variable, storing the name of the object's view and taking values from the multitude {VFRONT, VRIGHT, VLEFT, VTOP, VBOTTOM,VBACK}.

"F" - view name "Frontal".

"R" - view name "Right-side view".

"L" - view name "Left-side view".

"T" - view name "Plane view".

"B" - view name "Bottom view".

"All" -names of all views.

VIEW - keyword, after which the slide's type goes from the multitude {"Vids", "Hdr", "None"}.

"Vids" - implementation's slides of the object

"Hdr" - slide of the object's header (Header)

"None" - there are no slides

Examples:

```
LoadInCache(dr);
UniDialog (
   DBFLD, dr,
   VFLD,
   Lthread, "thread length",
   Lhole, "hole length",
   d_hole, "hole diameter",
   VIDS, strDesignName, "All",
   TVIDS, lViewType,"All",
   VIEW, "Vids"
);
LoadInCache(Dn);
UniDialog (
   DBFLD, Dn,
   BFLD, bCommon, "General purpose",
   RADIO, rd1, "Aluminium alloy", "45 steel", "Bronze",
   VIDS, strDesignName, "All",
   TVIDS, lViewType, "All",
   VIEW, "Vids"
);
.....     .....
```

Custom forms

```
ShowForm("FormName");
```

Activates the object's form with the ***FormName*** name.

MessageBox function

```
MessageBox(StrMessage[,mb_Buttons = MB_OK, mb_Icons])
```

The function activates the standard Windows pop-up message.
StrMessage - message string
mb_Buttons - parameter, defining the buttons in the MessageBox.
Can take one of the following values:
MB_OK - only OK button
MB_OKCANCEL - OK/Cancel buttons
MB_YESNO - Yes/No buttons
mb_Icons - parameter, defining the icons in the MessageBox.
Can take one of the following values:
MB_ICONWARNING - warning
MB_ICONINFORMATION - information message
MB_ICONERROR - error
MB_ICONQUESTION - question
The function returns the following values according to which button is pressed:
IDOK - OK button was pressed
IDCANCEL - Cancel button was pressed
IDYES - Yes button was pressed
IDNO - No button was pressed
Example:

```
MessageBox("Connection error!", MB_OK, MB_ICONERROR);
```

Message in the notifier

ShowMessage("Text"[, npIcon, objectID]);
Shows the notification message with the specified ***Text*** text.
npIcon can take the following values:
npSimple == -1, // without icon
npUsual == 0, // "page"
npWarning == 1, // " exclamation mark "
npCure == 2, // "cross"
npHint == 3, // "lamp"
npError == 4 //error
If the objectID was specified, the message with the button for editing the object is specified as the ***objecteID.***

Note:	Message with the ***objectID*** cannot be sent during the insertion process or the dialog activation. The setting "Show pop-up messages" does not affect the messages with the ***npError*** key.

ShowValue function

ShowValue(StrMessage,Value);
The function shows the values of the transferred (***value***) parameter as an argument, with the ***strMessage*** string to the window of the special debugger - InDebMon.exe. Any parameter or variable can be used as a value.
Example:

```
ShowValue("pntOrigin", pntOrigin);
ShowValue("vecDirection", vecDirection);
ShowValue("WorkPlane WP1", WP1);
ShowValue("Dn", Dn);
ShowValue("******************", 1);
ShowValue("OnConnect Start", 1);
```

Functions to work with dependences

Function	Comment
ResetAllConstraint(idFrom = NULL)	Switches off the dependences, set in the last OnConnect, when the insertion of the object with the selection of several objects is used. ***idFrom*** - object identifier, from which the dependences should be switched off. By default it equals NULL; in this case, all dependencies from all objects are switched off.
SetParamConstraint(Parameter, idFrom, TYPE, expr, bBidirect = FALSE, idTo = objectID	Sets the parametric dependence. ***Parameter*** - parameter name ***idFrom*** - object identifier from which the dependence starts ***TYPE*** - the type of parametric dependence (***EXPR*** is only used) ***expr*** - expression, setting the parameter of the connected object ***bBidirect*** - checkbox, TRUE - two-sided dependence, FALSE (by default) - one-sided dependence. ***idTo*** - object identifier to which the dependence is set (By default is ***objectID*** - identifier of the current object) ***Example:*** SetParamConstraint(rDn, obj, EXPR, "obj.rDnE2", TRUE); SetParamConstraint(LTPlug, obj, EXPR, "obj.rLT");
SetGeomConstraint(	Sets the parametric dependence.

TYPE, SUBTYPE, IdFrom, PlaneTo, PlaneFrom, expr, bBidirect = FALSE, idTo = objectID);	***TYPE*** - type of the geometric dependence. Can take the following values: INSERT - Insertion MATE - Coincidence by plane AXIS - Coincidence by axis DIRECTION - Coincidence by direction (angular dependence) ***SUBTYPE*** - Dependence subtype. Can take two values: ***CODIRECT*** - codirectional dependence ***CONTRDIRECT*** - opposite directional dependence ***idFrom*** - object identifier, from which the dependence is set. ***PlaneTo*** - plane of the inserted object. ***PlaneFrom*** - plane of the connected object. ***expr*** - expression, setting the dependence parameter (for insert the ***INSERT*** - distance between planes) ***bBidirect*** - checkbox, TRUE - two-sided dependence, FALSE (by default) - one-sided dependence. ***idTo*** - object identifier, to which the dependence is set (by default equals the ***objectID*** - the identifier of the current object) ***Example:*** SetGeomConstraint(INSERT, CODIRECT, obj, WP1, obj.WP2, 0);
IsFixedParam(Parameter, bOnlyFixed =FALSE)	Function as used in the ***OnChangeParameters*** event handler. Checks if the dependence for the parameter was set and if the parameter was changed, if the ***bOnlyFixed*** is TRUE. Activation by default (from FALSE) is recommended when checking the parameters of objects with permanent geometry (screw). In this case, the function returns 1 if the parameter is connected by the dependence and changed its value; and 0 otherwise. Activation (from TRUE) is recommended for objects with changing geometry (tubes, barrels); the function returns 1 if the parameter is connected by the dependence. ***Example:*** if (IsFixedParam(WP1, TRUE)) { pntOrigin = Point(new.WP1); vecDirection = Vector(new.WP1); }

Function - event handlers, activated by the base of the application

Function	Comment
ActHeader	Description of the object, list of the parameters, setting checkboxes. *Activated every time the object is opened.*
OnInitialization	Initialization of the object. *Activated every time the object is called up.*
BeforeConnect	Execution of the previous actions before the selection of the objects during the insertion. *Activated upon insertion of the object.*
OnConnect	Setting of the dependences when connecting to other objects. *Activated upon insertion of the object.*
OnChangeParameters	Setting of new values for the object parameters by another object. *Activated when connected by the dependences, the parameters are changed.*
OnDialog	Editing the object parameters. *Activated when the dialog opens.*

OnMakeParameters	Final preparation of all parameters. *Activated before closing the object.*
SetGripPoint	Preparing the grip-points. *Activated when the object is selected.*
OnMoveGripPoint	Reaction to changing the grip-points position. *Activated whengrip-points are moved.*
OnAddObject	Setting of the inverse dependences to the connected objects. *Activated when insertion with selection of another object is executed.*
OnDialogChanged	Interactive process of the input values. *Activated every time the object parameters are changed in the dialog.*
OnInitSelect	Initialization of the prompt string and pop-up menu. *Activated before every step of the dynamical selection cycle of the parameters.*
OnSelectParam	Process of the values input from the keyboard or mouse. *Activated in the body of the of dynamical selection cycle of the parameters. Align values by grid.*
OnMenu	Menu handler. *Activated after the menu item is selected, created by the SetMenu function.*

Function for working with objects' identifiers

Function	Comment

setWorkId(index, id)	Saves the selected identifier in the special storage with the specified index.
	index - integer value (index >= 0).
	id - object identifier.
	Every object of the application, when created, gets a unique identifier like GUID in Windows. To get access to the Obj1 identifier of the object in the script use the Obj1.***objectID*** expression.
	Example:
	id1 = getWorkId(0);
	id2 = getWorkId(1);
	If (id1 == id2) MessageBox("Attempt to connect to one object two times!", MB_OK | MB_ICONWARNING);

getWorkId(index);	Returns the saved identifier.
	index - integer value (index >= 0).
	Example:
	if (obj.strTheName == "MyEnabledObject") {
	setWorkId(0,obj.objectID);
	}
	....

getObjData(WorkId,ObjName)	Function gets the public-parameters of the object, having the ***id***identifier from the database.In the case of successful execution it returns 1, otherwise 0. To access the received parameters, use the ***ObjName***.Param1

	expression etc.
getObjectConnectedTo(idOut, objToSerch, strExpression, ParamName);	Searches the objects, connected with the dependence, with the ParamName variable of the objToSerch object, satisfying the strExpression condition. Condition can be null.

Other functions

setGlobalParam(var, val);	Sets the global ***var*** variable in the ***val*** value.
	Example:
	`if(rLen2 <= B/4){`
	`  SetGeomConstraint(INSERT, CODIRECT, obj, WP1, obj.WP2, 0);`
	`  setGlobalParam(rSubType, CONTRDIRECT);`
	`} else if(abs(rLen1) <= B/4) {`
	`  SetGeomConstraint(INSERT, CODIRECT, obj, WP1, obj.WP1, 0);`
	`  setGlobalParam(rSubType, CODIRECT);`
	`  } else {`
	`  SetGeomConstraint(INSERT, rSubType, obj, WP1, obj.WP1, rLen1);`
	`};`

GridRound(b)	Align the values by grid. The parameter of the grid is specified in the parameters or in prompt parameters (Ctrl+Shft+Q by default);
	Example:
	`a = GridRound(b);`
	If the grid step is 5 and, b ==12.4, the function returns 10; If the b >= 12.5 function returns 15.
	`rDiameter = GridRound(abs(rYcoord*2));`

paramByName(strName)	Returns the parameter value by its name: ***Example:*** `strName = GetNearestPlane(obj); plane = paramByName(strName);`
getChildrenCount()	Returns the number of child objects
getChildId(nChild);	Returns the identifier of the ***nChild*** child object.

Appendix 3. Object Classes

For some types of objects in the database has its own class reserved:

Number	Icon	Description
200		Default object. By default, when creating a new object, class 200 is assigned. The developer can assign his own class to created objects. All classes of objects, except for the ones described below, will have the same icon.
600		Blueprints. All object classes from 600 to 650, except for the ones described below, will have the same icon.
601		Tables, specifications
630		System object

Number	Icon	Description
631		Collection
632		Object project construction site
633		Object - building site signs
634		Master of Engineering
645		Batch file, when clicked, the command stored in the comment is sent
646		Bolted, riveted connections
647		Callouts
648		Markers
649		Templates
650		Object groups
670		Window
671		Wall
672		Wall modifier
673		Wall object
674		Architecture object
675		Column
676		Opening
680		Door
750		Piles
751		Additional crane objects
752		Load chart
806		Formats
807		Stamps
809		Cover Sheet and Cover Templates
820		Connectors
821		Track
822		Equipment with connectors

Line Type nanoCAD Mechanica

nanoCAD Mechanica It operates nine types of lines:

№	Title	Type in the platform

№	Title	Type in the platform
1	Solid thick main line	Solid
2	The solid thin line	Solid
3	The wavy line	GOST 2.303 3
4	Dashed thin (axial) line	GOST 2.303 5
5	Cross-hatching line	GOST 2.303 4
6	The dashed line is thickened	GOST 2.303 6
7	Open line	Solid
8	Continuous thin line kinked	GOST 2.303 8
9	Dashed with two dots thin line	GOST 2.303 9

Each line corresponds to its number in nanoCAD Mechanica.
Adjust the color and thickness of lines made *"Settings - General settings - LineType"*.

Examples

Object - centre holes GOST 14034-74

```
SVersion = 2; // The first line of the script determines the version of the interpreter
// Description. It will be displayed in the Properties pane
// You can also sort the objects in the field
// Tabular data and mcqs
ObjectDescription = "center hole";

// This is the function description of the object
function ActHeader {
  NPart = 2; // The number of requests to a dynamic selection of parameters

  // This section contains descriptions of the parameters open object
  // Visible to other objects
  Public (
    // Define the parameter size hole - the diameter
    d, "diameter",

    // Plane to install, depending on the shaft end
    WP1, "the beginning of the plane"

    rValType, "Shaft Type" // Type of shaft, which connect the hole
            // Set for overlap on
            // Kinds and with a cut without cutting

    Lhole, "The depth of the holes," // This is the total hole depth
    Lthread, "Thread length" parameter // length thread
    // Parameters specific performances
    DF, "shaft diameter to form F",
    DH, "The diameter of the shaft to form H"
  );
```

```
  // Private parameters used to plot
  // And calculate the limits of other parameters
  Protected (seted, freeLen, bBlind, rSuppressPointHeight);

  // This section lists the parameters of the open,
  // Which will be accessible from the outside for editing
  // This is only classified as specific actions under
  // Changing the parameters of external functions defined in
  // OnChangeParameters
  Changeable (d, WP1, DF, DH, rValType, Lhole, Lthread);

  // Show dialog box to insert dynamic selection parameters
  OnDlgBeforeSelectParam = 1;
  // Do not regenerate suppression circuit after insertion
  ContourOnLine = 0;
  // This is the part of the shaft member
  IsAValPart = 1;
  // And a built-in part of the shaft
  InsidePart = 1;

  // This element - opening (this variable is used to
  // Dimensioning in Inventor)
  IsAHole = 1;
};

// Initialize the object parameters
function OnInitialization {
  // Load the table into the cache
  // Parameters table of the current line in the script is always available
  // Therefore loaded only one parameter d
  LoadInCache (d);
  // Variable seted is the default value UnknownValue
  // As initialized with this value
  // So, if we put the following test, this unit
  // Operators will be executed only once:
  if (seted == UnknownValue) {
    seted = 1; // Set seted 1 to the condition is not fulfilled
       // Again

    // Set initial values
    rValType = 1; // The hole is placed on the shaft portion
    rZOrder = 4220; // Ranking - initial value of

    // Select from a table initial values
    SelectInCache ("kFirst", "Form", "~", "FH", "d", "~", 10);
    //T.e. according to the syntax of the operator - selected the first record
    // Table where Form parameter is "FH", and a parameter
    // Hole diameter d is approximately equal to 10

    // Set, pursuant to which the object will be initially
    // Displays
    strDesignName = "Form F";
```

```
    // Set the initial value of arbitrary parameters
    Lhole = 30; // The length of the hole
    Lthread = Lhole-d / 2; // This is the minimum possible length of the thread

    // The variable that determines whether the hole deaf.
    // Default - blind.
    bBlind = TRUE;

    // This variable determines the value of the parameter is relative to
    // Height of the end point of the cone of suppression. (See the drawing)
    rSuppressPointHeight = tg (30) * (d-1.082 * p) / 2;
  };
};

// Setting function handles
function SetGripPoint {
  NGrip = 3; // Total 3 pens

  pntGrip0 = pntOrigin; // The first - at the insertion point

  // The second handle - the handle length of the threaded portion
  pntGrip1 = pntOrigin + vecDirection * Lthread;

  // Third handle - handle is the total length of the hole
  pntGrip2 = pntOrigin + vecDirection * Lhole;
};



// This function determines the reaction of the object on the movement of the handles
function OnMoveGripPoint {
// Here comes the variable NMovingGrip, equal to the index with a slide
// Handle the numbering of the array starts from zero.

  // When editing the first handle just move the item to
  // Where the cursor is with this pen. Ie move the insertion point
  // Where is the first pen
  if (NMovingGrip == 0) {
    pntOrigin = pntGrip0;
  };

  // This we stretch the length of the (second pen)
  // This defines the length of the threaded portion, depending on the current
  // Execution of holes
  if (NMovingGrip == 1) {

    // Determine the distance stretching as vector length,
    // Defines a point ruchki1 and ruchki0
    rDistance = vecLen (pntGrip1-pntGrip0);

    // For the execution (shaped hole) "P"
```

```
    if (Form == "P") {
     // The length of the thread should be limited to reduce the amount of
     // (L-d / 2). Obviously, the length parameter thread Lthread
     // Will be set to the relative position of the cursor
     // If it is greater than (L-d / 2).
     Lthread = max (rDistance, L-d / 2);
    };

    // For the execution of "F H"
    if (Form == "F H") {
     // The same goes for the execution of the thread length must be
     // Size is bounded below by (l + d / 2).
     Lthread = max (rDistance, l + d / 2);
    };

    // Finally, in this condition, we limit the length of the thread
    // On, since the length of the thread can not be more
    // The length of the hole. Therefore, we will pull out all the hole
    // According to the new diameter of the thread.
    if (Lthread > Lhole-d / 2) {
     //T.e. when you change the length greater than the length of the thread holes
     // Set the new length of the holes equal
     // Thread length plus half the diameter
     Lhole = Lthread + d / 2;
    };
  };

  // This section examines the movement of the third handle.
  // The result of editing it is increasing the length of the hole,
  // Without affecting the thread.
  if (NMovingGrip == 2) {

    // In this case, the current distance is defined as
    // Length of the vector formed by dots and ruchki0 ruchki2
    rDistance = vecLen (pntGrip2-pntGrip0);

    // For the execution of "P"
    if (Form == "P") {
     // The length of the hole is limited below the parameter L
     Lhole = max (rDistance, L);
    };

    // Dlyaispolneniya "F H"
    if (Form == "F H") {
     // Length of the restricted orifice parameters (l + d)
     Lhole = max (rDistance, l + d);
    };

    // If you change the length of the hole is less than the length of the thread
    // Move thread abroad
    if (Lhole > Lthread + d / 2) {
     // That is, if the length of the opening is smaller than Lthread + d / 2,
```

```
    // Recalculate the new thread length which is less than
    // Current length of the holes in the d / 2
    Lthread = Lhole-d / 2;
   };
  };

};

// Function of the final calculation of parameters
function OnMakeParameters {
  // Name, surname and patronymic of the object
  strTheName = "CenterHole"; // Centering Hole
  strTheType = "ArborParts"; // Refers to detail shafts
  strTheSubType = "Simple"; // Type of opening - easy

  // Set the position of the plane WP1. It's just a plane
  // The base point at the insertion point and a normal vector,
  // Coincides with the vector direction of insertion
  WP1 = Plane (pntOrigin, vecDirection);

  // Realize the overlap for different occasions:
  // If the variable rValType, specifying the type section of the shaft, at the
  // Set that the hole is equal to 1 (this is a simple
  // Portion of the shaft), the order of the overlap is equal to 4220
  // (This number is given on the basis of the values of the order of
  // Portions of the shaft, so that the hole overlaps the shaft to its
  // Loop suppression
  if (rValType == 1) rZOrder = 4220;
  // For internal shaft portions and other portions where rValType not equal to 1
  else rZOrder = 2150;
  // The sequence will be different


  // This parameter constraints are found, depending on the version
  // For the execution of "P", more precisely, for the current record in the table,
  // Form that meets the hole "P".
  if (Form == "P") {
    // Limits the length of the holes parameter L
    Lhole = max (Lhole, L);
    // Limits the length of the thread parameter (Ld / 2)
    Lthread = max (Lthread, L-d / 2);

    // Define strDesignName in "Form P"
    //t.e. choose what the two-dimensional design, which corresponds to
    // The current parameters of the table
    strDesignName = "Form P";
  };

  // The same goes for the form "F H"
  if (Form == "F H") {

    // Limit the parameters of the holes and the length of the thread length
```

```
    Lhole = max (Lhole, l + d);
    Lthread = max (Lthread, l + d / 2);

    // This reassurance, if it does the user chose
    // Execution "P" in the two-dimensional forms, since the table
    // Exhibited parameter value in Form "FH".
    // In general, if you select the wrong version, the set
    // First correct.
    if (strDesignName == "Form P") {
     strDesignName = "Form F";
    };
   };

   // Limiting the length of the thread from the top, depending on the
   // Total length of the hole
   Lthread = min (Lthread, Lhole-d / 2);

   // The holes form P always deaf
   if (Form == "P") {bBlind = TRUE;};

   // This is to ensure that the correct path is drawn suppression
   // The schedule has a contour suppression, the position of one of the points
   // Which is specified by rSuppressPointHeight
   // So as to whether the user has selected the display
   // Blind or through-hole, set the parameter
   // RSuppressPointHeight desired value
   if (bBlind == TRUE) {// For blind hole
    rSuppressPointHeight = tg (30) * (d-1.082 * p) / 2;
   } Else {// For the through-hole (ie, not from the cone drill)
    rSuppressPointHeight = 0;
   };

   // This line simply sets specifications
   // Depending on the version adds the necessary letter
   // To the designated holes.
   if (strDesignName == "Form F") {strLetterForm = "F";};
   if (strDesignName == "FormaH") {strLetterForm = "H";};
   if (strDesignName == "FormaP") {strLetterForm = "P";};

   // Parameter strPartName - a string specification holes.
   // In addition, he also displayed hinte.
   strPartName = "Ed. Center." + strLetterForm + "M" + d + "GOST14034-74";
   // Format string automatically, ie
   // When adding a string value with
   // Numerical value is automatically converted to a string
};

// This function determines the type of dialogue the insertion hole
function OnDialog {
  // Load the parameters of the cache work table
  LoadInCache (d);
  // Call the dialogue with the key
```

```
  UniDialog (
    // This defines the fields of the table hole
    // The shape and diameter. Output as a table defining the parameters
    DBFLD, Form, d,
    // The diameter of the shaft to form F and Form H
    // Output only as reference parameters
    DBINF, DF, DH,
    // Parameters asked randomly (subject to restrictions, of course)
    // Display on the Properties tab
    VFLD,
     Lthread, "thread depth"
     Lhole, "The depth of the hole"
    // Tag blind or through hole
    // Display on the properties tab in the form of a tick
    BFLD,
     bBlind, "Deaf"
    // Choices performances. Displays all performance
    VIDS, strDesignName, "All",
    // Displays all recognized types
    TVIDS, lViewType, "All",
    // Displays preview
    VIEW, "Vids"
  );
};

// This function is performed before the dynamic selection of the parameters
function OnInitSelect {
  // This function comes variable rPart - sequence number
  // Prompt you to choose the dynamic parameters. counting begins at 0
  if (rPart == 0) {
    // For the first case indicate the diameter of the thread
    // Output a corresponding hint
    strPromt = "Enter the depth of the thread";
  };
  if (rPart == 1) {
    // For the second case indicates the depth of the hole (it zhe
    // Total length).
    strPromt = "Enter the hole depth";
  };
};

// This function defines the behavior of the object in the dynamic selection of the parameters
function OnSelectParam {
  // For the first cycle of selection parameters. In principle, the limitations
  // The same as for the handles.
  // These variables come rXcoord and rYcoord - relative
  // Abscissa and ordinate of the cursor
  if (rPart == 0) {

    // Parameter range is determined
    // Module relative abscissa
    rDistance = Abs (rXcoord);
```

```
    // Limitations on choice of parameters depending on the version
    if (Form == "P") {
     Lthread = max (rDistance, L-d / 2);
    };
    if (Form == "F H") {
     Lthread = max (rDistance, l + d / 2);
    };
    if (Lthread > Lhole-d / 2) {
     Lhole = Lthread + d / 2;
    };
   };

   // For the second request
   if (rPart == 1) {

    // Parameter distance
    rDistance = Abs (rXcoord);

    // Restrictions.
    if (Form == "P") {
     Lhole = max (rDistance, L);
    };
    if (Form == "F H") {
     Lhole = max (rDistance, l + d);
    };
    if (Lhole < Lthread + d / 2) {
  Lhole = Lthread + d / 2;
  };
  };
};

// This function is performed when the parameters in the dialog insert
function OnDialogChanged {
// Function adds interactivity in the form of dialogue.
// In this case we need to hide execution and daw "deaf"

// Operator ShowDesign hides or displays performance.
// Depending on the value of the first parameter (1 - Shows
// 0 - hides.
// Operator ShowBool hides and displays switches (tick)

// For a start, execution and show all daw
ShowDesign (1 "Form F");
ShowDesign (1, "Form H");
ShowDesign (1 "FormaP");
ShowBool (1, "deaf");

// If you choose such a line in the table where d3 is 0,
// It is necessary to hide the performance of "Form H". (As defined in the guest,
// That the performance is not used for such values
// Settings
```

```
if (d3 == 0) ShowDesign (0, "Form H");

// If the selected row corresponding to the shape of the hole "P", then
// To hide the other two performance (they can not be selected)
// And daw "blind" because the shape of P is always blind.
if (Form == "P") {
ShowDesign (0, "FormaH");
ShowDesign (0, "FormaF");
ShowBool (0, "deaf");
};

// But for the chosen form of "FH", you need only to hide
// Execution "Form F"
if (Form == "F H") {
ShowDesign (0, "Form P");
};
};

// Function OnChangeParameters performed under external influence
// Object. It is an attempt to establish a dependence or direct
// Change the parameters of the object properties panel
// This function determines the reaction of the object to such a change.
function OnChangeParameters {
// Here comes the object new, representing a
// The same object with the new value of open options.

// Random variable we can immediately assign new values
rValType = new.rValType;
Lhole = new.Lhole;
Lthread = new.Lthread;

// Here consistently given, how to choose a line from the table,
// The new value of the corresponding parameters.
// Check whether the changed parameter d
if (new.d! = d) {// ie. If the new value is not equal to the old
// Table is loaded into the cache
LoadInCache (d);
// And get out of it the first available record with a parameter
// D, approximately equal to the new value.
SelectInCache ("kFirst", "d", "~", new.d);
// Then we check not change any other parameters
} Else if (DH! = New.DH) {// change the diameter of the shaft H performance
// Load
LoadInCache (DH);
// Select the first record
SelectInCache ("kFirst", "DH", "~", new.DH);
} Else if (DF! = New.DF) {// change the diameter of the shaft version F
// Load
LoadInCache (DF);
// Select the first entry with an approximately equal value
SelectInCache ("kFirst", "DF", "~", new.DF);
};
```

```
// Default processing completed successfully.
//T.e. send the object to change the parameters that
// Set the necessary parameters.
// To do this, we assign the value of the variable Handled OBJ_HANDLED
Handled = OBJ_HANDLED;

// If after selecting from the table the shaft diameter is not established
// The new value, it may require the cyclical update
// Send a variable object variable
// Handled equal OBJ_WARNING
if (d! = new.d) {
Handled = OBJ_WARNING;
};

// In this block operators recalculated the new position of the object
// When you change its work plane

// Variable fix to test the ability to move.
// In general, in this case it could not be used
// Because the hole does not change when you change its geometry
// Planes.
// This block is generated script wizard, so it has
// This unified view

fix = 0;

// Check if the dependency has changed and whether the
// PloskostWP1
if (IsFixedParam (WP1)) {
// Increase the value of the variable fix, for the follow-up audit
fix = fix + 1;
// Save the old value of the vector insertion.
vecXOld = vecDirection;
// Assign new value to the vector inserts match
// To the new position of the plane WP1
vecDirection = Vector (new.WP1);
// And move the insertion point to a new position openings
// PloskostiWP1
pntOrigin = Point (new.WP1);

// Restore the coordinate system on the basis of
// Old and new values of the vectors
restoreBasis (vecXOld, vecPlane, vecDirection);
};
// If all of a sudden hole with fixed geometry
// Objects begin to stretch in different directions, then
// Return an error
if (fix > 1) {
   Handled = OBJ_ERROR;
  };
};
```

```
// This function is performed before each attempt to establish a relationship
function BeforeConnect {
  // Here we just reset according to the latest
  // The following function to establish new
  ResetLastConstraint ();
}

// This function sets dependency. More specifically,
// It determines exactly how to behave when an object
// Accession to another object
function OnConnect {
  // Check for the value of the variable rPart, defining
  // The sequence number of the request to join the parts.
  //T.e. if the relationship is set in two stages,
  // Or more, it is possible to install one piece
  // Depending on two or more parts.
  // But in this particular case, depending on established only
  // From one piece, so only one request and verification
  // RPart equal to 0
  if (rPart == 0) {
    // The function comes onConnect object obj - this is a detail,
    // To which we are trying to join. We can help qualifier
    // Access to the parameters of the parts to check whether
    // Automatically install dependencies or not.

    // Check whether the part to which are trying to join,
    // Shaft:
    if (obj.strTheType == "Arbor") {

      // Next - will set depending on the current execution
      // Holes. Ie different versions is implemented differently depending on the shaft.
      if (Form == "F H") {

        // Set for the opening of the shaft of the same type as that of the
        // Section of the shaft, which join (this is necessary for the implementation of
        // Overlap
        rValType = obj.rValType;

        // Find the nearest plane to the plane of the opening WP1:
        strNearestPlane = nearestPlaneName (WP1, obj.WP1, obj.WP2);
        // This takes into account only the plane WP1 and WP2 object
        //t.e. if these are not the nearest plane, strNearestPlane
        // Is uncertain.

        // Further - check the shaft type. for common areas (external):
        if (obj.rValType! = 1)
          // Set the geometric relationship
          // Type "insert" codirectional. By the object to which
          // Subscribe for planes WP1 details and found
          // The nearest plane. The distance between the planes - 0 mm.
          // Dependence unidirectional
```

```
        SetGeomConstraint (INSERT, CODIRECT, obj, WP1, strNearestPlane, 0, FALSE);

      // For the internal sections of shafts:
      else
        // Set the relationship with the same parameters,
        // But oppositely.
        SetGeomConstraint (INSERT, CONTRDIRECT, obj, WP1, strNearestPlane, 0, FALSE);

      // Set the parameter depending on the current execution
      // Holes.

      // To form the hole F must be equated
      // Parameter rDiameter (shaft diameter) to the diameter DF table hole
      if (strDesignName == "Form F") {
        // So if the test is performed, the
        // Set the parametric dependence on expression.
        // RDiameter equating to DF
        SetParamConstraint (DF, obj, EXPR, "obj.rDiameter");
      };

      // To form H is the same test, but establishes a relationship
      // Already diameter hole DH.
      if (strDesignName == "FormaH") {
        SetParamConstraint (DH, obj, EXPR, "obj.rDiameter");
      };

      NoVectorSelect = 1; // setting the variable NoVectorSelect 1
          // Disable the selection of the direction vector for details
          // Since been joined and established addiction
          // Then choose further it is not necessary.

      Handled = OBJ_HANDLED; // Set the variable Handled
        // Value OBJ_HANDLED, ie saying that according to
        // Set successfully.
    };

  };
 };
}
```

Blind GOST 16076-70

Script:

```
SVersion = 2;
ObjectDescription = "16076-70";

functionActHeader {
  NPart = 0;
  Public (
    // From outdoor settings need only determine the face
    // Diameter of a working surface for attachment
    // Material and reference weight.
```

```
    Dn,NOMINAL_DIAMETER,
    massa,MASS,
    sw0, @ MATERIAL,
    WP1, @ WORKING_PLANE1
  );

  Protected (seted, d, D, D1, D2, D3, D4, l, l1, L, type, type1);
  Changeable (Dn, WP1);
  OnDlgBeforeSelectParam = 1;
  ShowWhenSelPnt = 1;
  ContourOnLine = 0;
};

function OnInitialization {
  LoadInCache (Dn, d, D, D1, D2, D3, D4, l, l1, L, massa);
  if (seted == UnknownValue) {
    seted = 1;
    rZOrder = 100;
    // In the 6th version of all the titles in English versions
    // 1 = Implementation Ispolnenie1
    strDesignName = "Implementation 1";
    rd0 = 1;
    sw0 = 1;

    SelectInCache ("kFirst", "Dn", "~", 18);
  };
};

function SetGripPoint {
  NGrip = 1; // A handle at the insertion point
  pntGrip0 = pntOrigin;
};

function OnMakeParameters {
  // All localizable strings specified resource links
  // Starting with the dog.
  // Accordingly, they will be displayed depending on
  // The current application language.
  //
  strTheName = @ BLIND_GOST_16076_70;
  strTheType =ON_INTERNAL;
  strTheSubType =ON_INTERNAL;
  WP1 = Plane (pntOrigin + vecDirection * (0), vecDirection);


  if (rd0 == 0) type = "022";
  if (rd0 == 1) type = "012";

  if (sw0 == 1) type1 = "A"; else type1 = "";

};
```

```
function OnDialog {
  LoadInCache (Dn, d, D, D1, D2, D3, D4, l, l1, L, massa);
  UniDialog (
     // Type of dialogue - which determines the nominal diameter
     // And reference weight
     DBFLD, Dn,
     DBINF, massa,
     // Jackdaw overall performance
     BFLD, sw0,COMMON_IMPLEMENTATION,
     // Alternative Material
     RADIO, rd0, @ STEEL_45, @ 12X18H9T__X18H9T_,
     // These options affect the string specification covers
     VIEW, "Vids");
};

function OnDialogChanged {

};

function OnChangeParameters {
  if ((Dn! = new.Dn)) {
     LoadInCache (Dn, d, D, D1, D2, D3, D4, l, l1, L, massa);
     // When you change the diameter of the table, select the new value
     SelectInCache ("kFirst", "Dn", "~", new.Dn);
  };
  Handled = OBJ_HANDLED;
  if ((Dn! = new.Dn)) {
     Handled = OBJ_WARNING;
  };
  fix = 0;
  if (IsFixedParam (WP1)) {
     fix = fix + 1;
     vecXOld = vecDirection;
     vecDirection = Vector (new.WP1);
     pntOrigin = Point (new.WP1) - vecDirection * (0);
     restoreBasis (vecXOld, vecPlane, vecDirection);
  };
  if (fix > 1) {
     Handled = OBJ_ERROR;
  };
};

function BeforeConnect {
  ResetLastConstraint ();
}

strPartName = @ SPHERE_CAP_ + Dn + "-" + type + type1 + @ _ GOST_16076_70;
```

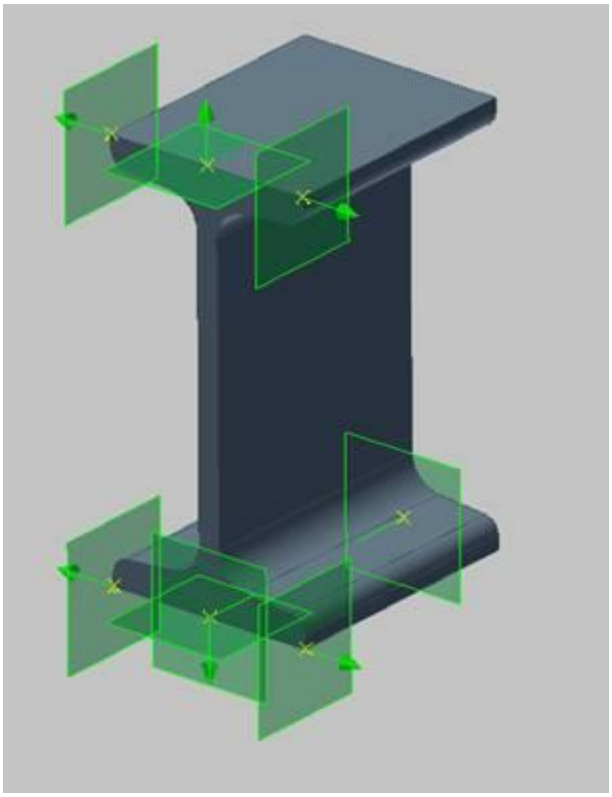
Steel welded neck GOST 12821-80

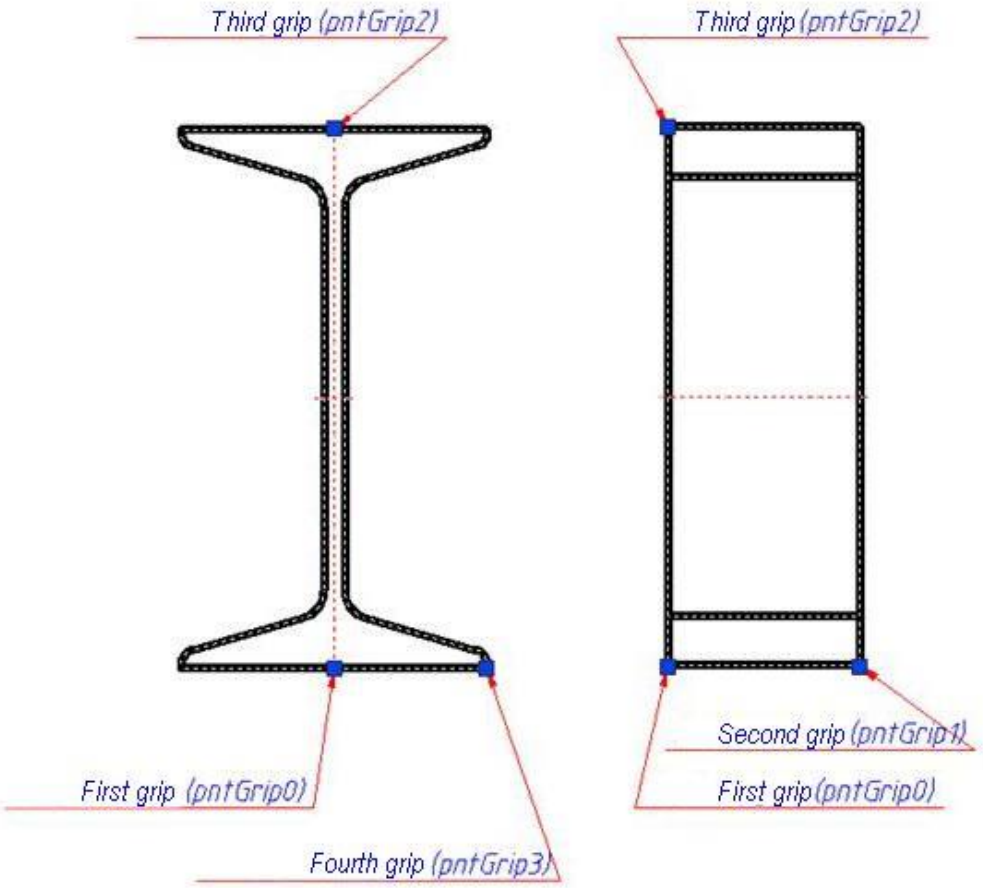
Script (only the BeforeConnect and OnConnect procedures)
Other can be seen in the database of the standard details.

```
// For flange necessary to geometric constraints
 // Have been installed on the gasket and parametric -
 // The second flange.
 function OnConnect {
   // When setting dependencies on multiple objects, we first
   // Store information about objects,
   // And then use it to install, depending

   // So, the first stage - the selection of the parameters
   // Here rPart is 0 (the serial number of the query begins
   // 0.
   // If this is the first request, and the selected object = laying GOST 15180,
   // Save the parameters of the object (pads) under the number 0
   if (rPart == 0 & & obj.strTheName == " GOST 15180-86 ") {
     setWorkId (0, obj.objectID);
     Handled = OBJ_HANDLED;
   };

   // For the second request - check further whether this
   // Object flange GOST 12821, and, if so, to save the parameters of this
   // Flange at number 1.
   if (rPart == 1 & & obj.strTheName == " GOST 12821-80 ") {
     setWorkId (1, obj.objectID);
     Handled = OBJ_HANDLED;
   };


   // Then consider the stored data objects:
   // If something was preserved in the first stage, that is, the data is not equal to 0
   if (getWorkId (0) = 0)
   {
     // Restore the data object variable obj 0
     getObjData (getWorkId (0), obj);
     // Closest to the insertion point plane
     strNearestPlane = GetNearestPlane (pntOrigin);
     // If the plane had been found,
     if (strNearestPlane! = UnknownValue) {
       // We set the geometric constraints of the insert.
       SetGeomConstraint (INSERT, CONTRDIRECT, obj, WP1, strNearestPlane, 0, TRUE);
       // Do not choose a direction vector
       NoVectorSelect = 1;
     };
   };

   // If in the latter case the data has been stored, then:
   if (getWorkId (1) = 0)
   {
     // Restore them back to the variable obj
     getObjData (getWorkId (1), obj);

     // Set a set of parametric dependencies
     // In nominal pressure, nominal size,
```

```
    // Form with a cut flange Series and a number of performances.
    SetParamConstraint (Py, obj, EXPR, " obj.Py ", TRUE);
    SetParamConstraint (Dy, obj, EXPR, " obj.Dy ", TRUE);
    SetParamConstraint (bSection, obj, EXPR, " obj.bSection ", TRUE);
    SetParamConstraint (R, obj, EXPR, " obj.R ", TRUE);
    SetParamConstraint (isp_row, obj, EXPR, " obj.isp_row ", TRUE);
    NoVectorSelect = 1;
  };

  // This block is realized when joining the flange to the pipe
  // The first request for a check type and subtype
  if (rPart == 0 & & obj.strTheType == " Tube " & & obj.strTheSubType == " Round ") {
    // Is the nearest plane
    strNearestPlane = GetNearestPlane (pntOrigin);
    if (strNearestPlane! = UnknownValue) {
      // Set geometric
      SetGeomConstraint (INSERT, CONTRDIRECT, obj, WP2, strNearestPlane, 0, TRUE);
      SetGeomConstraint (DIRECITON, CODIRECT, obj, WPnormal, obj.WPnormal, 0, TRUE);
      NoVectorSelect = 1;
    };
    // And parametric dependence
    SetParamConstraint (Dn, obj, EXPR, " obj.B ", TRUE);
    Handled = OBJ_HANDLED;
  };
};
```

I-profiles GOST 19425





I-profile script:

```
SVersion = 2;
ObjectDescription = @ GOST_19425_74;

function ActHeader {
  NPart = 1;
  Public (
     // Open options- defining the geometry: length, height
     // Width of the shelves.
     L,LENGTH,
     h,I_SHAPE_HEIGHT,
     b,FLANGE_WIDTH,
     // Reference: weight per meter and the total weight
     massa,MASS,
     mas1,RUNNING_METER_MASS,
     // Standard designation and a series of I-beam
     Designation, @ STANDART_DESIGNATION,
     seria, @ SERIES,

     // Working planes
```

```
    WP1,CROSS_PLANE,
    WP2, @ BOTTOM_PLANE,
    WP3, @ SIDE_BOTTOM_RIGHT_PLANE,
    WP4, @ SIDE_TOP_RIGHT_PLANE,
    WP5, @ TOP_PLANE,
    WP6, @ SIDE_TOP_LEFT_PLANE,
    WP7, @ SIDE_BOTTOM_LEFT_PLANE,
    WP8, @ CROSS_PLANE

  );


  Protected (seted, massa, bHid, seria);

  Changeable (L, b, h, WP1, WP2, WP3, WP4, WP5, WP6, WP7, WP8, Designation);

  OnDlgBeforeSelectParam = 1;
  ShowWhenSelPnt = 1;
  ContourOnLine = 0;
  NotStdBody = 1;
};

function OnInitialization {
  LoadInCache (seria, Designation, h, b);
  if (seted == UnknownValue) {
    seted = 1;
    rZOrder = 100;
    bHid = 0;
    L = 50;
    strDesignName = "Implementation 1";
    SelectInCache ("kFirst", "seria", "~", "C", "h", "~", 100, "b", "~", 10, "Designation", "~", "1") ;
  };
};

function SetGripPoint {
  // At four I-beam handle
  NGrip = 4;
  // Handles are situated at the extreme points of the model I-beam
  // The insertion point
  pntGrip0 = pntOrigin;
  // On the opposite end
  pntGrip1 = pntOrigin + L * vecDirection;

  // At the midpoint of the top shelf
  pntGrip2 = pntOrigin + h * vecPlane;

  // At the extreme point of the bottom shelf
  vecNormal = getLocalNormal (vecDirection, vecPlane);
  pntGrip3 = pntOrigin + vecNormal * b / 2;
};

function OnMoveGripPoint {
```

```
  // The first carrying handle entire I-beam
  if (NMovingGrip == 0) {
    pntOrigin = pntGrip0;
  };

  // The second handle stretches the length of the I-beam and determines
  // Direction vector insertion I-beam
  if (NMovingGrip == 1) {
    L = max (vecLen (pntGrip0 - pntGrip1), 1);
    vecDirection = pntGrip1 - pntGrip0;
    LoadInCache (seria, h, b);
    SelectInCache ("kFirst", "seria", "~", "20", "h", "~", h, "b", "~", b);
  };

  // Handle height of an I-beam
  // At its height stretched selected new I-beam,
  // Roughly equal relative distance from the cursor point
  // Insert parts.
  if (NMovingGrip == 2) {
    // Auxiliary variable height of the new I-beam.
    rNew_h = vecLen (pntGrip0 - pntGrip2);

    LoadInCache (seria, h, b);
    SelectInCache ("kFirst", "seria", "~", "20", "h", "~", rNew_h, "b", "~", b);
  };

  // Handle I-beam flange width
  // If it is stretched selected new I-beam flange width,
  // Approximately equal to twice the relative distance
  if (NMovingGrip == 3) {
    // A new auxiliary variable distances.
    rNew_b = 2 * vecLen (pntGrip0 - pntGrip3);

    LoadInCache (b, h);
    SelectInCache ("kFirst", "b", "~", rNew_b, "h", "~", h);
  };

};

function OnMakeParameters {
  strTheName = "19425-74";
  strTheType = "Profile";
  strTheSubType = "I-Shape";

  massa = mas1 * L / 1000;

  // Find the right third vector in the workpiece coordinate system,
  // As the vector product
  vecNormal = getLocalNormal (vecDirection, vecPlane);

  // We place the plane in order counterclockwise
  // The plane of the first end
```

```
  WP1 = Plane (pntOrigin + vecDirection * (0), vecDirection);
  // The plane of the second end
  WP8 = Plane (pntOrigin + vecDirection * L, -vecDirection);
  // Plane from the middle of the bottom shelf.
  WP2 = Plane (pntOrigin-vecPlane * (0), -vecPlane);

  // Right side
  WP3 = Plane (pntOrigin + vecNormal * (b / 2), vecNormal);
  WP4 = Plane (pntOrigin + vecNormal * (b / 2) + vecPlane * h, vecNormal);

  // The plane from the middle of the upper shelf
  WP5 = Plane (pntOrigin + vecPlane * (h), vecPlane);

  // Left Side
  WP6 = Plane (pntOrigin-vecNormal * (b / 2) + vecPlane * h, -vecNormal);
  WP7 = Plane (pntOrigin-vecNormal * (b / 2), -vecNormal);
};

function OnDialog {
  // For beauty
  strRefDataHeader = "------";
  LoadInCache (seria, Designation, h, b);
  // Type of dialogue
  UniDialog (
    // Define the parameters of a series of height and width of the shelves
    DBFLD, seria, h, b,
    // Reference parameters - designation and weight per meter
    DBINF, Designation, mas1,
    // Arbitrary length parameter I-beam
    VFLD, L, @ I_SHAPE_LENGTH__MM,

    // Making reference block settings page
    // Arbitrary properties
    strRefDataHeader, @ REFERENCE_DATA_,
    A, @ A__CM2,
    Ix,IX__CM,
    ix,IX__CM,
    Wx, @ WX__CM3,
    Sx, @ SX__CM3,
    Iy, @ IY__CM4,
    Wy, @ WY__CM3,
    iy,IY__CM,

    // Jackdaw display hidden lines
    BFLD, bHid,HIDDEN_LINES,
    // Display types AnyWBK generation means
    // Mirror species, if they are not overridden performances
    TVIDS, lViewType, "AnyWBK",

    VIEW, "Vids"
  );
};
```

```
function OnDialogChanged {
  ShowBool (0,HIDDEN_LINES);
  // Check the "invisible line" is only required for those species
  // For which they have.
  if (lViewType == VLEFT || lViewType == VRIGHT)
     {ShowBool (1,HIDDEN_LINES)};
};


function OnChangeParameters {
  // To change the settings:
  // Arbitrary simply assigned new values
  L = new.L;

  // For table or selected by a combination of the defining parameters
  if (new.b! = b || new.h! = h) {
     LoadInCache (h, b);
     SelectInCache ("kFirst", "h", "~", new.h, "b", "~", new.b);
  // Or by defining a standard notation field
  } Else if (Designation! = New.Designation) {
     LoadInCache (Designation);
     SelectInCache ("kFirst", "Designation", "~", new.Designation);
  };


  Handled = OBJ_HANDLED;
  fix = 0;

  // The response to the change of position of the planes.
  // Is to recalculate the new position of the insertion point
  // And directions

  // Under the conditions IsFixedParam units can be grouped into
  // Options direction of the plane.

  // Surface normal collinear vecDirection
  if (IsFixedParam (WP1)) {
     fix = fix + 1;
     vecXOld = vecDirection;
     vecDirection = Vector (new.WP1);
     pntOrigin = Point (new.WP1) - vecDirection * (0);
     restoreBasis (vecXOld, vecPlane, vecDirection);
  };
  if (IsFixedParam (WP8)) {
     fix = fix + 1;
     vecXOld = vecDirection;
     vecDirection = -Vector (new.WP8);
     pntOrigin = Point (new.WP8) - vecDirection * (L);
     restoreBasis (vecXOld, vecPlane, vecDirection);
  };
```

```
  // Surface normal collinear vecPlane
  if (IsFixedParam (WP2)) {
     fix = fix + 1;
     vecYOld = vecPlane;
     vecNormal = getLocalNormal (vecDirection, vecYOld);
     vecPlane = -Vector (new.WP2);
     pntOrigin = Point (new.WP2) + vecPlane * (0);
     restoreBasis (vecYOld, vecNormal, vecPlane);
     vecDirection = getLocalNormal (vecPlane, vecNormal);
  };
  if (IsFixedParam (WP5)) {
     fix = fix + 1;
     vecYOld = vecPlane;
     vecNormal = getLocalNormal (vecDirection, vecYOld);
     vecPlane = Vector (new.WP5);
     pntOrigin = Point (new.WP5) - vecPlane * (h);
     restoreBasis (vecYOld, vecNormal, vecPlane);
     vecDirection = getLocalNormal (vecPlane, vecNormal);
  };


  // Surface normal collinear vecNormal
  if (IsFixedParam (WP3)) {
     fix = fix + 1;
     vecZOld = getLocalNormal (vecDirection, vecPlane);
     vecZNew = Vector (new.WP3);
     vecYNew = getLocalNormal (vecDirection, vecZNew);

     restoreBasis (vecZOld, vecPlane, vecZNew);

     pntOrigin = Point (new.WP3) - vecZNew * (b / 2);
     vecDirection = getLocalNormal (vecZOld, vecYNew);
  };

  if (IsFixedParam (WP4)) {
     fix = fix + 1;
     vecZOld = getLocalNormal (vecDirection, vecPlane);
     vecZNew = Vector (new.WP4);
     vecYNew = getLocalNormal (vecDirection, vecZNew);

     restoreBasis (vecZOld, vecPlane, vecZNew);

     pntOrigin = Point (new.WP4) - vecZNew * (b / 2) + vecYNew * h;
     vecDirection = getLocalNormal (vecZOld, vecYNew);
  };

  if (IsFixedParam (WP6)) {
     fix = fix + 1;
     vecZOld = getLocalNormal (vecDirection, vecPlane);
     vecZNew = -Vector (new.WP6);
     vecYNew = getLocalNormal (vecDirection, vecZNew);
```

```
    restoreBasis (vecZOld, vecPlane, vecZNew);

    pntOrigin = Point (new.WP6) + vecZNew * (b / 2) + vecYNew * h;
    vecDirection = getLocalNormal (vecZOld, vecYNew);
  };


  if (IsFixedParam (WP7)) {
    fix = fix + 1;
    vecZOld = getLocalNormal (vecDirection, vecPlane);
    vecZNew = -Vector (new.WP7);
    vecYNew = getLocalNormal (vecDirection, vecZNew);

    restoreBasis (vecZOld, vecPlane, vecZNew);

    pntOrigin = Point (new.WP7) + vecZNew * (b / 2);
    vecDirection = getLocalNormal (vecZOld, vecYNew);
  };

  if (fix > 1) {
    Handled = OBJ_ERROR;
  };
};

function BeforeConnect {
  ResetLastConstraint ();
};

function OnSelectParam {
  // The dynamic selection of parameters.
  if (lViewType == VTOP || lViewType == VBOTTOM)
    // Do not choose the parameters for the types of side
    {BreakAll = 1;}
  else {
    // For other species to choose the length of an I-beam in the relative
    // Cursor abscissa
    L = max (abs (rXcoord), 1);
    };
};

// Formation of the line specifications, as the sum of substring
strPartName =I_PROFILE_N_ + Designation + "x" + L + @ _MASS_ + massa + @ _GOST_19425_74;
```

Panel HB

Script of the detail:

```
SVersion = 2;
ObjectDescription = @ MCS_STRING6;

function ActHeader {
  NPart = 1;
  Public (
```

```
    // Options- exterior length and width of the plate
    L,LENGTH,
    W,WIDTH,
    // Design load and weight plates
    P, @ MCS_STRING9,
    massa,MASS
  );
  Protected (seted, obozn, H);
  Changeable ();
  OnDlgBeforeSelectParam = 1;
  ShowWhenSelPnt = 1;
  ContourOnLine = 1;
};

function OnInitialization {
  LoadInCache (obozn, L, W, H, massa, P);
  if (seted == UnknownValue) {
    seted = 1;
    rZOrder = 100;
    // Set the initial values of parameters and execution
    SelectInCache (" kFirst ", " L ", " ~ ", 2700, " W ", " ~ ", 1190);
    strDesignName = " Implementation 1 ";
  };
};

function SetGripPoint {
  NGrip = 2;
  Two handles // - at the insertion point and with the opposite end
  // Panel
  pntGrip0 = pntOrigin;
  pntGrip1 = pntOrigin + L * vecDirection;
};

function OnMoveGripPoint {
  // When editing handles the following behavior:
  // Move the entire object when you drag over
  // First handle. And stretch the length and change the vector
  // Direction when stretching a second handle
  if (NMovingGrip == 1) {
    // Intermediate variables to define the geometry
    rW = W;
    rP = P;
    // Relative length
    rL = vecLen (pntGrip0 - pntGrip1) / rScl;
    // The new direction of the vector inserts
    vecDirection = pntGrip1 - pntGrip0;

    LoadInCache (P, W, L);
    // Select the new settings plate.
    SelectInCache (" kFirst ", " P ", " ~ ", rP, " W ", " = ", rW, " L ", " ~ ", rL);
  } Else {
    // If you edit the handle to move the 0 point
```

```
    // Insert an object
    pntOrigin = pntGrip0;
  };
};

function OnMakeParameters {
  // Classifications plates
  strTheName = " Panel NV ";
  strTheType = " Plita ";
  strTheSubType = " Plita ";
};

function OnDialog {// insert type of dialogue
  LoadInCache (obozn, L, W, H, massa, P);
  UniDialog (
    // The table control fields
    DBFLD, L, W, P,
    // The table reference field
    DBINF, obozn, H, massa,
    // Types - front, top, left
    TVIDS, lViewType, " F ", " T ", " L ",
    VIEW, " Vids ");
};

function OnDialogChanged {
  ShowDesign (0, " Preview ");
};

// Parameter changes
function OnChangeParameters {
  // Just when controlling parameter changes
  // Select from a table those which correspond to the new
  // Values
  if (new.L! = L || new.W! = W)
  {
    LoadInCache (L, W);
    SelectInCache (" kFirst ", " L ", " ~ ", new.L, " W ", " ~ ", new.W);
  }

  Handled = OBJ_HANDLED;
};

function OnSelectParam {
  // Function of the dynamic selection of parameters
  rW = W;
    // For type VTOP do not select dynamically
  if (lViewType == VTOP) {
    BreakAll = 1;
  } Else {
    // For other species
    // Selected variable rL - how
    // Abscissa the relative position of the cursor.
```

```
    rL = abs (vecCoord: x);
    // Selected values from the table with the same width,
    // But with a new value for the wavelength plate.
    SelectInCashe (" kFirst ", " W ", " = ", rW, " L ", " ~ ", rL);
  };
};

function BeforeConnect {
  ResetLastConstraint ();
}

function OnConnect
{
// Automatic installation of dependencies
// Set only parametric dependence.
// Between the plates along the length and width.
  if (rPart == 0)
  {
    if (obj.strTheType == " Plita ")
    {
      SetParamConstraint (L, obj, EXPR, " obj.L ");
      SetParamConstraint (W, obj, EXPR, " obj.W ");
      NoVectorSelect = 1;
      seted = 1;
    };
  };
}
// The name plate is formed on the basis of the table box
// Obozn
strPartName = obozn;
```

Specification object

In this example, we will consider setting up an object to correctly add it to the specification.
There is a script for inserting a rectangle:

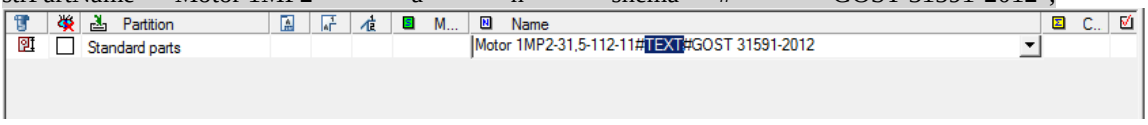
```
.....  .....  .....  .....  SVersion = 2;
.....  .....  .....  .....  ObjectDescription = "Example";

.....  .....  .....  .....  function ActHeader {
.....  .....  .....  .....  NPart=0;
.....  .....  .....  .....  Public(  );
.....  .....  .....  .....  Protected( seted, a, b );
.....  .....  .....  .....  Changeable( );
.....  .....  .....  .....  OnDlgBeforeSelectParam = 0;
.....  .....  .....  .....  ShowWhenSelPnt = 1;
.....  .....  .....  .....  ContourOnLine = 0;
.....  .....  .....  .....  };

.....  .....  .....  .....  function OnInitialization {
.....  .....  .....  .....  if(seted == UnknownValue) {
.....  .....  .....  .....  seted=1;
.....  .....  .....  .....  rZOrder=100;
```

```
.....  .....  .....  .....  a = 100;
.....  .....  .....  .....  b = 100;
.....  .....  .....  .....  };
.....  .....  .....  .....  };

.....  .....  .....  .....  function SetGripPoint {
.....  .....  .....  .....  NGrip = 1;
.....  .....  .....  .....  pntGrip0 = pntOrigin;
.....  .....  .....  .....  };

.....  .....  .....  .....  function OnMakeParameters {
.....  .....  .....  .....  strTheName = "Rectangle";
.....  .....  .....  .....  strTheType = "Rectangle";
.....  .....  .....  .....  strTheSubType = "Rectangle";
.....  .....  .....  .....  };

.....  .....  .....  .....  function OnDialog {
.....  .....  .....  .....  UniDialog( VFLD, a,"a", b,"b", TVIDS,lViewType,"All",VIEW,"Vids");
.....  .....  .....  .....  };

.....  .....  .....  .....  function OnDialogChanged {
.....  .....  .....  .....  ShowDesign(0, "Implementation1");
.....  .....  .....  .....  };

.....  .....  .....  .....  function OnChangeParameters {
.....  .....  .....  .....  Handled = OBJ_HANDLED;
.....  .....  .....  .....  };

.....  .....  .....  .....  function BeforeConnect {
.....  .....  .....  .....  ResetLastConstraint();
.....  .....  .....  .....  }
.....  .....  .....  .....
.....  .....  .....  .....  SpecPartition="Details"; strPartName = "Rectangle "+a+"x"+b;  .....
       .....  .....
```

For the correct addition of an object to the specification, added: a section of the specification and the name of the object in the specification.

The specification section is specified using the "SpecPartition" parameter. If the specified section is missing in the specification, the object will be added to the specification without the section. It is recommended to set the value using the resource table.

The object name in the specification is specified using the "strPartName" parameter. The value can be composite and generated using the *"Part list entry wizard"*.

When you add the "#" symbol to the value of the name of the object, in the position editor it will be possible to insert text between two lattices with subsequent saving. When exporting a part list, the "#" symbol will not be displayed.

```
strPartName = "Motor 1MP2" + "-" + a + "-" + n + "-" + shema + "#"+ " " + "GOST 31591-2012";
```