Table of contents

New features of nanoCAD 21.0 Platform

*The most relevant and full description of new features of nanoCAD 21.0 is available online.

License search information

Now the information on checking a license is displayed in the welcome window:

Paste special

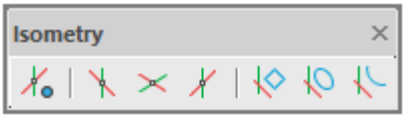
The **Paste special** command pastes objects in the current document, which allows managing data format.

Source	Contains information about the data copied to the clipboard. The kind of information depends on the type of data. For example, for Excel document data, it displays the layout name and the range of copied cells: Source: Layout1!R1C1:R3C3
Paste	The clipboard content will be pasted to the specified location in the drawing as an embedded object. Embedded data cannot be updated in the drawing workspace if modified in the original document from where they were inserted.
Paste Link	The clipboard content will be pasted to the specified location in the drawing as linked data. If the source application supports OLE or a data link, a link to the source file will be created. Linked data can be updated in the drawing workspace if modified in the original document from where they were inserted. OLE object links are updated and configured in the **Update OLE-links** dialog box of the **Update Data Links** command
As:	Formats in which you can paste the clipboard content into the current drawing. Depend on the type of data in the clipboard and the type of insertion (embed or bind).
Display As an Icon	Instead of data, the corresponding application icon will be placed in the drawing. To view or edit data, double-click the left mouse button on the icon.

Isometric drafting mode

A toolset has appeared that allows you to draw two-dimensional drawings in an isometric projection. A flat isometric drawing emulates a three-dimensional view of an object from a specific perspective, being, in fact, a flat representation of an isometric 3D-projection. The isometric drawing tool is convenient to use, when you need to create several simple isometric views in a two-dimensional drawing or edit an existing isometric drawing.

To create isometric drawings, it is convenient to use the **Isometry** toolbar.



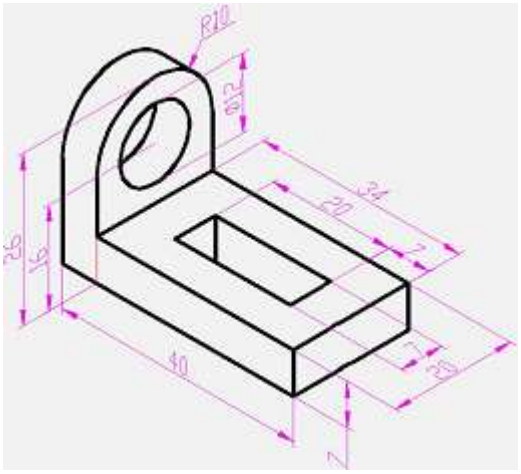
It contains:

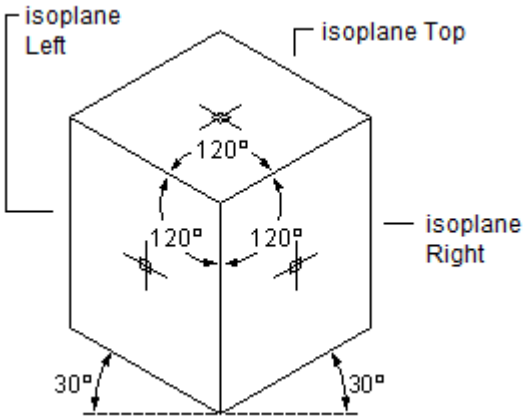
- The button to enable the isometric drawing mode;
- Three buttons to set isometric planes (ISODRAFT1, ISODRAFT2 and ISODRAFT3 commands or an option on ISODRAFT command);
- Buttons to create isorectangles, isocircles and isoarcs (rectangles, circles and arcs in an isometric projection).

Isometric drafting mode

For isometric drawing, use the Isometric drafting mode. The mode is enabled by entering the ISODRAFT command, by  button on the **Isometry** toolbar or by 

button in the status bar.





The isometry mode allows for quick switch between three isometric planes (isoplanes) by F5 key or by clicking an appropriate button on the **Isometry** bar.

-  Isoplane Left. The plane allows you to create objects oriented along the axes of 90 and 150 degrees.
-  Isoplane Right. The plane allows you to create objects oriented along axes of 30 and 90 degrees.
-  Isoplane Top. The plane allows you to create objects oriented along the axes of 30 and 150 degrees.

Impact of the mode on drawing settings and precision tools

Enabling (disabling) the isometric drawing mode, as well as setting this or that isoplane changes a number of related drawing parameters and settings, for example, drawing precision tools. The following settings and modes change automatically:

- orthogonal directions;
- snap orientation;
- grid orientation;
- polar tracking angles;
- orientation of isometric circles, arcs and rectangles when they are created.

Creating geometric entities

The following commands are used to create circles, arcs, rectangles in the current isometric plane:

- To create a circle in the current plane, use  ISOCIRCLE command that starts the ELLIPSE command with the **Isocircle** option;
- To create a rectangle in the current plane, use  ISORECTANGLE command that starts the RECTANGLE command with the **Isorectangle** option;
- To create an elliptical arc in the current isometric plane, use  ISOARC command that starts creating an elliptical arc by the ELLIPSE with the **Arc** > **Isoarc** option.

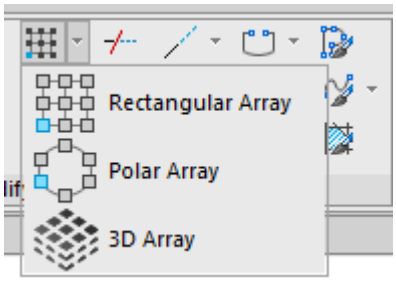
System variables

The SNAPSTYL system variable allows you to change the drawing mode. The variable values are integers.

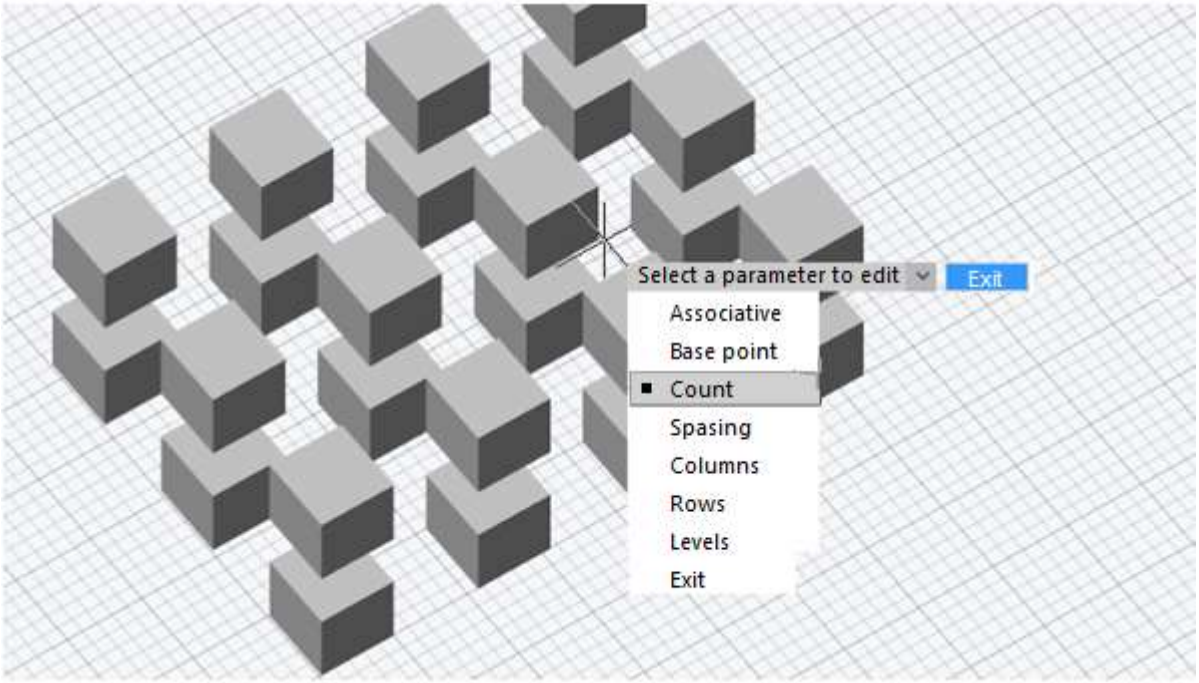
The SNAPISOPAIR system variable allows you to change the current isometric plane in the current viewport. The variable values are integers.

Associative arrays

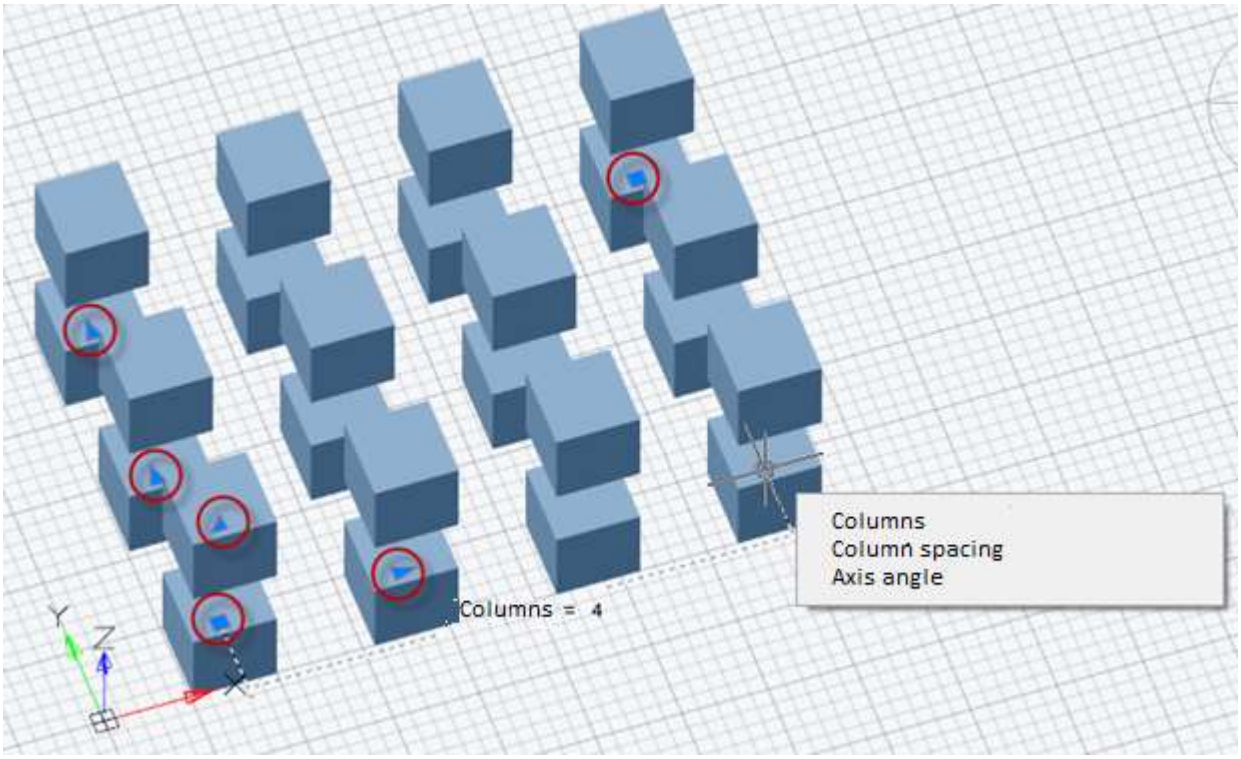
The commands for constructing rectangular and circular arrays have been completely redesigned.



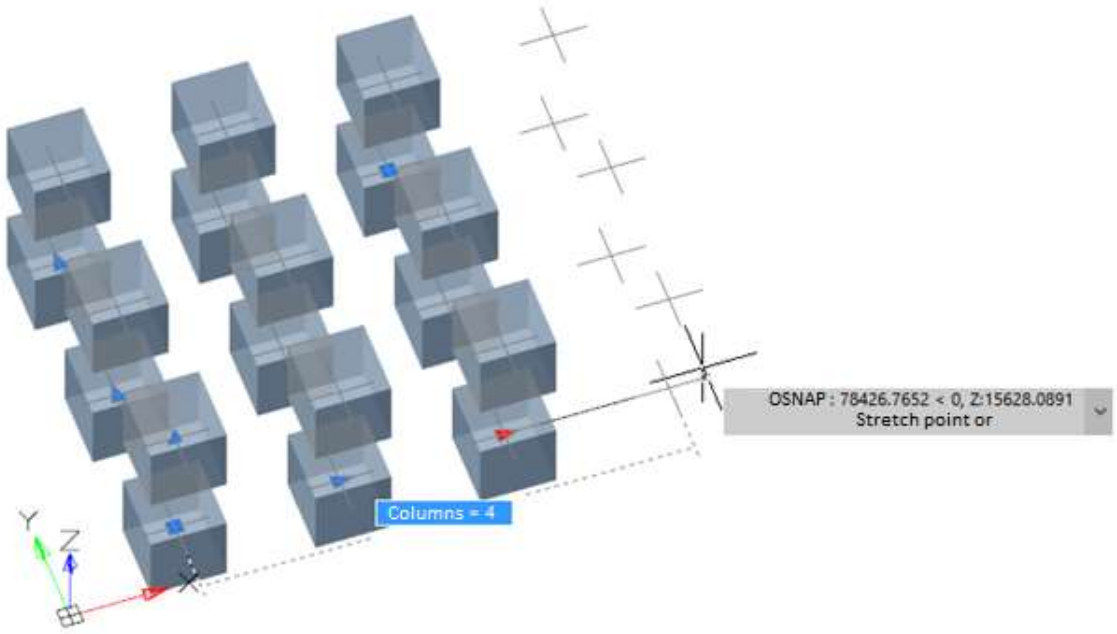
The commands work in a non-dialogue interactive mode.



The possibility to create associative arrays have been added.



The **Array - Associative** command option allows you to specify whether arrays are associative or non-associative ones. Elements of an associative array are stored in a single object – an array. In an associative array you can change the number of elements and the distance between them. Properties of an associative array, such as the number of elements and the distance between them, can be changed using the array grips or in the **Properties** toolbar.



Block Editor

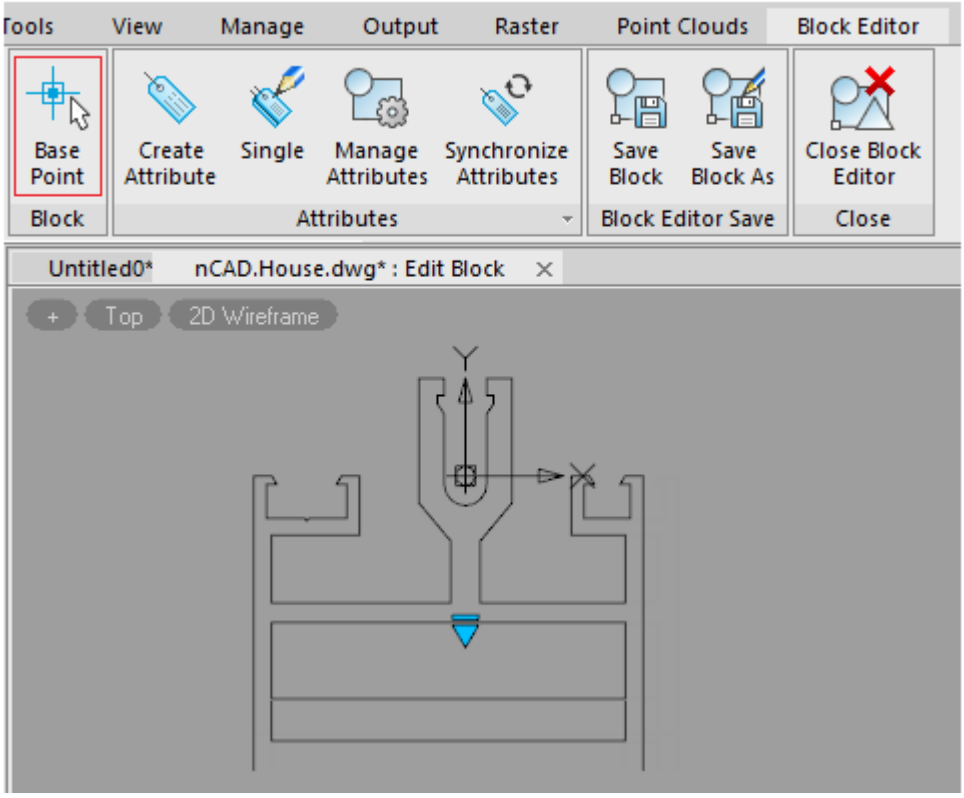
Now it became possible to:

- add remove and modify block objects while maintaining the integrity of dynamic block parameters. Actions concern objects that are not involved in the block parameterization;
- convert any block to dynamic;
- replace the insertion point.

Most of these features are available in the block editing mode by the **Block editor** command (BEDIT).

Specifying a base point for the block


Base point of the block (BBASEPT) – specifies the variable position of the block definition base point. Corresponds to the position of grip of the block reference insertion point.

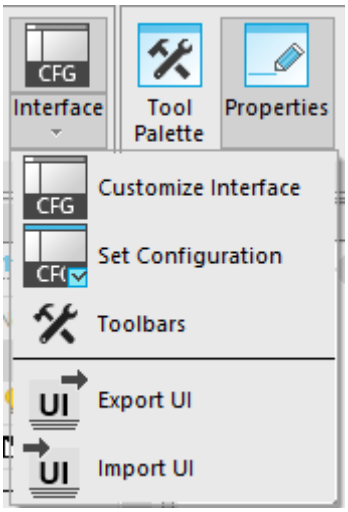


Reset block

During active work and modification of dynamic block inserts, followed by repeated editing of this block definition, distortions of the appearance of its inserts may occur. The command **Reset block** (RESETBLOCK) resets the parameters of the selected dynamic block inserts to default values. It is opened from the context menu of the selected inserts or from the command line.

Transfer of interface configuration

Now there is a possibility to transfer and save user interface configuration. To transfer settings between versions or from one computer to another, use two commands: **Import UI** (UIIMPORT) and **Export UI** (UIEXPORT).



The **Export UI** command allows you to save a package of individual configuration, while the **Import UI** command applies it to the current version of the program.

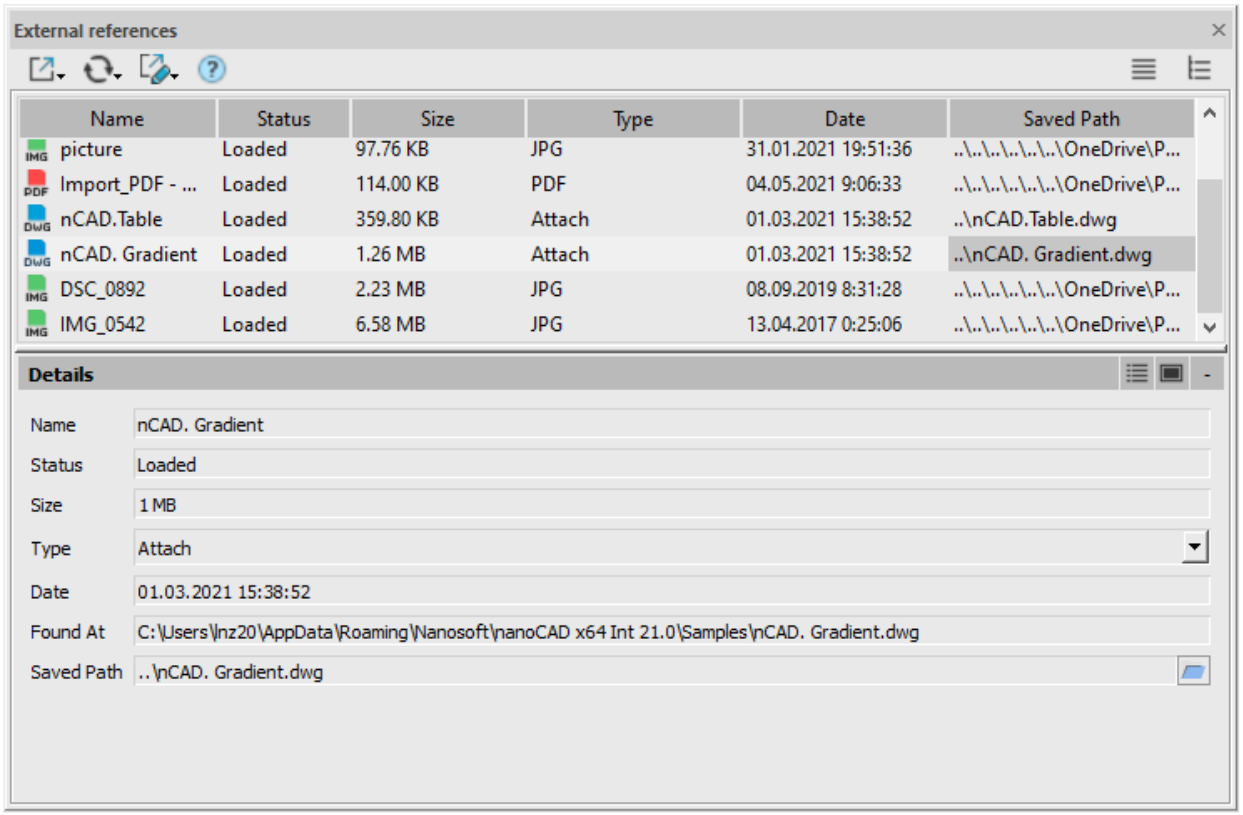
Export UI allows you to save both the composition of interface elements (ribbon, toolbars, etc.), configured in the **Customize user interface** dialogue box, and the display with location of interface elements (location of toolbars, palettes, command line, visual style of interface, way of the ribbon display, size and location of the program window).

Thus, you can quickly and easily transfer user configurations for convenient work with the program, as well as save the configurations package to prevent them from being lost.

If necessary, you can also edit LSP-file, which describes **Import UI** and **Export UI**, commands and edit the list of file formats included in the configuration package being exported.

Managing external references

The External reference manager has been redesigned in the form of modeless dialogue box (functional panel). This makes possible to work with external references in parallel with editing the drawing in real time mode.



An external reference selected in the drawing is marked in the dialogue box of external references.

The **External references** palette allows you to perform the following operations with the referenced drawing files:

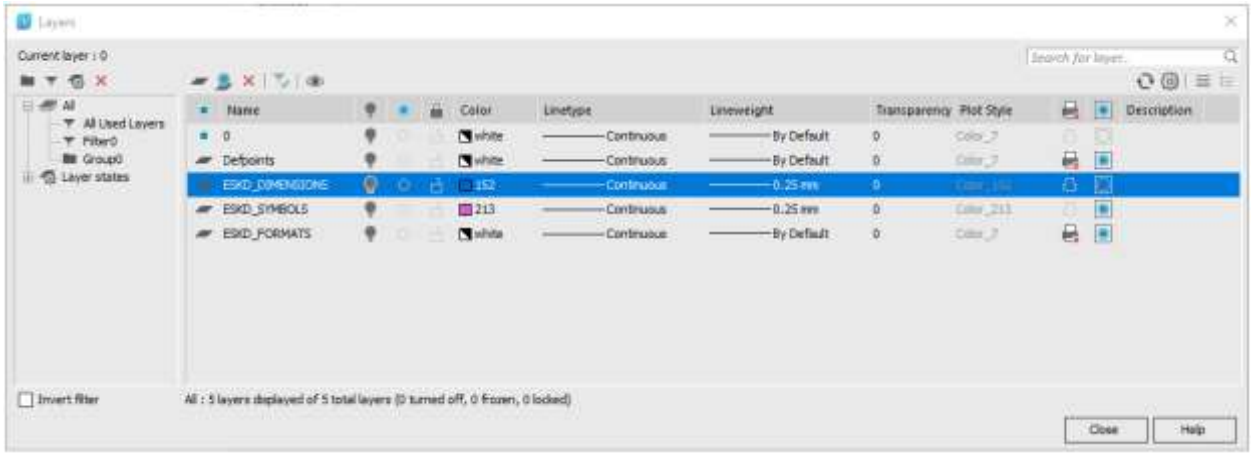
- Loading/unloading an external reference in the current drawing;
- Binding an external reference to the current drawing;

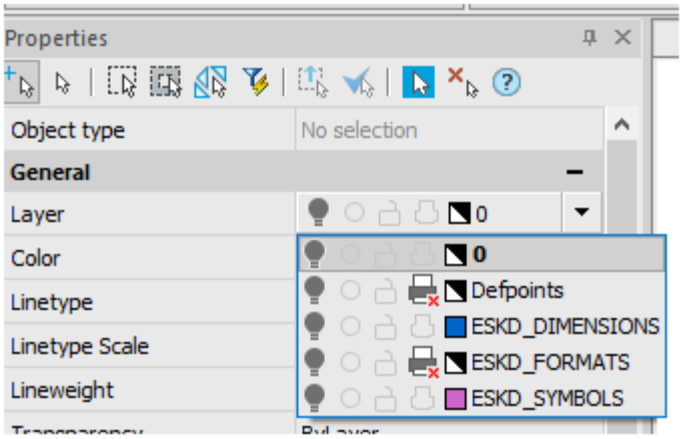
- Updating an external reference in order to display in the current drawing the latest changes made to the external reference file (without reloading the current drawing);
- Complete removing an external reference insert from the current drawing with all associated data. It is not sufficient to simply remove a reference from the drawing, since such removal would not lead, for example, to removal of layers associated with the external reference. To remove the external reference completely, use the **Remove** parameter on the **External references** palette;
- Changing the name of the reference file and its location (path);
- Changing the file type and the format settings of the file of reference to the raster image.

The number of missing external references displayed in a balloon message is now limited to twenty.

Managing layers

Changes in design



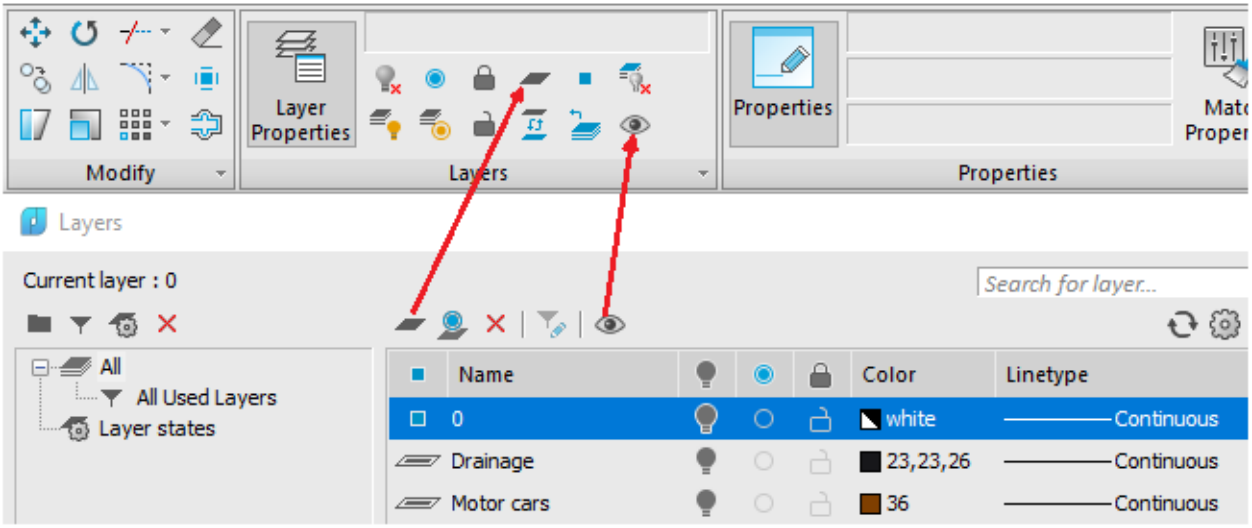


- All icons have been redesigned.
- Instead of a pair of icons for each state (for example, freeze/thaw), now there is one, the most important (frozen). The second state is made semi-transparent to show that you can click there to change the state.

- In the layer control the indents between rows and columns have increased, and the current layer is additionally marked with a square.
- Instead of a double arrow to enable-disable the tree, a switch has been made in the right side of the bar.

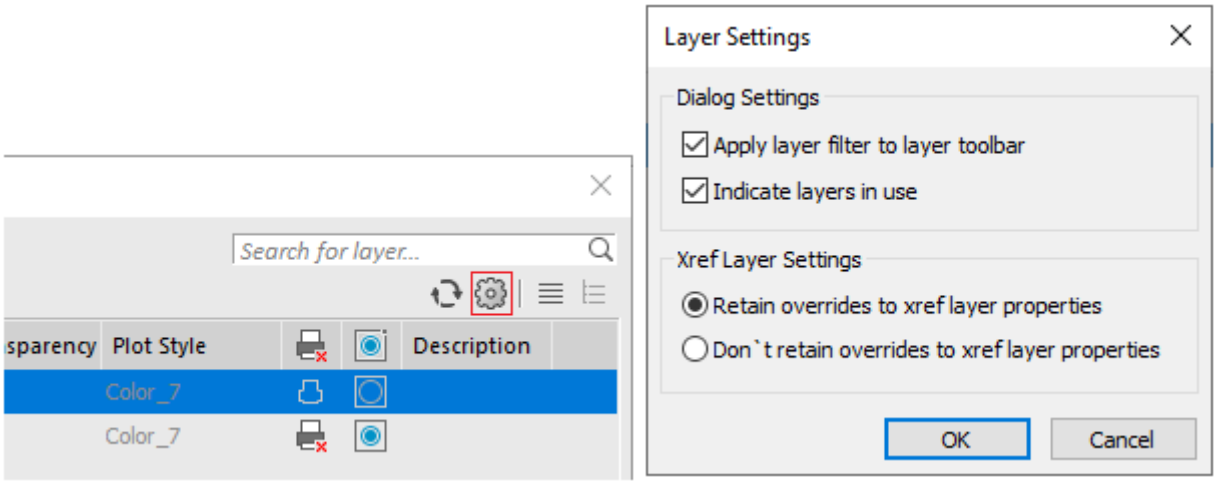
New commands

Creation of a new layer and the layer walk mode have been designed as separate commands **New layer** (NEWLAYER) and **Layer walk** (LAYWALK), which can be called both from the ribbon and the menu, and from the command line.



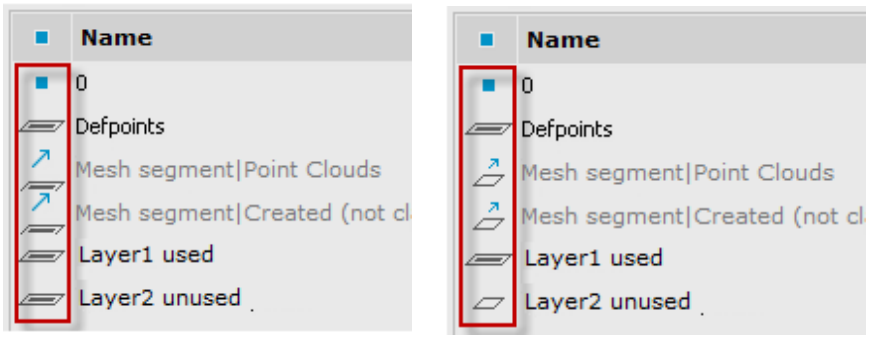
Display of layer setting in the manager

The  **Layer settings** button allows you to manage display settings in the **Layers** dialogue box and properties of layers included in external references:



- **Apply layer filter to layer toolbar** – whether the current filter is taken into account or not when displaying the list of layers on the toolbar. In previous software versions, the list was always displayed taking the filter into account.
- **Indicate layers in use** – whether to mark layers in use or not in the layer manager. Recalculation occurs when you open the manager, add/delete a layer. If this option is enabled, then opening

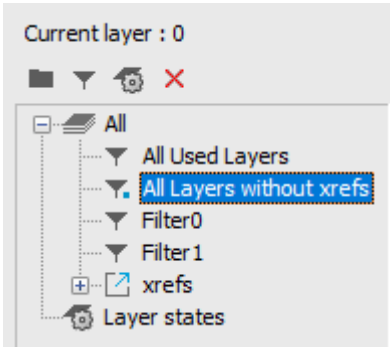
the layer manager in large files can take longer than expected. In previous software versions, recalculation was always performed.



- **Retain/Don't retain overrides to xref layer properties** – whether save or not the changes in properties of layers included in the external references.

Filtering xref layers

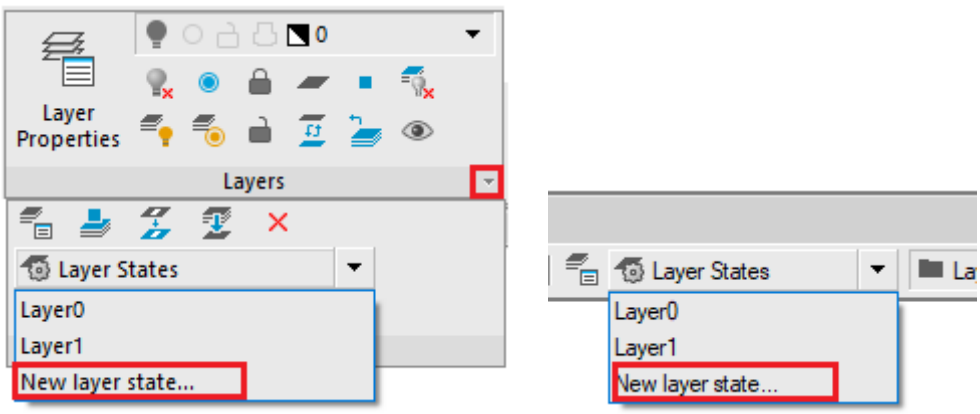
If there are xrefs in a drawing, the filter **All layers without xrefs** becomes available:



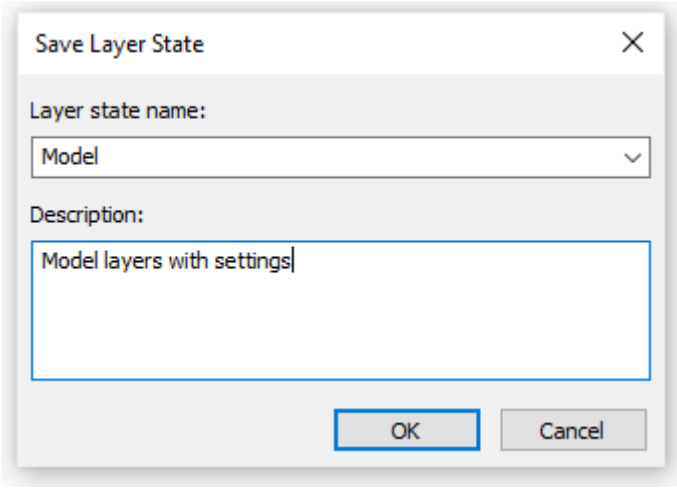
Additional selection of xref layers.

New layer state

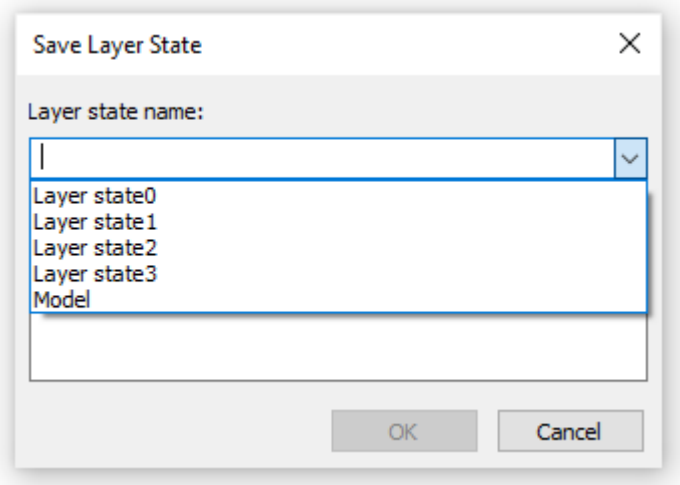
In the drop-down list of layer states on the ribbon and on the **Layers-2** bar, the command to create a new configuration is available.



The **New layer state** command allows you to quickly save layers settings in a new configuration without opening the **Layers** dialogue:

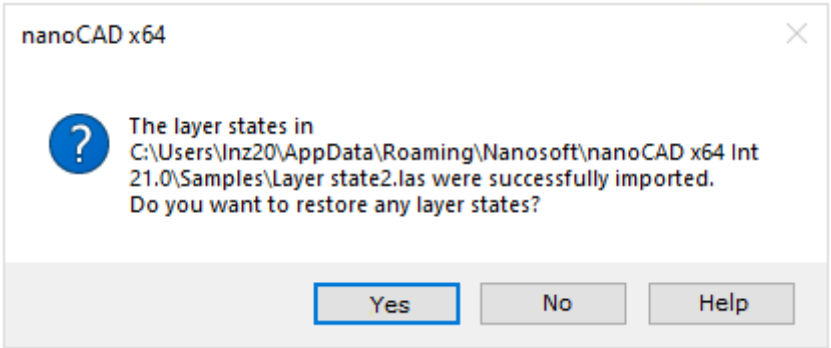


The opening list of the **Name of layer state** field provides an opportunity to save changes to the already existing state:



Possibility to restore a layer state when importing it

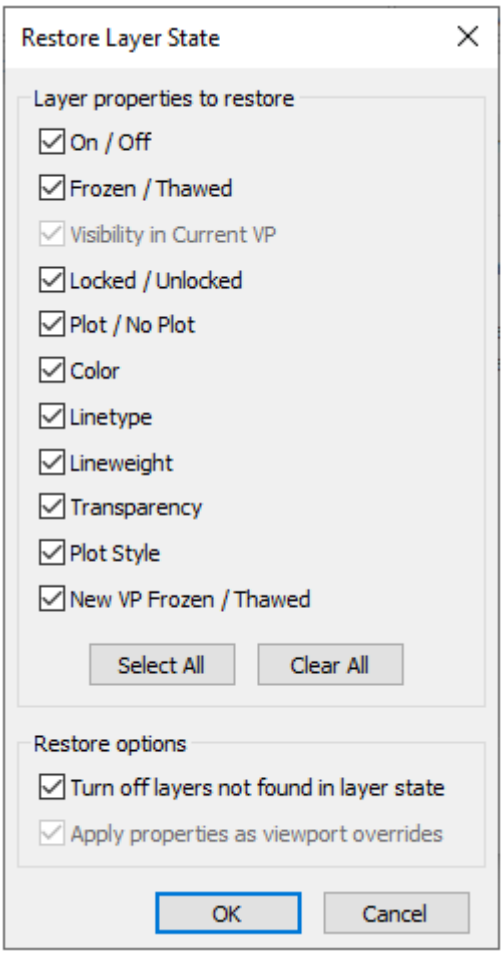
When importing a layer state, now a request to restore it appears automatically.



In previous version, to do this it was necessary to additionally run the restore command from the context menu of the imported state.

Displaying layers and their properties when restoring states

Now there is an opportunity to choose which layer properties to restore when restoring the state. When restoring layer state in the Layer Manger Dialogue, as well as when restoring layer state during import, the Dialogue is displayed with the layer properties to be restored. Unmarked properties will not be restored.



- **Turn off layers not found in layer state** – layers added to a drawing after a later state is created and not saved in it can be disabled when restoring the state.
- **Apply properties as viewport overrides** – when restoring a layer state saved from a layout viewport, you can choose whether to use such state settings as global properties of drawing layers or as overrides for that viewport.

Possibility to create a new layer in the HPLAYER variable

It is possible to create a new layer directly in the HPLAYER variable. To do this, in response to the request, you just need to specify the name of a new layer, for example:

```
Command: HPLAYER
Enter new value for HPLAYER or . for use current <"Layer1">: Layer1
```

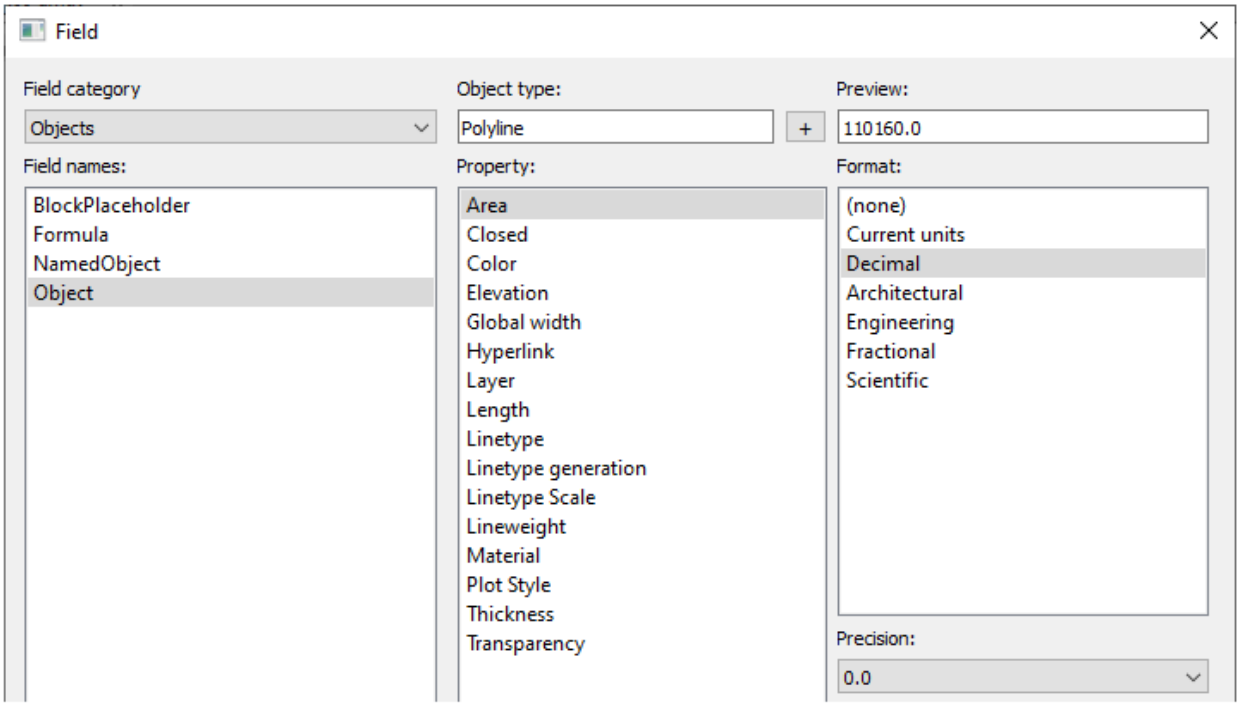
A new layer named Layer1 will be created after the creation of the first hatch and will inherit properties of layer 0.

Clearing layer property overrides in the -VPORTS command

The **LAyer** option has appeared in the -VPORTS command to clear overrides of the viewport layer properties.

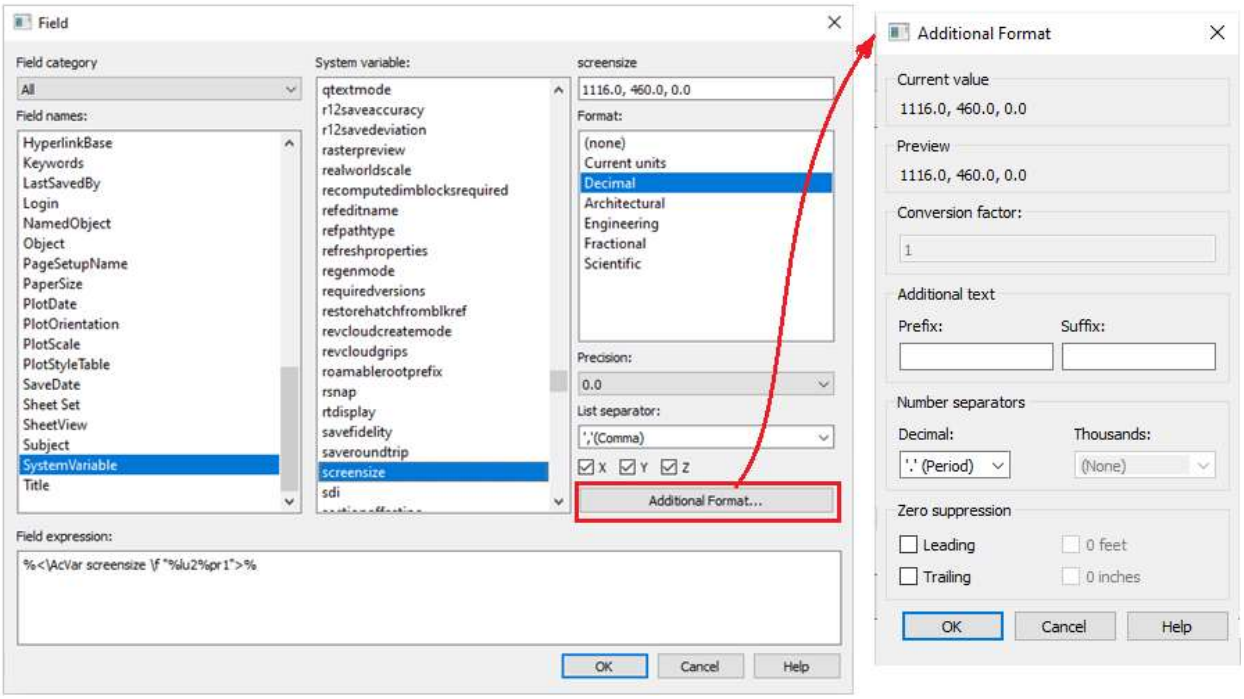
Tools to work with fields

The FIELD dialogue box has been completely redesigned.



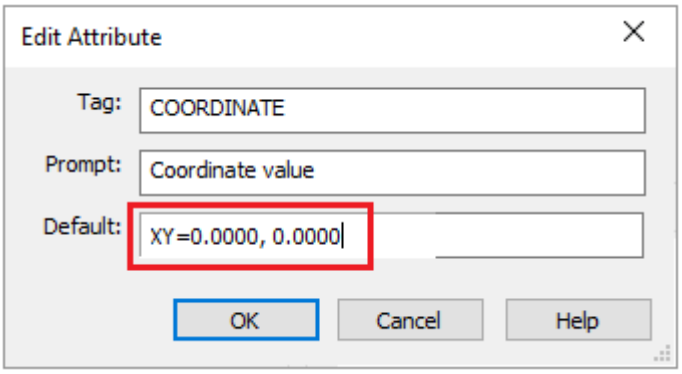
Additional format dialogue

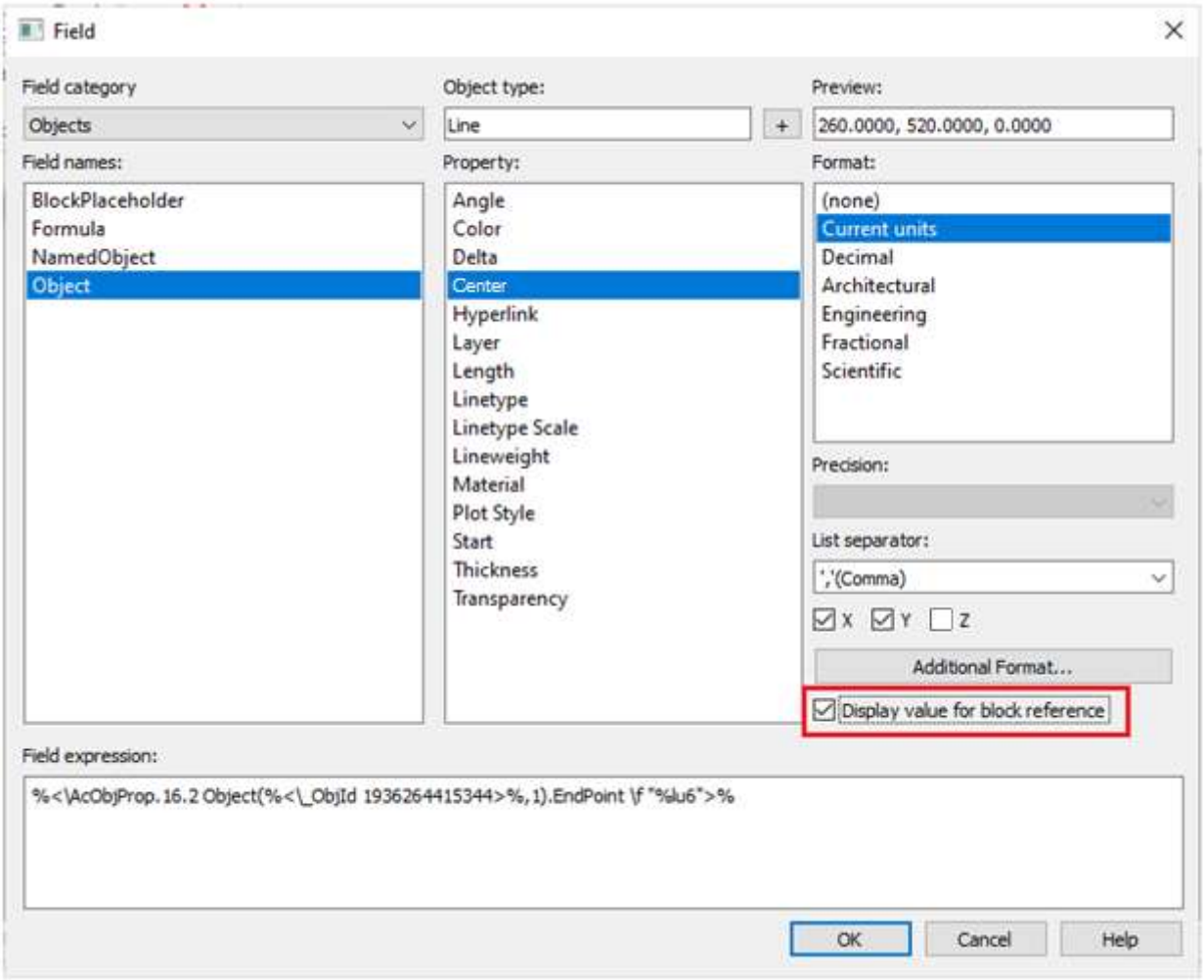
More detailed formatting of numeric values is now available in the field editor. The **Additional format** button opens the dialogue box, where you can set the conversion factor, zero suppression, additional text, numeric separator.



Option for fields inside a block

Display value for block reference – the checkbox is displayed only for object properties fields that are located inside the block. For example, when editing a field inside a block attribute definition.

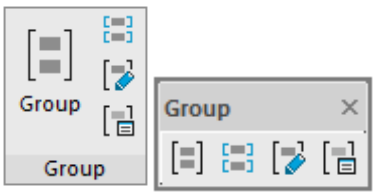




The parameter obliges the property to calculate its value relative to the size and orientation of the block reference in the drawing, and not relative to the internal block description space. So, for example, the coordinates of the center of the object included in the block will be calculated relative to the drawing coordinate system, and not relative to the coordinates of the block definition.

Tools to work with groups

Commands to work with groups



For more convenient grouping and ungrouping objects, adding and removing objects from a group (without the need to launch the **Object grouping** dialogue box) and for possibility to create unnamed groups, the tools for work with groups have been detached as separate commands in the **Group** ribbon section (**Main** tab) and to the **Group** toolbar.

New group (GROUP)

A new command to create a group of objects. Unlike the **New** option, the GROUP command allows you to create both name and unnamed group.

Exploding a group (UNGROUP)

Exploding a group or ungrouping.

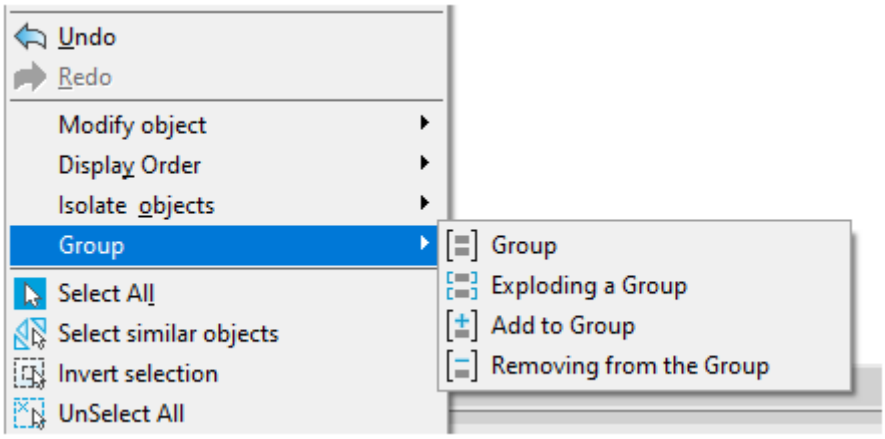
Editing a group (GROUPEDIT)

Adding or deleting objects from a group.

Dialogue for creating groups (CLASSICGROUP)	Displays the **Object grouping** dialogue box to manage groups. In the previous program versions this dialogue box was called by the **Group** command (GROUP).

Group context menu

In the context menu that opens by the right mouse click on the selection of objects or a group, now the **Group**, **Exploding a Group**, **Add to Group**, **Removing from the Group**: commands have appeared:



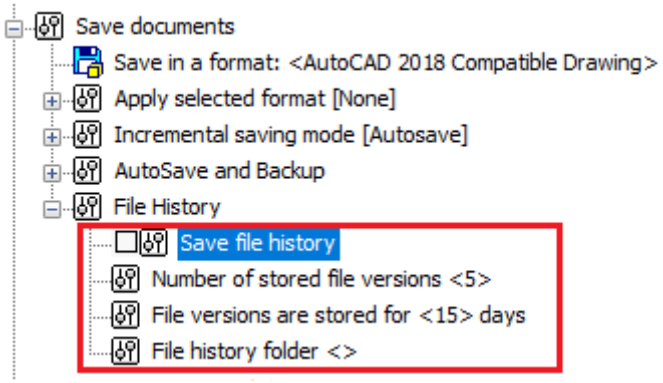
Only unnamed groups can be created via the context menu.

Saving unsaved files and autosave files

A mechanism has appeared for preventing data loss in case of thoughtless refusal to save a file. Such annoying accidents occur when closing a file or exiting the program, especially if several files are closed at the same time. Enabling the “rescue” mechanism for such files allows you to restore them later from a temporary storage.

It happens so that in the process of work some data were deleted, to which later you’d like to return. But the “Save” command has already been performed, the file is closed and therefore, the operations undo buffer is cleared. And the intermediate file copy was not saved. In this case, an autosave archive available for some time can help.

It is based on the file autosave mechanism and is configured in the **Options** dialogue box – **Save documents** section – **File History**:



- **Save file history** (FILEHISTORY variable) – whether maintain the autosave history or not (0 – disabled or 1 – enabled);

- **Number of stored file versions** (FILEHISTORYMAX variable) – the maximum number of autosaved version of a file;
- **File versions are stored for <15> days** (FILEHISTORYDURATION variable) – storage time for autosaved file versions in the storage (in days);
- **File history folder** – folder for storing autosaved copies.

Files are deleted from the storage:

- When trying to save a file version in the storage, which already contains a number of version equal to FILEHISTORYMAX (the earliest version is deleted);
- When closing nanoCAD, the storage period for files in the storage is checked. If the file is in the storage longer than FILEHISTORYDURATION, then it is deleted.

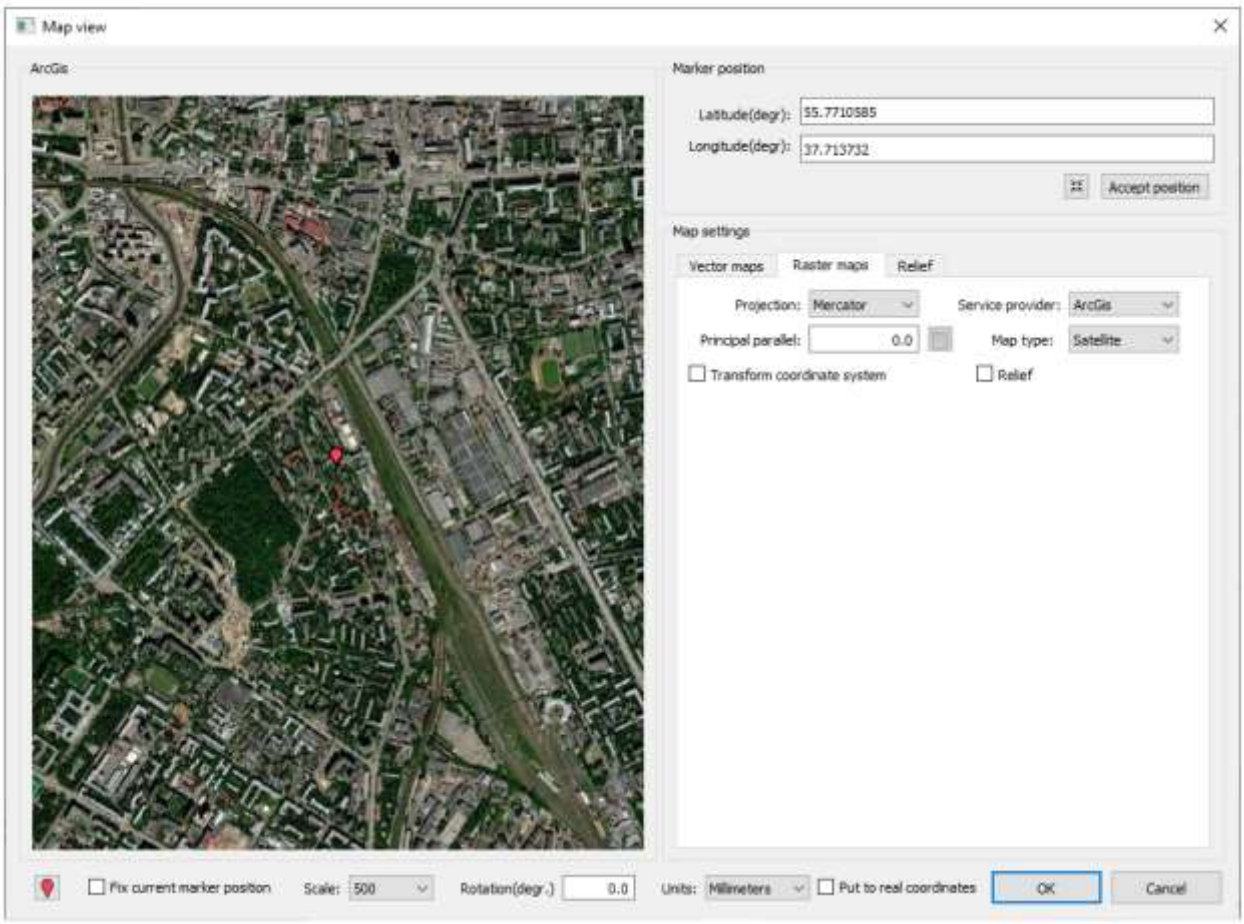
The **Open file history** command (OPENHISTORY) has been added to open a file from the storage.

Underlays from cartographic services

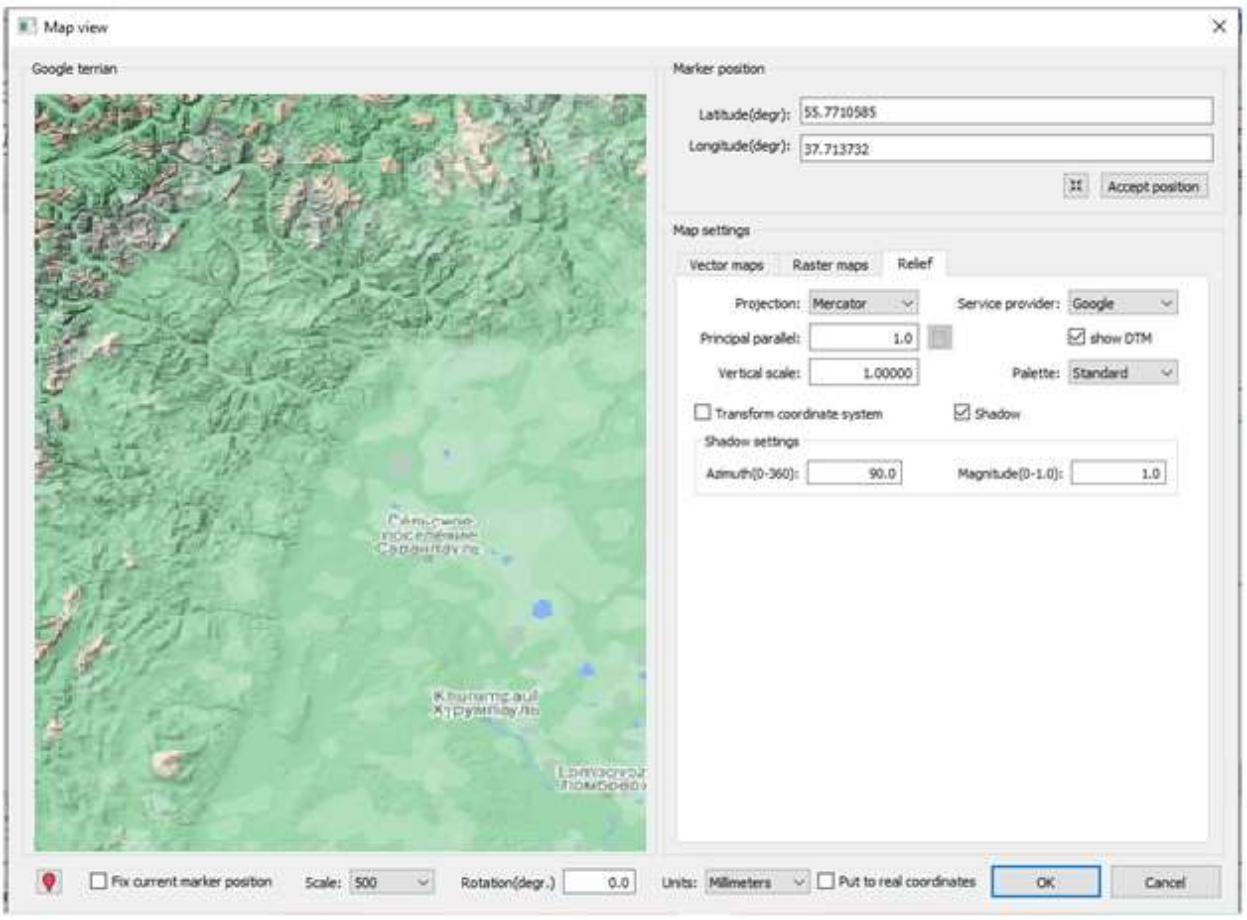
Possibility to insert underlays from different cartographic services in different formats.

Possibility to insert:

- OpenStreetMap vector maps;
- raster maps from services: OpenStreetMaps, Google Maps, Yandex Maps, OSM Topo, MapBox, ArcGis, Bing. In addition, support of different types of maps for some providers: street maps, satellite, hybrid, relief, topographic ones.



It is possible to use a 3-dimensional terrain model as an overlay.


Map view
Google terrian
Marker position
Latitude(degr):
55.7710585
Longitude(degr):
37.713732
Accept position
Map settings
Vector maps
Raster maps
Relief
Projection:
Mercator
Service provider:
Google
Principal parallel:
1.0
show DTM
Vertical scale:
1.00000
Palette:
Standard
Transform coordinate system
Shadow
Shadow settings
Azimuth(0-360):
90.0
Magnitude(0-1.0):
1.0
Fix current marker position
Scale:
500
Rotation(degr.)
0.0
Units:
Milimeters
Put to real coordinates
OK
Cancel

Tools to work with point clouds

Improvements of import and export

- Import and export of clouds of E57 and ReCAP (*.rcs) format have been added;
- Import got a new improved algorithm for importing cropped areas;
- Possibility to import into a drawing (including by dragging and dropping into the drawing field) of the own nanoCAD format of point clouds (*.npc);
- The import dialogue can set drawing measurement units;
- Support of point clouds when exporting to 3D-PDF format has been implemented.

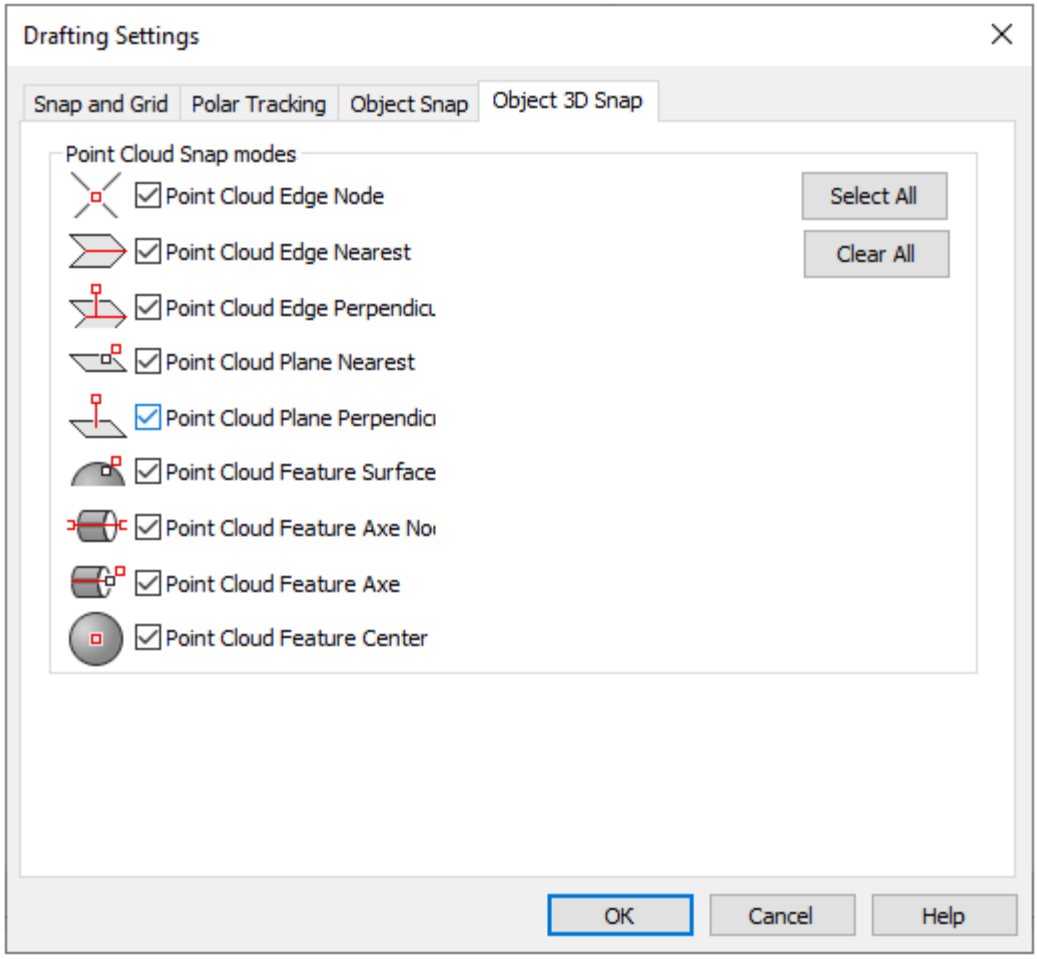
Cloud clipping commands

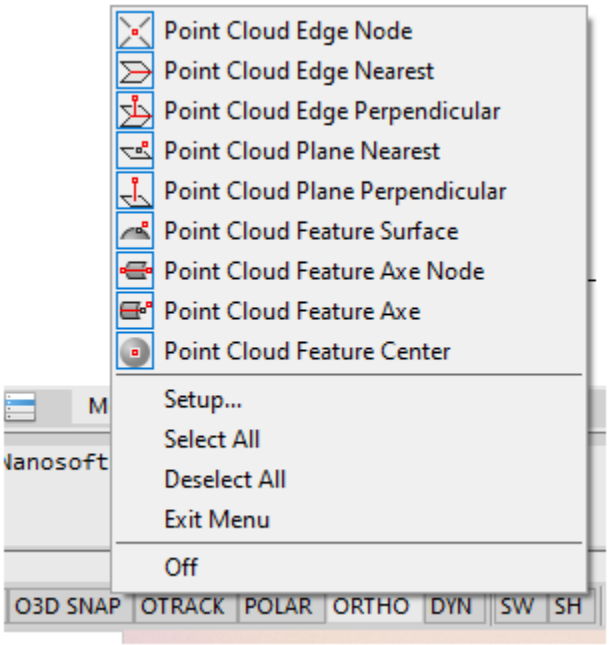
- Commands for clipping a cloud by sphere and cylinder have been added;
- Clipping commands interface has been improved;
- A command has been added to invert a point cloud clipping has appeared. A possibility to invert clipping inside/outside directly when the clipping command works through choosing the option.

3D snaps to recognized geometry

There are three-dimensional snaps to earlier recognized geometry in a point cloud. The following snaps are available:

- Point Cloud Edge Node;
- Point Cloud Edge Nearest;
- Point Cloud Edge Perpendicular;
- Point Cloud Plane Nearest;
- Point Cloud Plane Perpendicular;
- Point Cloud Feature Surface;
- Point Cloud Feature Axe Node;
- Point Cloud Feature Axe;
- Point Cloud Feature Center.


Drafting Settings
Snap and Grid
Polar Tracking
Object Snap
Object 3D Snap
Point Cloud Snap modes
Point Cloud Edge Node
Point Cloud Edge Nearest
Point Cloud Edge Perpendicu
Point Cloud Plane Nearest
Point Cloud Plane Perpendicu
Point Cloud Feature Surface
Point Cloud Feature Axe No
Point Cloud Feature Axe
Point Cloud Feature Center
Select All
Clear All
OK
Cancel
Help


Point Cloud Edge Node
Point Cloud Edge Nearest
Point Cloud Edge Perpendicular
Point Cloud Plane Nearest
Point Cloud Plane Perpendicular
Point Cloud Feature Surface
Point Cloud Feature Axe Node
Point Cloud Feature Axe
Point Cloud Feature Center
Setup...
Select All
Deselect All
Exit Menu
Off
O3D SNAP
OTRACK
POLAR
ORTHO
DYN
SW
SH

Opening .DWG files in the specified layout or view

Now the possibility appeared to open **.dwg** files from the command line “in the right place”. You can specify a name for the sheet, and it will be activated after opening. When specifying a named view, the model space will be activated and the desired view will be displayed.

The syntax for the command to open the file with the desired sheet from the command line:

<path>/ncad.exe [<path>/drawing_file] [-list "sheet name"] [-v "view name"]

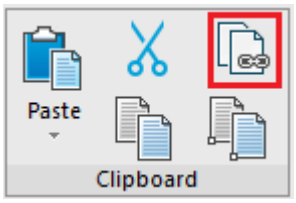
or

<path>/ncad.exe [<path>/drawing_file] [/list "sheet name"] [/v "view name"]

OLE-server to integrate .DWG files into alternate editors

Copy OLE-link command

A new command **Copy OLE-link** (COPYLINK) copies the current view to the clipboard to link with other OLE applications. Its task is to copy to the clipboard a special indicator by which an object of **nanoCAD drawing** type will be created in another program.



Thus, for example, now it is possible to insert nanoCAD drawing into MS Word as an OLE-object with possibility of its further opening for edit in nanoCAD by the mouse double-click.

New command names

Names of OLE-commands are changed to more intuitively comprehensible in the classic interface.

Menu **Insert**. **Object…** to **OLE-Object…**

Menu **Edit**:

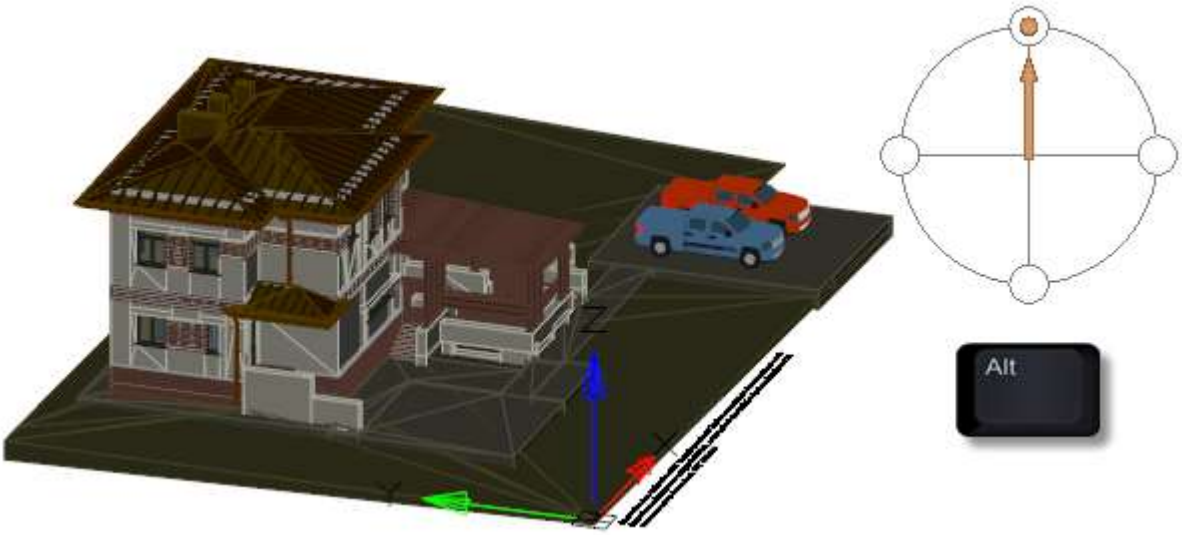
- **Copy view** to **Copy OLE-link**
- **Links…** to **OLE-links…**
- **Update all links** to **Update all OLE-links**

Options in the Properties bar

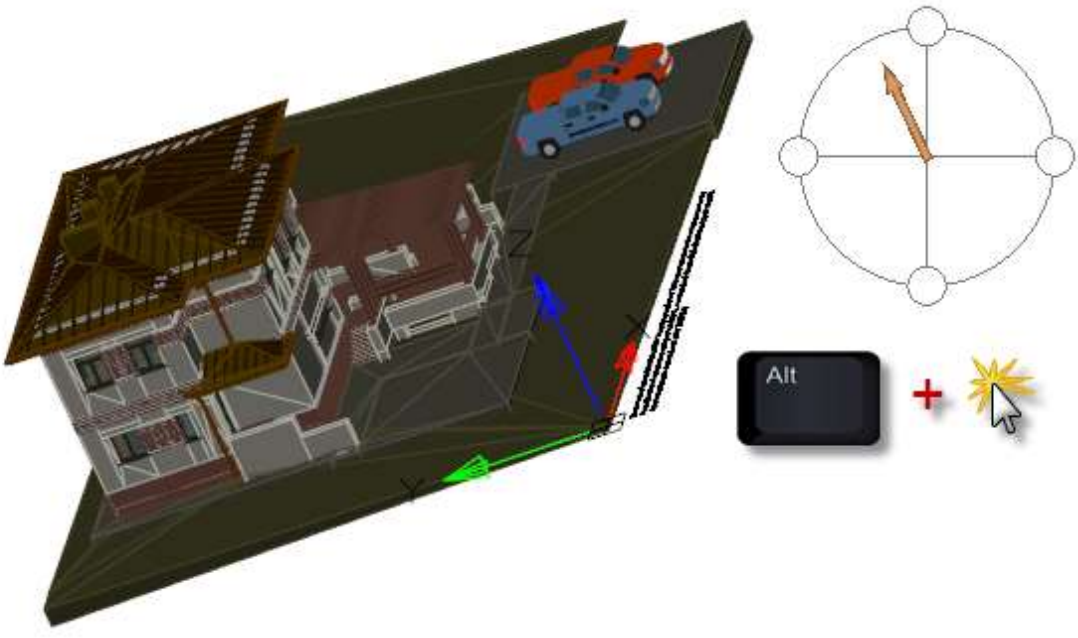
Properties have been added in the **Properties** bar.

Expanding the locator functionality

Now with help pf the **Locator** tool you can rotate the current view of the model. To do this, press ALT and move the cursor over the locator. The locator will switch to the view rotation mode. To rotate, select an arbitrary point on the locator circle or one of the four marked points to rotate a view by 90◦. The arrow position indicates the current view rotation position.



To rotate, select the arbitrary point within the locator or one of four circles for accurate rotation of the view at the angle divisible by 90 degrees. The arrow position indicates the current position of the view rotation.



Creating and modifying geometric objects

Creating arcs (including elliptical ones) in the direction of the mouse cursor movement

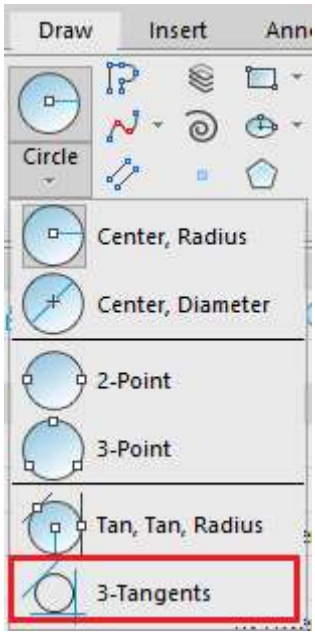
Now, while creating arcs by the methods of:

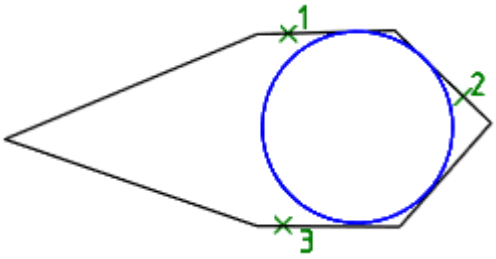
- Center-Start-End,
- Start-Center-End,
- as well as an elliptical arc,

the arc is automatically created in the direction of the mouse movement. The command itself determines the construction direction: clockwise or counterclockwise. You can still forcibly change the direction for constructing the arc by CTRL key.

Creating a circle by three tangents

A new command allows you to create circles by three tangents (menu **Draw** > **Circle**).





Creating a circle by tangent to spline and ellipse

Now the command to create a circle by tangent, tangent and radius, allows you to specify splines, ellipses and elliptical arcs as tangents.

Also, the circle creation by tangent to arc or other circle has been improved.

Breaking ellipse and closed polyline by two points

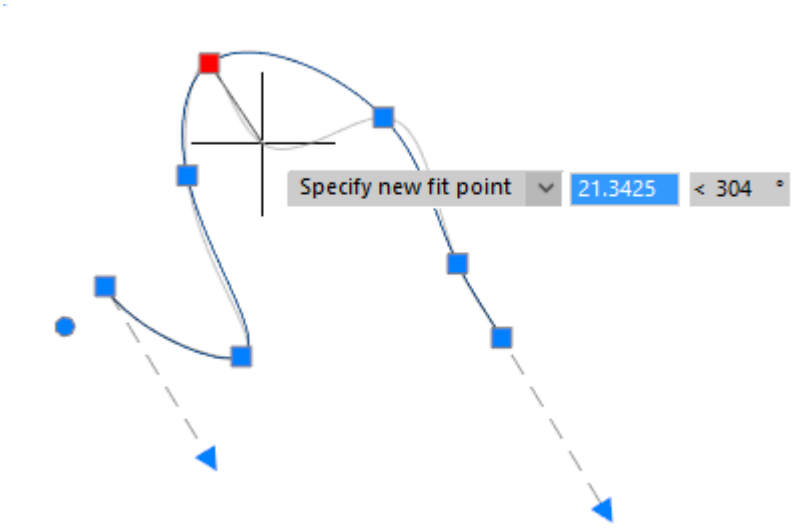
Now in breaking an ellipse, as well as a closed polyline by 2 points, the sequence of specifying points is taken into account to determine which part of the primitive should be removed. Previously, always the same part remained regardless of the order the points were specified.

Lengthening in two directions by the LENGTHEN command

The **Lengthen** (LENGTHEN) command received a switchable mode-option **Two sides/One side**, which allows you to lengthen objects simultaneously (symmetrically) in two directions for all options of the command work. It can be useful, for example, when increasing centerlines in two directions.

Editing spline fit points using Ctrl

Editing spline in the **Fit points** mode is available using the smart grips (using CTRL), as it is done for polyline. The context menu has also been changed.



Selecting and editing objects

Selecting object using lasso

The “lasso” method has been implemented for selecting drawing objects by moving the cursor with the left mouse button pressed.

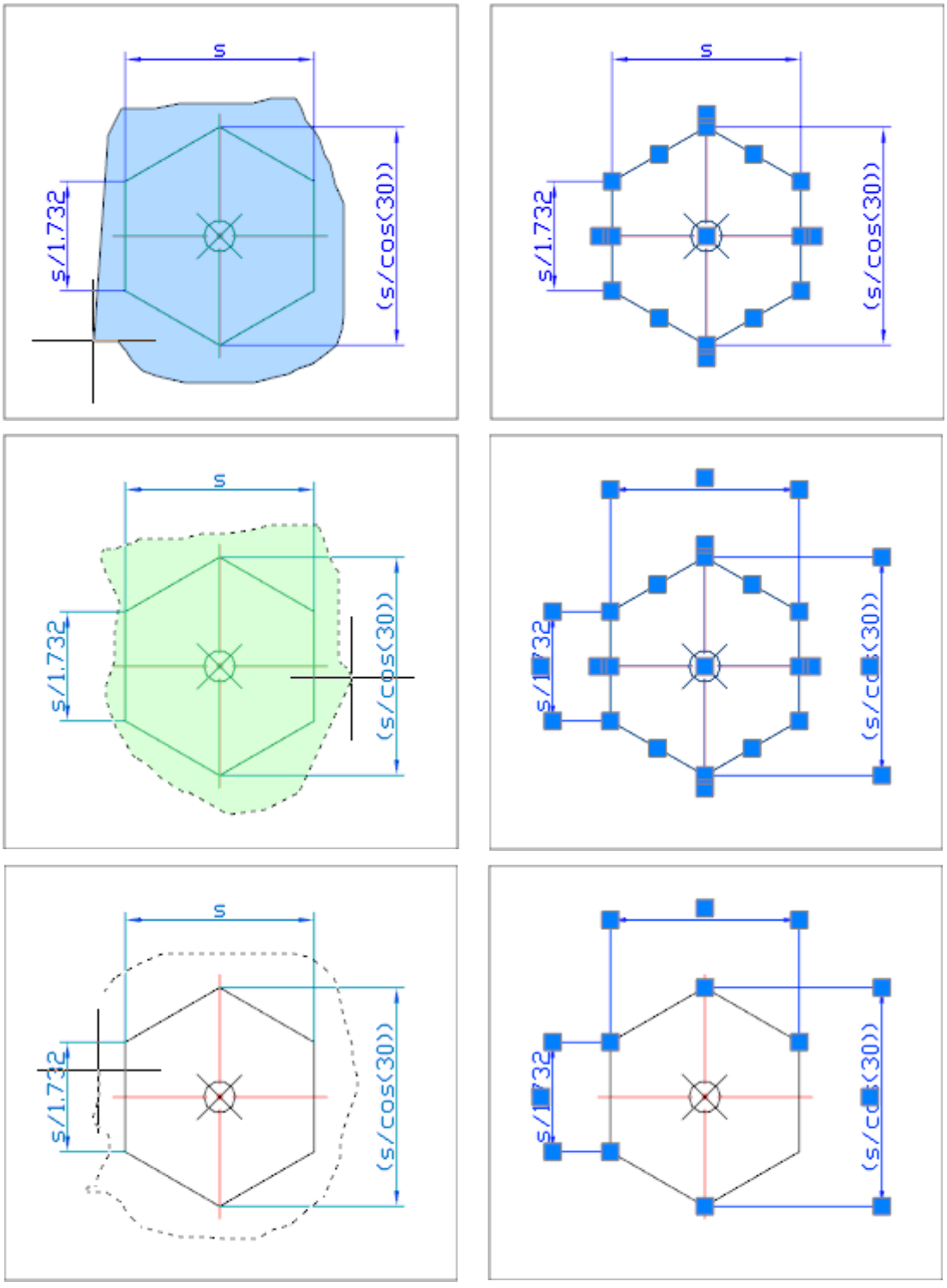
A group of objects is selected by a rectangular selection frame or using a lasso:

- If you move the cursor in the drawing area while holding down the mouse button, the lasso outline will be drawn. To end the selection, just release the mouse button.
- In case of a single click on an empty space in the drawing area, a rectangular selection window will begin to draw. The second click will mark the second vertex of the frame and complete the selection process.

If after starting a group selection, the cursor moves to the right, a selection window or lasso with a blue semi-transparent fill are used. Only those objects that got entirely inside the specified area will be selected.

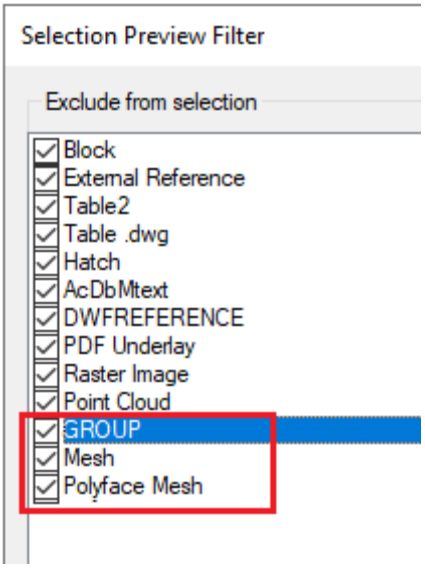
If after starting a group selection, the cursor moves to the left, a crossing selection or lasso with a green semi-transparent fill are used. This will select objects both completely inside, and crossed by the border of the specified area.

In the process of selecting objects by lasso method, you can cycle through the three lasso modes (inside the lasso area, lasso cutting edge, lasso line), by pressing the **Space** key



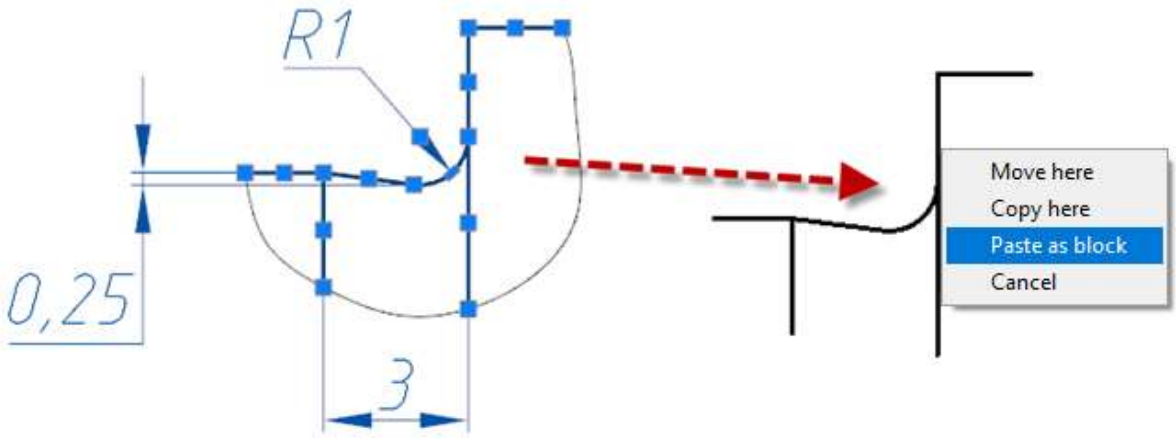
Selection preview filter

The following types of objects have been added to the **Selection preview filter** dialogue box: **Group**, **Mesh** and **Polyface mesh**.

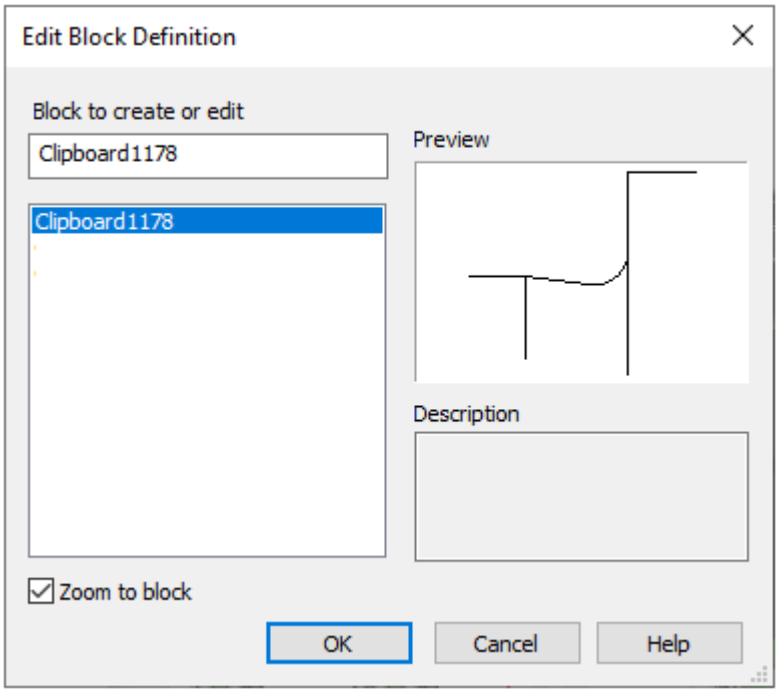


Copying and creating a block by dragging selected objects by the right mouse button

Possibility to drag-and-drop selected objects by the right mouse button with further selection of the required action in the context menu: Move here, Copy here, Paste as block.



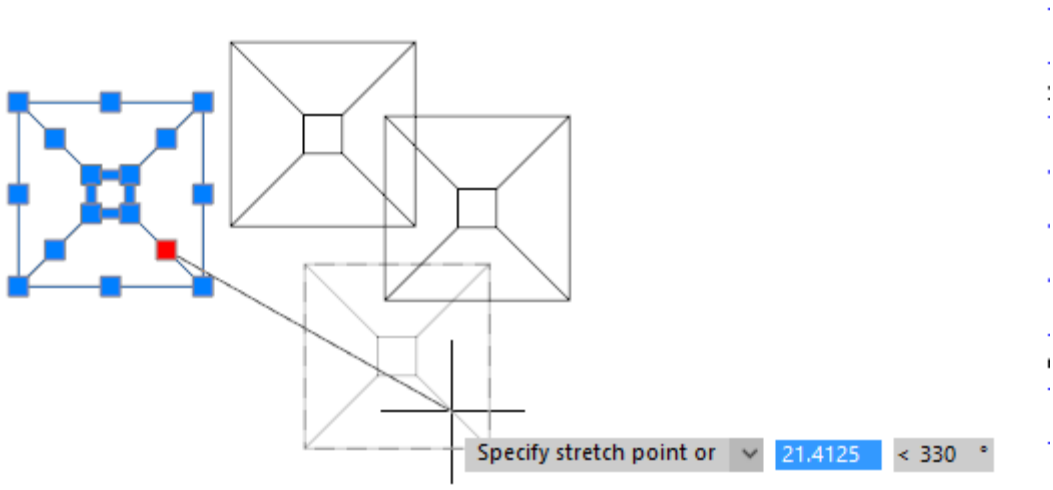
If you select **Paste as block** option, a new block with insertion in this place will be created in the document.

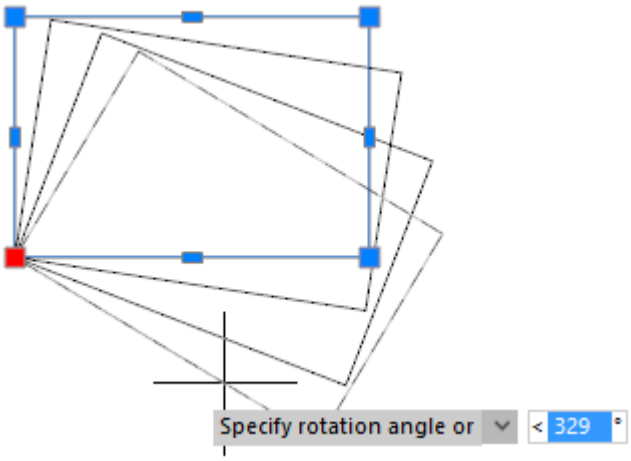


Multiple copies by holding down Ctrl while moving/rotating/scaling

When moving, rotating, scaling and mirroring with grips, you can use CTRL to create the object copies:

- Select one or more objects;
- Click the grip;
- Press the SPACE key a required number of times to transfer to the selection copy, rotate, scale or mirror mode;
- While holding down the CTRL key, create modified copies of editable objects by clicking the mouse in the desired place in the workspace.





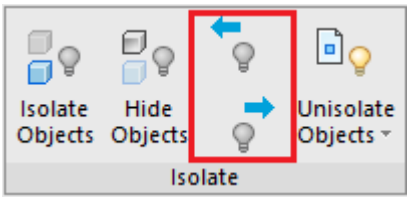
This mode of creating modified copies using ***CTRL*** is also available for multifunctional grips with their edit modes.

ROTATE command

The **Key point** option has become available. It combines requests for a base point and a reference angle: allows to specify the reference angle, the first point of which will be taken as the base point.

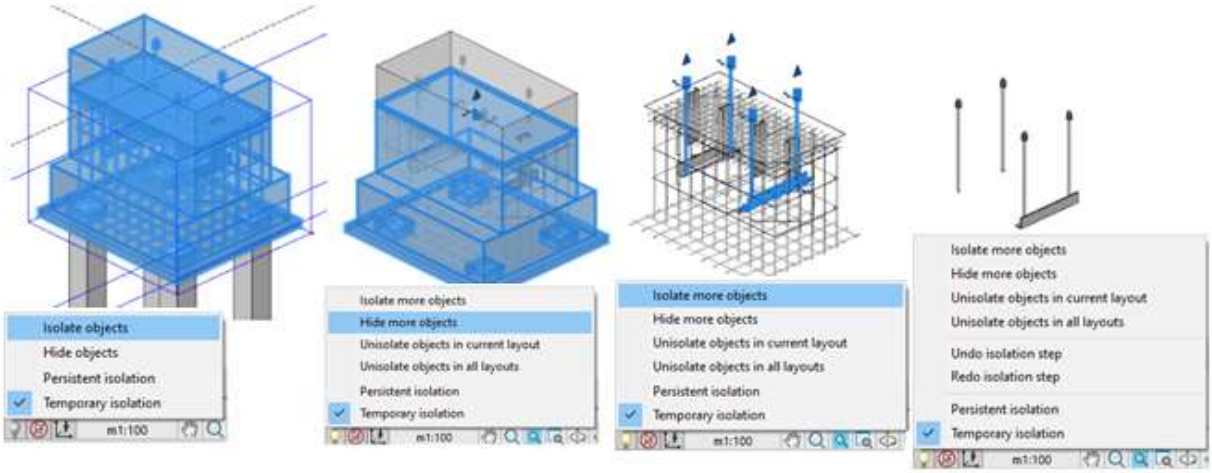
Undo/redo commands

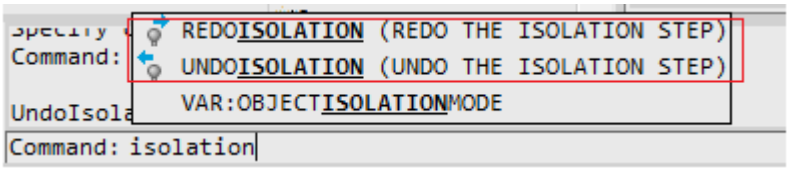
Two-way list of objects isolation/hiding history



New commands **Undo the isolation step** and **Redo the isolation step** represent UNDO/REDO functionality to isolate objects, which allows you to navigate through a sequence of isolation steps.

Isolation and its undo are independent of the general undo-redo mechanism (UNDO/REDO). For example, you can isolate a set of objects, edit them, and then return the isolation to a previous visibility state. In this case all changes to the edited objects will remain.

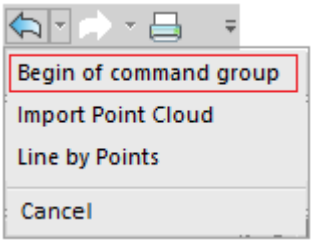




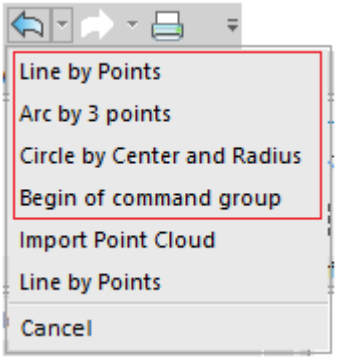
Grouping actions in the undo/redo list

BEgin and End options in the **Undo** and **Redo** commands (UNDO/REDO) allow you to combine a sequence of actions to be cancelled as one operation. Commands performed between setting the **BEgin** and **End** options will be cancelled simultaneously as one action. By using **BEgin** and **End** options, you can create several groups of performed operations, which will be cancelled in sequence.

After setting the **BEgin** option, the **Begin of command group** item will appear in the command list.

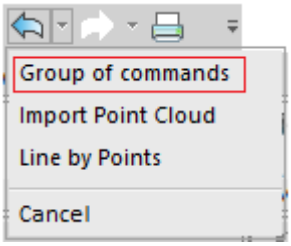


Next actions continue to be added in the undo list of operations one by one.



Till the **End** option is applied, the commands still can be cancelled by one. But no further than the **Begin of command group** item.

After the **End** option is applied, all actions in the undo list between the beginning and the end will collapse in a single group of commands.



Now these actions can be cancelled only together.

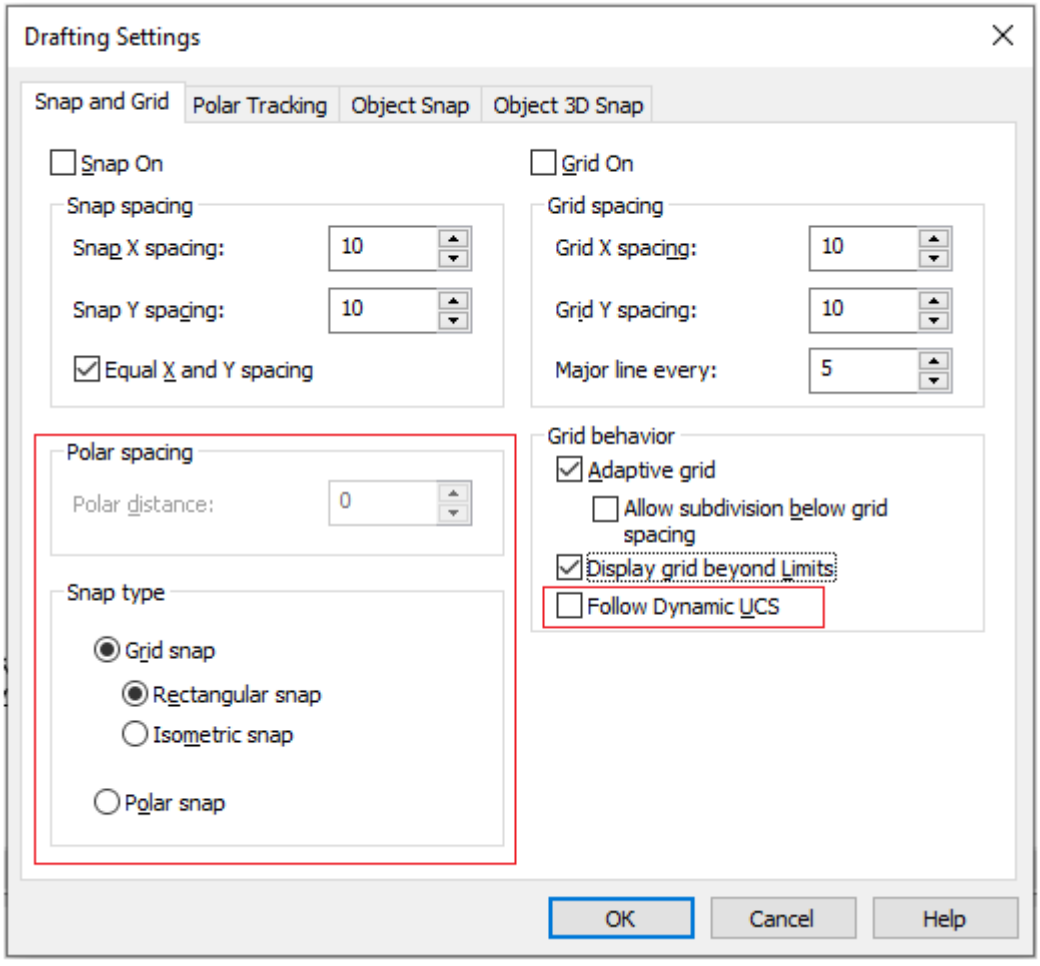
Internal undo in the Copy object properties command (MATCHPROP)

A local undo (internal Undo) has appeared in specifying the next destination object in the **Copy object properties** command (MATCHPROP).

```
Select destination object(s) or [?/Settings/Undo]:
```

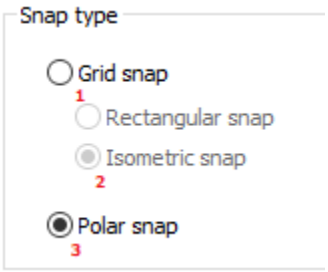
Previously, when copying properties to a large number of objects, in case of accidentally copying to the wrong object, you had to cancel the command entirely and start copying again.

Drafting Settings dialogue (DSETTINGS)



Snap type (Grid/Polar, Rectangular/Isometric)

- **Grid snap** – specifies the snap spacing for moving the cursor to horizontal and vertical grid points.
    - **Rectangular** – sets a default rectangular snap mode as a snap style. With a step type and rectangular snap style, the cursor moves along the nodes of a rectangular structure.
    - **Isometric** – sets an isometric snap mode and activates the isometric drawing mode. With a step type and isometric snap type, the cursor moves along the nodes of an isometric plane.
- **Polar** – sets the polar snap type. With a polar snap type and enabled polar tracking, the cursor is snapped to tracking lines drawn from the base point at angles specified on the **Tracking** tab. To use the polar snap type in the isometric drawing mode, you should first set the grid isometric snap, and then switch to the polar snap.



Polar spacing

Manages the spacing distance of the polar snap.

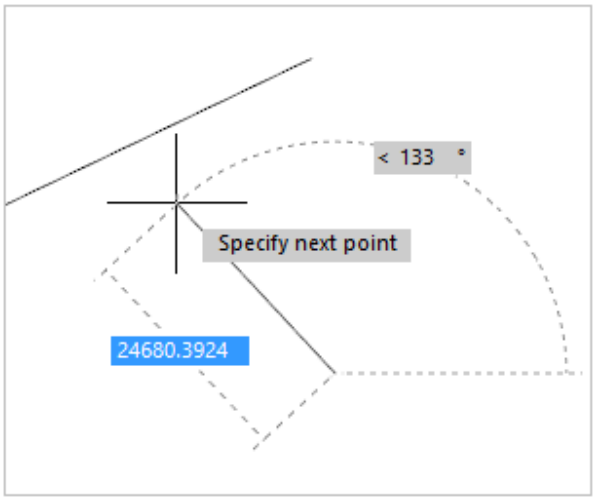
Spacing – polar snap spacing. Becomes available, if the **Polar snap** (POLARDIST system variable) is set. In case of zero value, the polar spacing is equal to the snap spacing interval by X. Works when polar or object tracking mode is enabled.

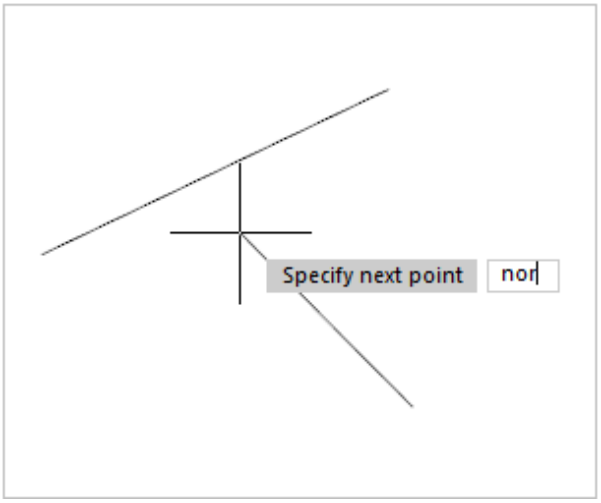
Follow dynamic UCS

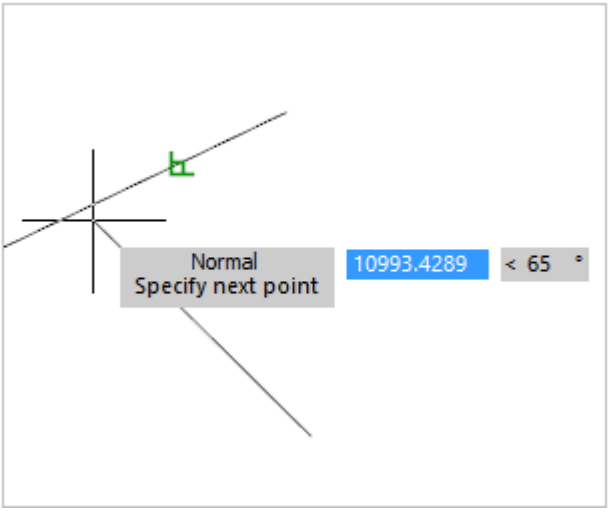
Follow dynamic UCS – the setting brings the grid plane to follow XY plane of dynamic UCS (GRIDDISPLAY system variable).

Single object snap

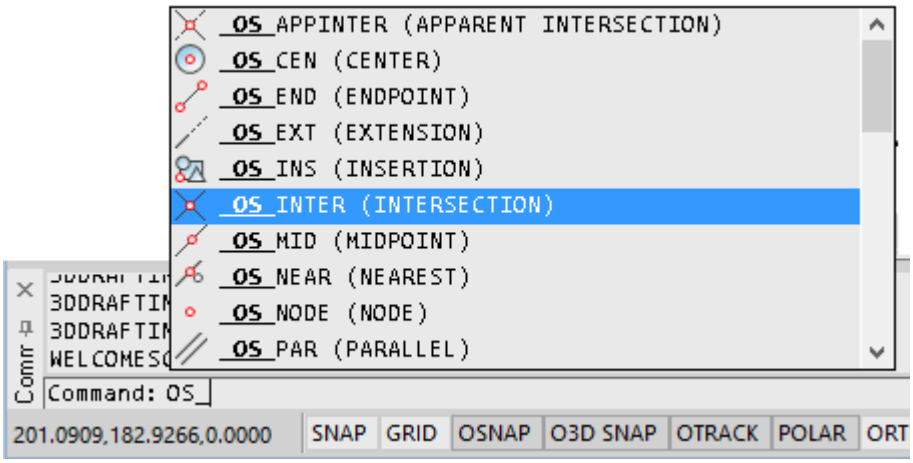
Now you can use keywords to call a single object snap (prompt snap). To do this, enter a snap keyword when prompted for a point and press Enter. The snap will become temporary available.







Full list of snap keywords is available by typing an **OS_** in the command line.



The list of snaps and keywords is given in the table below

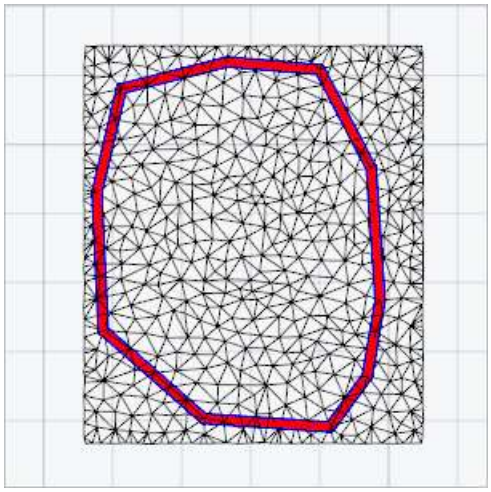
Keyword	**Snap type**
TT	Tracking point
FROM	From
M2P	Mid between two points
END	Endpoint
MID	Middle
INT	Intersect
APP	Apparent intersection

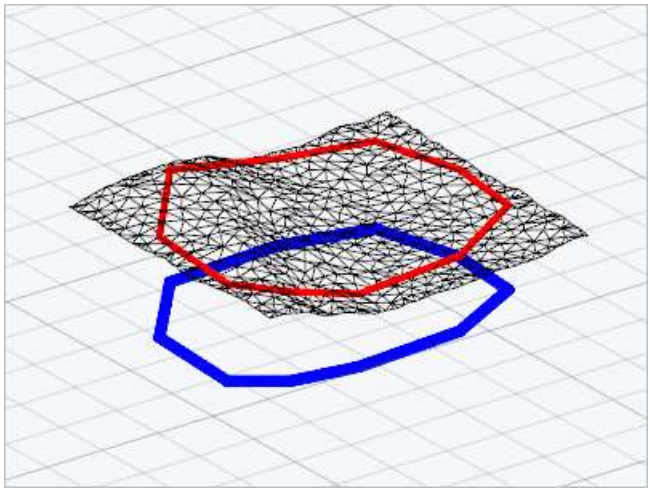
EXT	Line extension
CEN	Center
QUAD	Quadrant
TAN	Tangent
PERP	Perpendicular
PAR	Parallel
INS	Insertion point
NODE	Node
NEAR	Nearest
NONE	Disabling object snap

Projecting z-coordinate of the object snap to the XY plane of the current UCS

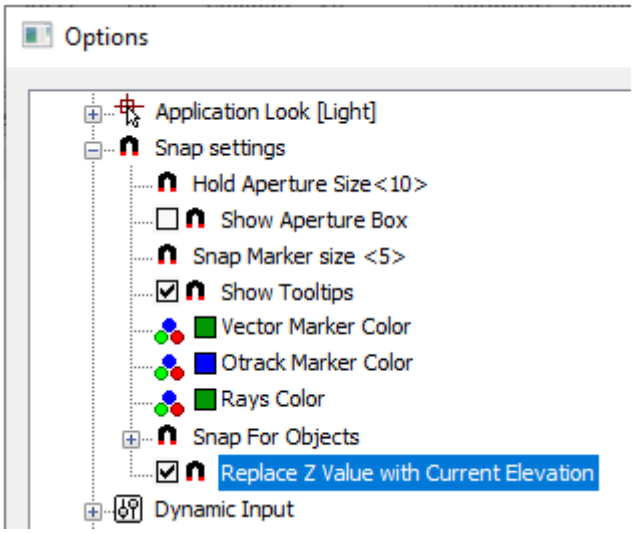
A new snap option **Replace Z Value with Current Elevation** allows you to create objects on XY plane of the current UCS by snapping to points above or below this surface. In this snap mode, the Z coordinate value is replaced with the value of its projection to XY plane of the current UCS. Or with the value of its projection to the plane parallel to the XY plane at the level specified by the ELEVATION variable (if it has a non-zero value).

Below there are two 3D-polylines drawn with w snap to mesh vertices. Red – in a normal snap mode, blue – with enabled option of Z-coordinate replacement.



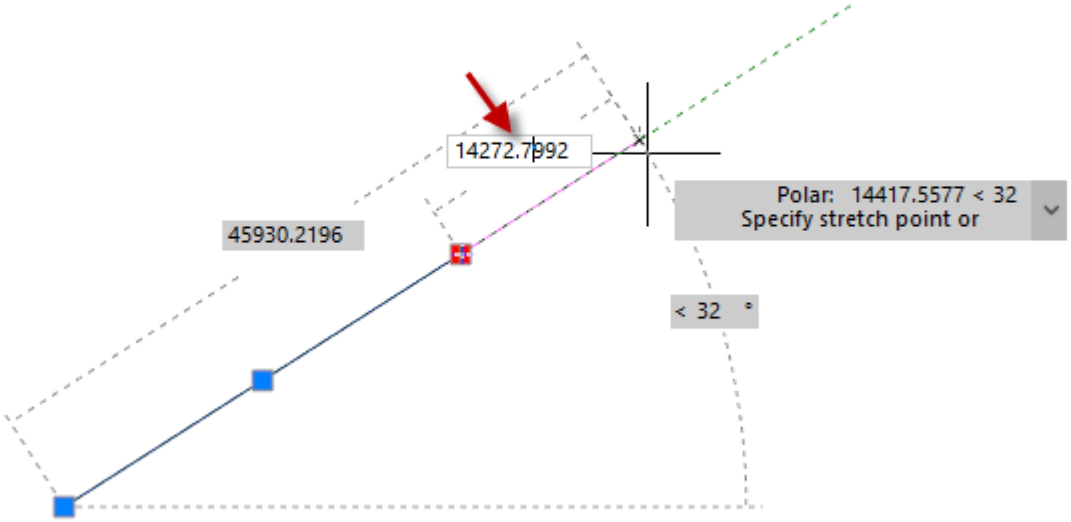


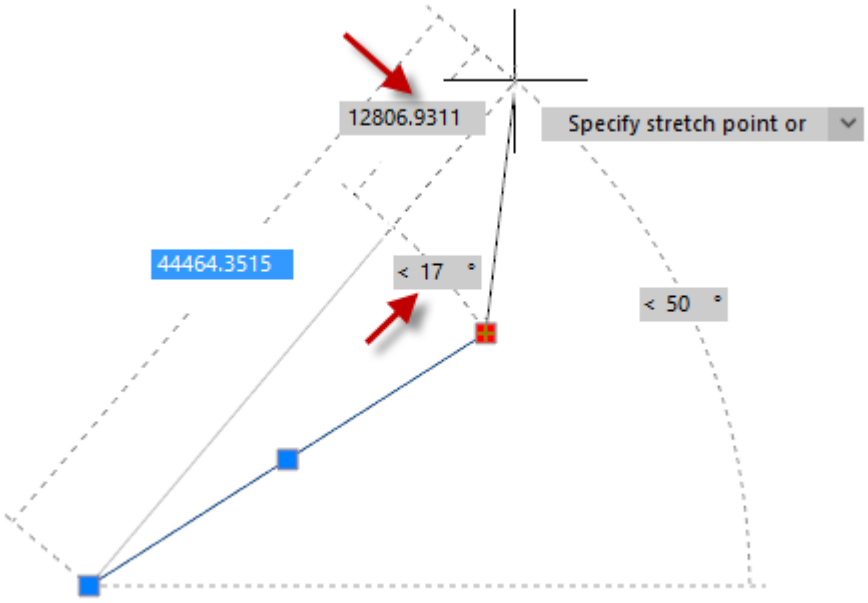
The option is enabled in the **Options** dialogue box in the **Snap settings** section. The option is synchronized with the OSNAPZ variable.



Dynamic input

Fields are added for entering the increment of length and angle when editing objects by grips;

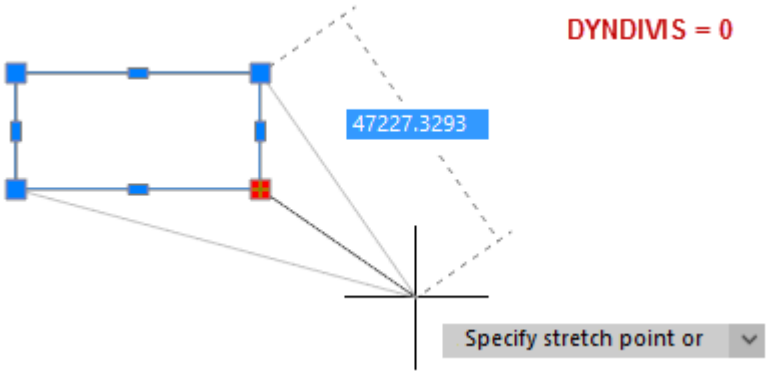




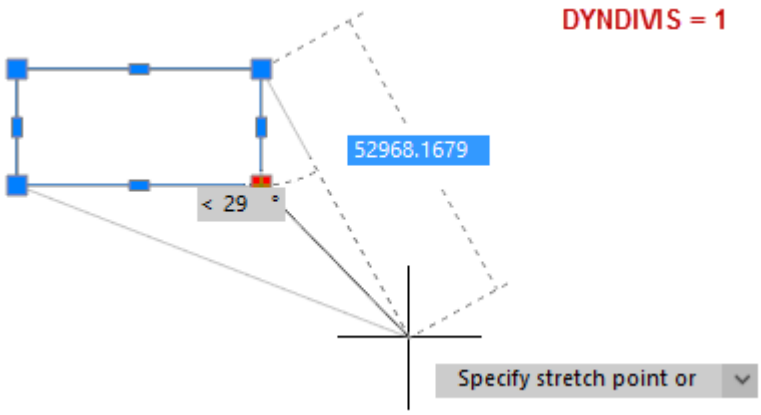
Support is added for the DYNDIVIS variable when editing objects by grips;

Not all dynamic dimensions can be seen on the screen simultaneously. The number of simultaneously displayed dimensions is controlled by the DYNDIVIS variable. Depending on its value, the following can be displayed simultaneously:

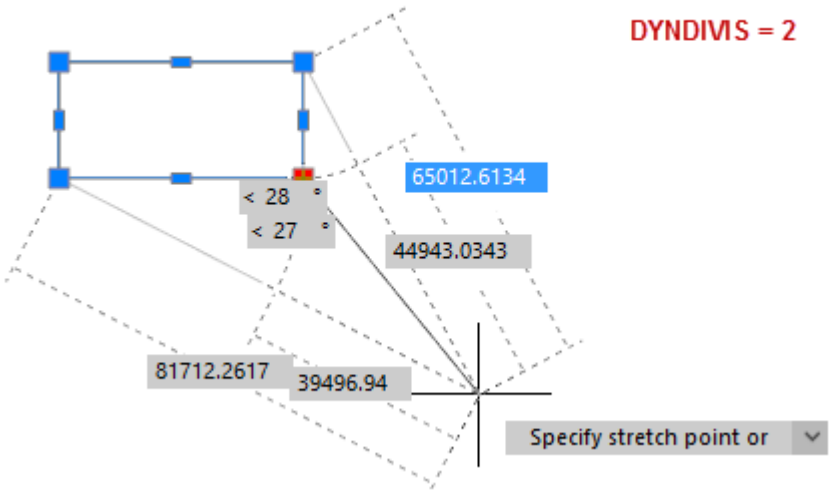
- Only one dimension (DYNDIVIS = 0),



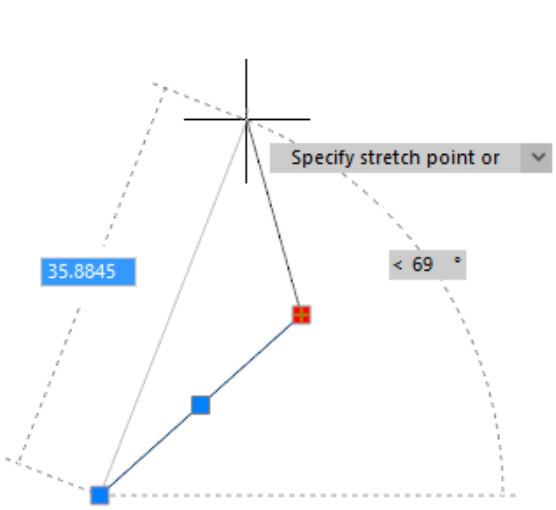
- Only two dimensions (= 1),

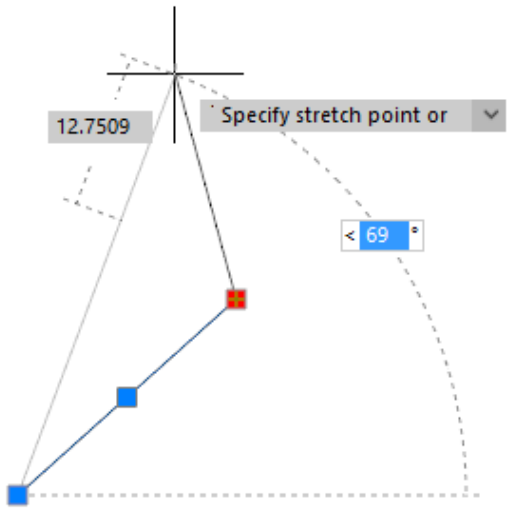


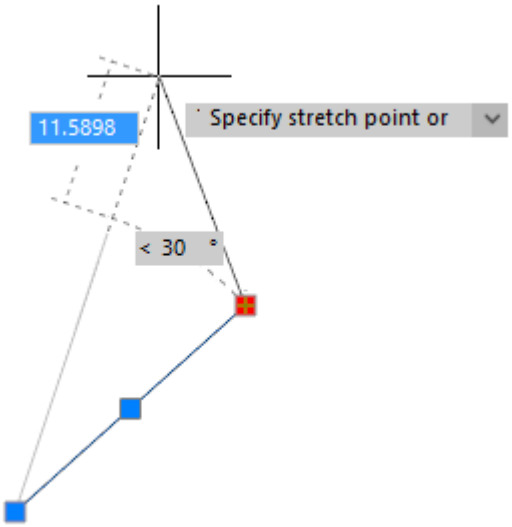
- or all dimensions (= 2).



If only one or two dimensions are displayed simultaneously, and you need to display others, this can be done by cyclically pressing the **Tab** key. Press **Tab** until the desired dimensions are visible. On subsequent cursor movement, the selected dimensions will continue to be displayed.







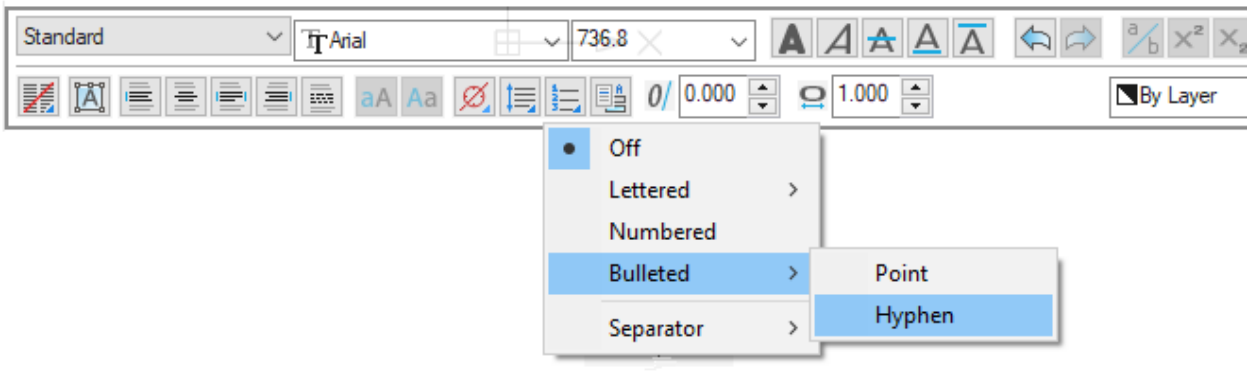
In addition:

- Dynamic dimensions when creating an ellipse has been implemented;
- Dynamic input fields have been added for the second and the next points in the process of creating various objects in UCS.
- Dynamic input has become available for coordinate values for 3D-polyline in UCS.
- Input and editing of dynamic dimensions of 3D-polyline in UCS have become available.
- Observations on the work of dynamic input in diverse variants of constructing lines and circles have been corrected.

Multiline text

Using hyphen in bulleted lists

It became possible to create bulleted lists using dashes.

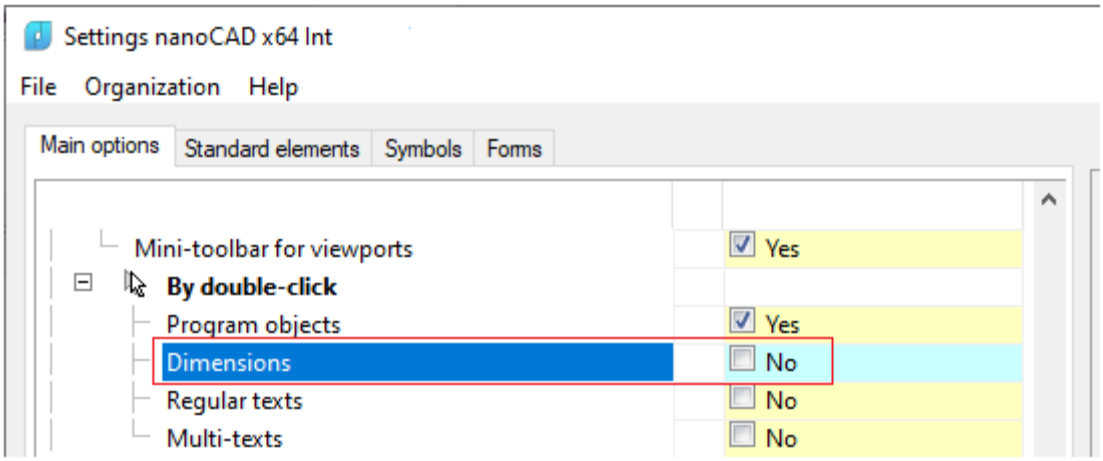


Aligning the last line

If you add Tab in the end of the last line of distributed multiline text, then it is aligned to the left and remains so after editing is completed.

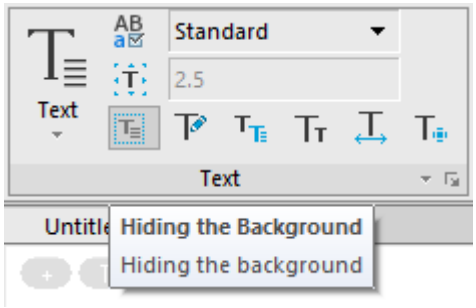
Blocking subdimensional text

Now the cleared **Dimensions** box in the **Main options** tab of the **Design Settings** dialogue box (Section **Edit** > **by double-click**) also blocks editing by double-click and subdimensional multiline text.



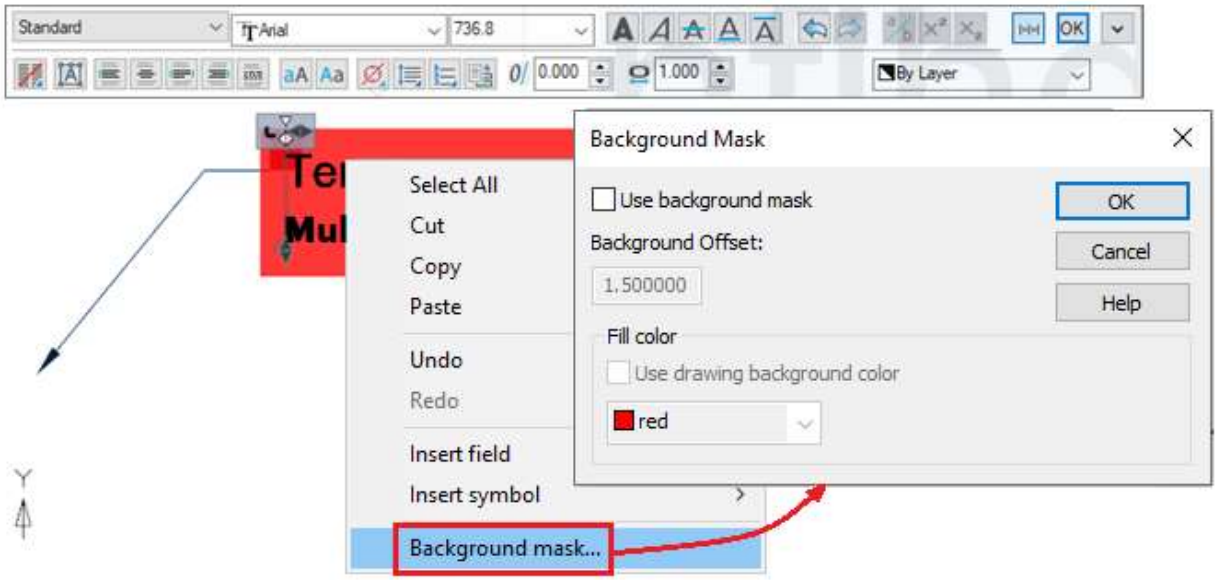
Hiding the text background for different type objects

Calling the dialogue to hide background for multiline text, multiline attributes and multileader multiline text was placed as a separate command  on the ribbon.



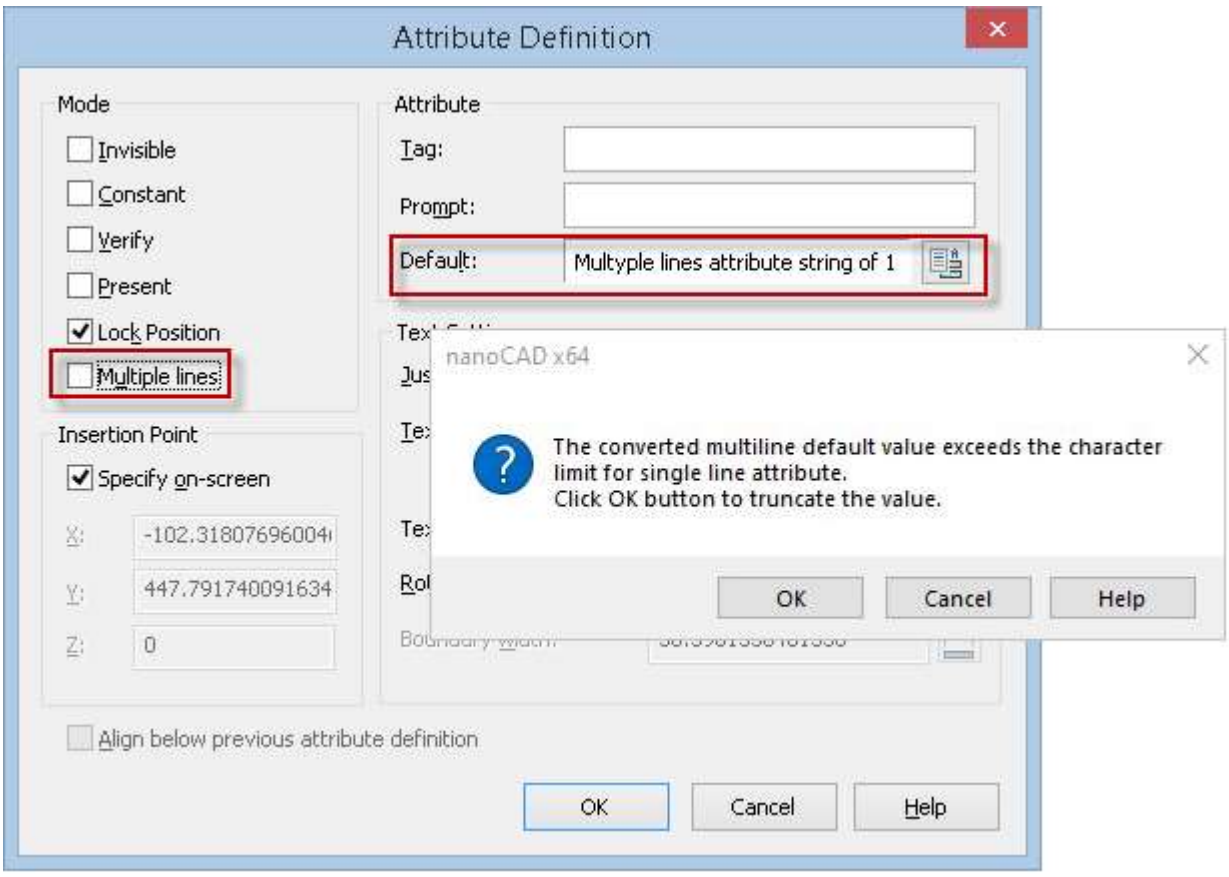
Objects can be selected before or after the command is performed. This allows you to apply the command to several selected different type objects.

The command displays the **Background mask** dialogue box, which will display parameters for hiding background for multiline text in the first selected object.



Saving the value when changing the attribute type

When creating an attribute in the **Attribute definition** dialogue box of the ATTDEF command, the switch from attribute type from a multiline to a single-line one displays a warning message on the need to shorten too long text of a previously set value (**Default** field). In previous versions of the program, such text was completely ignored.



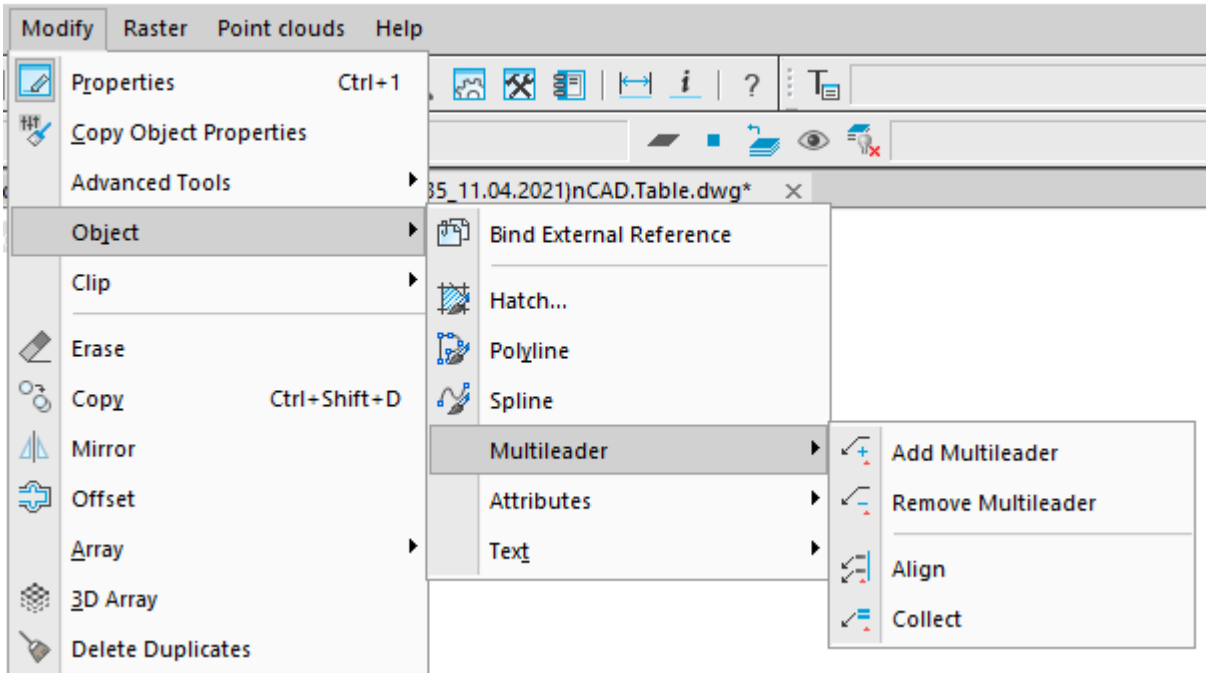
When switching from a single-line attribute to a multiline one, its text is now also preserved.

Multileaders

The following errors of drawing multileaders have been corrected:

- Incorrect underlining text in the multileader with **Bottom attachment** with **Underline** and **Middle** has been corrected.
- Incorrect underlining text in the multileader with **Top attachment** with **Overline** and **Middle** has been corrected
- In multileaders with **Top attachment**, **Left** align of text changed to **Right**.
- If TTF font was used, then spaces were not underlined.

A multileader modify commands have been added to the menu



Maintaining associativity of dimensions and multileaders

For multileaders and dimensions created in other CAD-systems, associativity with objects is supported while working with them in nanoCAD.

Breaking associative links of dimensions can be carried out by shifting grips of extension lines bases.

Break and restore dimension command (DIMBREAK)

The **Break dimension** command DIMBREAK has been renamed to MDIMBREAK.

The **Restore dimension** command DIMUNBREAK has been renamed to MDIMUNBREAK.

A new command **Break and restore dimension** command DIMBREAK has been added. It is called from the command line and combines the functionality of both commands. It is identical to the same name CAD-commands.

Sheet list in the Sheet Set

Now there is a possibility to create a sheet list for a sheet set.

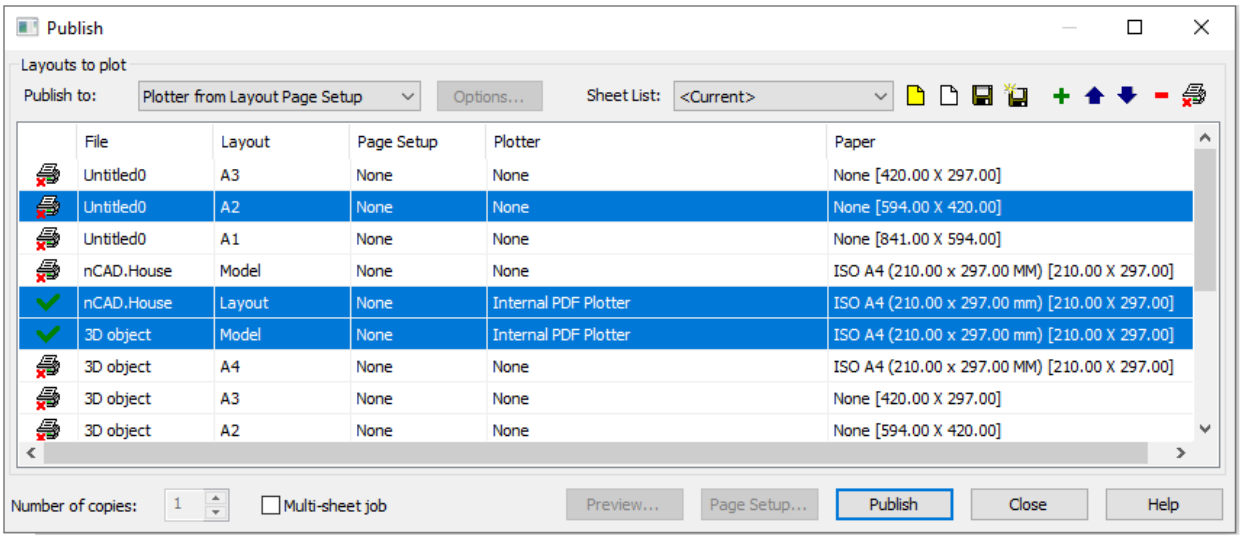
The command to insert a sheet set inserts a table. The provided templates of sheet sets contain all the required settings.

In the **Sheet Set** toolbar, select the name of the sheet set, sheet group and in the context menu select the **Insert Sheet list** item

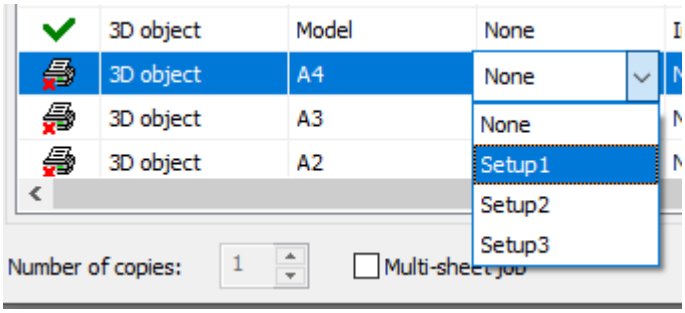
If you need to use a different style for design of a sheet list, it is first necessary to set the required table style in the table style dialogue box or by the CTABLESTYLE variable.

Batch plot. Multiple choice and lists of values

A possibility to simultaneously select multiple sheets using CTRL and SHIFT buttons has been added in the **Batch Plot** dialogue box (PUBLISH).

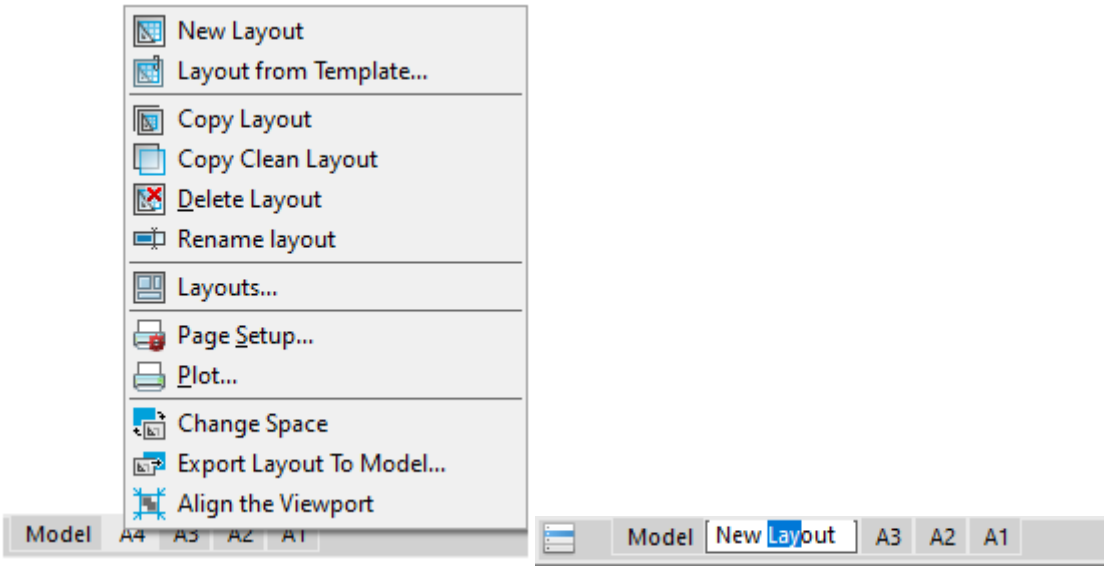


Now you can select directly in the batch plot dialogue box an available value for each sheet from the drop-down list of the **Page setup** option.



Layout tabs

The **Rename layout** command does not require re-entry of the layout name for renaming, but allows you to change the name of the current layout directly on its tab.



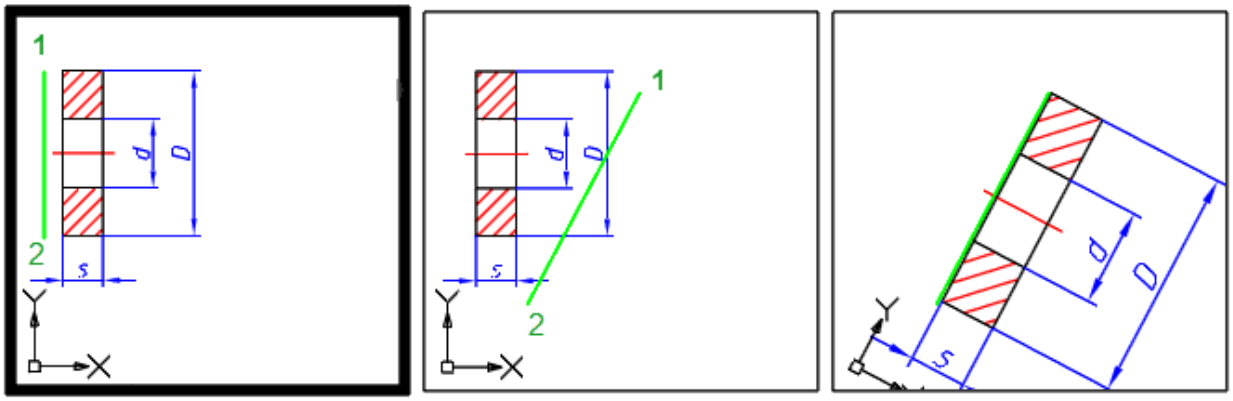
The order of the layout tabs can be changed just by drag and drop. The Step left Step right have been removed from the layout tab context menu as unnecessary.

Display of layout content

Layout content is displayed in accordance with the plot style applicable to this layout. For example, if the plot style is set to black and white in the Page Setup Manager, the layout content will also be displayed in black and white.

Viewport align command (ALIGNSPACE)

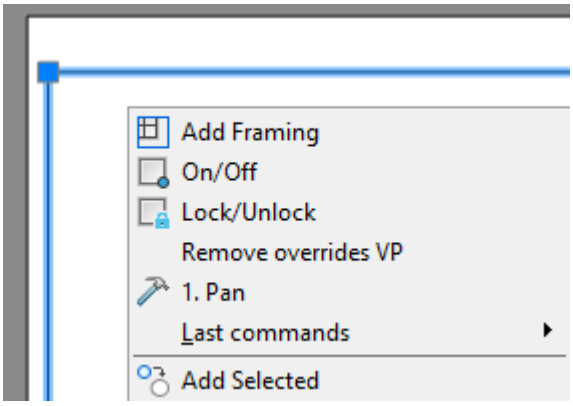
A new command **Viewport align** (ALIGNSPACE) allows adjusting pan and zoom factor of objects in the layout viewport using an indication of alignment point in the model space and paper space.



Specifying one point in the model space and one point in the paper space will offset the view. By specifying two points in the model space and two points in the paper space, you can set scaling, moving and rotation. Points specified in the model space are aligned by points set in the paper space. The zoom factor and UCS rotation are adjusted in accordance with the specified points.

Viewport context menu

The context menu for paper space viewports has been expanded:



Import PDF files data

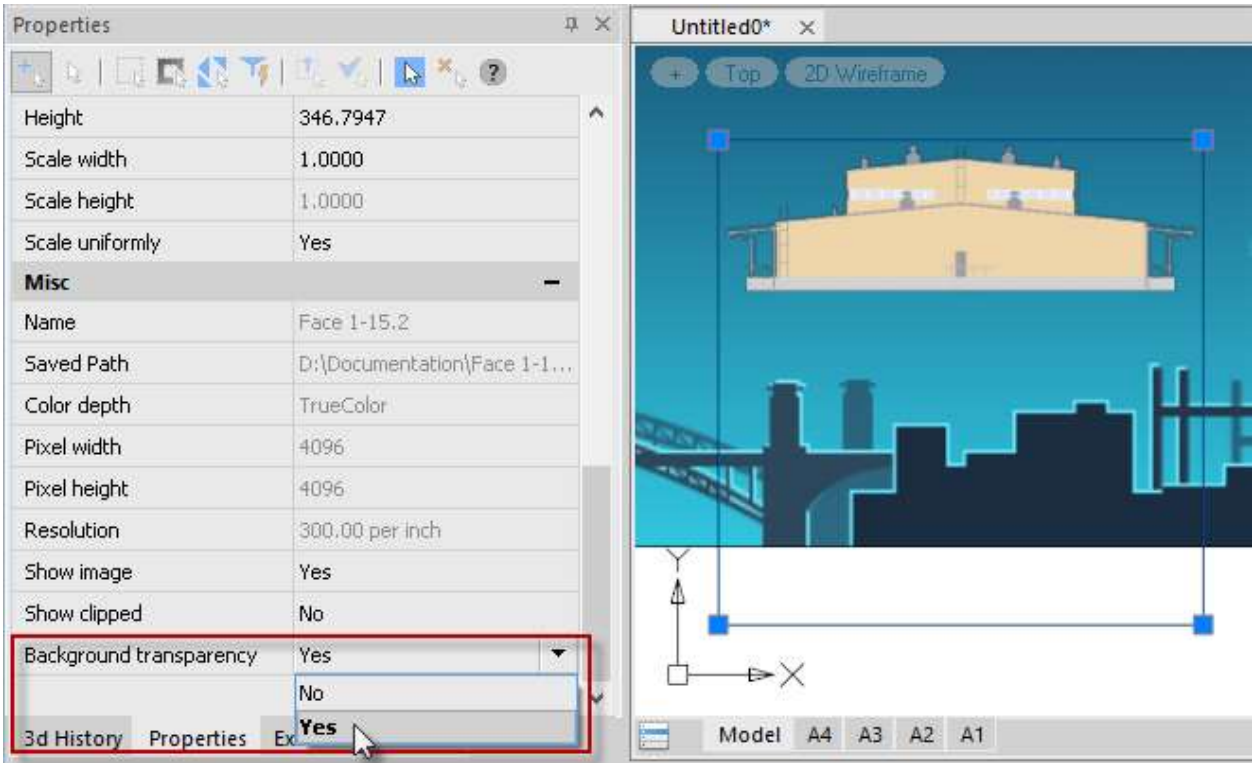
Now when importing PDF files using the **Import PDF** (PDFIMPORT) command, text data are placed on the same layers, on which they were when creating a PDF file. Names of layers will start with **PDF** prefix. For this, in the **Import PDF** dialogue box, when selecting layers parameters, select the **Use PDF layers** option;

Solid fills in nanoCAD Platform are now imported by a fill, and not as a contour as before;

A bug with importing gradient fills from PDF, when some types of gradient fills were not imported into the drawing has been fixed. Gradient fills are imported as raster images.

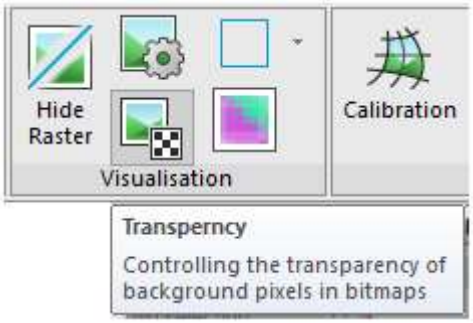
Alpha-channel for PNG files

The support of transparent background for PNG files (alpha channel) has been implemented.



Transparency command

A new command **Transparency** (TRANSPARENCY) controls the transparency of background pixels of raster images. The command is available from the ribbon (**Visualization** group), **Raster** menu and from **Raster** toolbar.



Sorting linetypes in an alphabetical order

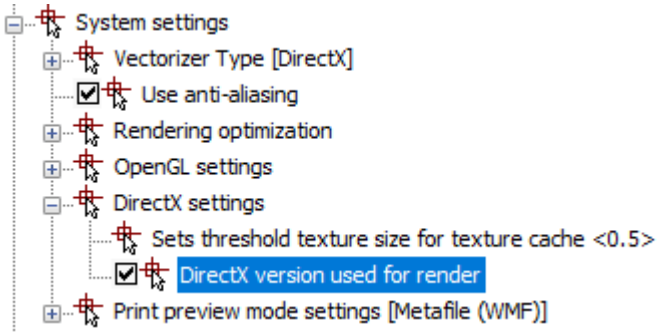
In the **Linetype** dialogue and in the drop-down list on toolbars, sorting of linetype names in an alphabetical order has been implemented.

Formulas in .dwg tables

Now, when opening files containing .dwg tables created in other CAD systems, the values in cells with formulas are recalculated and displayed correctly.

DirectX compatibility mode

You can optimize the work in the program when connecting to a workstation remotely via RDP by using the enhanced DirectX compatibility mode in the **Options** dialogue.



If the box is checked, DirectX 9 is used. When cleared, the default DirectX version is used. In most cases it is DirectX 11, but if it is not supported by the system, then it is DirectX 9. You can also switch DirectX version by the NCGS_TOGGLE_DIRECTX command.

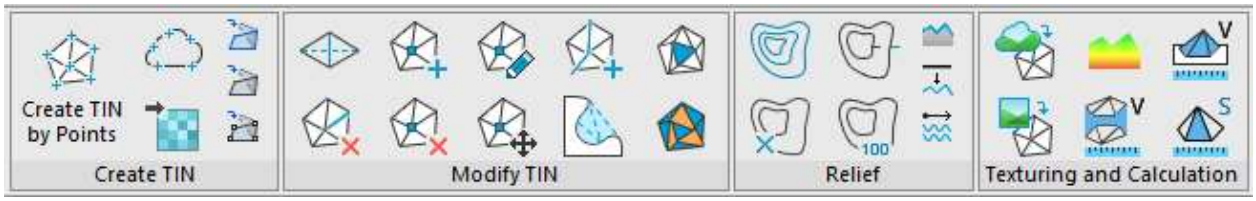
Continuation of work during an hour after the loss of license

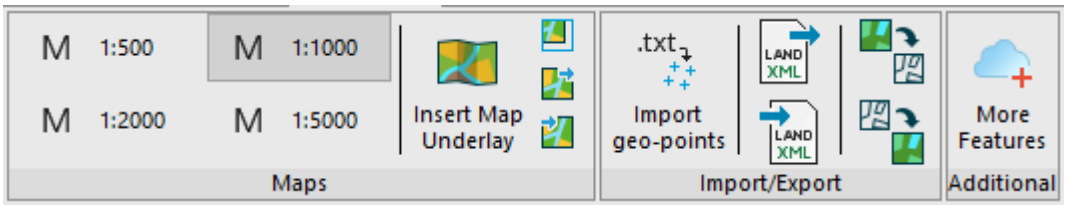
It is possible to continue working at a loss of license for one hour with periodic reminders.

Topoplan module

The **Topoplan** module contains tools necessary for topographers, which allows creating digital terrain models (DTM) based on the engineering survey data. The scope of application is any civil and industrial facilities.

The commands are collected in the **Topoplan** ribbon tab and **Topoplan** menu of the classic interface.



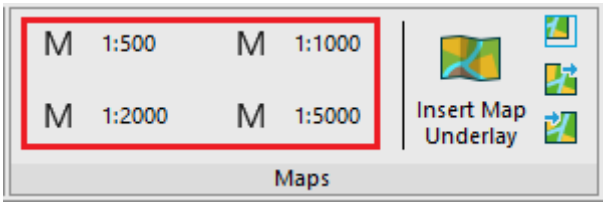


Drawing units and topographic scale

To work with geodetic survey data, there is a special drawing template with a set value for a drawing units – meters. The nanoCAD_EarthWork_metric.dwt template is located in the Templates folder. Find more information in the User guide in the section **Work with documents** – **Create a new document** – **Use template**.

Also you can set the drawing units – meters in the same name dialogue, description is in the section **Tuning nanoCAD** – **Drawing units**.

For correct creation of topoplan objects (bergstrichs and contour labels), you should set a topographical scale first. It is possible to set the desired scale quickly through special buttons displayed on the toolbar and on the ribbon – separately for each scale: 1:500, 1:1000, 1:2000, 1:5000.



Creating surfaces

Create TIN by points

Create TIN by points – allows creating surfaces based on Points and GeoPoints.

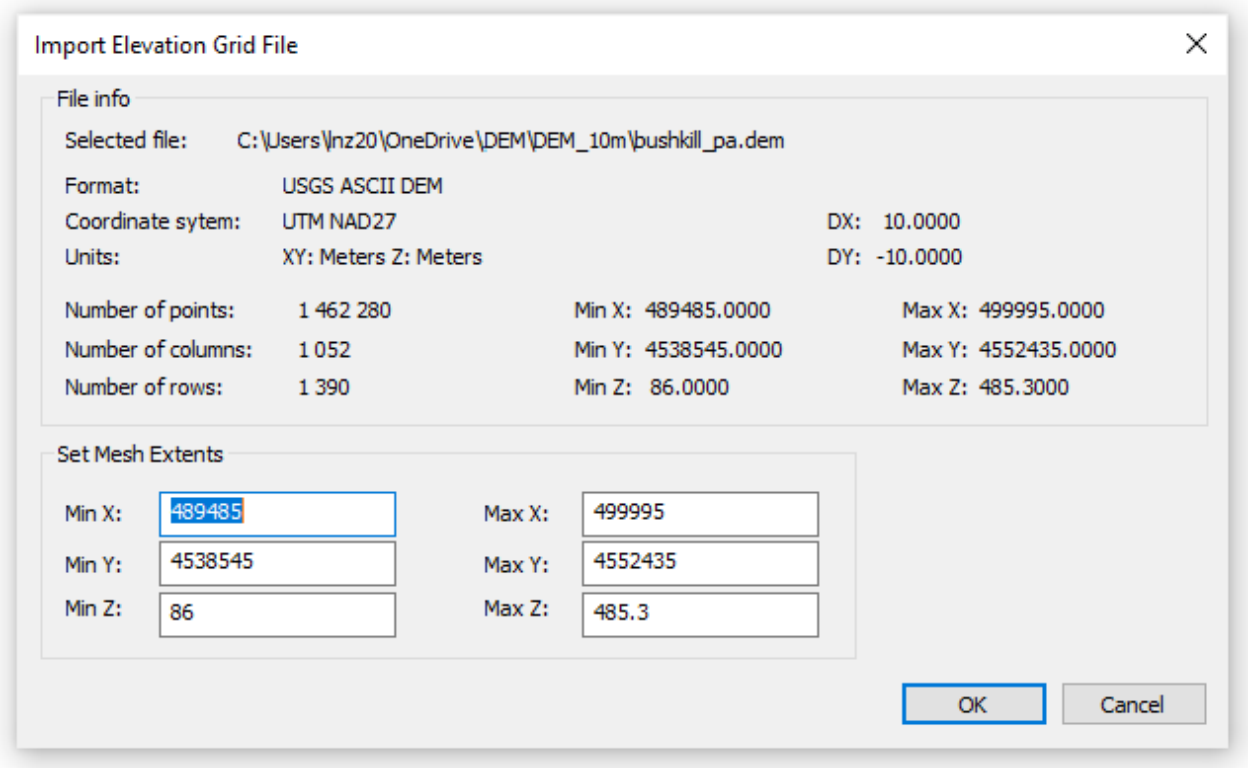
Break cloud into points

Break cloud into points – on the basis of a small point cloud in a drawing you can create **Point** or **GeoPoint** objects for further use in creating topoplans.

Keep in mind that for large point clouds this command may be resource intensive – both time and computer memory. For effective work with point clouds, it is recommended to use a specialized application. You can find more details by choosing the More features command.

Import elevation matrix

Import elevation matrix – allows creating surfaces based on imported elevation matrices of various formats.



Conversion commands

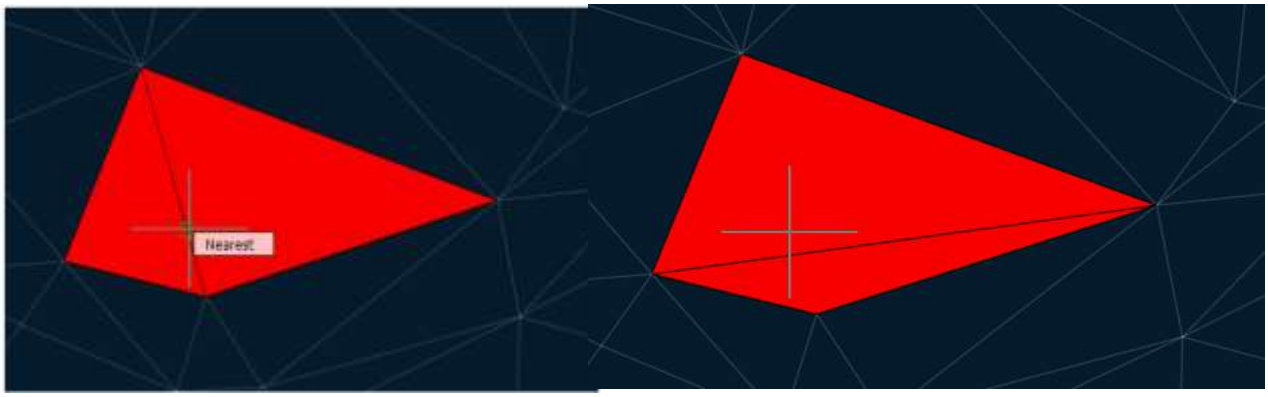
The conversion commands convert surfaces to **3D Faces**, **Submesh**, **Polyface mesh**.

Tools to edit surfaces

Edge flip

Surface before flip

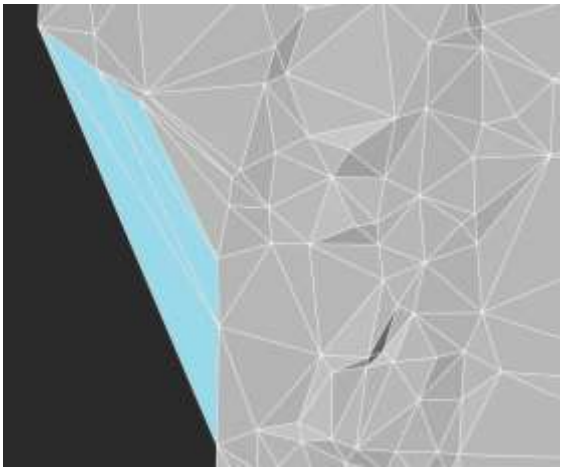
Surface after flip



Edge removal

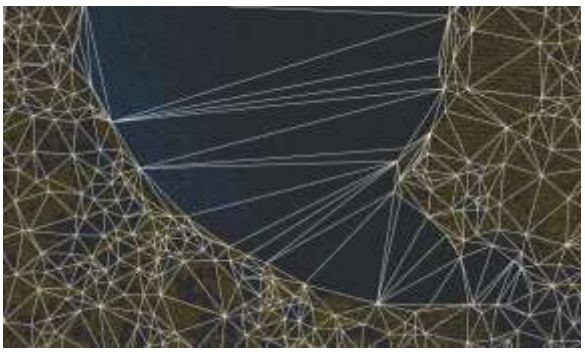
Faces to be removed

Surface after faces removal



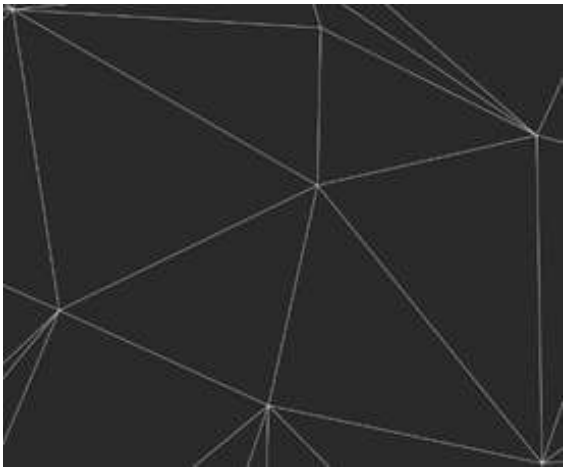
Face along the water surface

Faces are removed

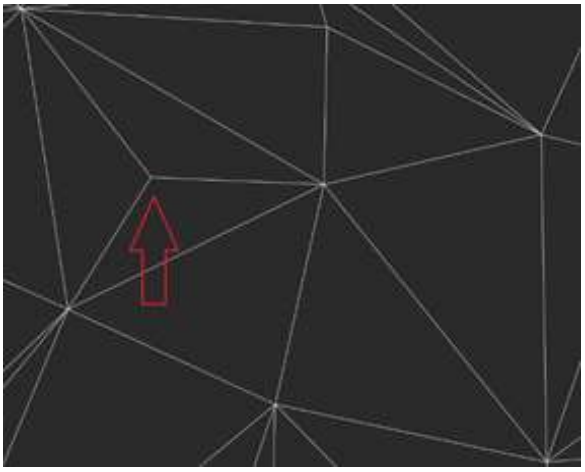


Adding a vertex

Surface before adding a vertex



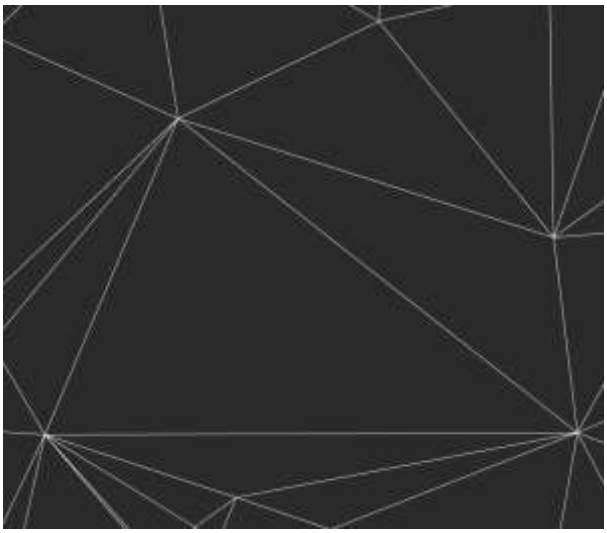
Surface with an added vertex



Vertex removal

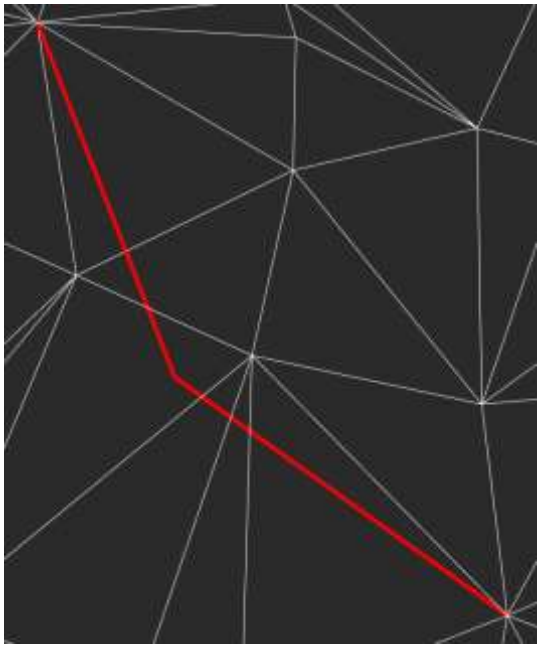
Surface before a vertex removal

Surface after a vertex removal

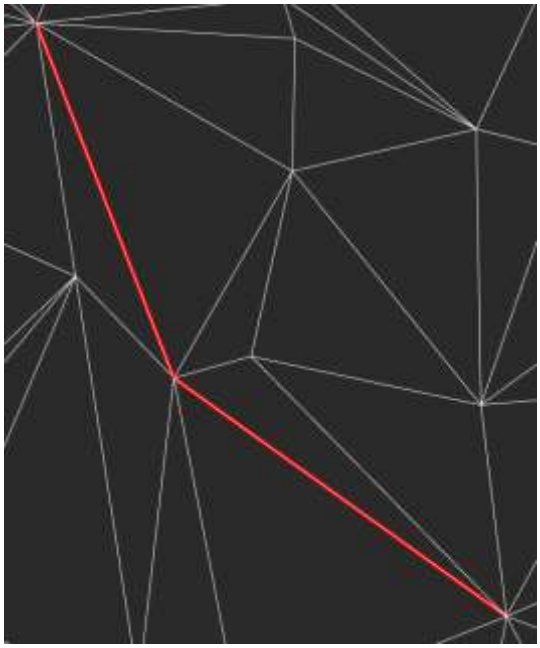


Adding a structure line

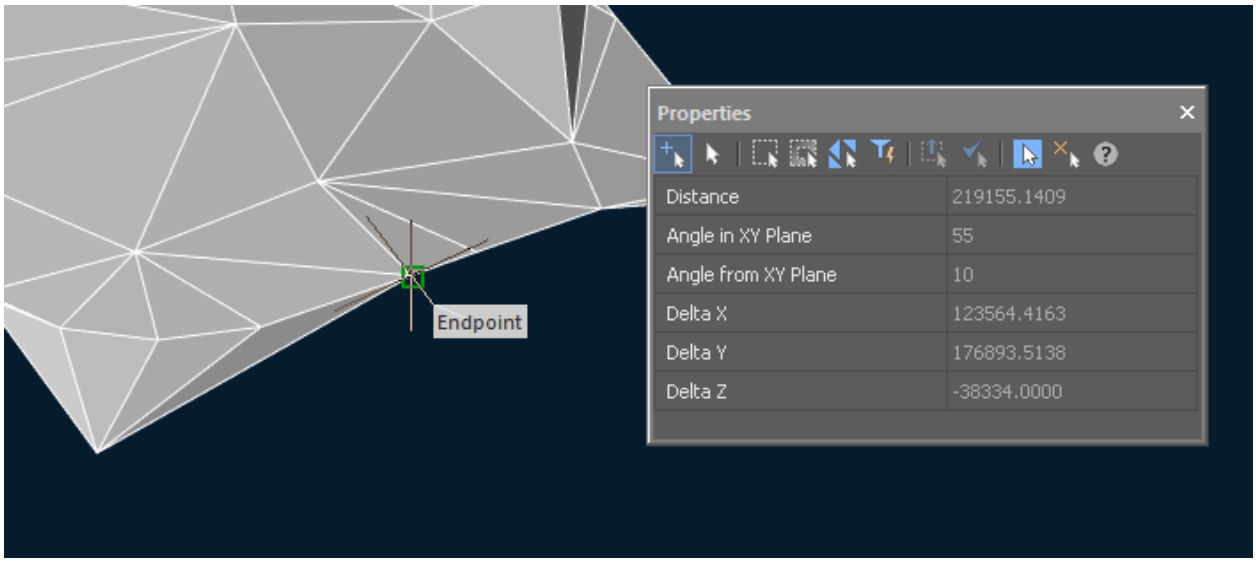
Surface before adding a structure line



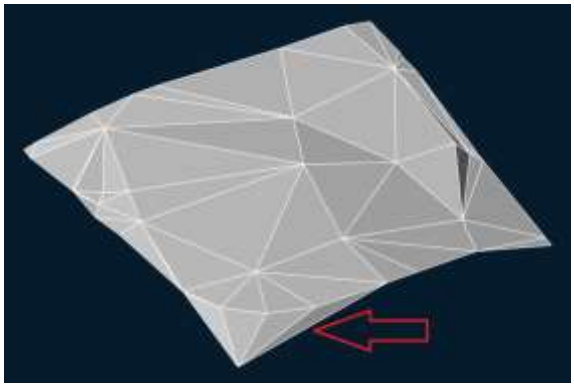
Surface after adding a structure line



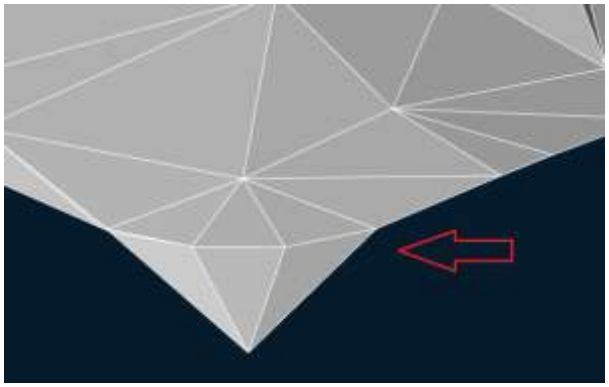
Mesh boundaring (removing edges along the surface boundary)



An edge that interferes with the correct construction of contours

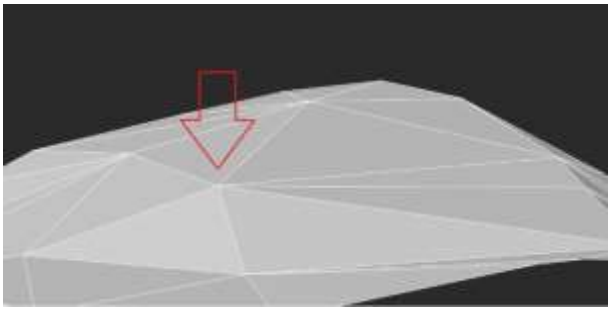


Result of mesh boundaring

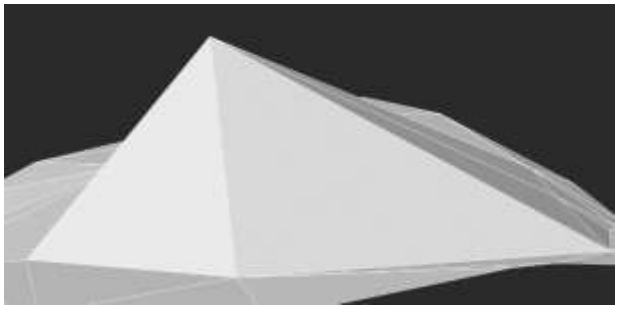


Elevation change

Vertex of the surface, whose elevation should be changed

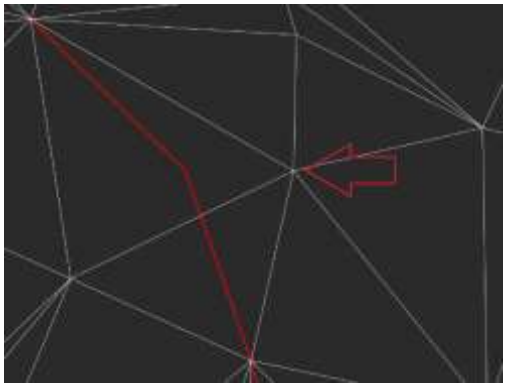


Surface after vertex elevation change

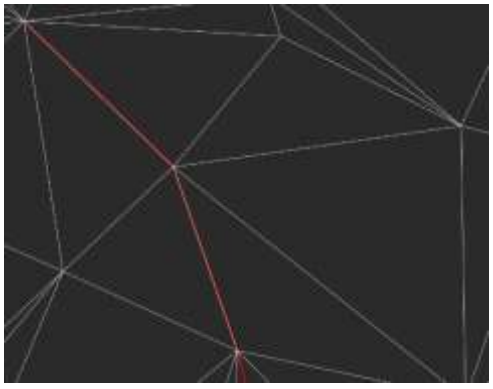


Moving a point

Node to be repositioned



Node position changed



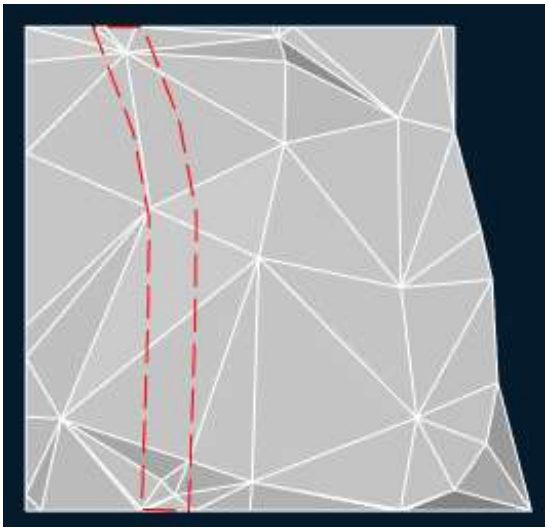
Cut mesh

Cut mesh – the command is used to divide a mesh into sections.

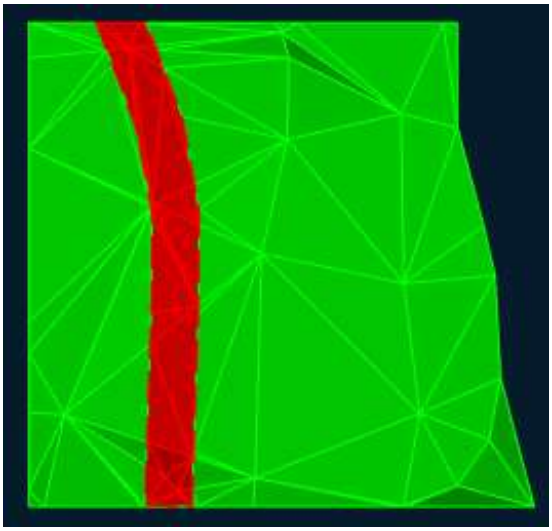
Mesh classification

Mesh classification – the command is used to divide a mesh into classes: for example, if you need to separate road surface from lawns.

Mesh before classification



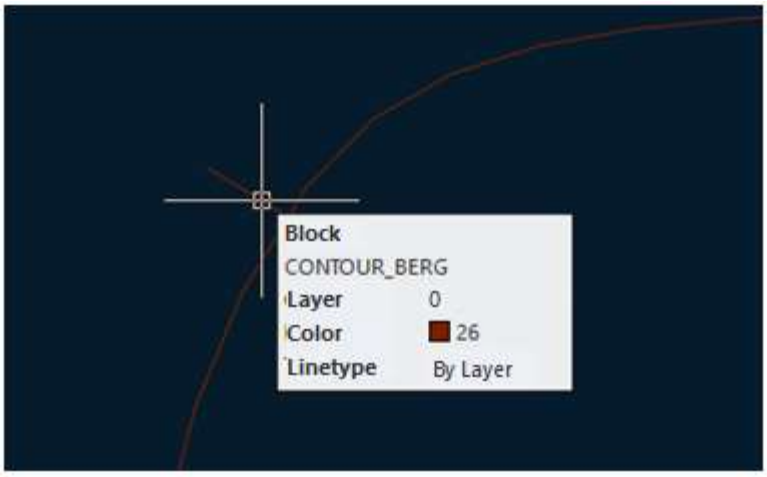
Mesh after classification



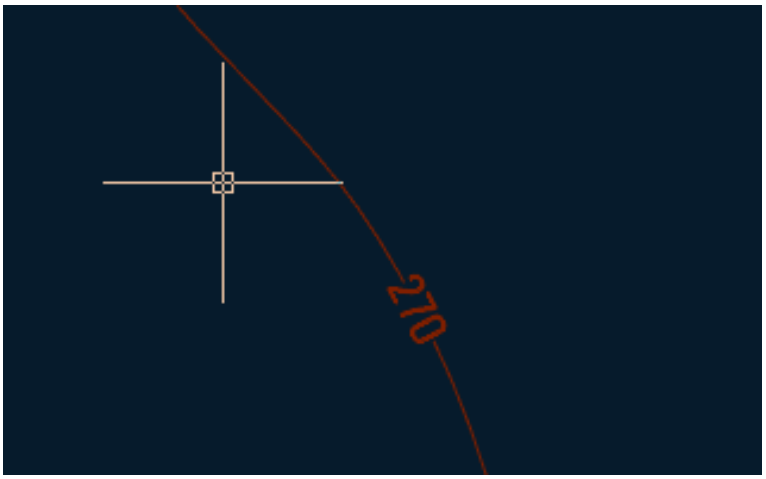
Tools to create terrain relief

- **Constructing contour lines**;

- **Deleting contour lines**;
- **Bergstrichs**;



- **Contour labels**.



Draw a profile line

Draw a profile line – constructs a profile 3D-polyline on the surface.

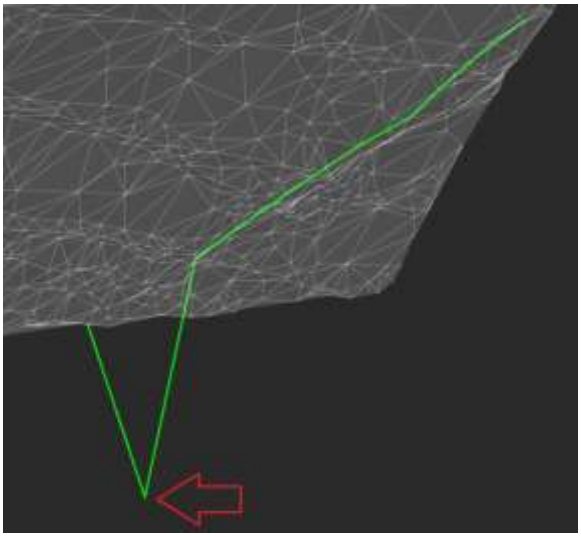
Projecting a line on a mesh

Project a line on a surface – the command is intended to project the existing plan elements on the surface.

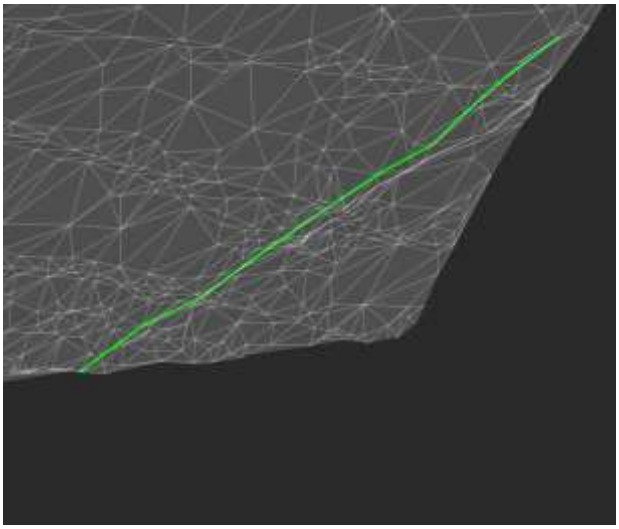
Correcting zero elevations

Correct zero elevations – the command approximates zero (insignificant) elevations (Z coordinates) of a 3D-polyline vertices based on the data from the nearest vertices with non-zero elevations.

Node of 3D-polyline to be corrected



3D-polyline after correcting elevation



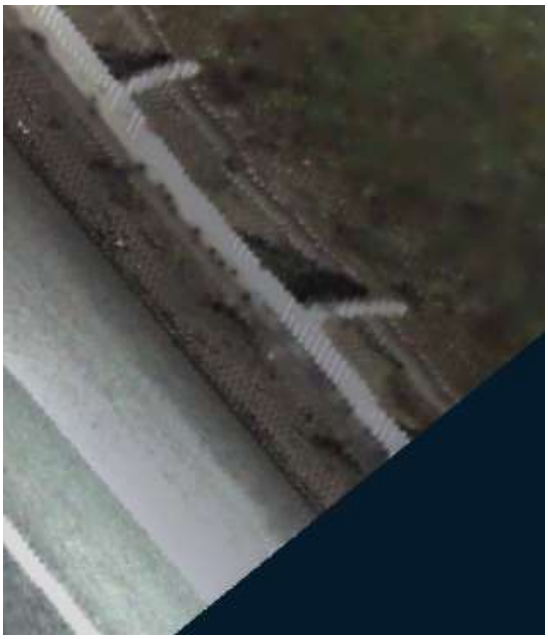
Texturing meshes and performing calculations

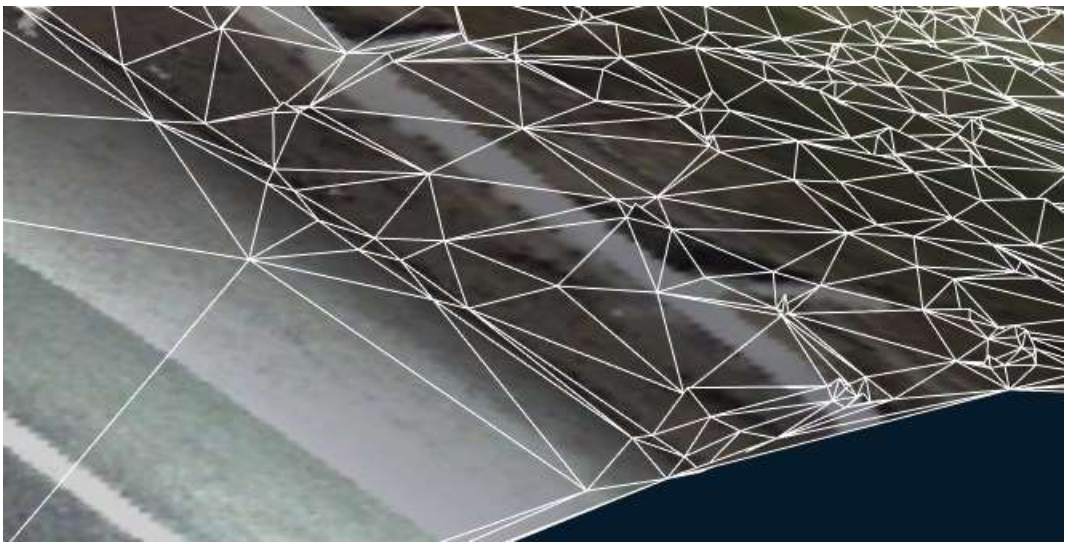
Applying planar texture

Apply planar texture – applies texture from a point cloud to the surface.

Mesh before applying texture

Mesh after applying texture

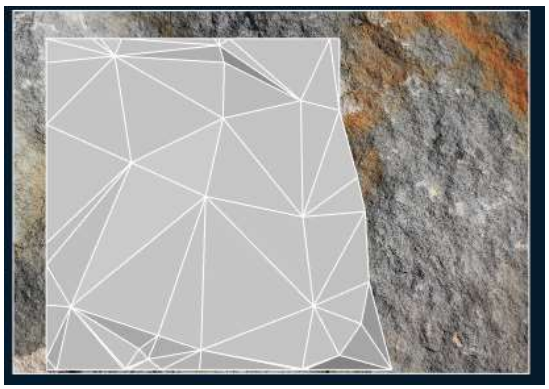




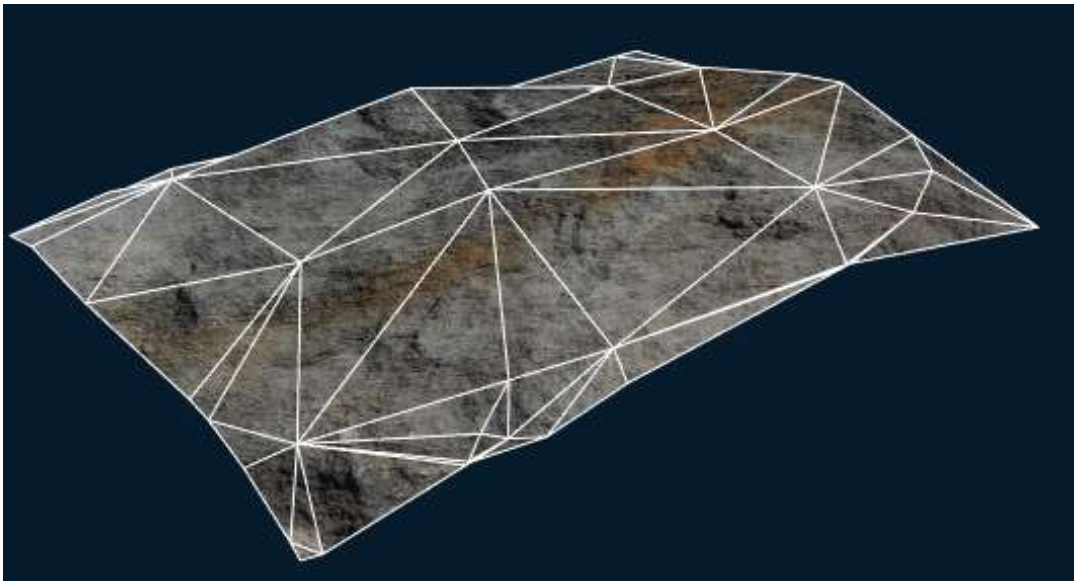
Applying raster texture

Apply raster texture – the command is used to texture a surface using raster images.

Mesh before applying texture

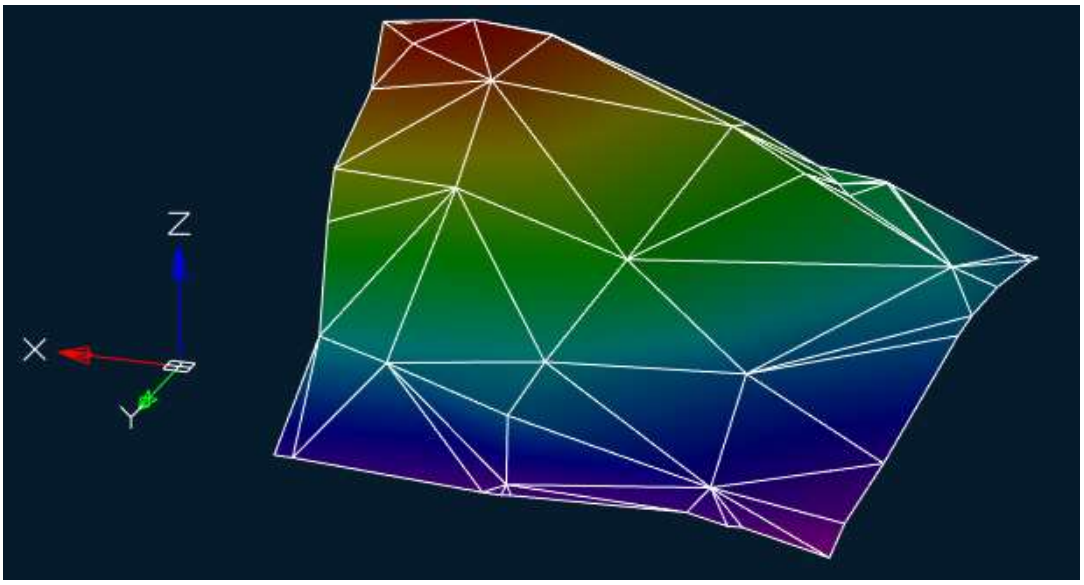


Mesh after applying texture



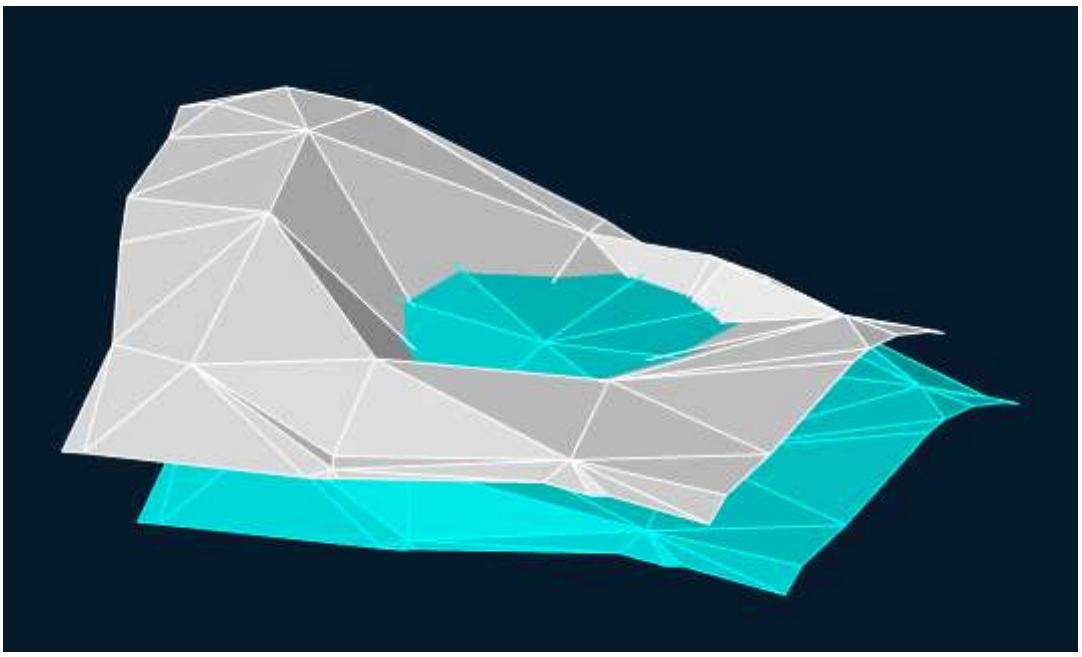
Coloring mesh by height

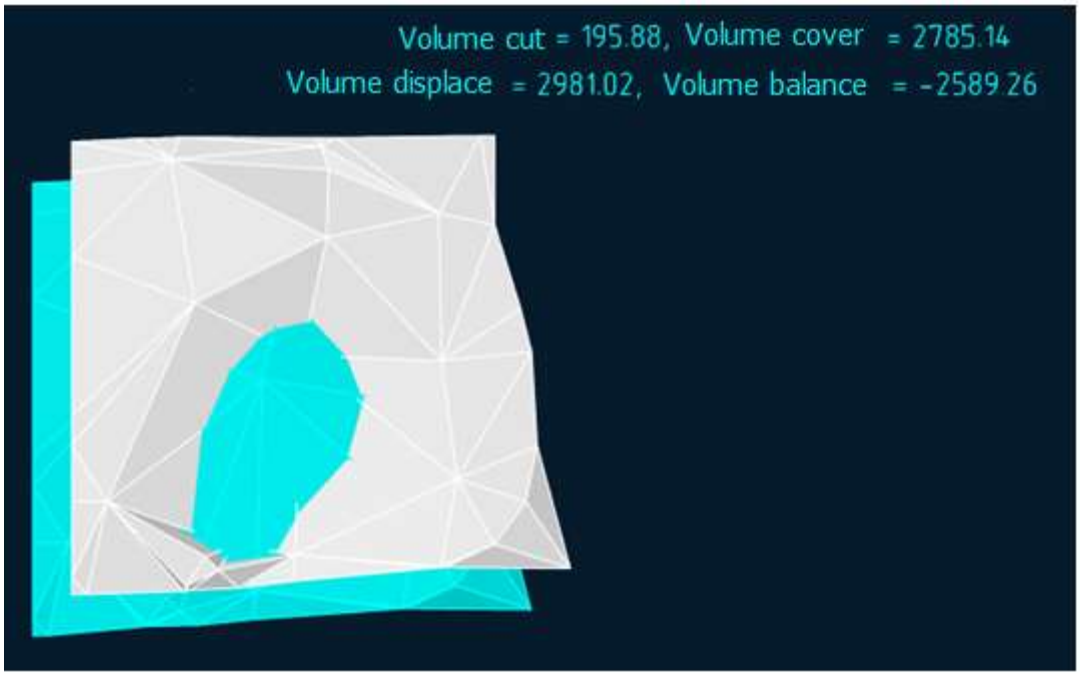
Color mesh by height – colors surfaces with a gradient color.



Calculating volume between models

Calculate volume between models – the command allows calculating the volume of intersecting surfaces: Common, Differing and Balance.





Calculating surface volume

Calculate surface volume – the command calculates the surface volume (complete or within a given contour).

Calculating surface area

Calculate surface area – the command to determine the area of surface plot.

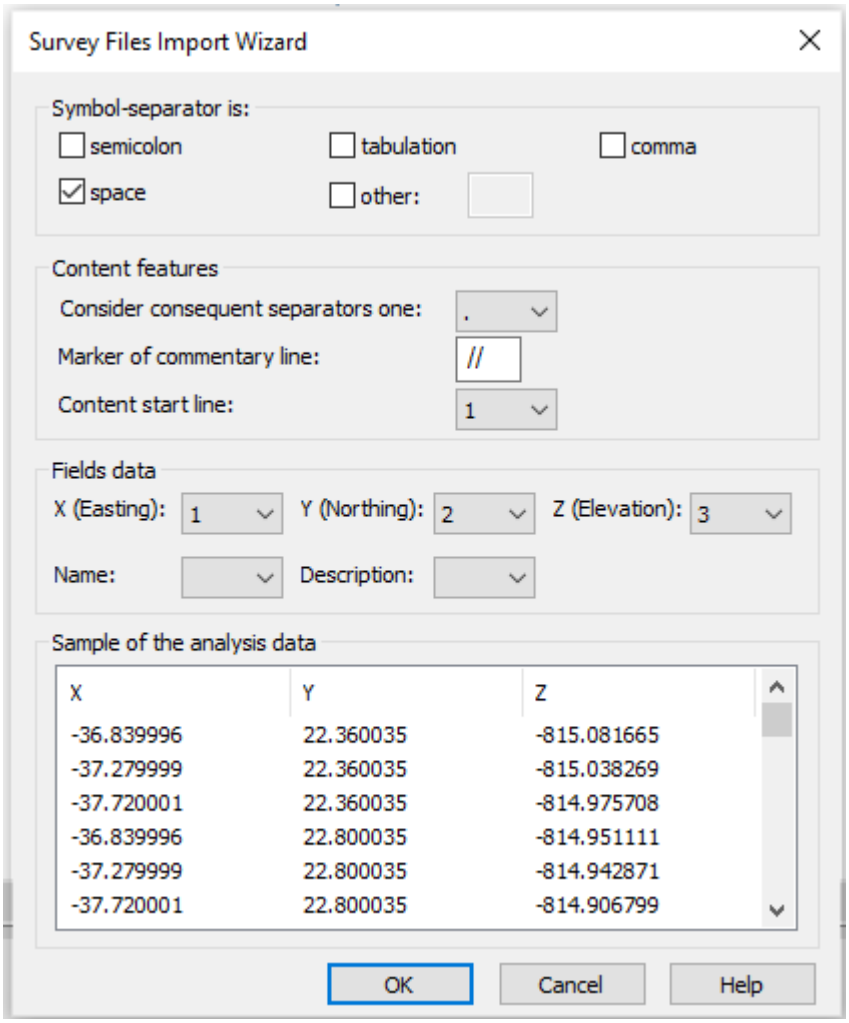
Support of Civil 3D objects

The **Topoplan** module includes support of Civil 3D objects.

Now, if a drawing contains such objects, they will not be displayed as proxy graphics. With help of **Properties** bar you can edit the style of their display, as well as view some characteristics.

Functionality of import and export of objects

Geopoints import – loading points from *.TXT and *.XYZ using the Import Master and creating geopoints with attributes in a drawing.

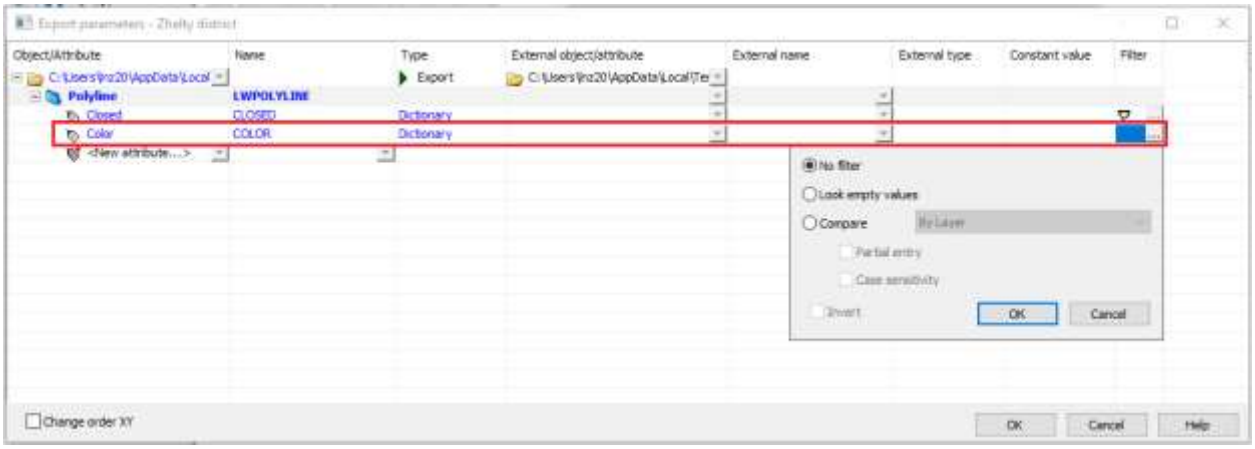


Import from LandXML format – the command allows you to import a surface from **LandXML-1.2** file format. As a result, a Mesh object will be created in a drawing.

Export to LandXML format – the command exports a surface (Mesh object) to **LandXML-1.2** file format for use in other applications.

Import from GIS – importing MIF and SHP files and creating polylines and point objects in a drawing.

Export to GIS – creating MIF and SHP files on the basis of drawing objects.

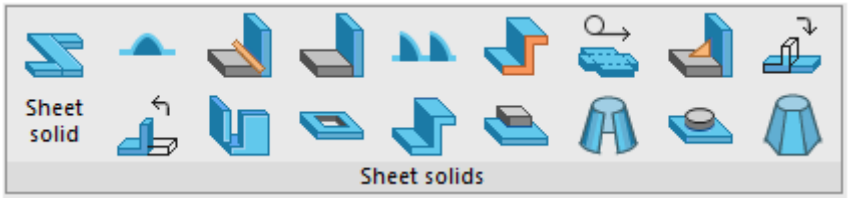


New functionality of 3D module

Modeling parts from sheets

A large number of tools for designing parts from sheet material in 3D appeared in the new version.

Sheet modeling commands:



Basic tools (**Sheet solid**, **Bend edge**, **Bend over segment** and **Fold by sketch**) allow you to create standard parts from sheet material. Any of the folds of a sheet body can be bent and unbent at the right time.

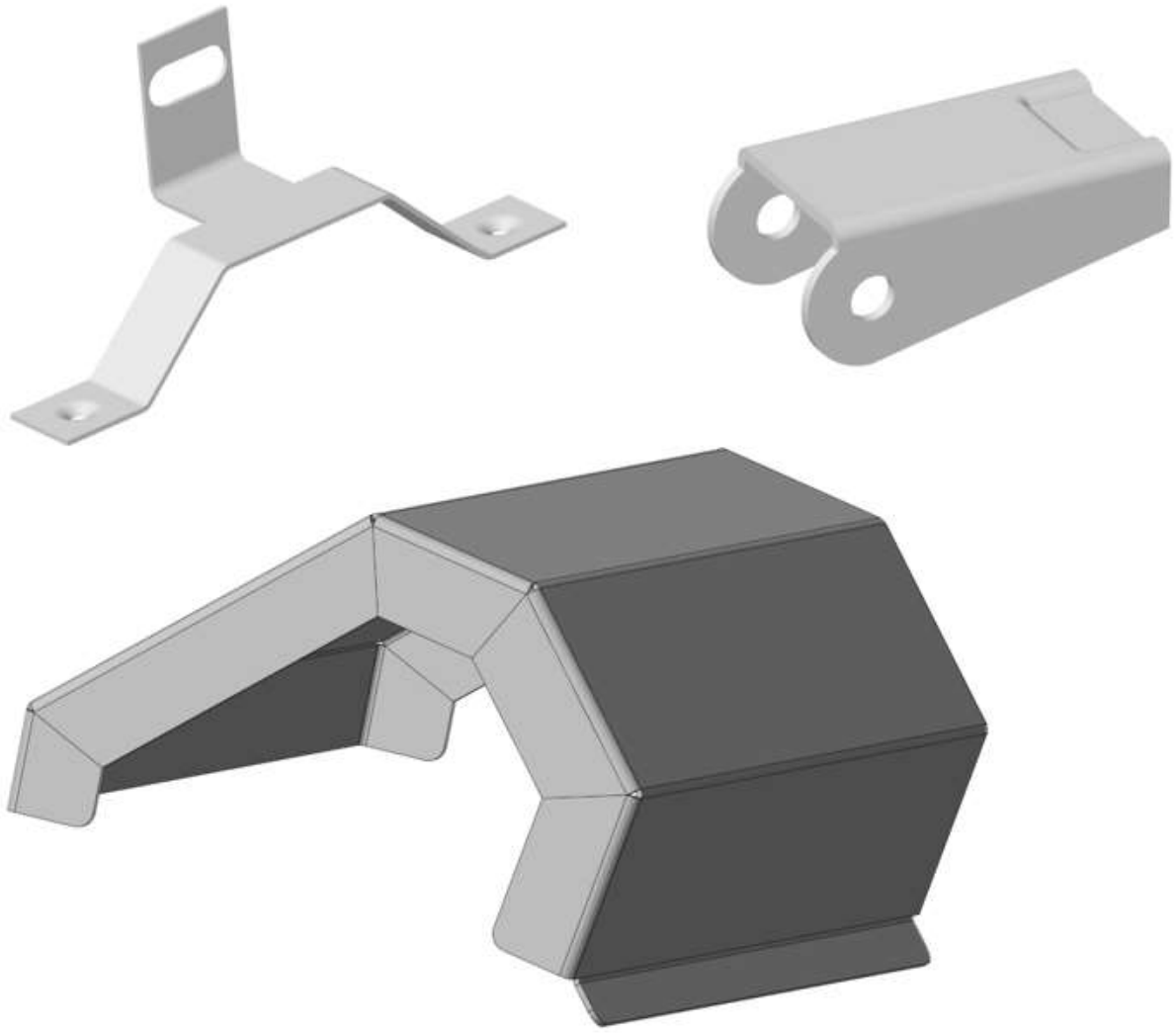
For specific sheet parts you can use the **Shell** and **Ruled shell** tools.

There are also special tools for quick construction of sheet part elements that are frequently met in practice:

- collar;
- shutters;
- stiffener;
- hole;
- undercutting;
- plate;
- stamping.

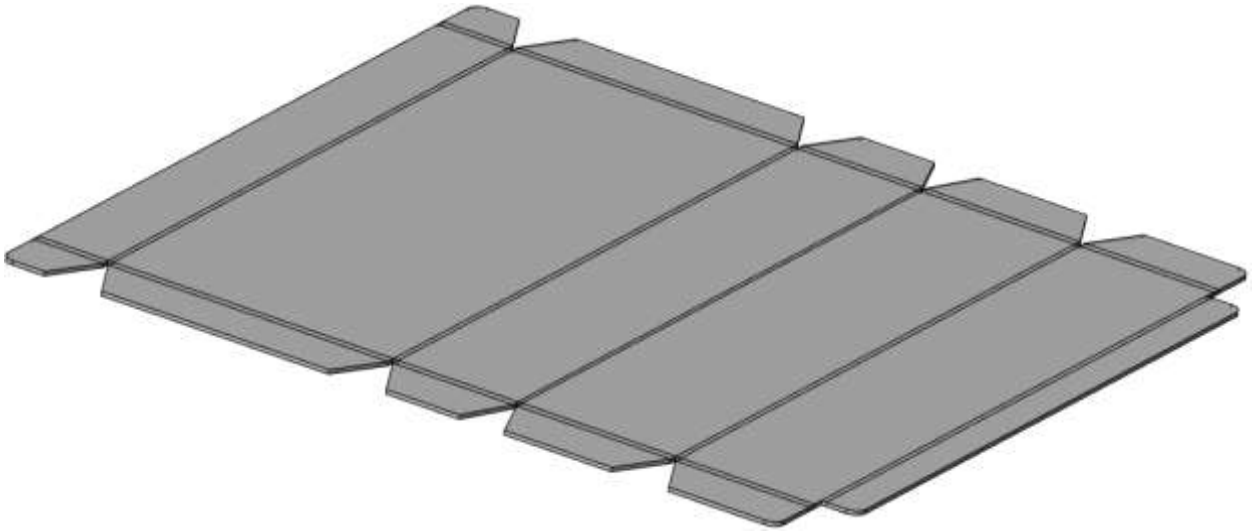
You can make diverse variants for closing corners.

Parts designed by sheet modeling:



The **Unbend** tool allows you a flat drawing of a part from a sheet, from which you can make a 2D view to create a drawing.

Sheet tool flat drawing:



3D Modeling mode

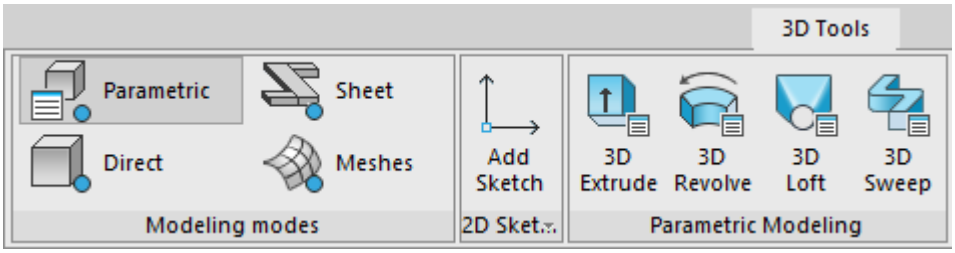
With each new version, more and more new commands appear. To make it more comfortable for users to work in all the variety of different functions, and most important, to make it easier to focus on the main things, the main 3D modeling commands have been grouped by 4 types of modeling:

- parametric;
- direct;
- sheet;
- meshes.

When you switch the modeling, the ribbon is automatically modified and it displays the commands specific for the selected type of modeling.

I.e., when switching to parametric modeling, a user will see commands for creating 3D geometry based on parametric sketches.

Parametric modeling tools:

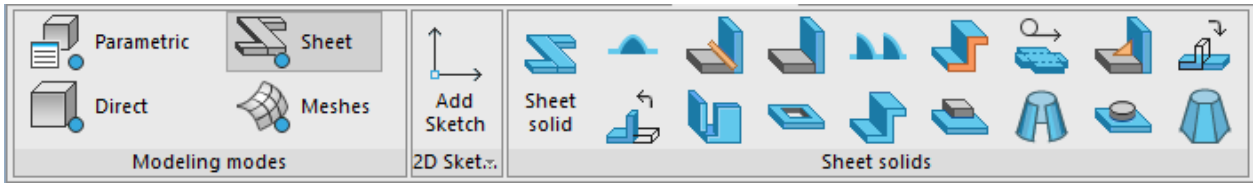


When switching to direct modeling, it will display commands for creating typical 3D shapes and operations related to direct modeling.

Direct modeling tools:

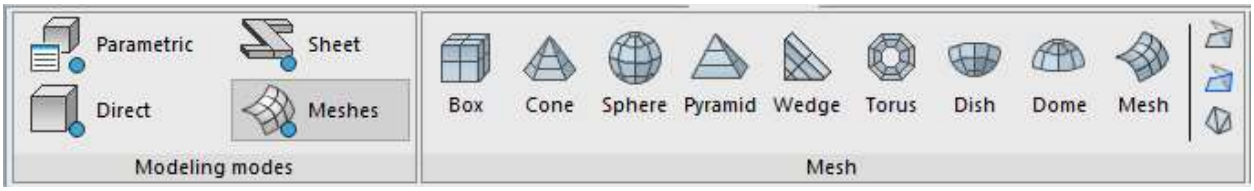
When switching to sheet modeling, the main commands will be those specific for such mode **Sheet solid**, **Shell**, **Unbend** and others.

Sheet modeling tools:



In the **Meshes** modeling, standard commands for creating 3D surfaces become available.

Surface modeling tools:



Commands that apply to all types of modeling such as, for example: Chamfers and Rounds, Boolean operations, commands of construction geometry, manipulation and others are always visible to a user.

Updated sketch mode

Now it is much more convenient to work in the sketch mode, if you use the ribbon interface style. There is no need to switch between the ribbon tabs. After launching the 2D sketch mode, all the tools necessary for creating parametric sketches are displayed to the user. At that, the most frequently used commands are always at hand and there is nothing superfluous.

Ribbon view in the sketch mode:

Tangency and symmetry 3D constraints

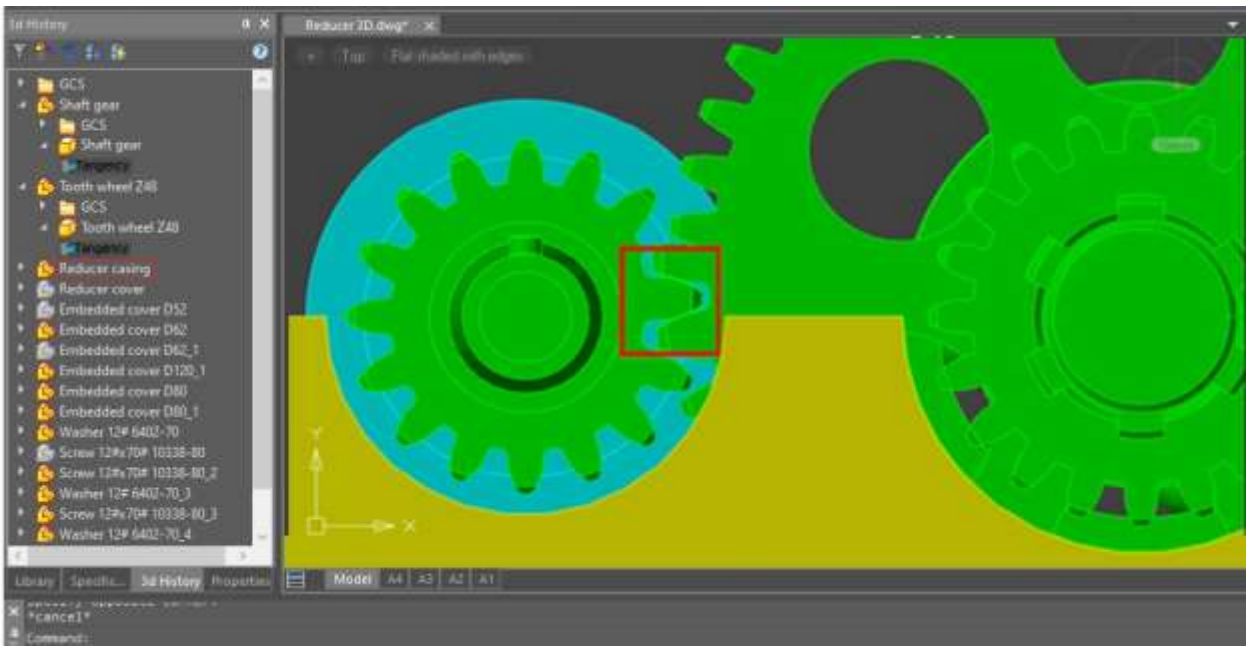
In addition to **3D-insert**, **3D-merge** and **3D-corner** constraints, two more appeared: **3D-tangency** and **3D-symmetry**.

Icons of 3D constraints commands:

3D-tangency constraint allows you to create more complex surface tangents than **3D merge** constraint. You can, for example, create tangents of:

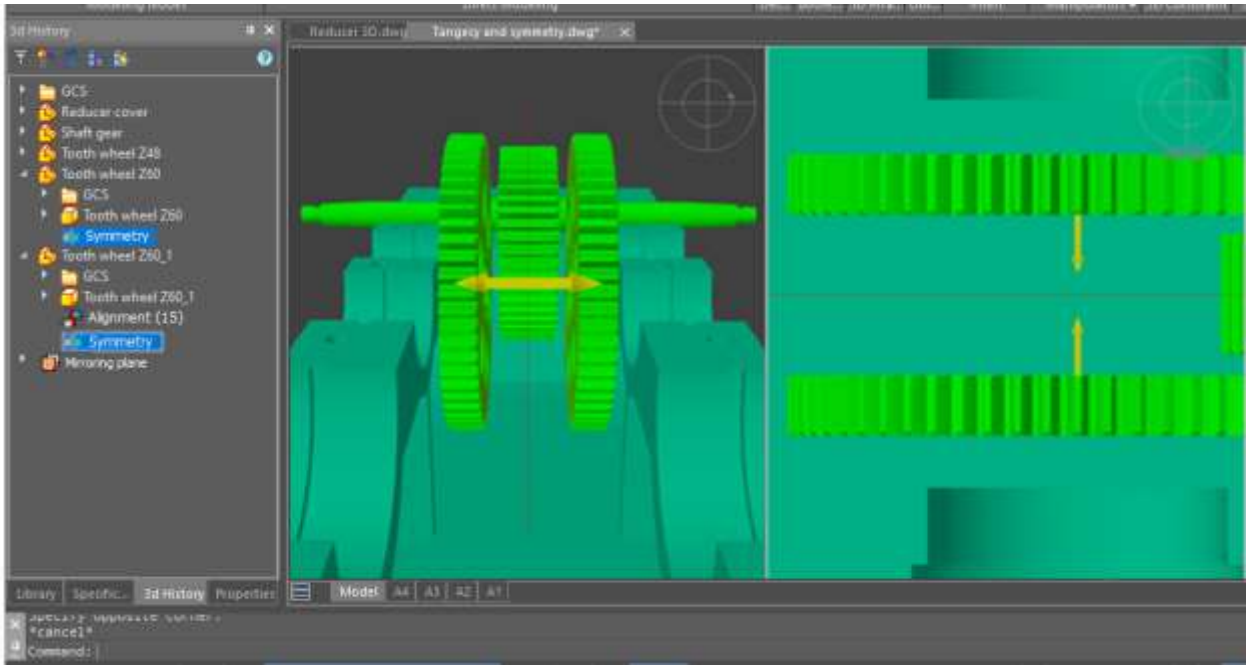
- cylinder to plane;
- cylinder to cylinder;
- cone to plane;
- sphere to cylinder;
- sphere to plane;
- circular edge to straight edge;
- circular edge to circular edge.

3D-tangency constraint:



3D-symmetry constraint allows you to align 3D solid elements symmetrically related to a selected plane.

3D-symmetry constraint:



New functionality of Construction module

The calculation of volume of wall materials of layers and the creation of automatic materials list for the selected walls have been implemented.

New report generation options have been added to the Sheet of Decoration:

- The areas of premises with the same decoration can be summed or not;
- The premises number and name can be placed in separate columns.

New functionality of the Mechanica module

Database of materials with elements from a common base

In connection with the transition to a new database management system and taking into account wishes of users concerning the database of materials, a new database of materials now includes elements from the general database of objects.

Compared to the old database of materials, the following improvement have appeared:

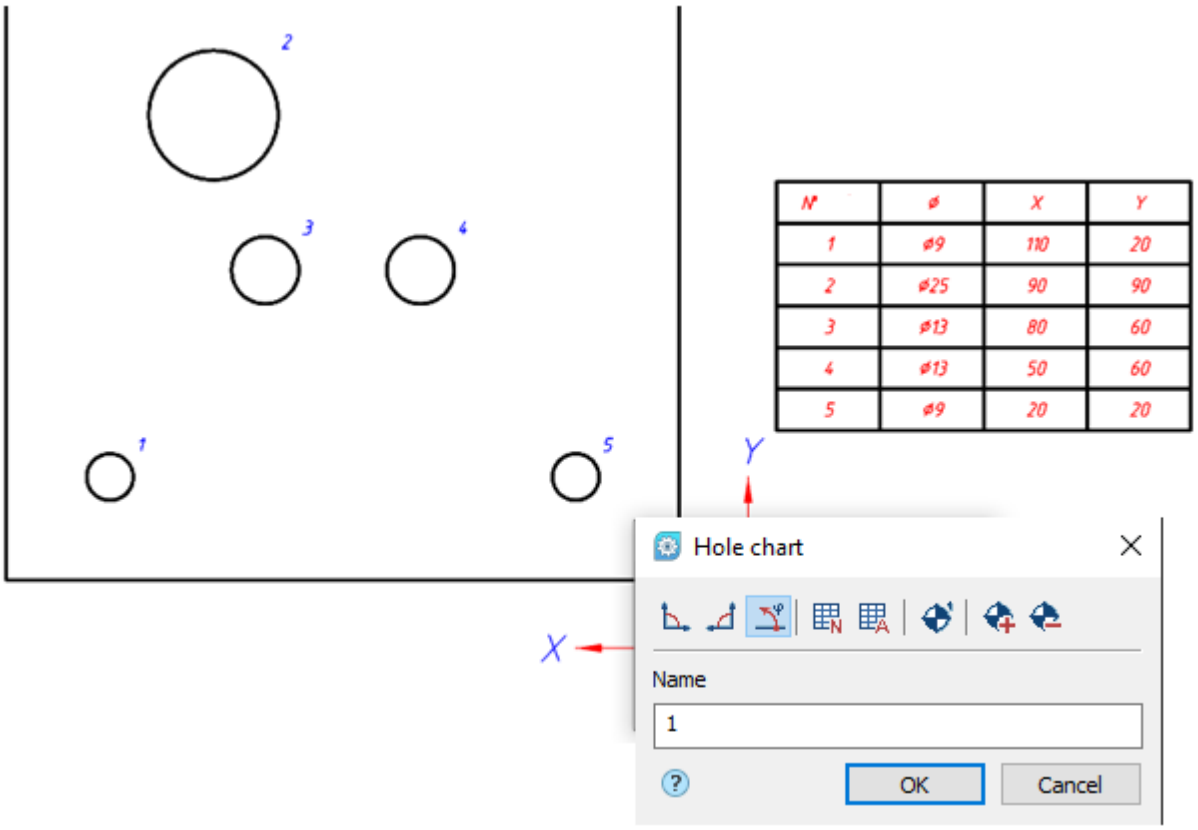
- more complex and varied materials are available in the database on the basis of standards;
- it is possible to form legend of materials in several lines;
- it is possible to create custom materials.

Improving the functionality of filling holes and tables of holes

In the **Hole fill** tool, the conversion to a threaded hole has been improved:

- the hole diameter becomes the outer diameter of the thread;
- the internal thread diameter is constructed depending on the selected snap;
- the name of metric thread is set.

In the **Tables of holes** tool, the possibility has been added to construct the left coordinate system of holes.



Other changes

- Dimensioning in UCS has been corrected.
- Symbols for various surfaces in the drawing have been corrected.