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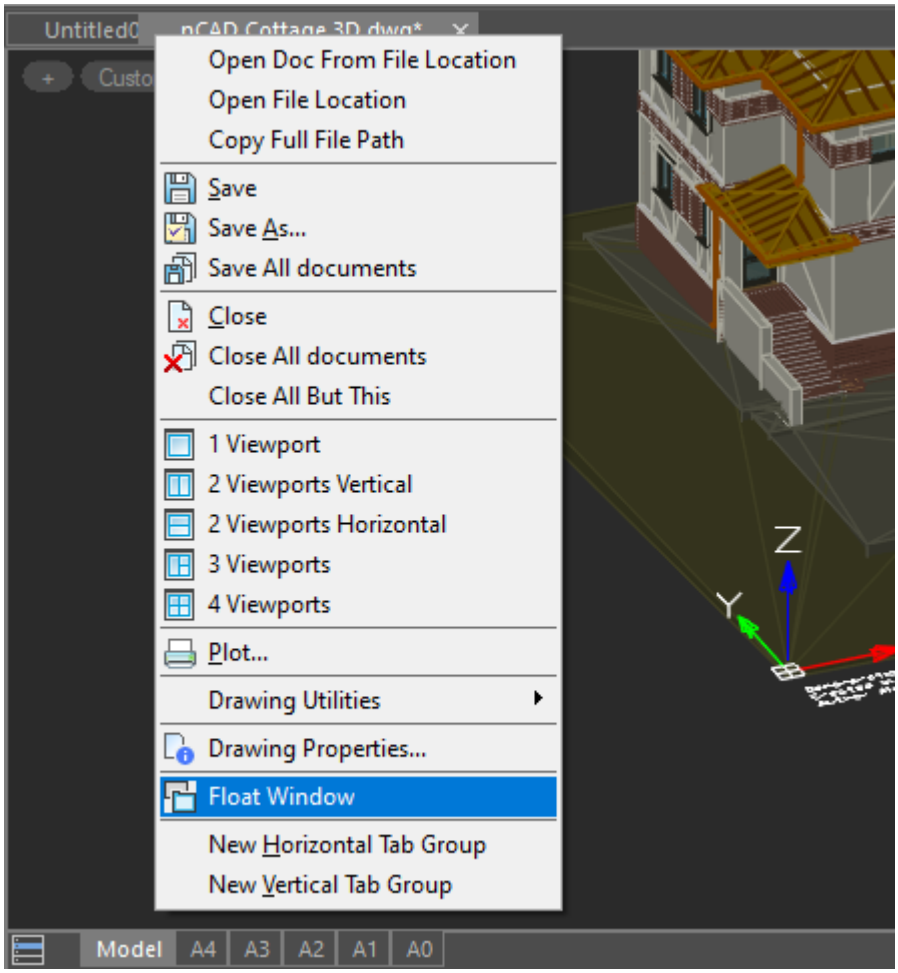
New Features of nanoCAD 22.0

* The most relevant and complete description of the new features of the nanoCAD 22 Platform is available online.

New Functionality of the Platform

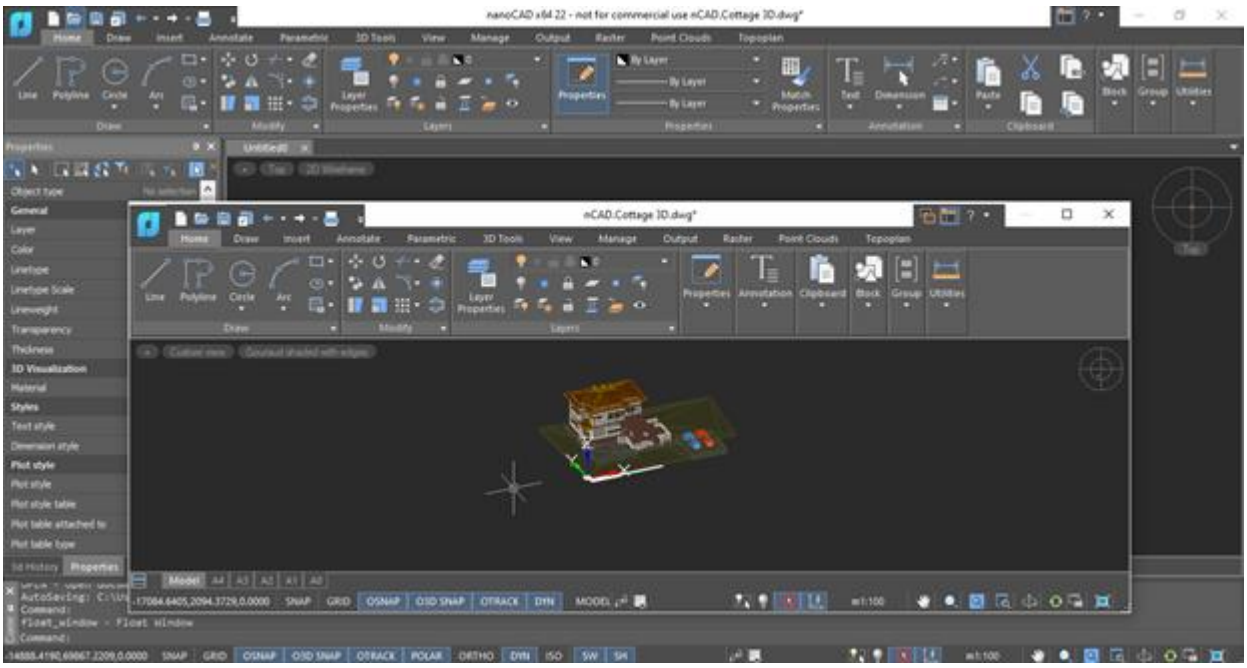
Float Windows

Now a document window (drawing file tab) can be detached from the nanoCAD Platform window by making it a float window using the **Float Window** (FLOAT_WINDOW) command.



The command can be called from the context menu of the document tab or from the command line.

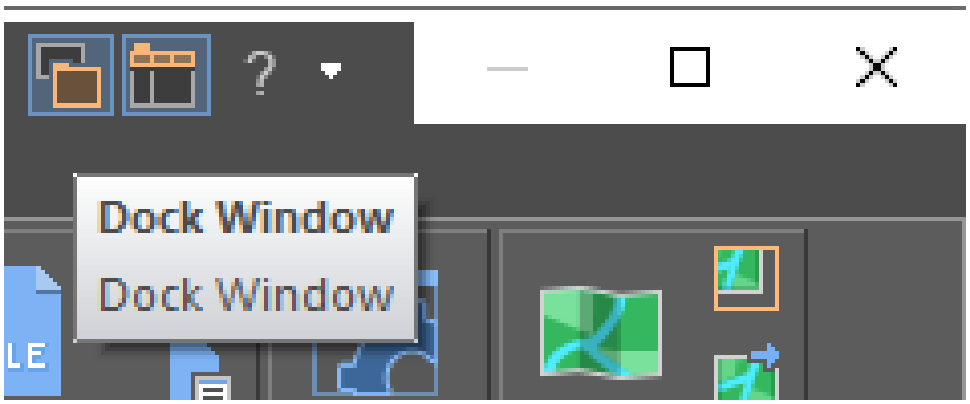
As a result, the document tab will be undocked from the platform window as a floating window.



The floating document window is always in the foreground. It does not contain a command line, it cannot contain document tabs. You cannot attach functional bers to it.

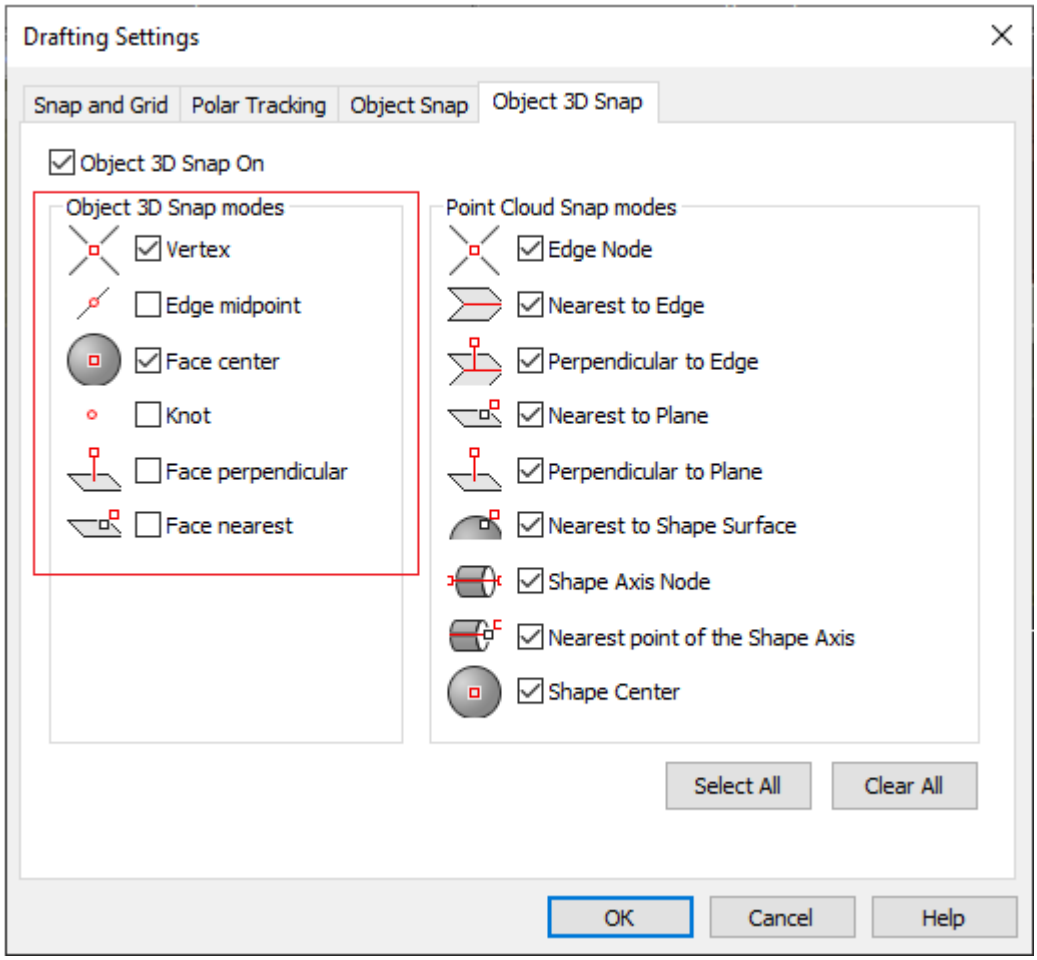
In the **Options** dialog you can specify: whether the floating window should contain a ribbon, a menu, and a status bar.

You can attach a floating document window back to the main platform window as a tab using the **Dock Window** (DOCK_WINDOW) button on the right side of the window title bar.



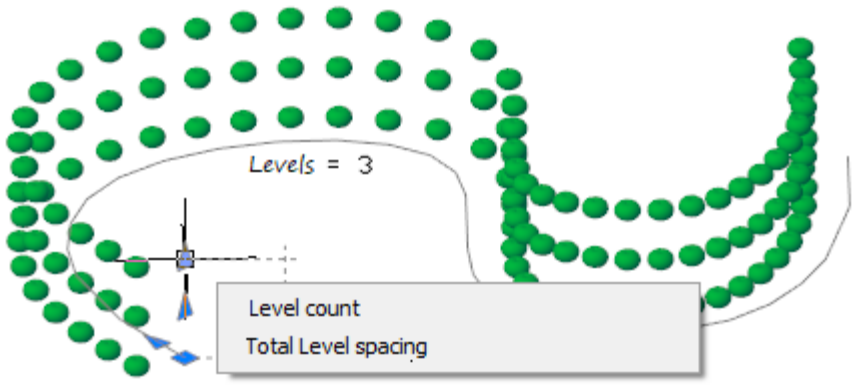
3D Snaps

3D snaps for solid objects (objects of the **3D Solid** type) have become available. Their setting is separate from 3D snaps to recognized point cloud geometry in the **Drafting Settings** dialog box and the context menu.

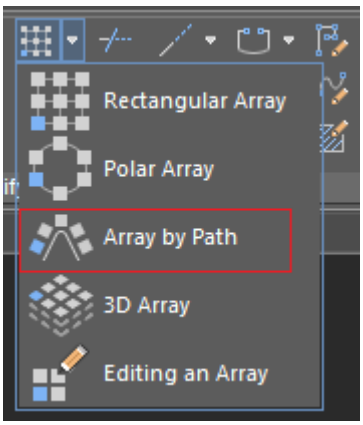


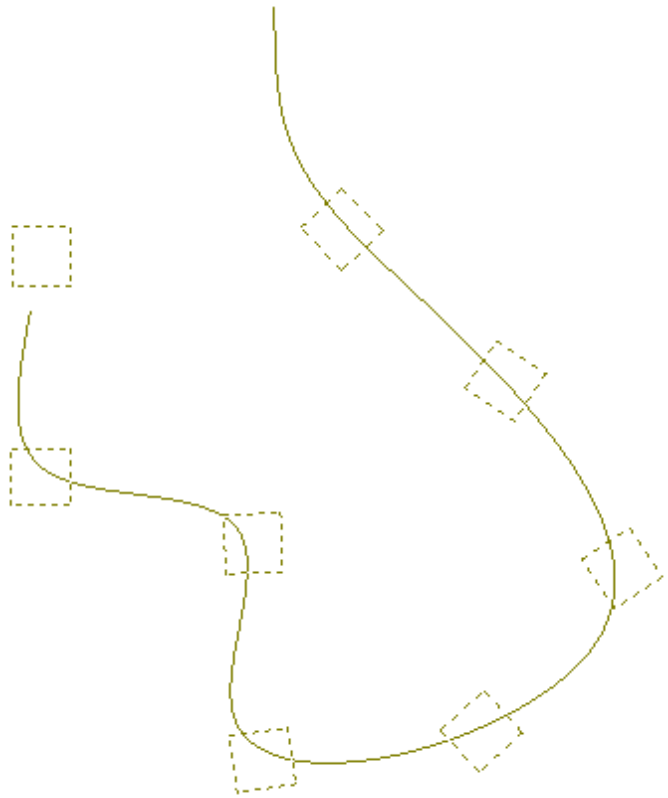
Associative (Dynamic) Arrays

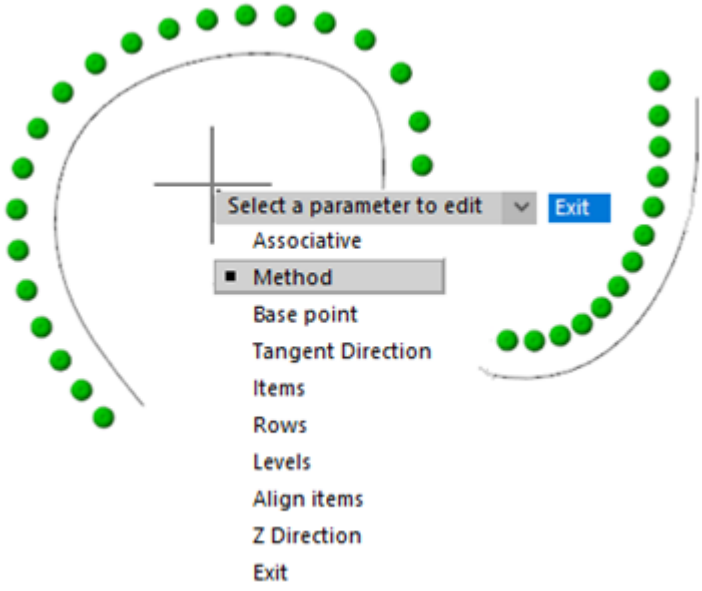
The functionality has been added for creating and editing a new type of associative (dynamic) arrays – **Array by Path**. Previously, such arrays were perceived as blocks.



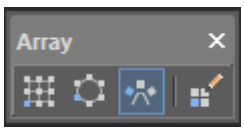
The **Array by Path** (ARRAYPATH) command allows you to evenly distribute the object copies along a path or a part of it. The path can be a line, polyline, 3D polyline, spline, helix, arc, circle, or ellipse.



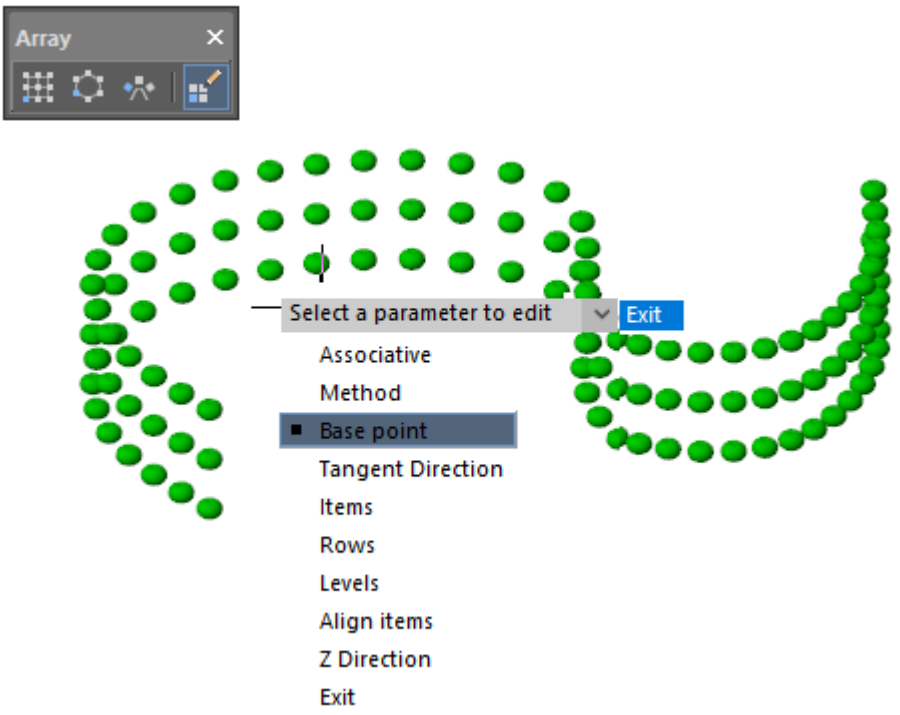


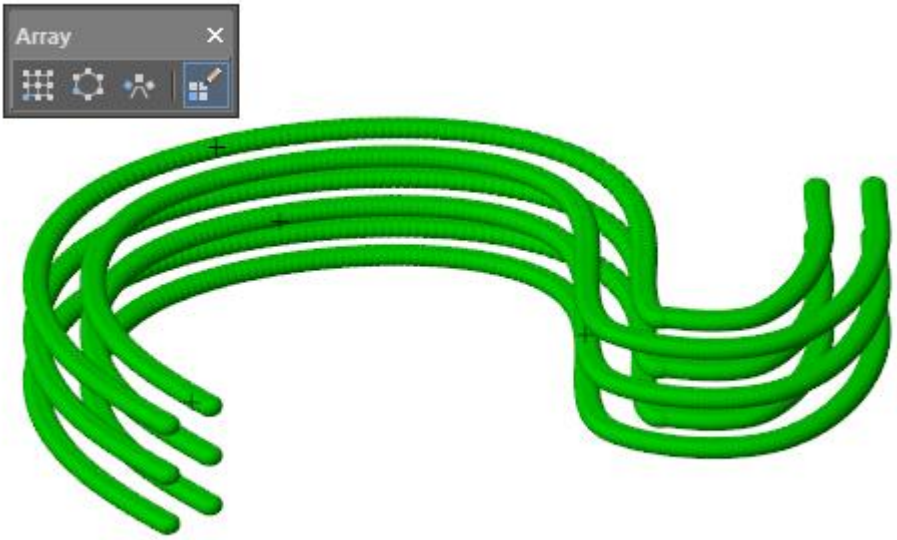


The **Array** toolbar appeared in the classic interface.

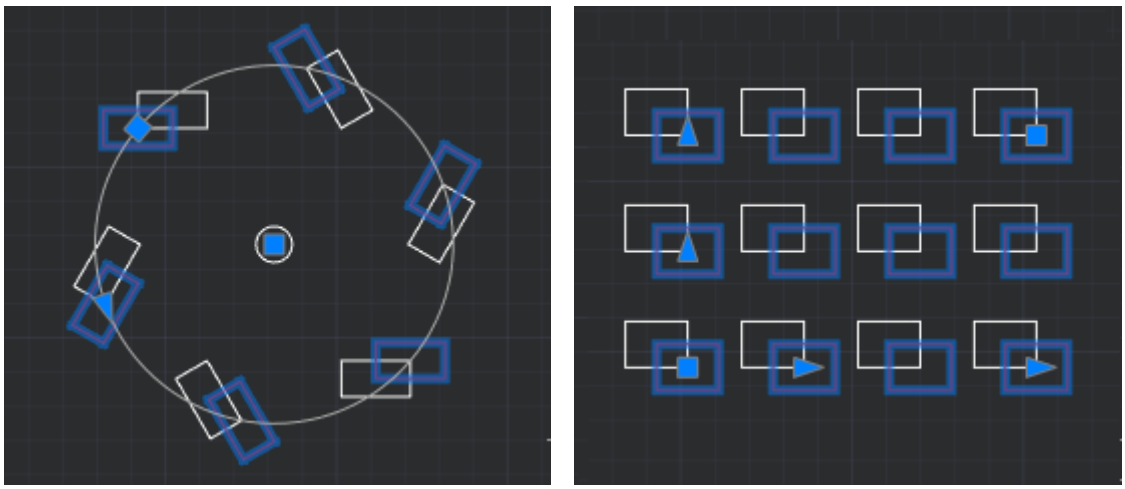


The **Editing an Array** (ARRAYEDIT) command allows you to edit the source of an associative array in place. It became possible to modify existing associative arrays by editing their specific parameters and source objects.



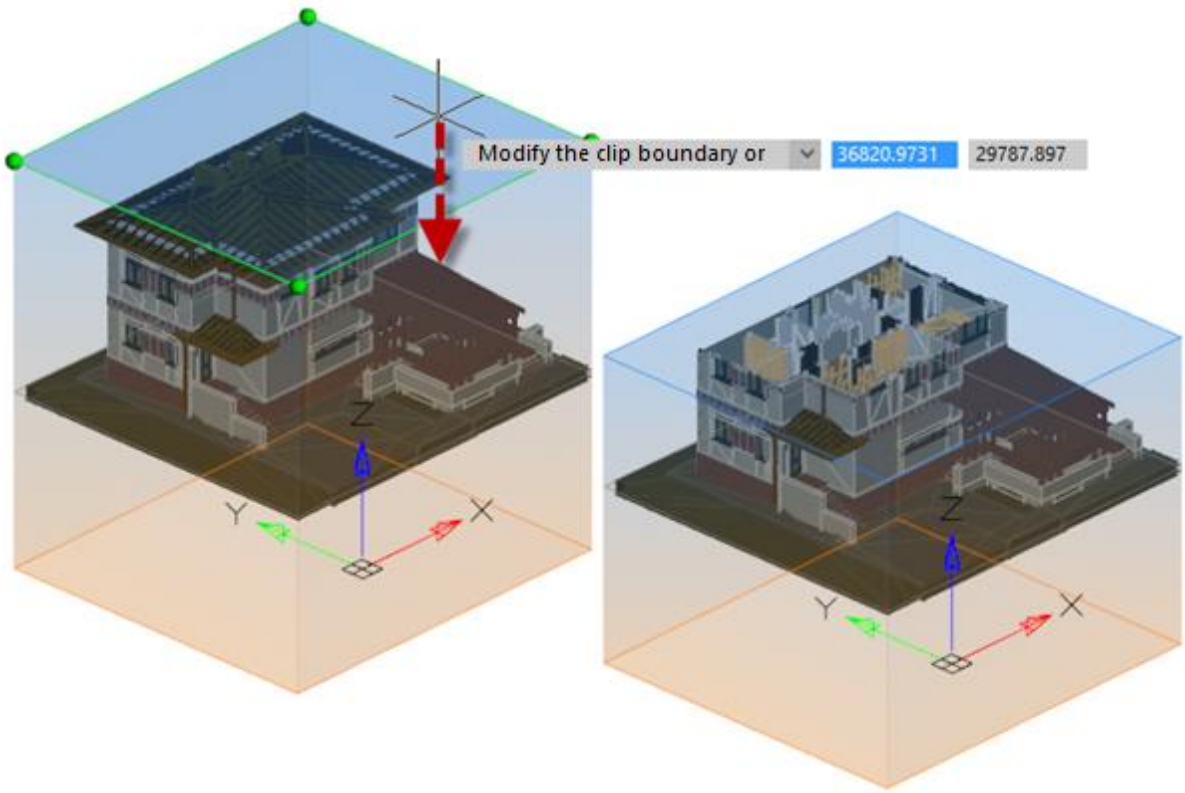


The error that occurred when changing the base point in building an associative array has been fixed. As a result, the array elements were shifted relative to the specified point instead of the base point.



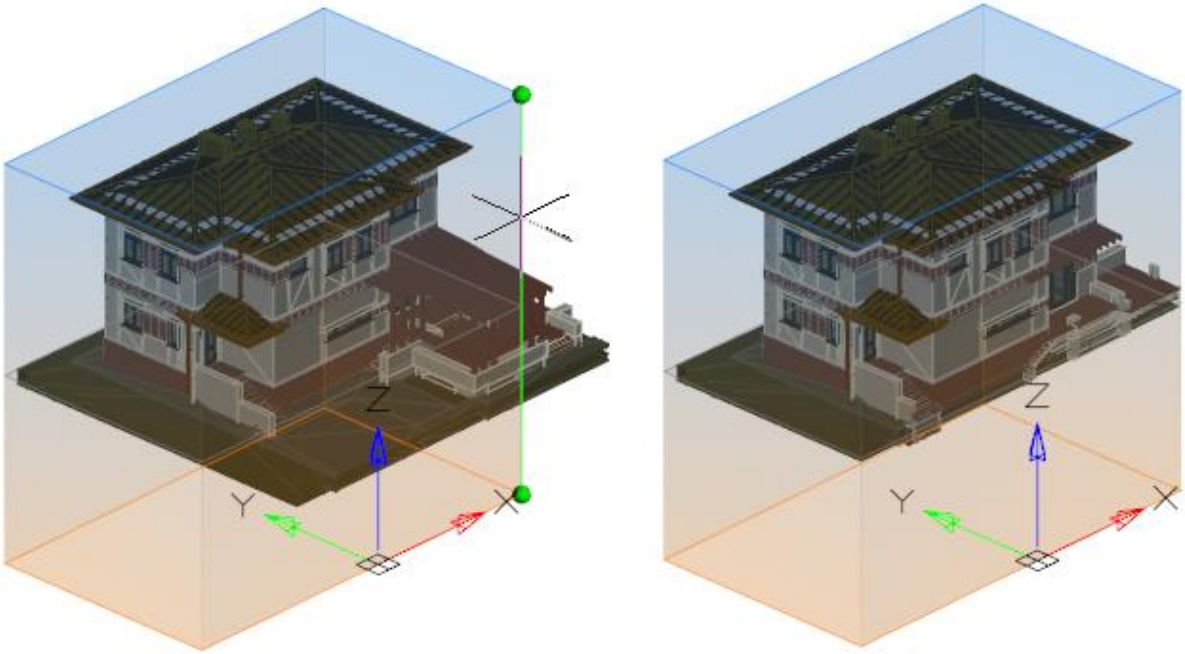
Bounding Prism

The mechanism for creating and managing a view bounding prism (MCLIP) has been completely redesigned and optimized. Prism usage has become easy and intuitive.

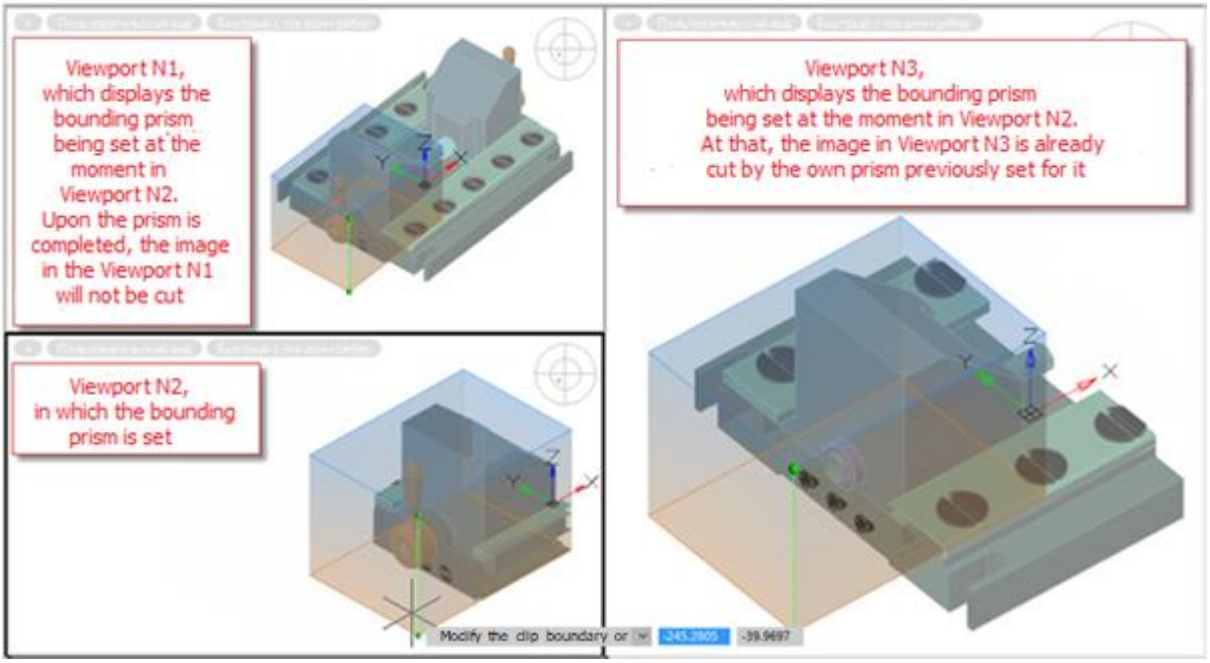


The prism can be stretched, compressed, and modified by dragging any edges and faces in any place. In this case, it changes its appearance in accordance with the user's expectations.

Thus, you can grab any area of the visible face and move it along its normal (in the case of a rectangular prism) or in any direction (in the case of a polygonal one). Snaps and orthogonal drawing modes can be used. Hidden faces can be moved by grabbing one of their edges. The entire prism can be dragged while holding CTRL.

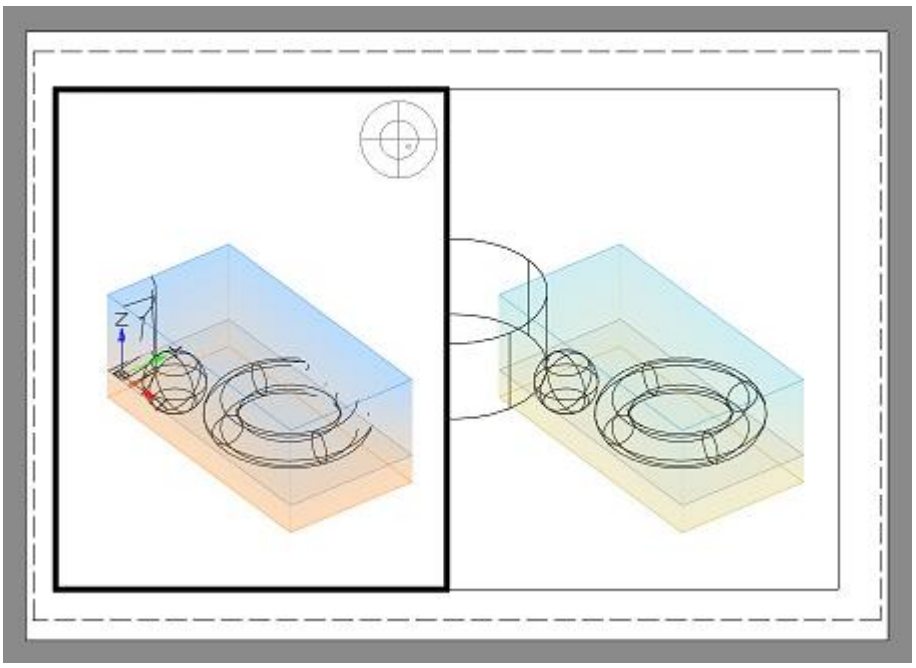


At the moment of creating or editing a prism, it is displayed in all viewports, however, its effect extends only to the screen in which it was created or was selected for editing. When the prism is set up, it will crop the image only in the viewport for which it was created.



To create a new or edit an existing prism, go to the desired viewport and run the **Bounding Prism** command.

While editing a prism, you can switch to another viewport to continue editing it from a more convenient angle. In this case, in the new viewport, editing will continue on the prism that belongs to the previous viewport (in which the **Bounding Prism** command was run), and not to the current one. A prism in its own viewport is displayed brighter than in others, so that it is always clear which viewport it belongs to, no matter which viewport is currently active.



Drawing Explorer

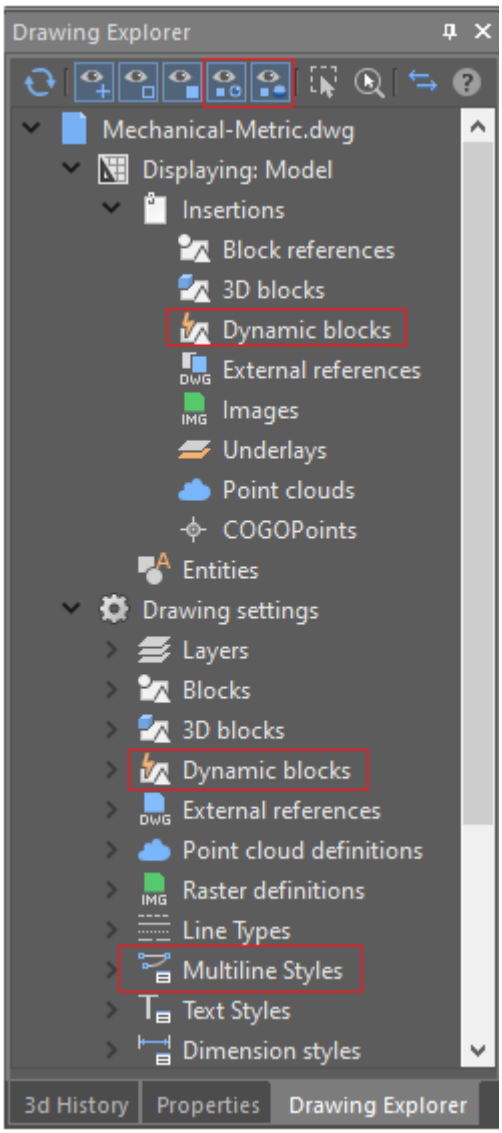
The **Drawing Explorer** bar has been updated

The following filters have been added:

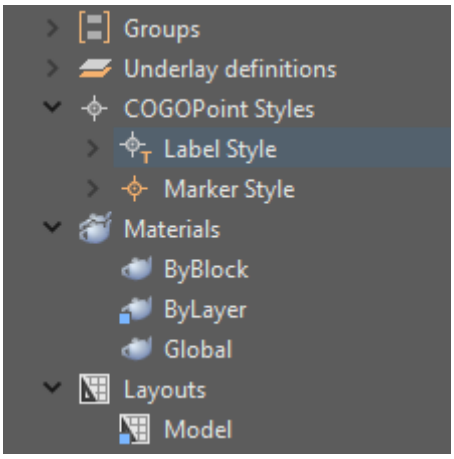
- **Show visible objects**;
- **Show invisible objects**.

Standard blocks, 3D blocks and dynamic blocks are now displayed separately.

Multiline styles are displayed.



Such Topoplan objects appeared as groups of points of coordinate geometry, as well as sets of styles of point markers and styles of their labels.



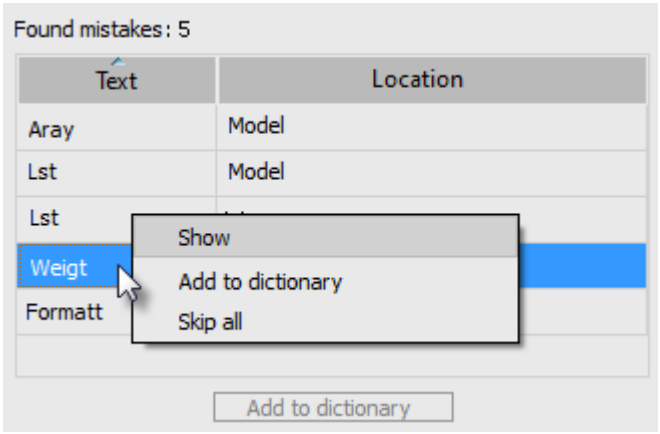
The styles of labels and markers of coordinate geometry are edited by selecting the desired style in the drawing explorer and then calling the command from the context menu. The properties of the selected style are displayed and changed in the **Properties** bar.

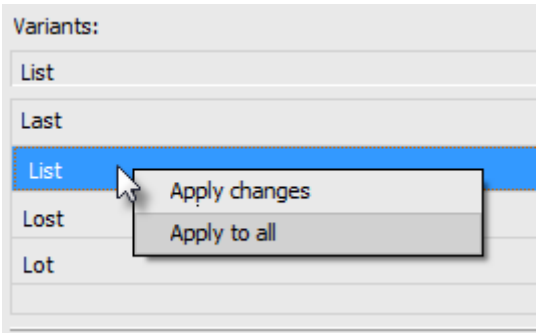
Spell Check Bar

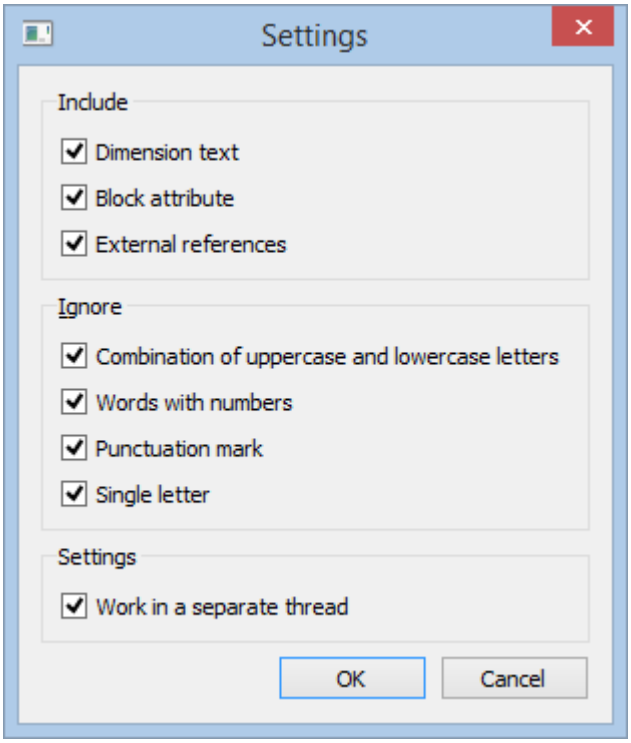
The updated **Spell Check** bar, in addition to improving ergonomics, provides a new functionality.

Now it is possible to check text objects of a drawing for spelling mistakes not only on demand, but also in real time. All new and modified objects will be checked for spelling. If mistakes are found, the words will be underlined in the drawing field and added to the list of found mistakes in the bar.

In this case the bar allows you to work with several documents at the same time without losing the results of spell check of the previous document when switching to the next document.







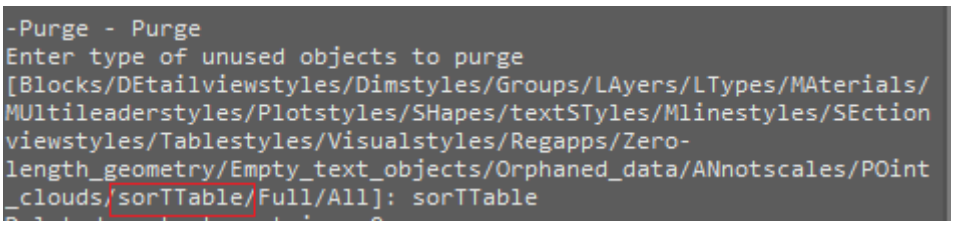
In the spell check settings you can:

- Add or exclude particular types of objects from the check.
- Exclude certain types of text from the check.
- Instruct the spell check command to run on a separate thread to process information independent of the platform thread.

Purging a Sort Table

Now the version of the **Purge Document** command to work from the command line (-PURGE) allows you to clear the object sorting table. The sort table contains lists of object order (DRAWORDER).

Sometimes there are so many such records that the program starts working with the document much more slowly. Cleaning the table removes empty records, greatly speeding up the work.



Prompt on the Need to Purge a Document

Now, when opening documents, the need to purge annotative scales and empty entries in the object sorting table is checked. Depending on the check result, the following prompts may be displayed in the command line:

- Attention: the file contains a large number of annotative scales (number). It is recommended to remove the scales by running the -PURGE command and selecting the ANnotscales option.
- Attention: the file contains a large number of empty entries in the sort table (number). It is recommended to remove them by running the -PURGE command and selecting the sorTTable option.

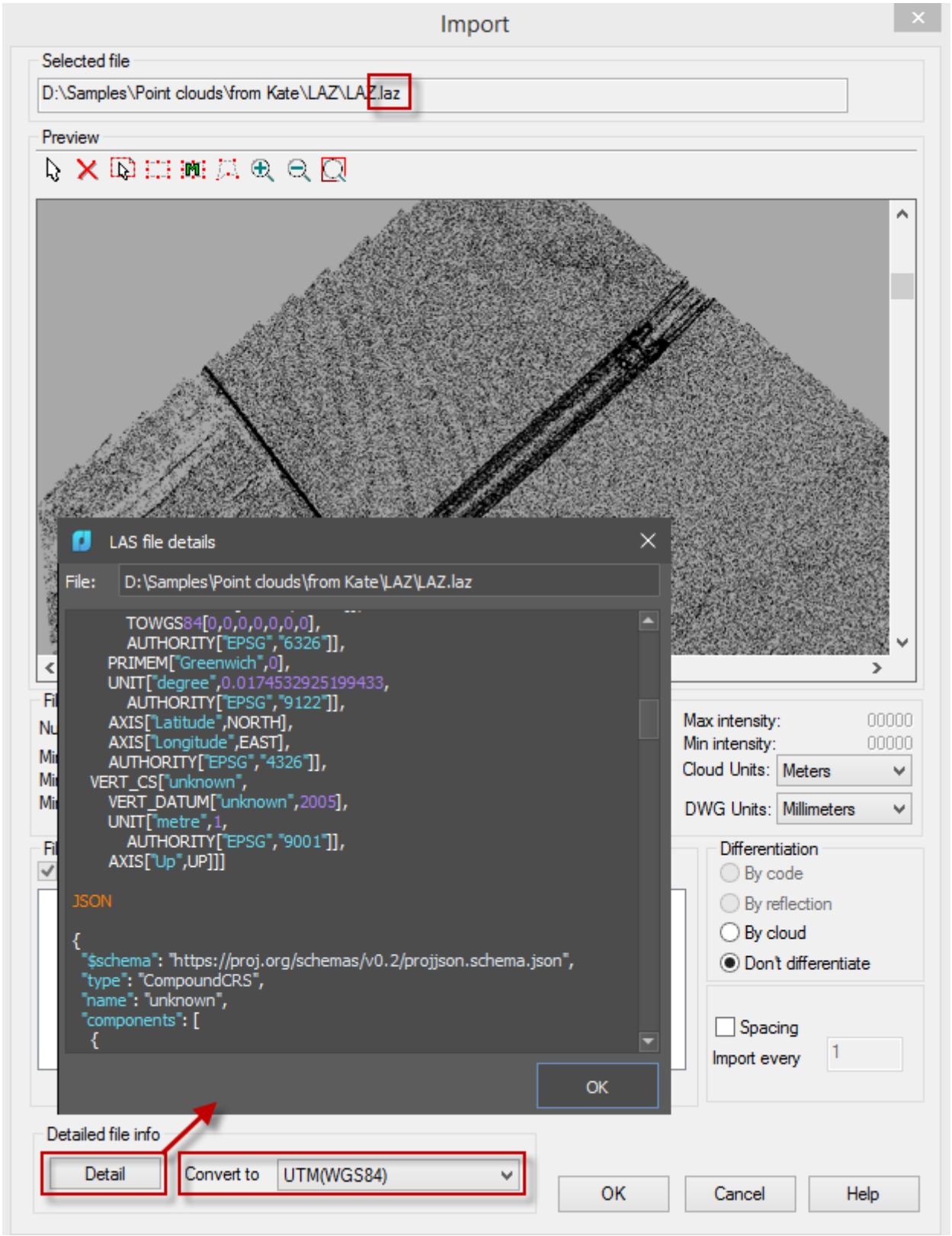
Acceleration of Opening Files

To speed up the opening of files, multi-threaded loading of DWG files has been implemented. The multithreading mode is controlled by the MTMODE system variable. A program interface has been opened that allows application developers to use multithreading to load objects, which gives a significant gain, including when opening files with a large number of specialized objects.

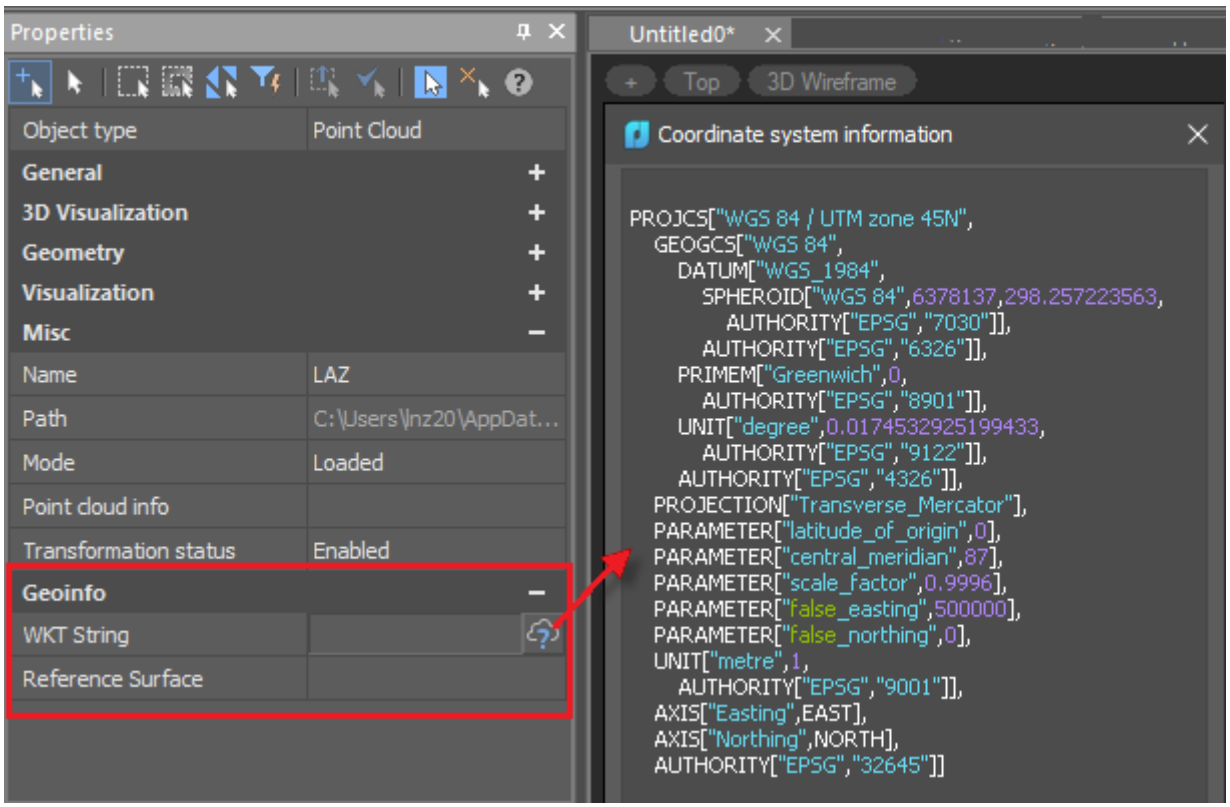
Import of Point Clouds

Import Point Clouds (NPC_IMPORT)

- The import of clouds in the LAZ format (LAS zipped format) has appeared.
- Recalculation of coordinates during import was supplemented by GCS 2011 geosystem.
- A new dialog has appeared that shows extended information about the parameters of the imported file.



- The **Point Cloud** object received two geoparameters. When recalculated, the cloud is marked with the geo-coordinate system to which it was recalculated.



- Support for EPSG geotagging.
- Unlimited set of coordinate geosystems (the list can be supplemented with your own CS). Possibility of more complete control of recalculation parameters: to which CS and with what parameters to recalculate.
- Improved import of point clouds from PTS format. It became possible to import such clouds with intensity in the ranges [-2048..2048], [0..255] and [0..65535], and not only [-4096..4096] as it was before.
- Import of normals from PTS format has been fixed.

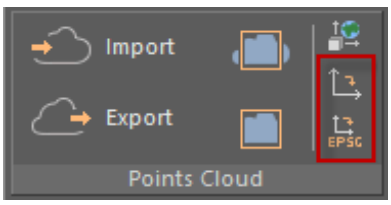
Export of Point Clouds

Export Point Clouds (NPC_EXPORT)

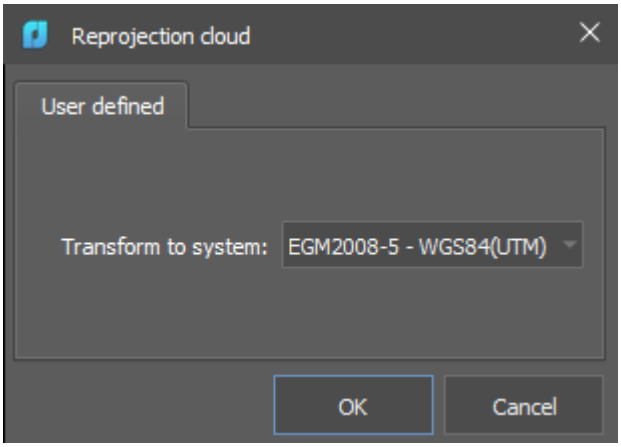
The PTS format has been added to the export command.

Commands for Recalculating Cloud Coordinates

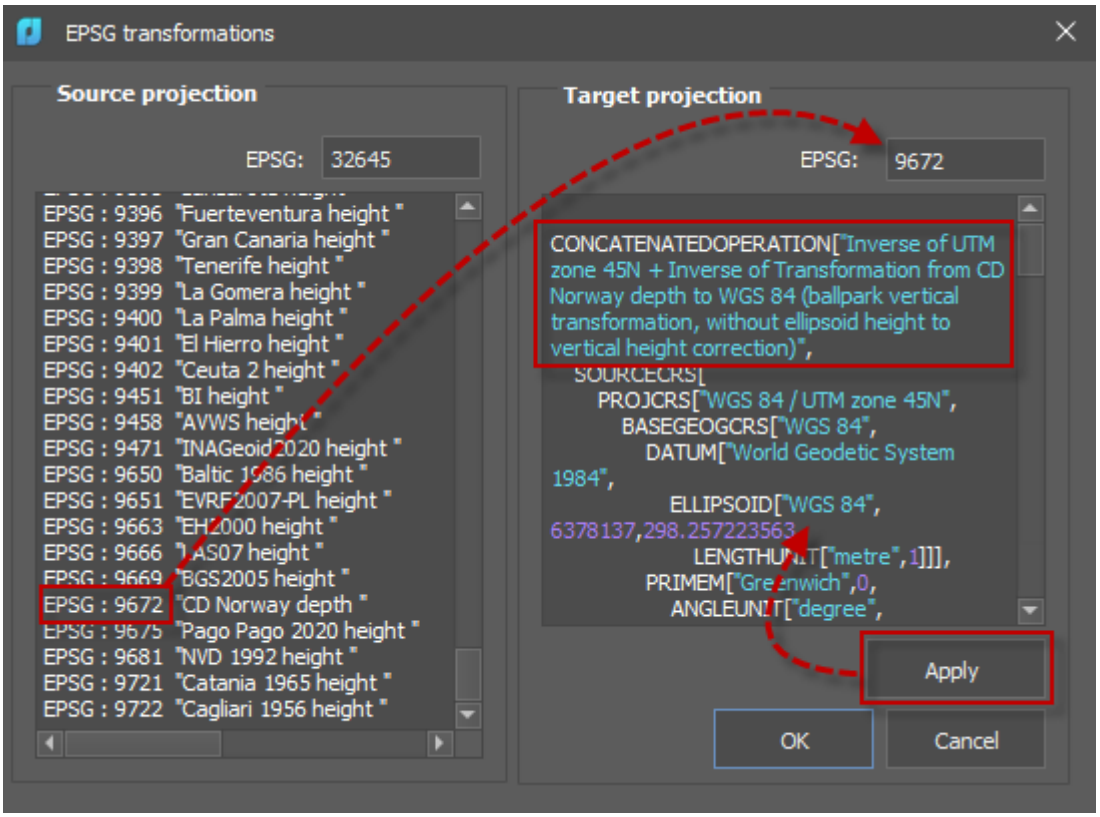
New coordinate recalculation commands have appeared that allow you to recalculate the coordinates of already loaded point clouds. This creates a new cloud for the desired coordinate system.



- NPC_REPROJECTION – recalculation of coordinates of a point cloud with settings from GCSTrans.xml file;



- NPC_EPSG_REPROJECTION – recalculation of a point cloud coordinates by EPSG codes.

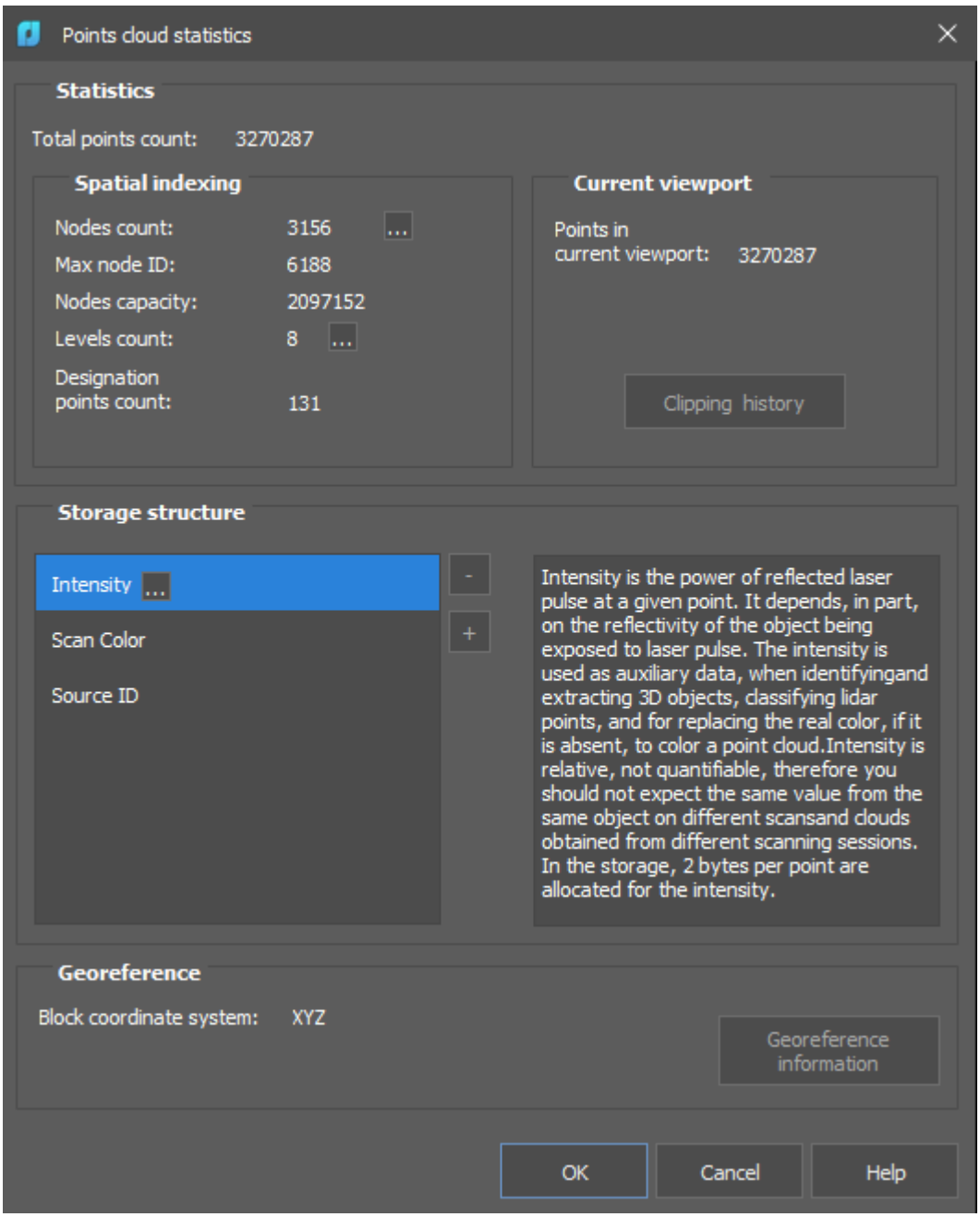


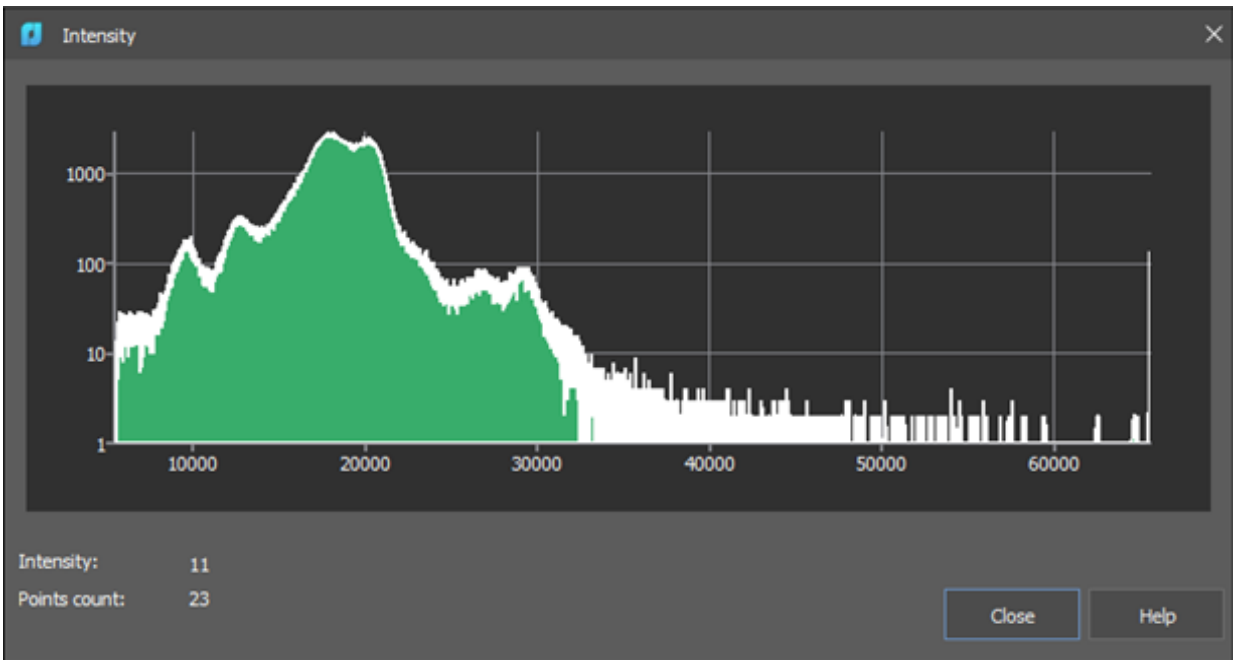
These commands are mutually exclusive: you can use either one or the other.

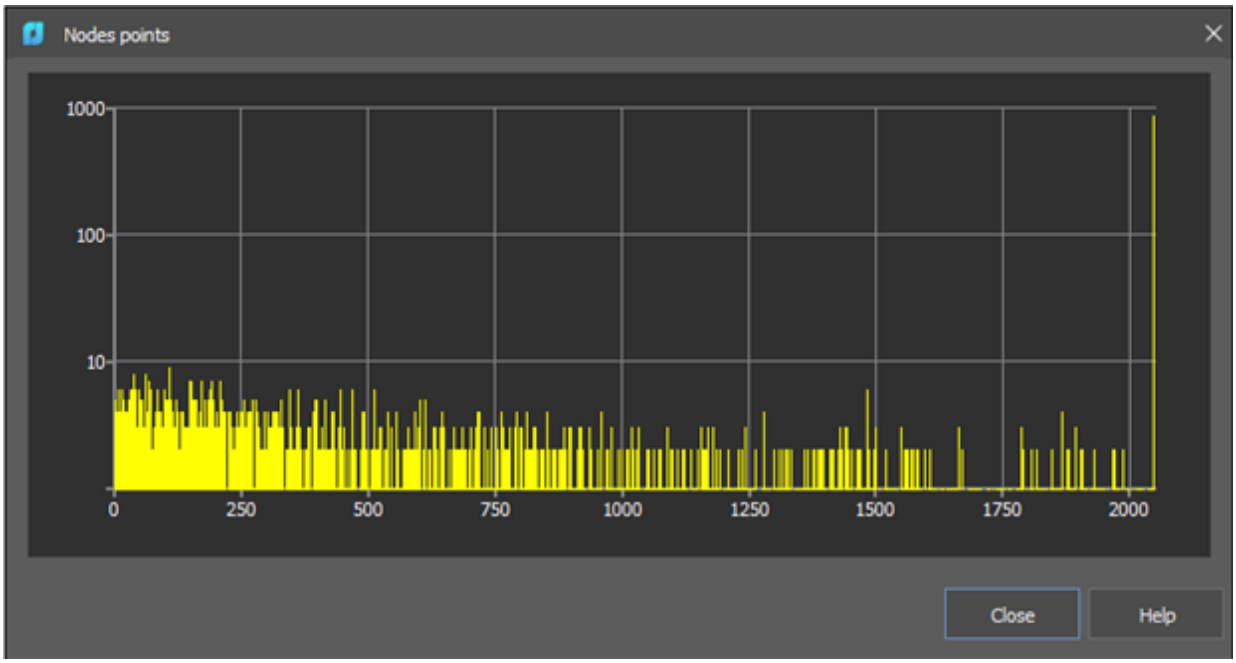
Point Cloud Information

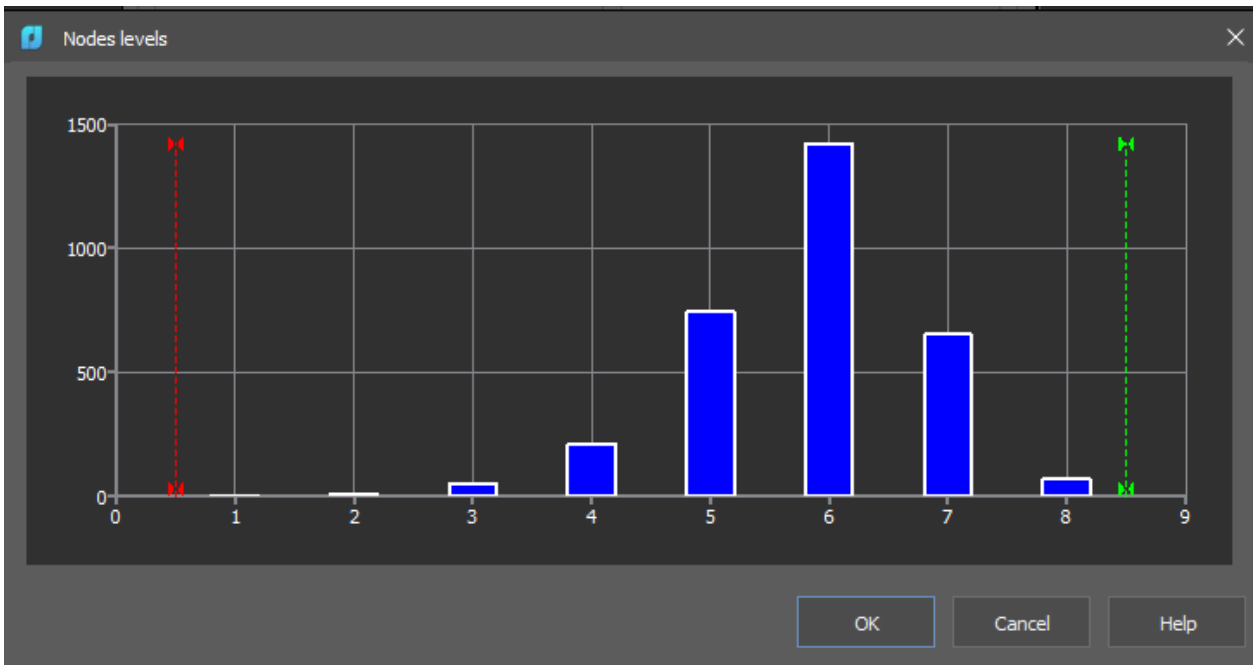
Point Cloud Information (NPC_INFO)

The dialog interface has been completely redesigned. Individual interactive charts are opened by clicking the ... button next to the corresponding parameter.


Points cloud statistics
Statistics
Total points count: 3270287
Spatial indexing
Nodes count: 3156
Max node ID: 6188
Nodes capacity: 2097152
Levels count: 8
Designation points count: 131
Current viewport
Points in current viewport: 3270287
Clipping history
Storage structure
Intensity
Scan Color
Source ID
Intensity is the power of reflected laser pulse at a given point. It depends, in part, on the reflectivity of the object being exposed to laser pulse. The intensity is used as auxiliary data, when identifyingand extracting 3D objects, classifying lidar points, and for replacing the real color, if it is absent, to color a point cloud.Intensity is relative, not quantifiable, therefore you should not expect the same value from the same object on different scansand clouds obtained from different scanning sessions. In the storage, 2 bytes per point are allocated for the intensity.
Georeference
Block coordinate system: XYZ
Georeference information
OK
Cancel
Help

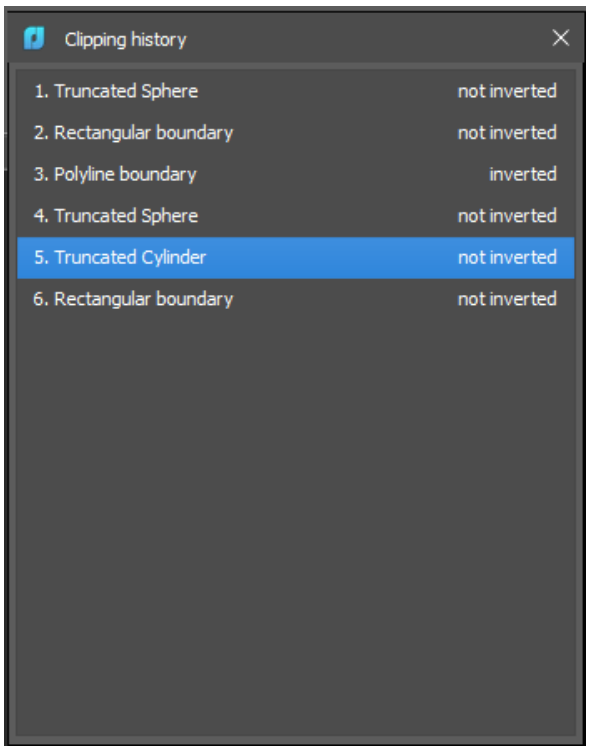

Intensity
1000
100
10
1
10000
20000
30000
40000
50000
60000
Intensity: 11
Points count: 23
Close
Help


Nodes points
1000
100
10
0
250
500
750
1000
1250
1500
1750
2000
Close
Help



New features have appeared:

- Clipping history. If the clip and section commands were applied to the point cloud, you can see the full list.
- Information about the geosystem coordinates of the point cloud. If the cloud coordinates were recalculated during import, it is possible to view its parameters.



The clipping history displays all cloud clipping manipulations with their attributes. The initial state of the cloud is displayed at the top of the list. When you select a clipping state from the list, all subsequent clippings become inactive for the point cloud.

API for Point Clouds

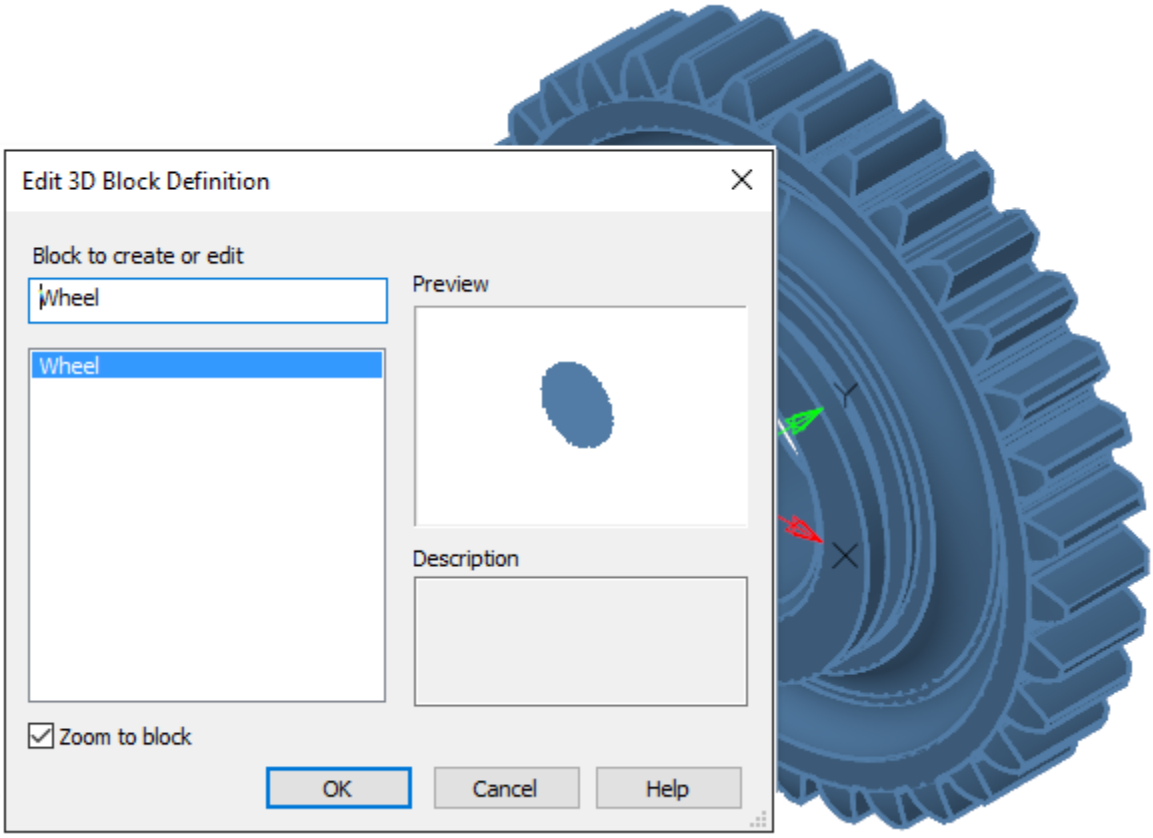
An open C++ API for working with point clouds has been implemented. An example is available that demonstrates the following steps:

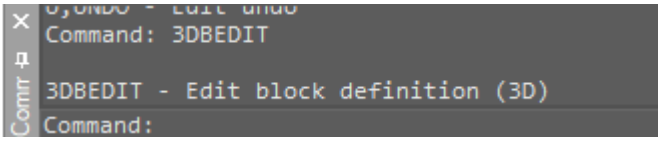
- Loading a cloud from a file;
- Replacing the standard coloring of the cloud with the coloring by height;
- Cropping the cloud by clip overlay;
- Exporting the cloud to another format.

New in the 3D Module

3D Block Editor

Blocks containing solid objects (3D solids) or parametric bodies is now edited in a separate editor, which is called by double-click on such a block with the 3DBEDIT command.





The **3D Block Editor** context tab appears on the ribbon with additional tools for 3D editing, buttons for saving the result and exiting the editing mode. The tab lacks a number of commands that are present in the standard block editor, for example, for working with attributes. However, these commands can still be called from other ribbon tabs, menu, or command line.

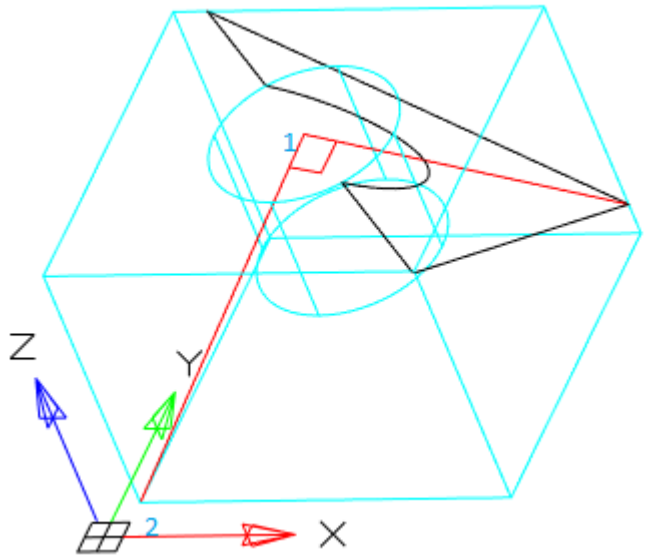
If there is no license for the 3D module, editing 3D blocks using standard methods will still be available, but without the ability to use 3D editing commands.

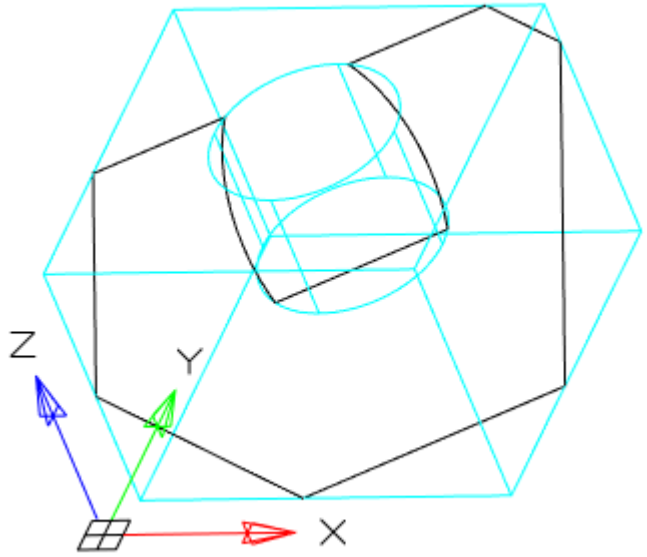
In the drawing explorer, 3D blocks have been moved to a separate section.

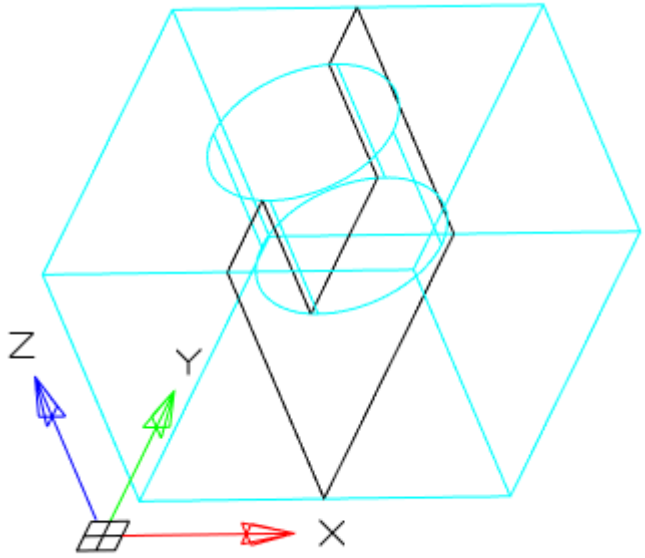
The 3DREFEDIT command is used to edit references.

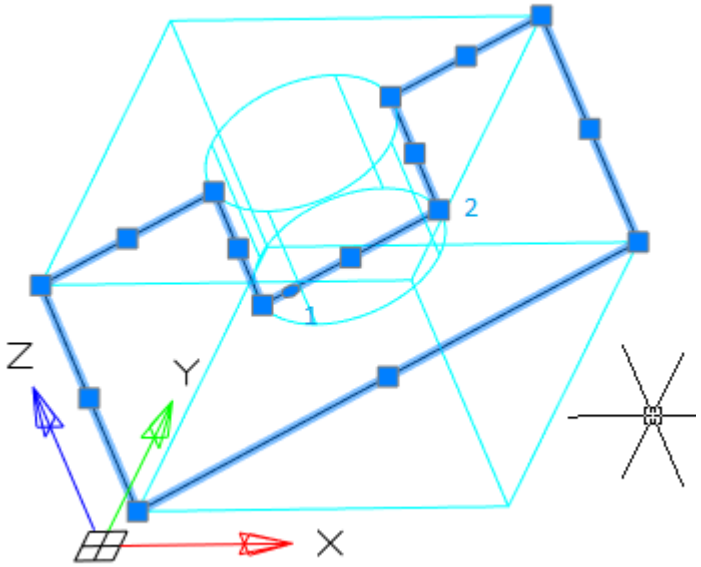
Section of 3D Objects

A new **Section** command of 3D-module in the **3D Tools** ribbon tab of the **Direct modeling** section creates and object of **Region** type that represents a 2D cross section of 3D objects, including 3D solids, surfaces and meshes.


1
Z
Y
2
X

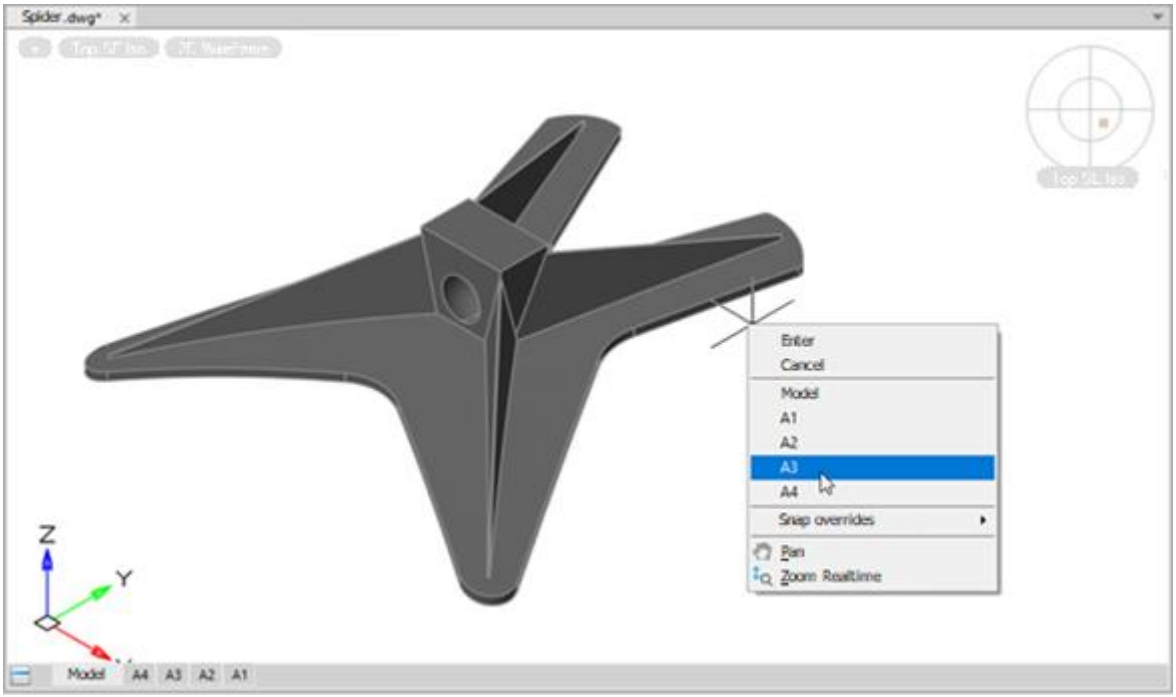

Z
Y
X

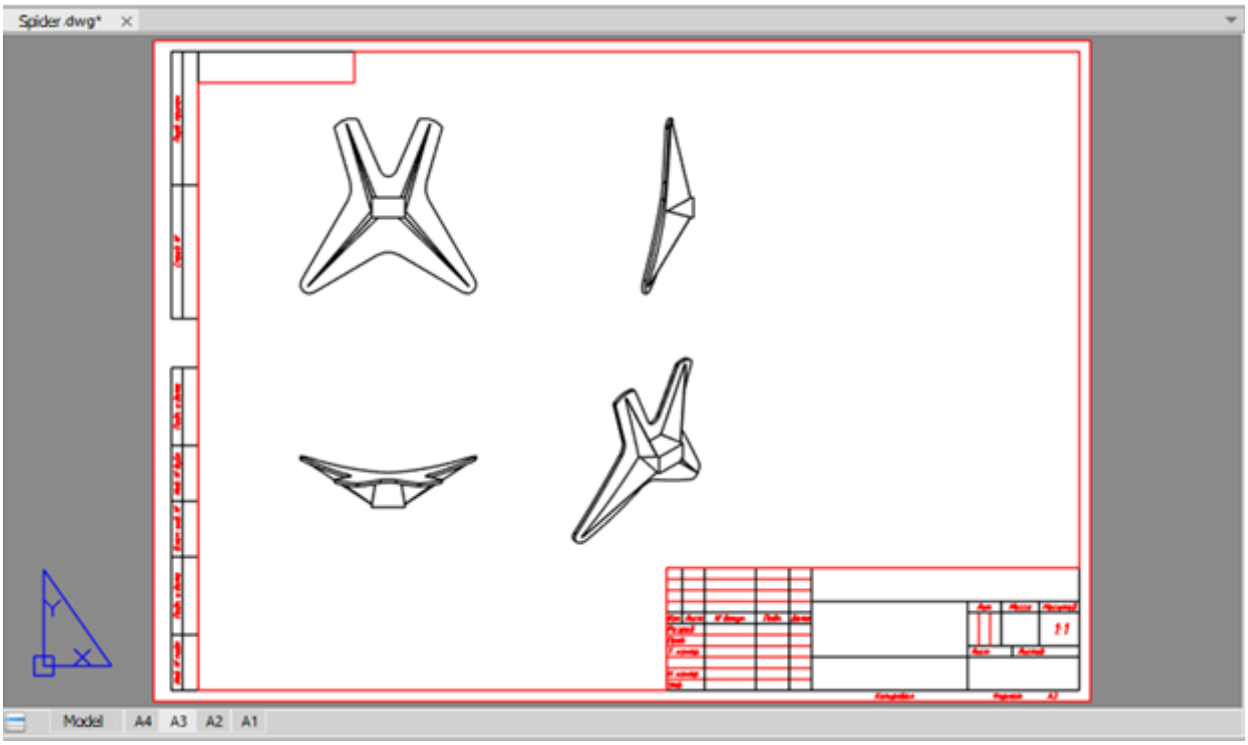

Z
Y
X



Direct Insertion of 2D Views into Layouts

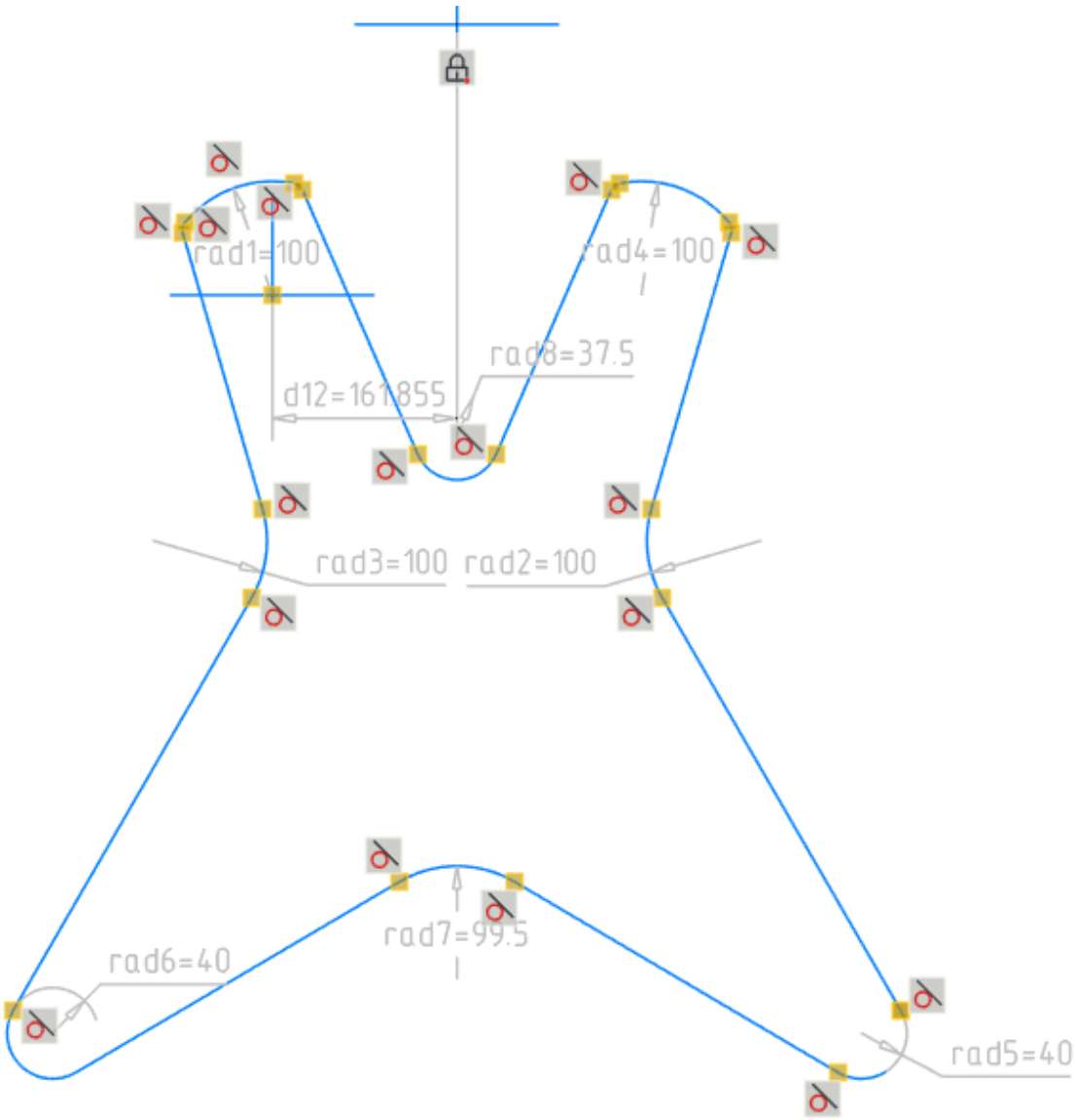
Previously, in order to create a drawing in a layout, it was necessary to create projections of 3D objects in model space and then insert them into layouts using viewports. An additional option has been implemented in the nanoCAD 22, using which you can create 2D views directly in layouts without duplicating them in the model space.





Parametric Mode in Sketch

The mode of auto-applying constraints is implemented in parametric modeling, when building flat sketches of future models. Now you don't have to waste time creating geometric constraints - the system independently parameterizes the object.

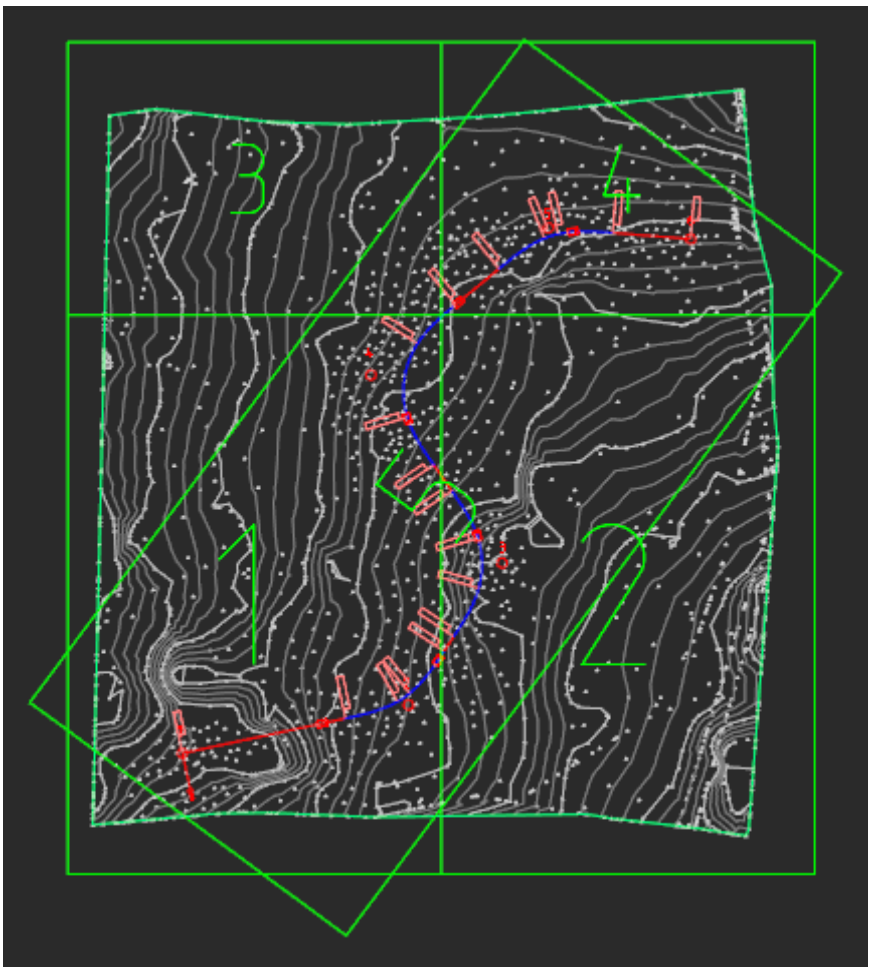


New in the Topoplan Module

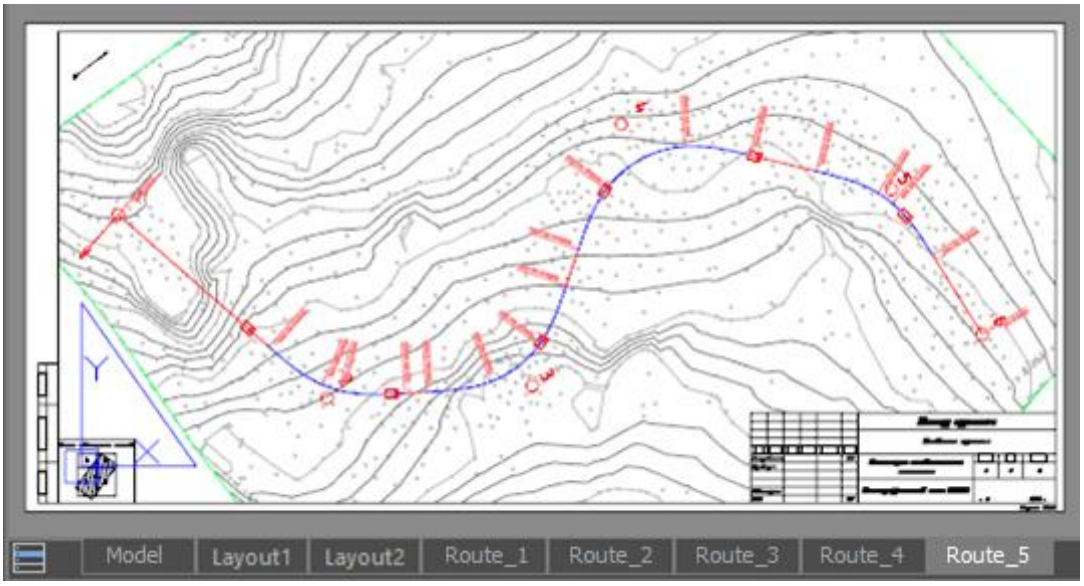
Sheet Design

The ability to split a drawing into prepared sheets for subsequent printing has been implemented. The **Creating Frames** command allows you to place view frames of future sheets on the drawing. The **Creating Layouts by Frames** command creates layouts based on frames with the desired design and scale.

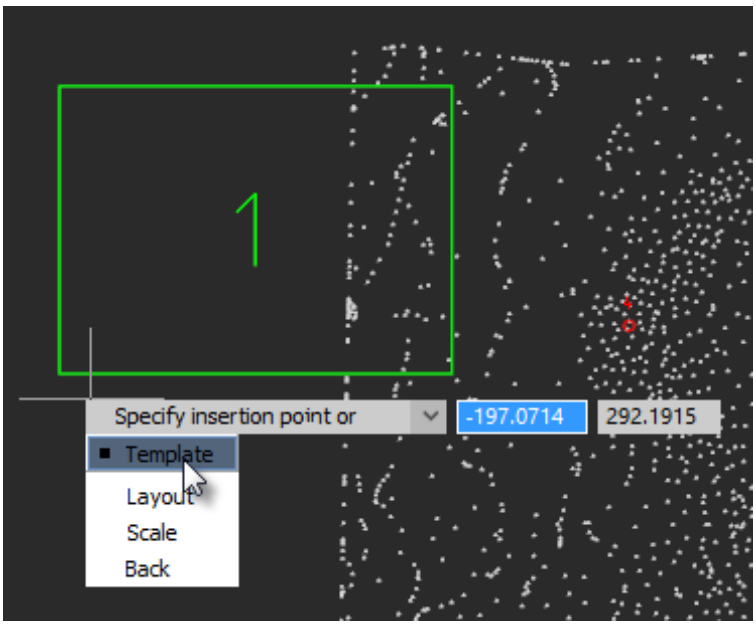
Creating Frames (CREATEVIEWFRAME). The command allows you to create frames in the model space for the subsequent formation of designed sheets. Each frame is a viewport of such a sheet.



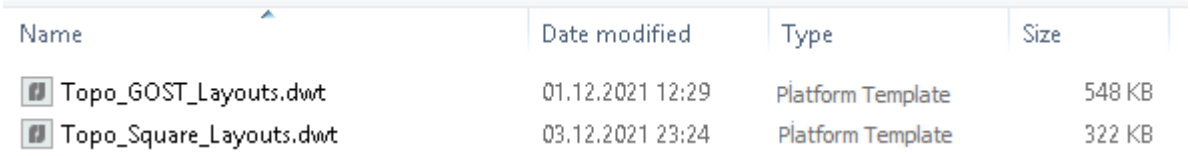
Each frame carries information about the template, format and scale of the sheet. The design of the future sheet will be taken from the template in accordance with the format.



After the **Creating Frames** command starts, you can configure the options of future sheet for the frame:



Template – a template file containing a list of available formats with frame design. You can specify a template for formats according to standards or for a topographic tablet.



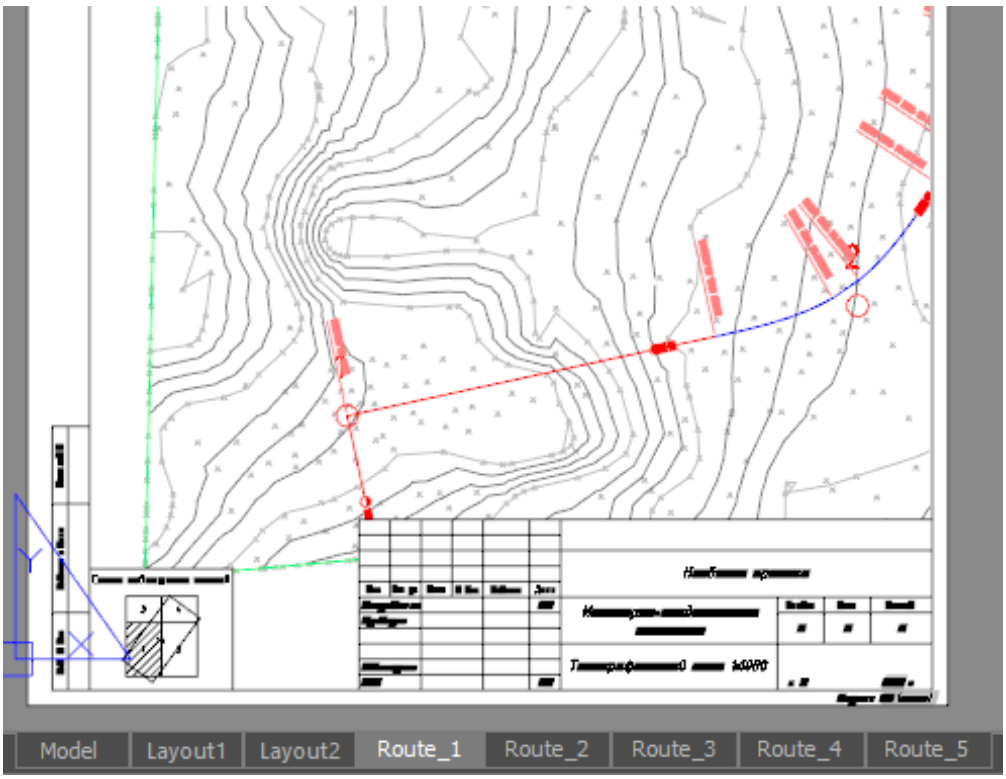
After selecting a template, specify the sheet size for the inserted frame.

Layout – Allows you to assign a layout format without reassigning the template.

Scale – selection of the topographic scale. By default, the scale of the created sheet is set according to the current TOPOSCALE:

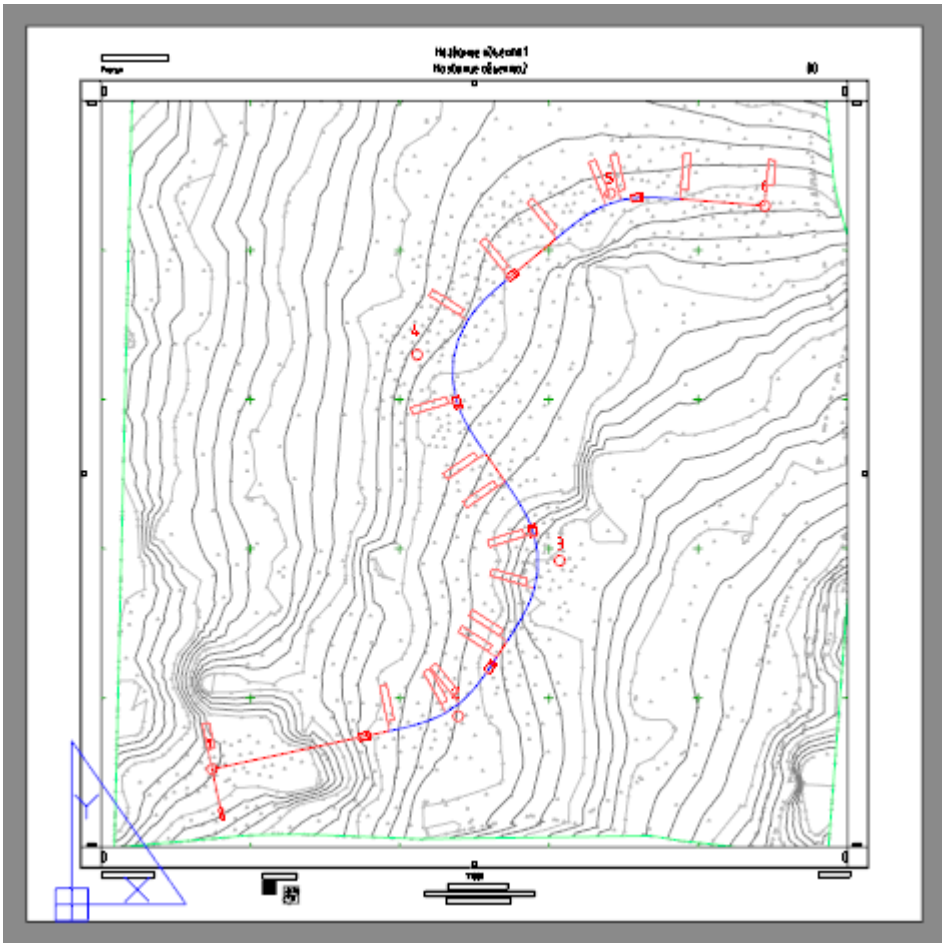
After setting all the parameters, specify the position of the view frame in the drawing. In this case, you can use the nanoCAD precise positioning tools to snap to axial, auxiliary lines, mesh, points, or existing view frames. You can rotate frames and use other standard mechanisms for placing objects in the drawing workspace.

Creating Layouts by Frames (CREATEVIEWFRAMELAYOUT). The command creates layouts for designing a drawing according to previously created frames. Each frame has its own sheet, with the same design, format and scale that were specified when creating the frame.

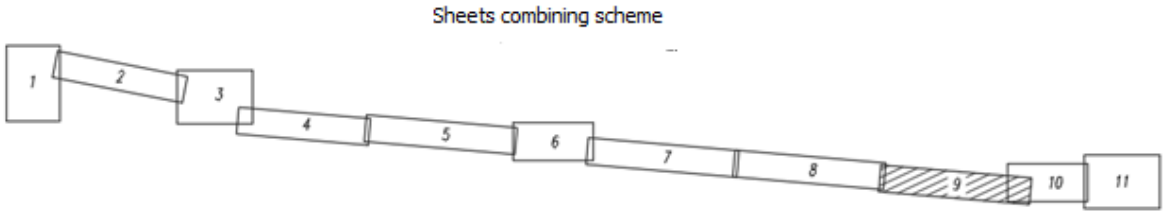


The sheet's frame and stamps are a dynamic block with parameters and attributes that can be edited in the **Properties** bar or using grips.

It is possible to design in form of GOST and topographic tablets.



For tablets, the **Survey plot scheme**, or the **Sheets combining scheme** are created

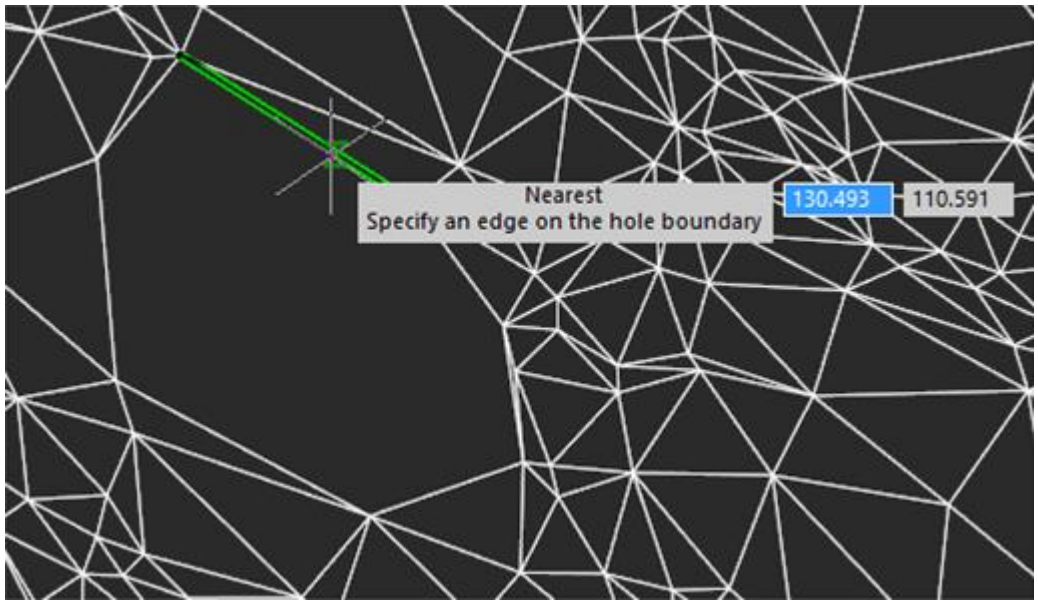


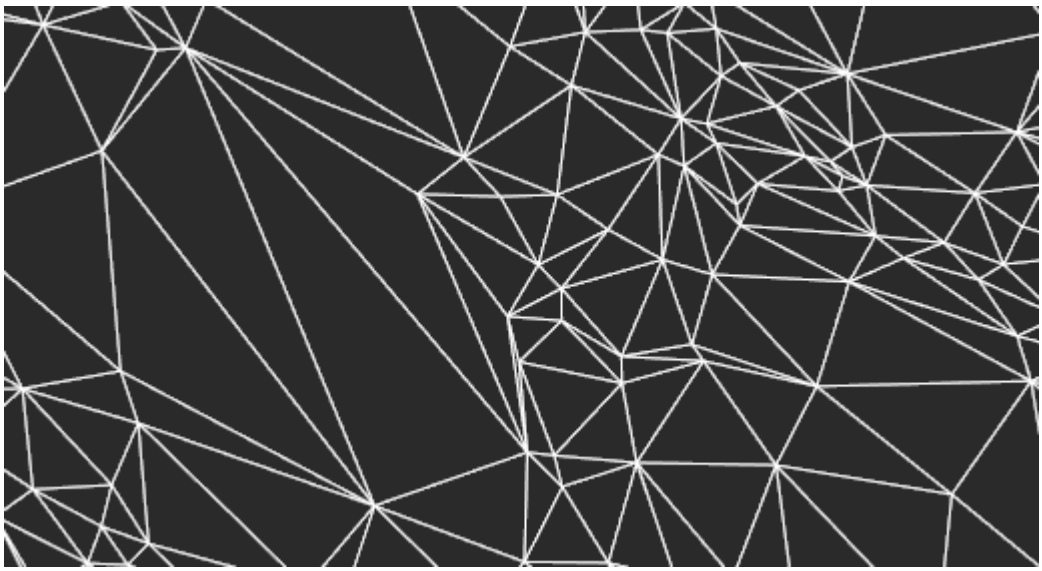
Hide Frames (HIDEVIEWFRAMES). Frame hiding mode. Turns off the display of all created frames in the model space. Repeated execution of the command returns the display of frames.

Delete Frames (DELETEVIEWFRAMES). Deletes all frames created by the **Creating Layouts by Frames** command (CREATEVIEWFRAME).

Mesh Seal Holes

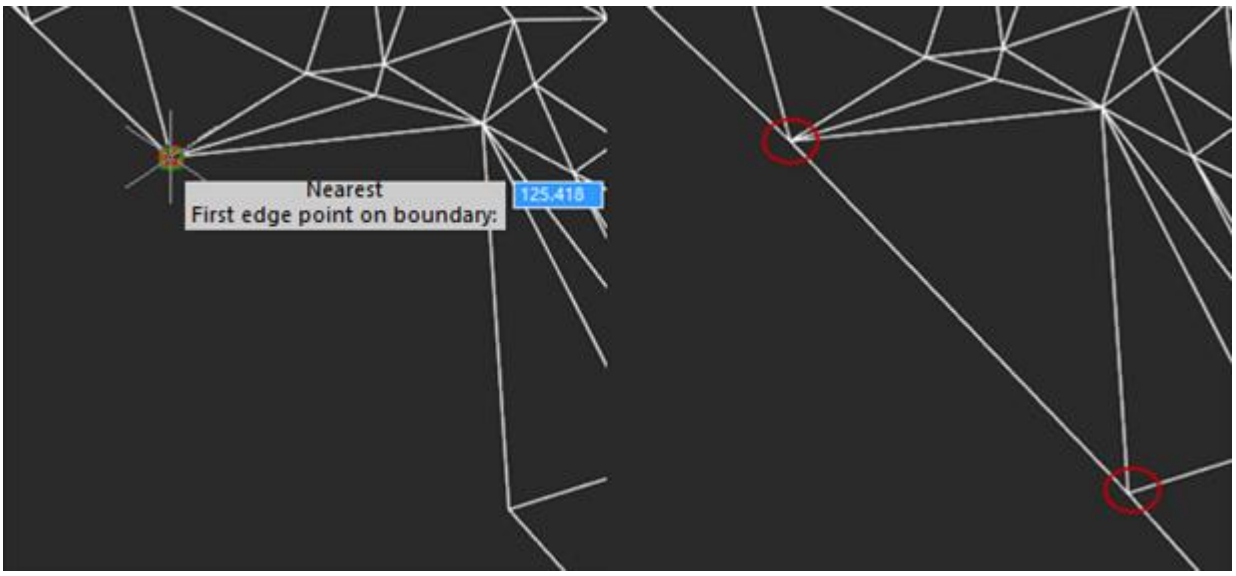
Mesh Seal Holes (NG_MESH_SEAL_HOLES). If during creating a TIN surface or during its editing, surface discontinuities have formed, the command allows you to fill them with missing faces.





Adding Mesh Edge

Add Mesh Edge (NG_MESH_EDGE_ADD). The command allows you to add edges to an existing surface at its boundary. An edge can only be added on a non-convex boundary (inner or outer) so that a new triangular face can be created without disturbing the 2.5D mesh.



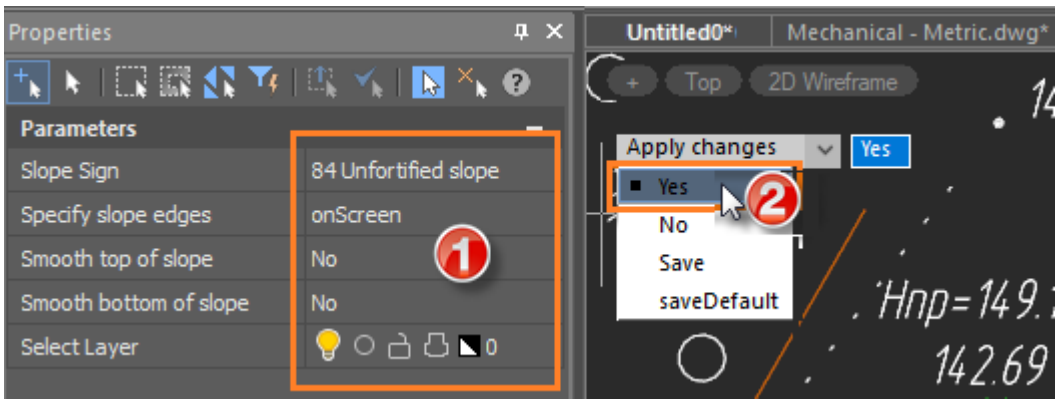
Creating a Slope

Create Slope (NG_CREATE_SLOPE). The command makes it possible to create various types of conventional signs for slopes, cliffs, retaining walls:

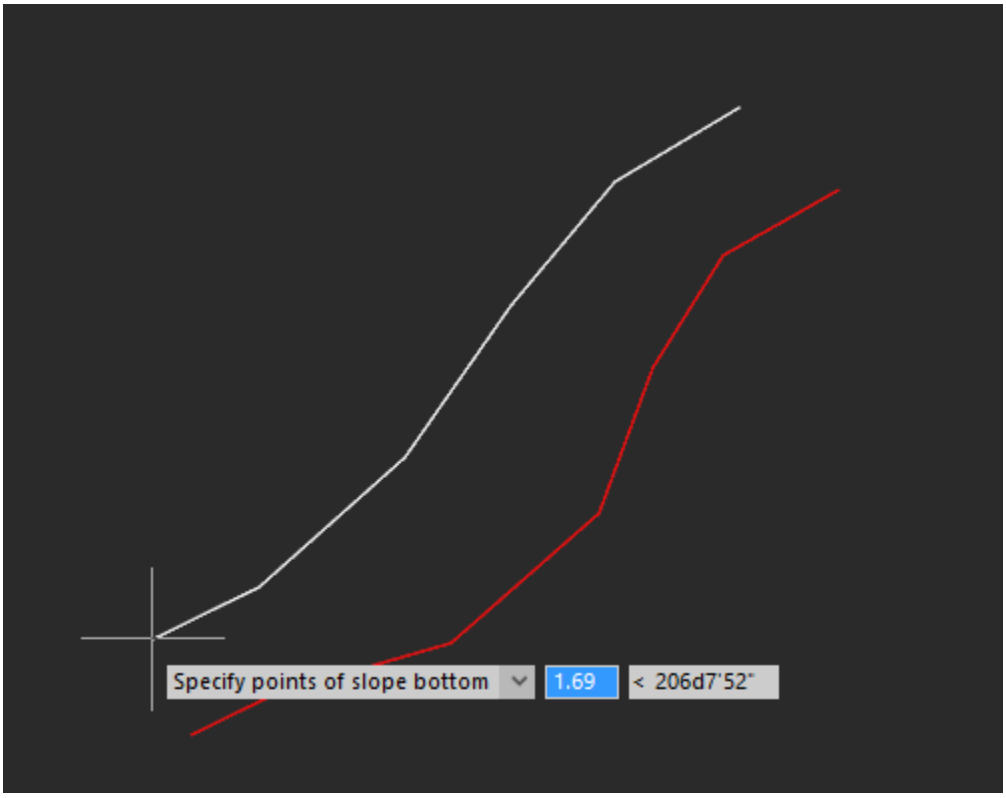
- Unfortified slope;
- Fortified slope;
- Steep coast with a beach;
- Steep coast without a beach;
- Stone retaining walls;
- Wooden retaining walls;
- Ground steep.

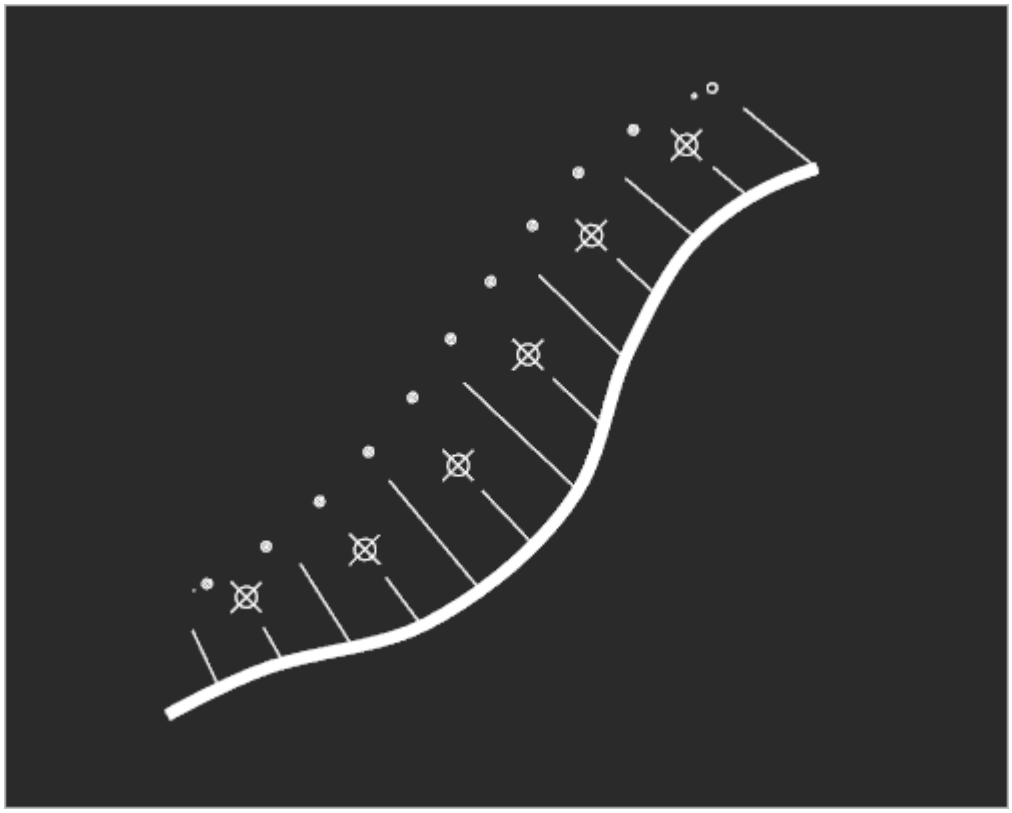
Before starting, the command checks the set topographic scale and offers to switch to the 2D-wireframe visual style.

After running the command, set the slope type and characteristics in the **Properties** bar. The options vary depending on the type of slope and how its edge is specified.

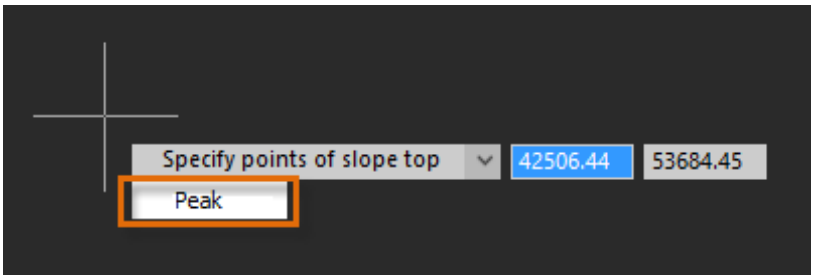


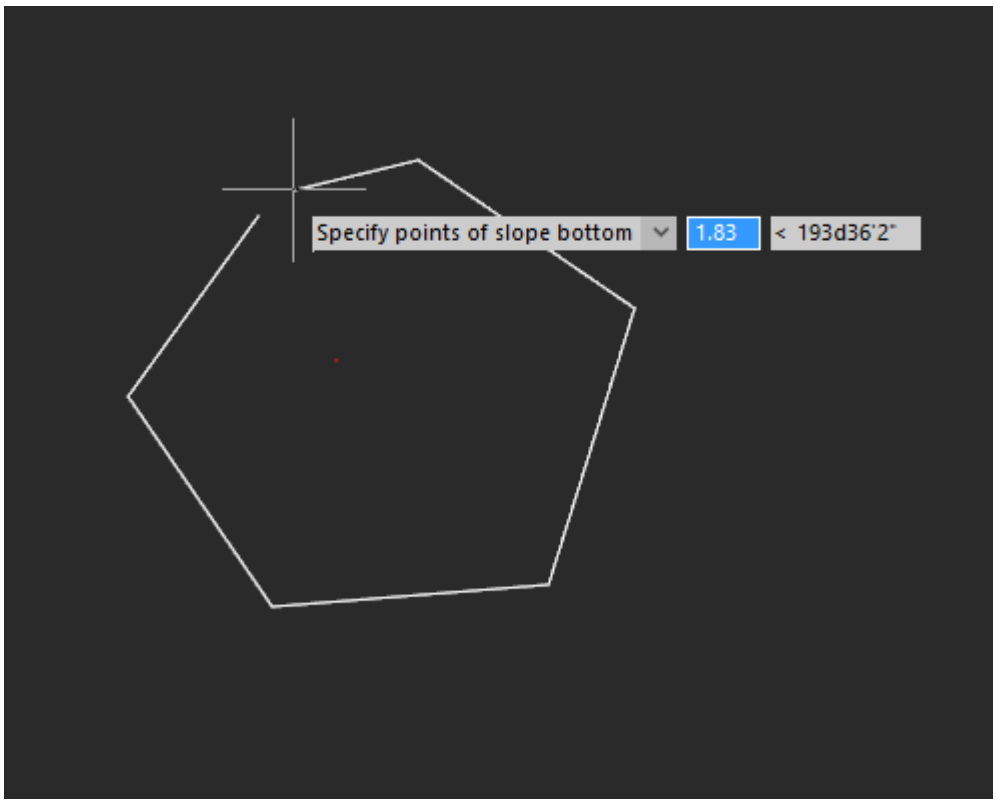
Next, indicate the location of the slope in the drawing. Slope edges can be drawn manually or created from existing drawing lines, depending on the value of the **Specify slope edges** option. The top and bottom of slope can be closed lines.

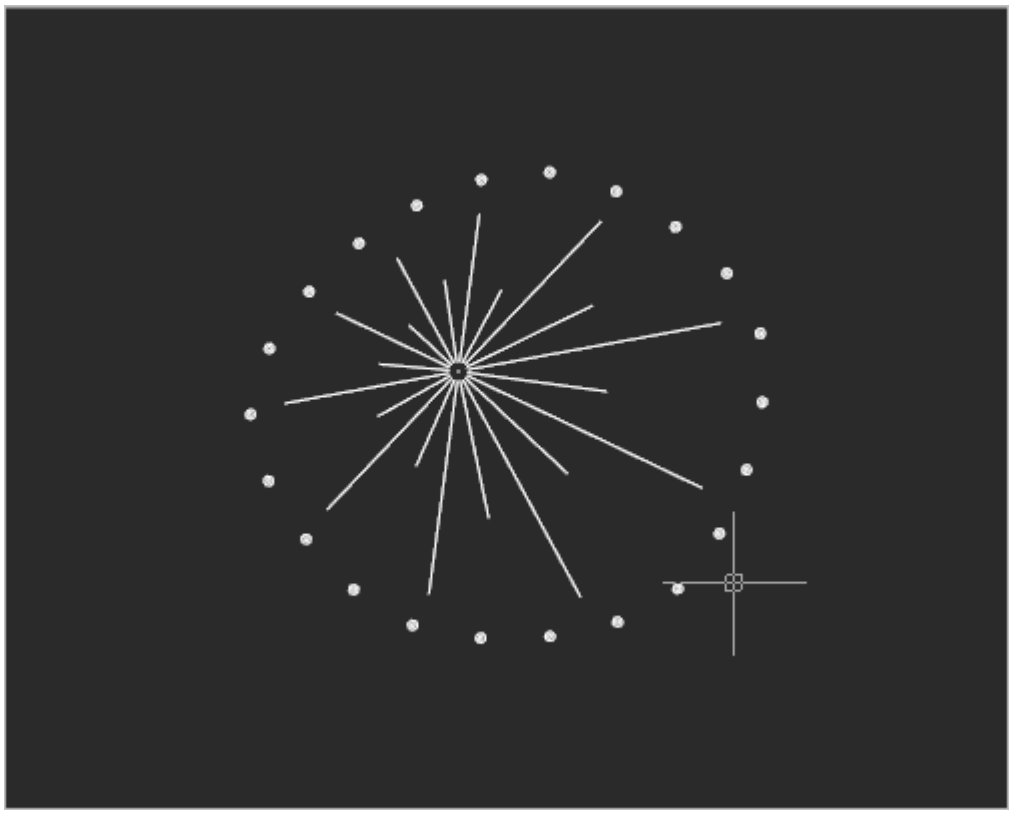




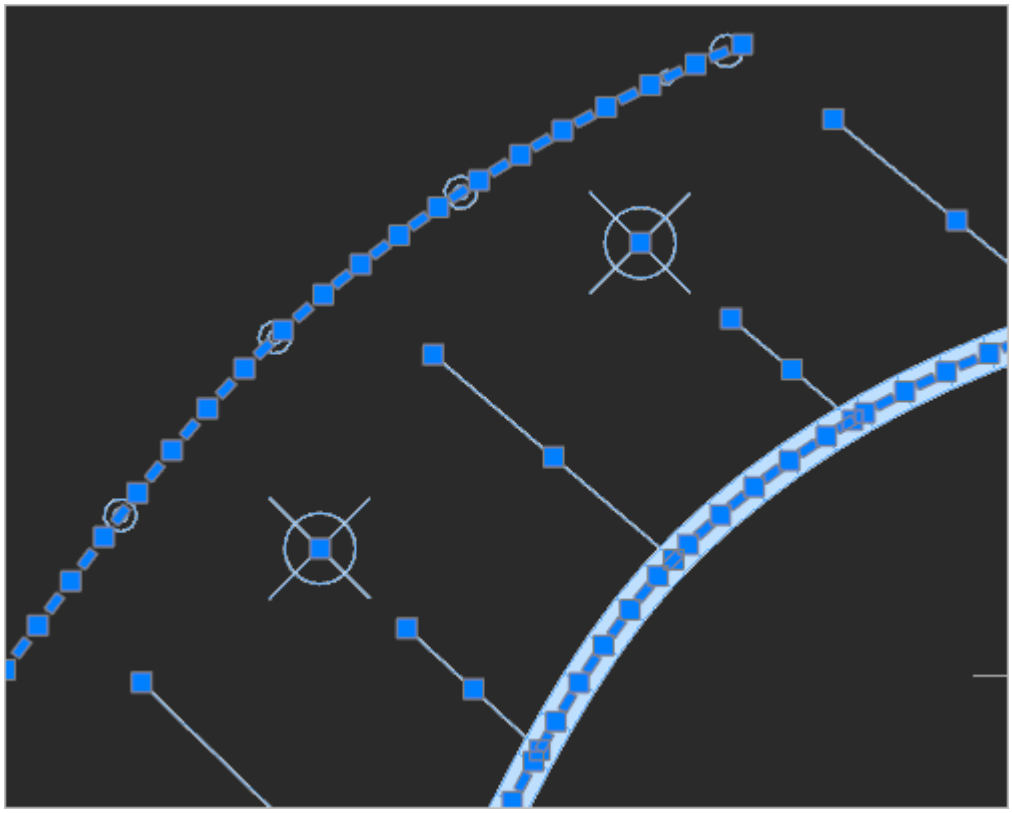
You can create a slope from one top point (hill). To do this, instead of specifying points for the slope top, select the **Peak** option from the context menu or command line.







In the drawing, the slope is selected as a single set of objects, parts of which can be moved.



Updated Commands

Move point (NG_MESH_VERTEX_MOVE_XY is renamed to NG_MESH_VERTEX_MOVE).
The command now has the ability to choose when moving a point whether snap to drawing objects or take the height of the mark from the surface;

Add point (NG_MESH_VERTEX_ADD)

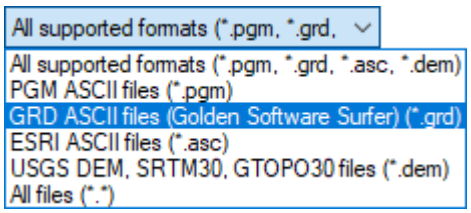
The command can now take elevation either from a drawing object or from a surface.

When adding surface points, the position of contour lines is now updated, if any have been drawn.

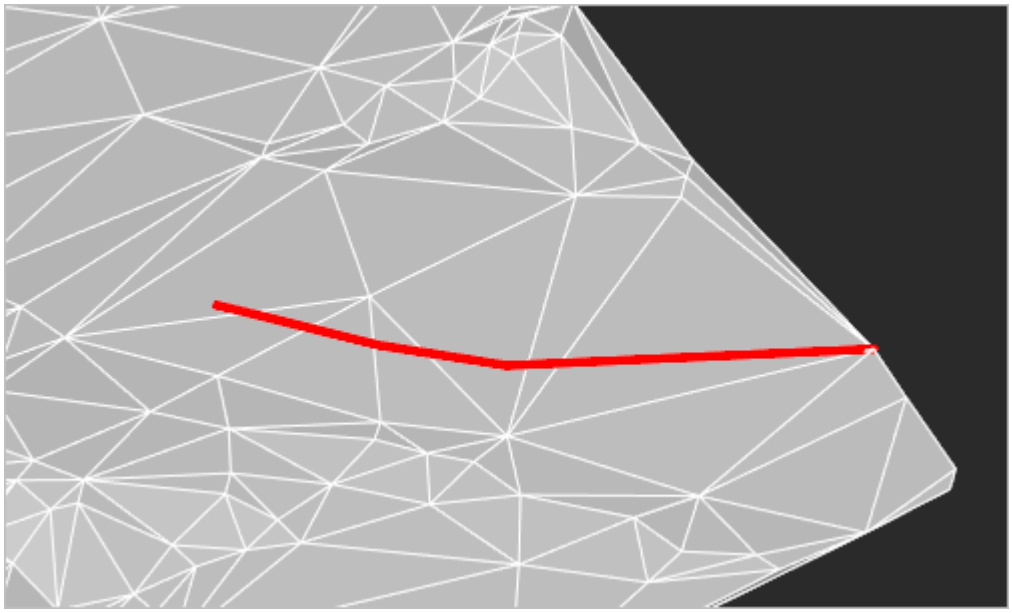
Delete point (NG_MESH_VERTEX_DELETE).

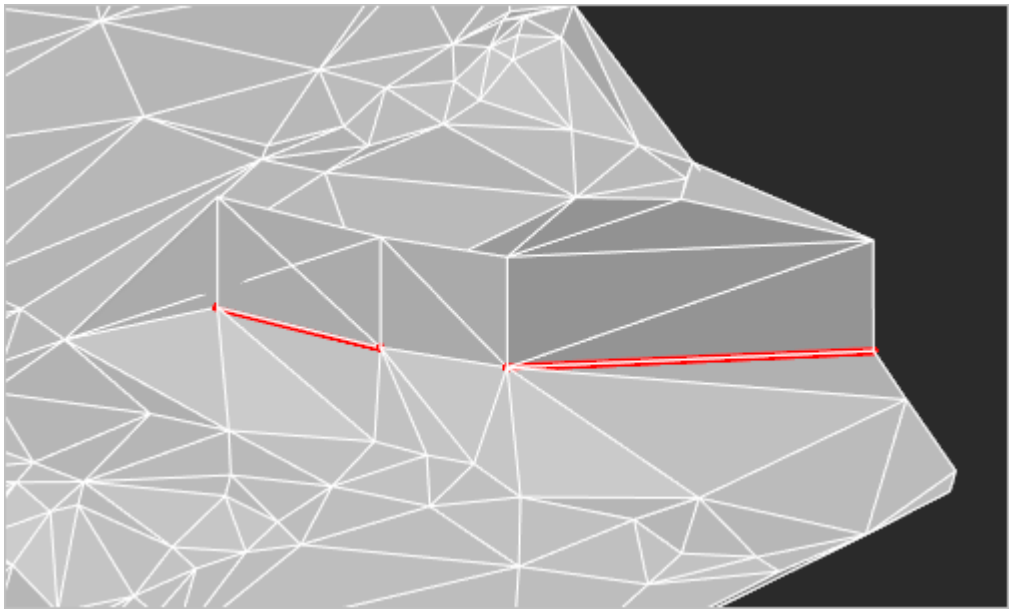
Deleting surface points now updates the position of the contour lines if they were drawn.

Import elevation grid (NG_IMPORT_ELEVATIONGRID). The GRD (Surfer 7 Grid File Format) binary format has been added to the list of supported formats.



Add structure line (NG_MESH_STRUCTURAL_CREATE). The ability to build retaining walls has been added to the command.



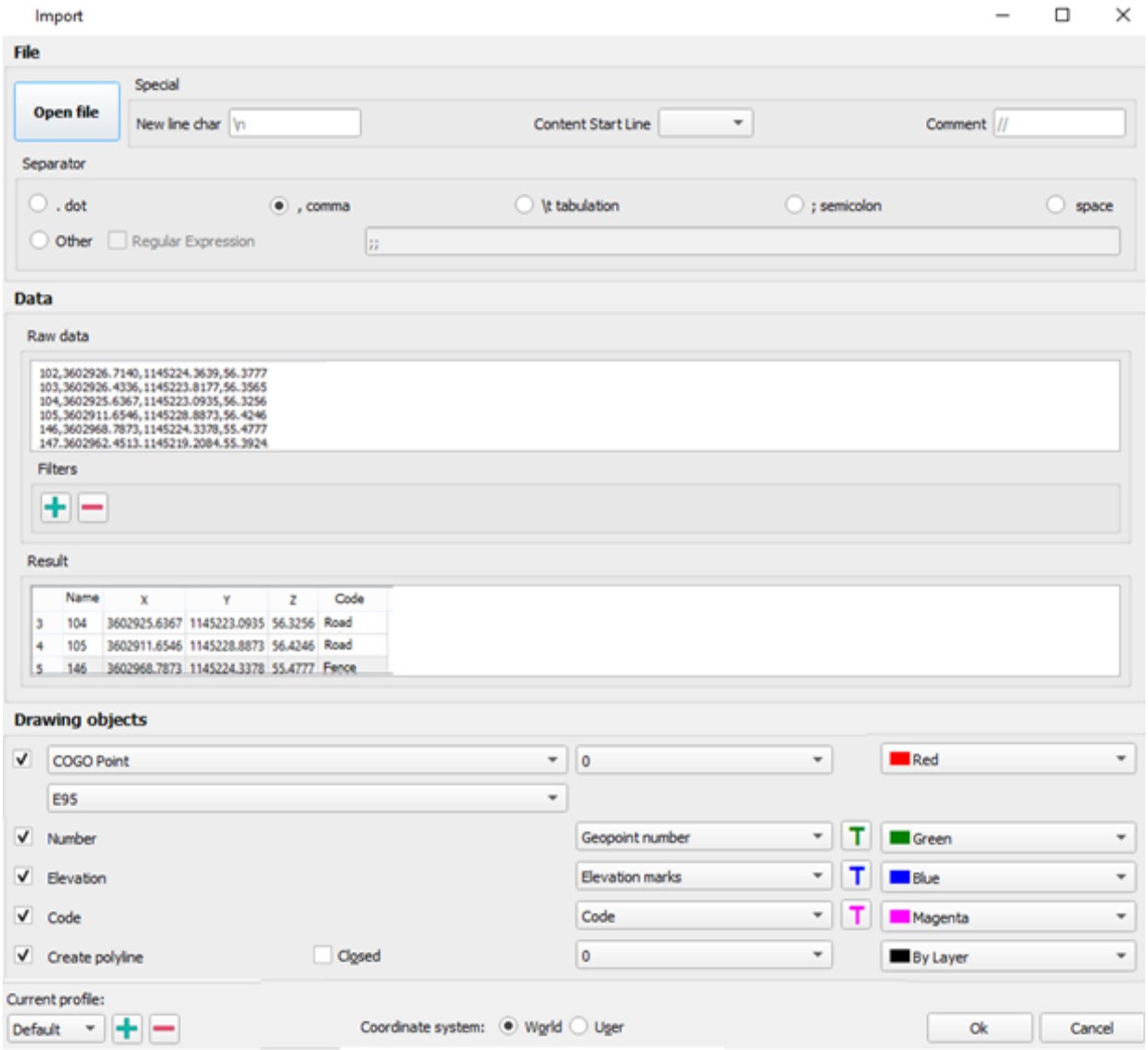


Export to LandXML (NG_EXPORT_LANDXML).

It became possible to export geopoints to the LandXML format. When exporting, the belonging of points to a group is saved. Export of geopoints and surfaces is disabled and enabled in the command parameters in the **Properties** bar

Import Geopoints

Import Geopoints (NG_IMPORT_POINTS). The command for importing geodetic measurements from external text files has been significantly improved.



Now you can open multiple files for import at the same time.

New formats became available for import: CSV, SDR (Sokkia).

You can choose the type of imported objects: blocks \ geopoints \ primitives.

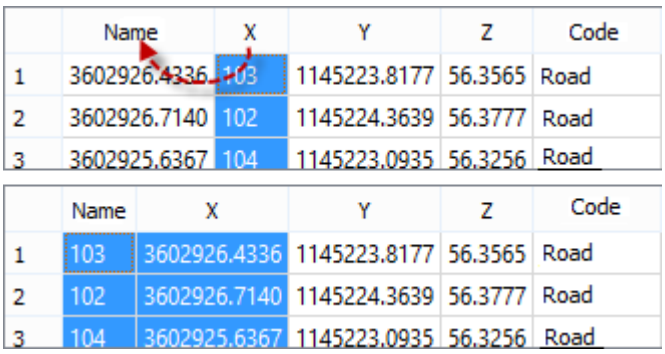
When importing survey files, it is possible to create blocks with the signature of number, mark, code (if available). The scale of blocks depends on the value of the current topographic scale.

If an existing block is selected when creating a point, then a new block with attributes is created, where the point sign is the block that the user has selected. If a new non-existent block is selected, then after the dialog settings, the block editor opens to edit the point sign (circle by default).

You can set layers for each element.

The result table has received additional features:

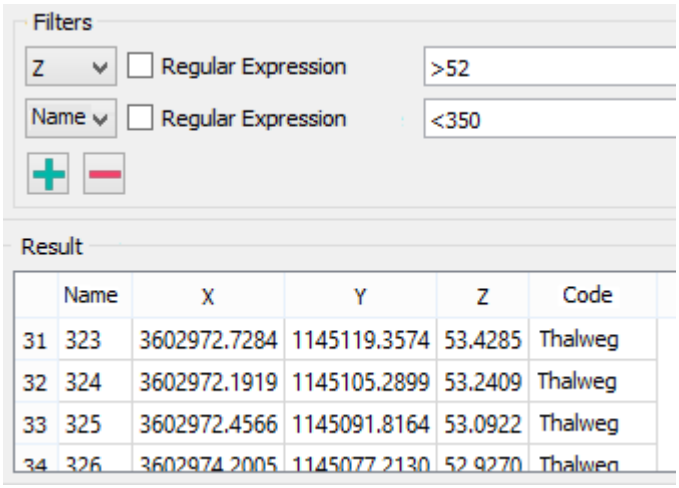
- Changing the order of points, changing the order of columns. In case of incorrect initial data typing, you can drag one column to another by the header, mutually changing data in these columns;



	Name	X	Y	Z	Code
1	3602926.4336	103	1145223.8177	56.3565	Road
2	3602926.7140	102	1145224.3639	56.3777	Road
3	3602925.6367	104	1145223.0935	56.3256	Road

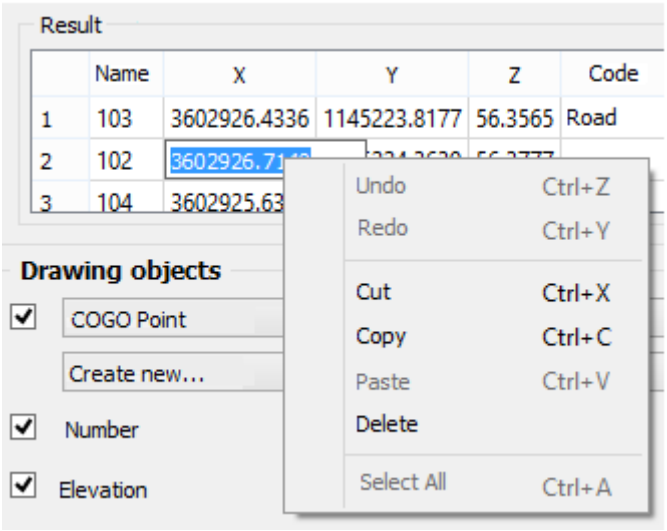
	Name	X	Y	Z	Code
1	103	3602926.4336	1145223.8177	56.3565	Road
2	102	3602926.7140	1145224.3639	56.3777	Road
3	104	3602925.6367	1145223.0935	56.3256	Road

- If the corresponding box is checked, it is possible to create a polyline using filtered points;
- Data filtering by any column is available;



	Name	X	Y	Z	Code
31	323	3602972.7284	1145119.3574	53.4285	Thalweg
32	324	3602972.1919	1145105.2899	53.2409	Thalweg
33	325	3602972.4566	1145091.8164	53.0922	Thalweg
34	326	3602974.2005	1145077.2130	52.9270	Thalweg

- It is possible to edit point data (name, coordinates, code);



- If the import is to geopoints, it is possible to create a new group.
- Multiple editing of cells is available using the SHIFT and CTRL keys. All selected cells are edited simultaneously. For example, you can clear a column if the data is not needed.

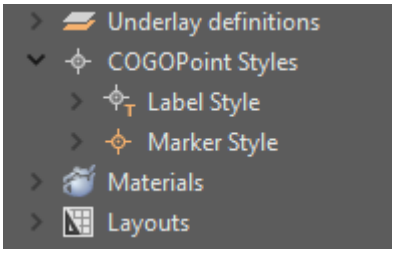
Saving of user settings (separators, colors, layers, fonts, etc.) to a file has been implemented.

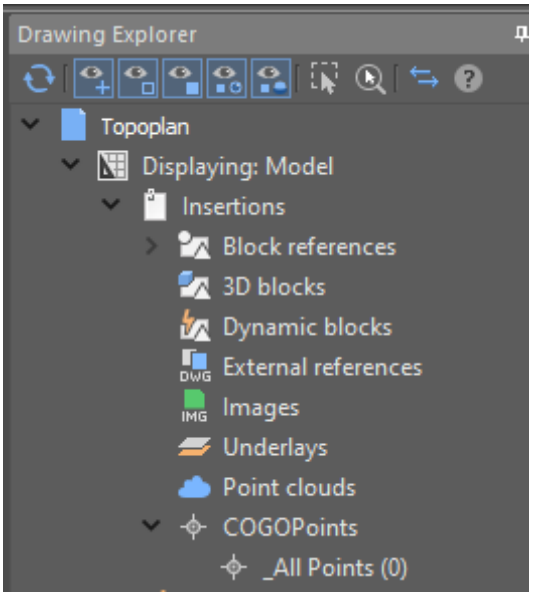
Ergonomics Update

Now the commands for surface editing and working with relief can be launched sequentially, without the need to complete the previous one.

Civil 3D Objects

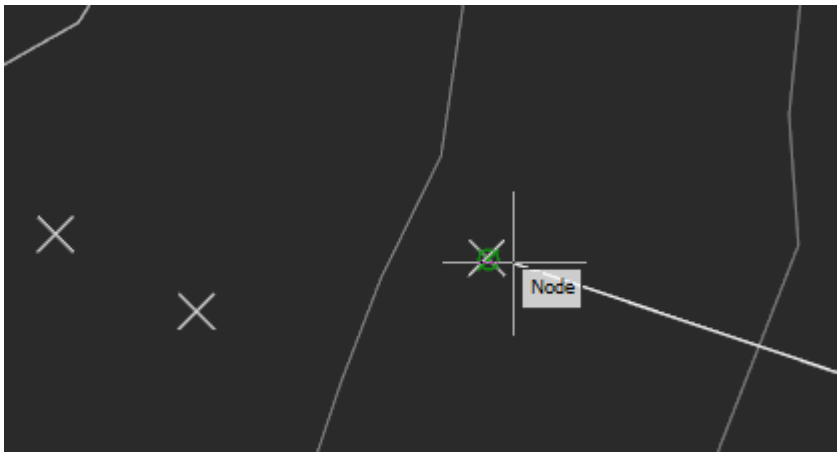
The drawing explorer now displays groups of coordinate geometry points (geopoints), as well as sets of point marker styles and point label styles.





Styles of labels and markers of coordinate geometry are edited by selecting the desired style in the drawing explorer and then calling the command from the context menu. The properties of the selected style are displayed and changed in the **Properties** bar.

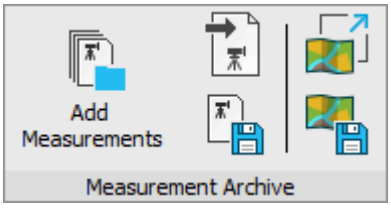
It became possible to bind to geopoints in the **Node** mode.



Now a geopoint can be moved with transformation commands: MOVE, ROTATE, ALIGN, etc.

Work with the Measurement Archive

Commands to work with the measurement archive have appeared.



The work with measurements can be divided into 2 stages: collection of a measurement files archive and formation of a project underlays collection from this archive.

An archive of measurement files contains raw or preprocessed data, while a set of prepared underlays should be groups of geopoints ready for use in projects.

Filling the archive of measurement files

Measurement files are stored in the **Geofiles** folder. The path to the folder is defined in the **Standard directories** section of the **Options** dialog.

To add a file to the archive, the **Add measurements** command is used, with the help of which the required file on the disk is specified, if necessary, a new name is given to it, after which it is copied to the measurement archive.

If you need not only to save a raw file, but to carry out a full import at this stage, so as not to return to setting parameters later, then you should use the **Save to measurement archive** command. You need to configure the parameters in it, after which the result in the form of a DWG file is placed in the archive.

The measurement archive is displayed in the file explorer in **Geofiles** folder. By expanding it, you can view all archive measurement files

Creating selections and forming a set of underlays

To work with points, it is necessary to form selections from the measurement archive.

Selections are formed by special commands in the form of separate DWG files, which are subsequently placed as external references in working drawings. Thus, the same selection (underlay) can be inserted into several different working DWG files.

To form a selection:

1. Create a new underlay with the **Attach geounderlay** command.

2. Import the required measurements into it from the archive using the **Import measurements** command.

3. Configure groups of points, forming them, if necessary, and setting their parameters (styles). Enable/disable the visibility of the desired groups.

4. Save the drawing to the collection of underlays as a separate named entity using the **Save as geounderlay** command.

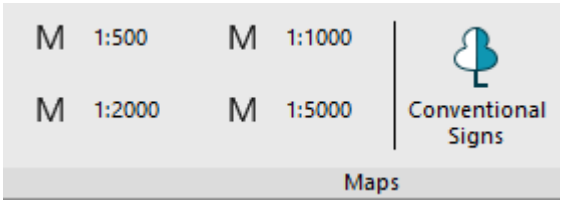
The underlay collection is a separate **Geofiles** folder. The path to the folder is defined in the **Standard directories** section of the **Options** dialog.

It is also visible in the File Explorer as the **Geofiles** folder. By expanding it, you can view all the underlay files.

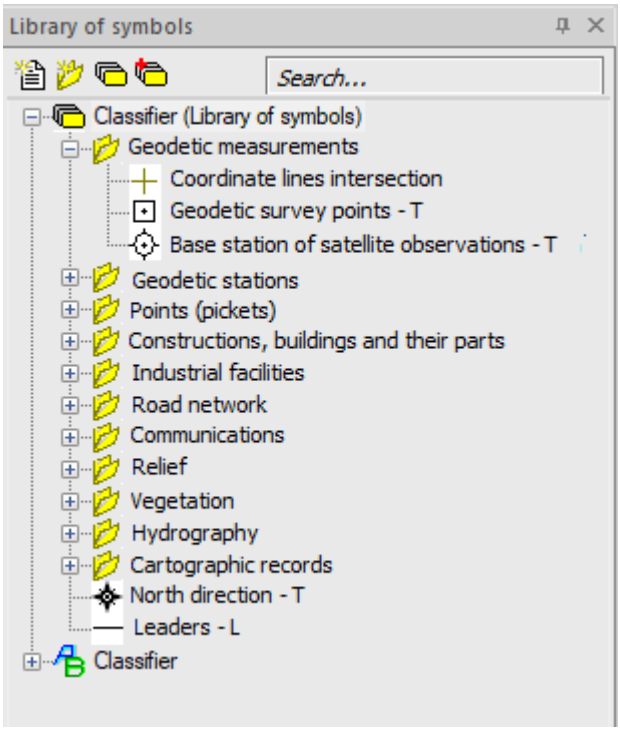
Drawing Context. Conventional Signs

A new **Conventional signs** functional bar appeared. It is used to insert conventional signs into a drawing (for drawing topoplans of scales 1:500, 1:1000, 1:2000, 1:5000).

Before drawing conventional signs, select one of the topographic scales: 1:500, 1:1000, 1:2000 or 1:5000, in accordance with the required scale of the finished drawing.



The bar elements are structured by thematic sections and in alphabetical order.



The capabilities of the **Conventional signs** bar are not limited to inserting signs from an existing classifier. It is also possible to create your own signs or connect third-party libraries. To do this, use the buttons at the top of the bar.

Create element. Creating a new element of the library.

Create section. Creating a folder for thematic ordering of library items.

Create library. Prompts for a name in the command line and creates a new empty library with two ordering options: thematic and alphabetical.

Attach/detach library. Opens a dialog box that allows you to enable/disable the display of existing libraries in the **Conventional signs** bar. The source library cannot be disabled.

It is also possible to create new thematic sections, edit the composition of elements included in them, as well as delete any sign or section from the library.

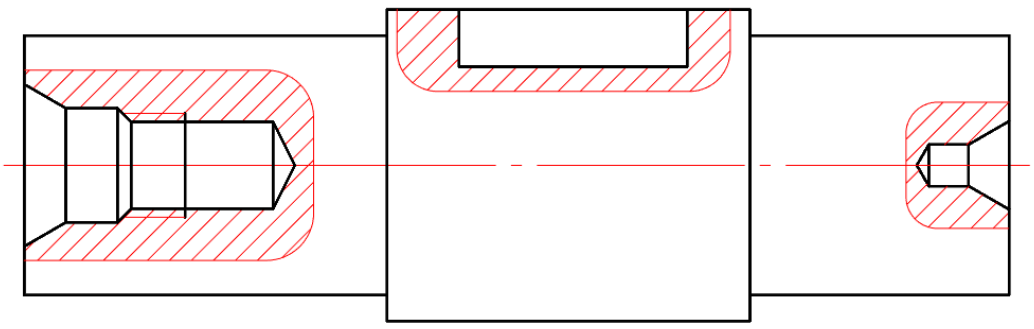
New in the Mechanica Module

Broken-out Section Views

The **Broken section view** command allows you to automatically obtain images of database items such as keyways and center holes on shafts. On parts and flanges, this command can be used to make half cuts. On the bottoms and shells with reduced dimensions, their thickness can be shown.

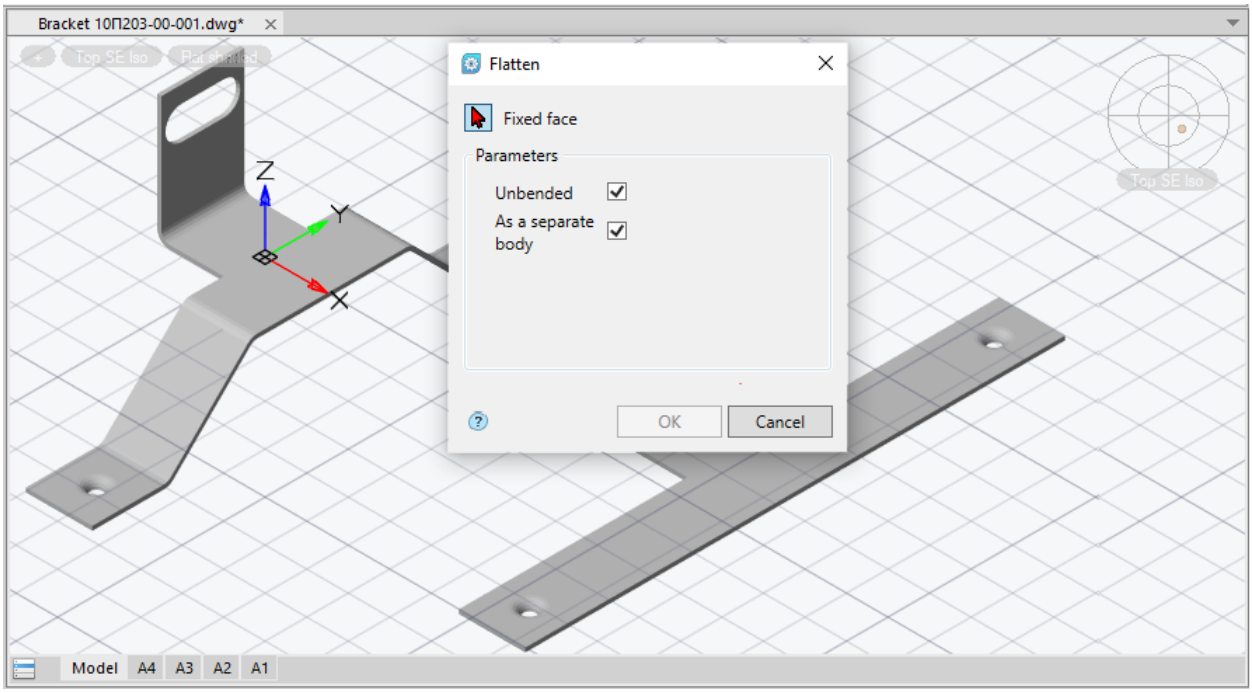
When creating a broken section, all database items that fall within its frame are shown in section. In this case, if the database item does not have a section view, its normal view will be displayed.

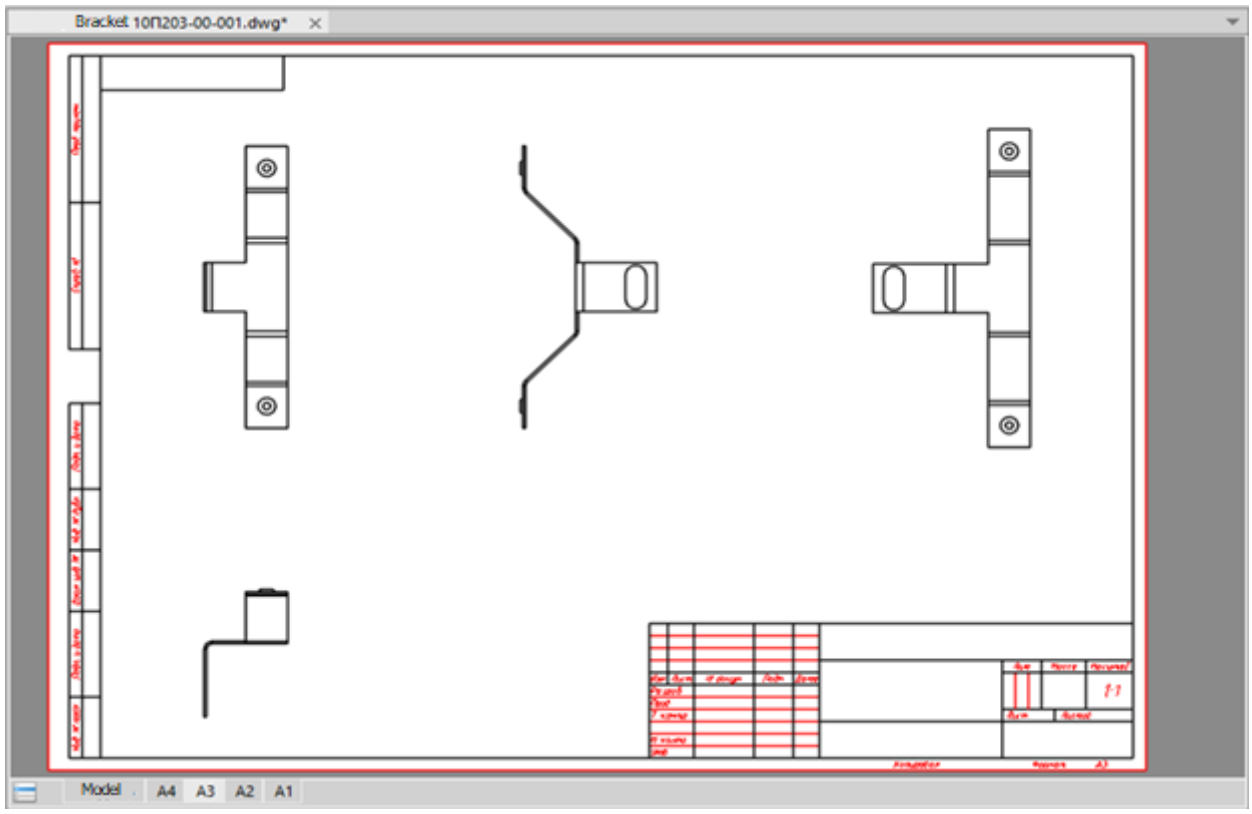
When using geometry that falls into a local section, it is reshaped by the **Refresh view** frame button or by mcregenobj command.



New Features for Working with Sheet Bodies

When working with sheet models, the ability to create a flat view as a separate body is implemented. This functionality gives you more freedom when drawing from sheet metal 3D models. Flat drawings can now be drawn in two states simultaneously: unbended and bent, while maintaining associativity with the 3D model.





Properties of Fasteners

The properties of fastener parts have been worked out.

For old standards, the following properties have been added:

- Thread direction;
- Thread tolerance field;
- Material;
- Strength class/group;

- Type/grade of steel;
- Coating;
- Coating thickness.

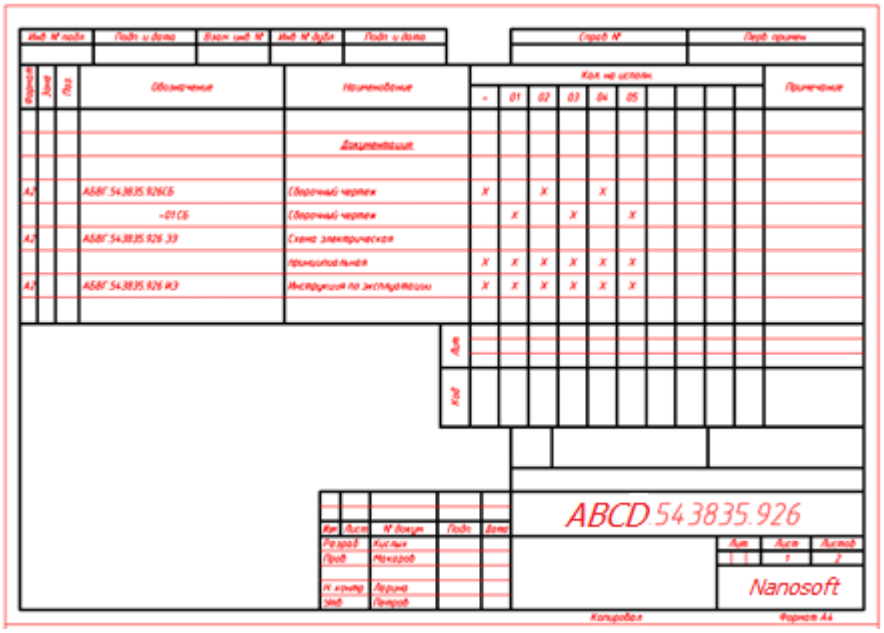
For new standards, the following properties have been added:

- Material;
- Strength class;
- Coating;
- Coating element;
- Coating thickness;
- Finishing of the coating;
- Coating method;
- Coating lubrication;
- Duration of trials.

Improvement of Specification Functionality

The file to replace the singular with the plural is added. This file named **SpecRecsGroupingNameReplace.txt** is installed in the C:\ProgramData\Nanosoft\nanoCAD x64 Int 22.0\NanoMechanical\UserTemplates\en-US folder and is available for users to modify in order to add own words that should be changed during grouping. If, when grouping specification lines, you come across a replacement word (for example, "Screw"), then it is replaced with a substitute word (for example, "Screws"). The file's action also applies to custom grouping of lines with **$$** tags.

Additional specification templates with the customer's stamp have been made for all specification templates available in the database (simple, group type A and type B, group type A up to 3 versions and plasma). Columns have been added in these templates for the customer sign, decision numbers, approval years, and customer index.



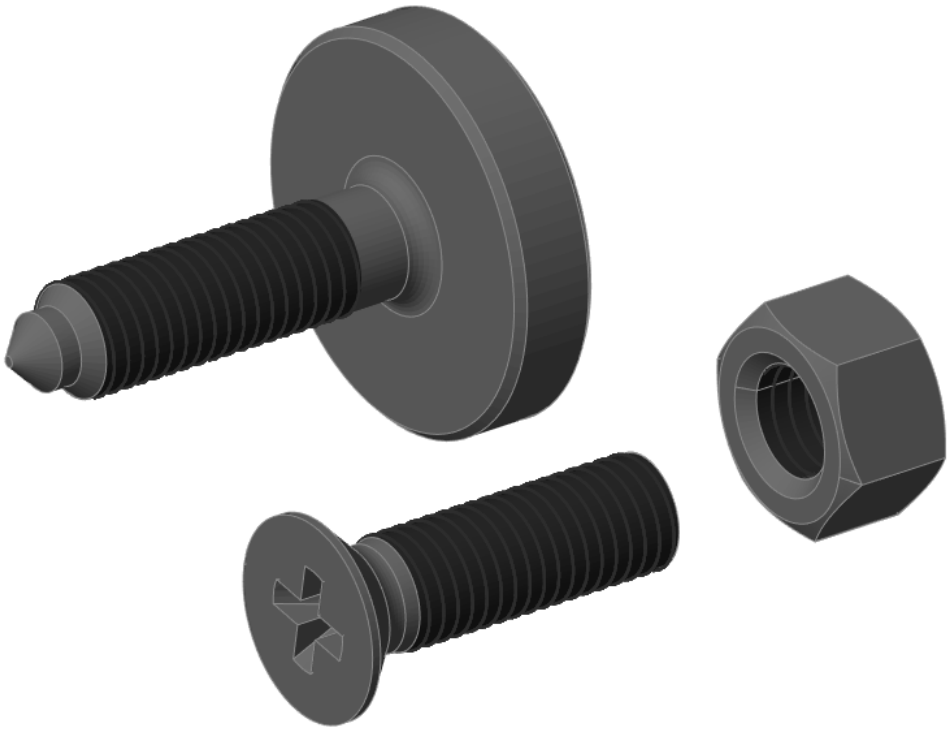
Additional sections of the electrical installation specification begin on a new sheet preceded by an entry like “**Installed according to XXXX.XXXXXX.XXXME**” or “**Installed according to XXXX.XXXXXX.XXXTB**” or “**Installed during electrical fitting**”. All components (including materials) included in the specification in additional sections are assigned with position numbers. The numbering of positions is end-to-end within the entire specification.

New Materials in the Database

Materials according to EN, DIN and ASTM with the possibility of their designation both by chemical composition or brand, and by numbers.

Other Changes

3D thread on standard fasteners.



New in the Construction Module

New Features of Fasteners

The properties of fastener parts have been worked out.

For old standards, the following properties have been added:

- Thread direction;
- Thread tolerance field;

- Material;
- Strength class/group;
- Type/grade of steel;
- Coating;
- Coating thickness.

For new standards, the following properties have been added:

- Material;
- Strength class;
- Coating;
- Coating element;
- Coating thickness;
- Finishing of the coating;
- Coating method;
- Coating lubrication;
- Duration of trials.

Corrections

The hangup when adding templates to the **Library of templates** for elements of utilities network has been fixed.

The work of the report and the name of the wall templates if they are absent in the database of elements has been fixed.

The configuration error in settings of **Construction axis** and **Separate axis** dialogs have been corrected.

The wall thickness overlapping with the background when inserting a wall template with layers into a drawing has been fixed.