

# WHAT'S NEW IN NANOCAD 3DSCAN 2.0

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## New Point Clouds Commands and Features

- Automatic recognition of ground surface has been significantly improved;
- Possibility to classify vegetation and objects at a given distance from the ground (or any other surface) has appeared;
- Support of WGS84 coordinate system;
- Significant corrections and improvements.

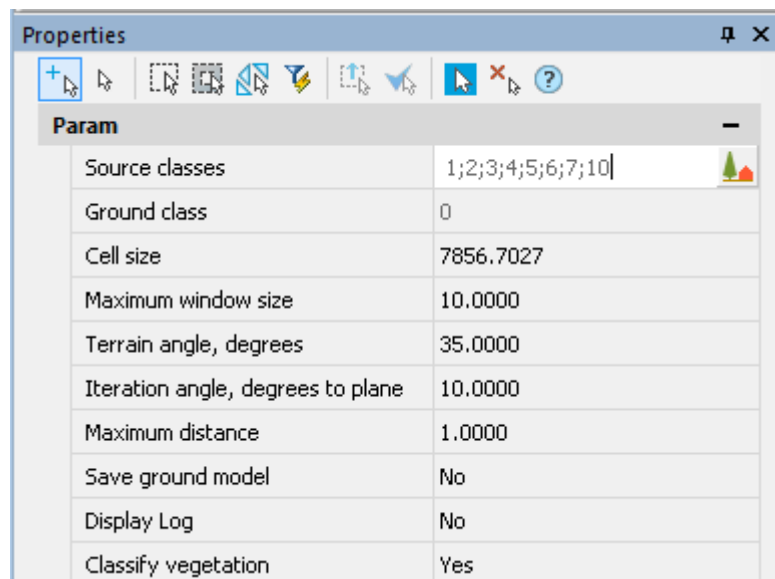
### Classification of Ground and Vegetation

The command  **Classification of Ground** (NPC\_CLS\_GROUND) has been revised.

#### Command Options on the “Properties” Bar


Now the command options are displayed and configured not in the command line, but on the **Properties** bar.

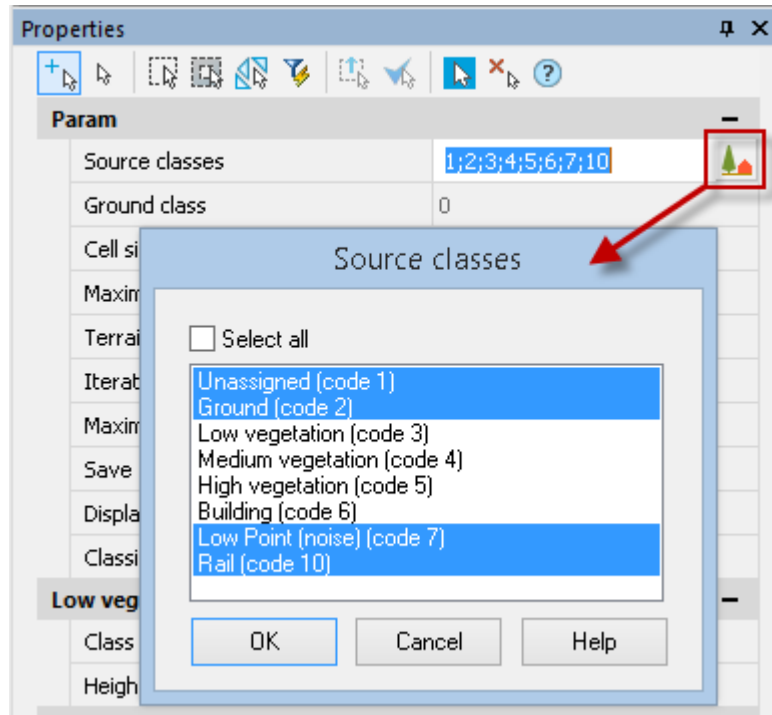
New command options:



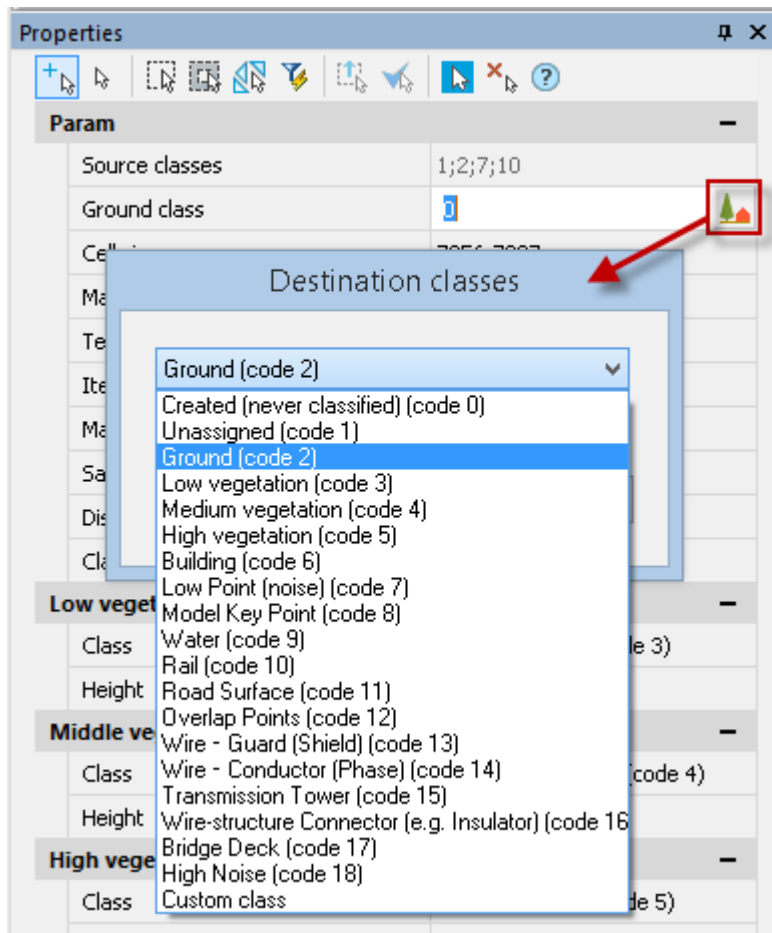
#### Filtration Based on Source Classes

The command allows you to specify certain classes of points for the ground classification process. It can be useful if a cloud was earlier classified by these or those criteria. For example, to exclude from the recognition process the points of noise, man-made objects, etc.

The **Source classes** parameter defines the classes of those points, that will be used for ground classification. Points of other classes will not be reclassified. The list opened by button  displays all classes of the current point cloud.



The **Ground class** parameter is the target class, which should be assigned to recognized points of the ground.



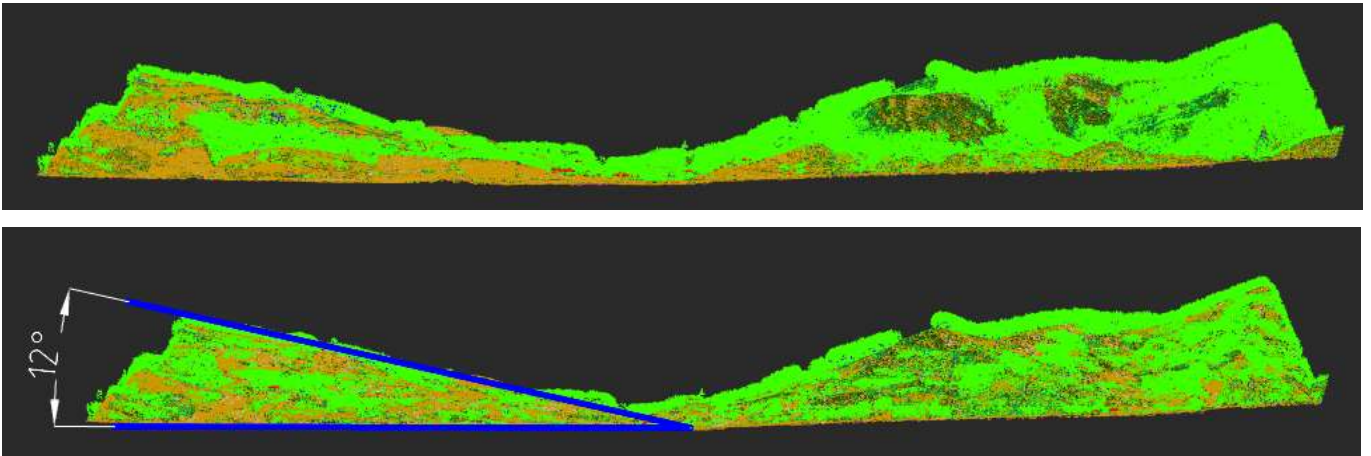
The target class should coincide with none of the source classes, otherwise the command will stop ahead of time with an error message in the command line.

## Automatic Calculating of Surface Detalization During Ground Classification

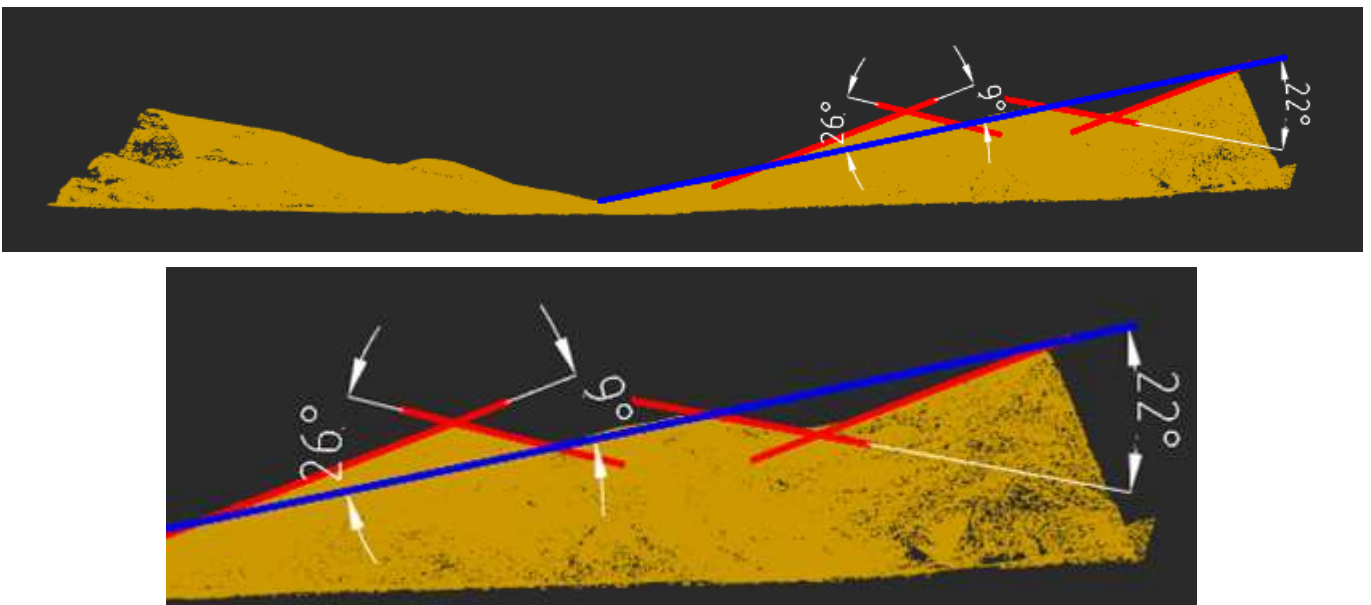
Now the **Cell size** parameter is calculated automatically, based on the size of a point cloud in the drawing. As a rule, it should not be changed. In previous version of a program, the value of this parameter was taken from the **Point clouds** section of the **Options** dialogue. The **Cell size** defines the length of sides of square cells that divided surface of point cloud. The smaller the value, the more accurately relief elements are reflected. Parameter value is set in the units of drawing.

### Surface Irregularities

We preserved the parameter **Terrain angle, degrees** – maximum angle of slope in degrees relative to XY plane in WCS. It can be measured at some characteristic area. Points on surfaces that exceed this angle will not be classified as ground surface points.

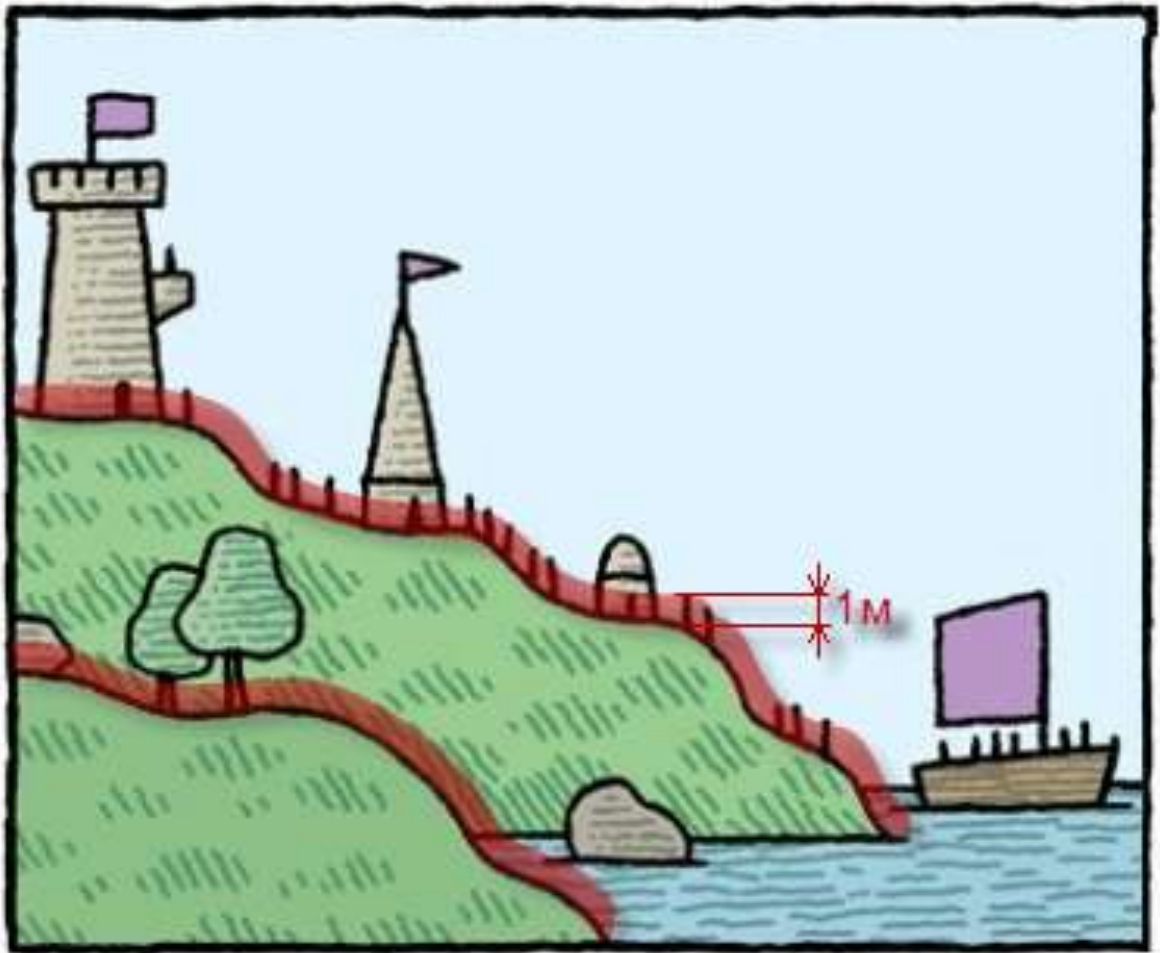


Parameter **Iteration angle, degrees to plane** – maximum angle of deviation of local irregularities from the main direction of the ground surface.



## *Filtration by Distance from Ground Surface*

The **Maximum distance** parameter determines the maximum distance above the ground surface, above which points will not participate in the classification process and, therefore, will not be classified as ground points. The distance is specified in the units of drawing.



## *Exclusion of False Surfaces*

A new parameter **Maximum window size** allows you to exclude low buildings and constructions with flat wide roofs from ground recognition.



It is necessary to find the widest low object and measure a half of its shorter side, increasing the resulting value by 20 percent. For each cloud the parameter should be set manually in the units of drawing.

**Note:** correctly measured parameter **Maximum window size** is always larger than **Cell size** parameter.

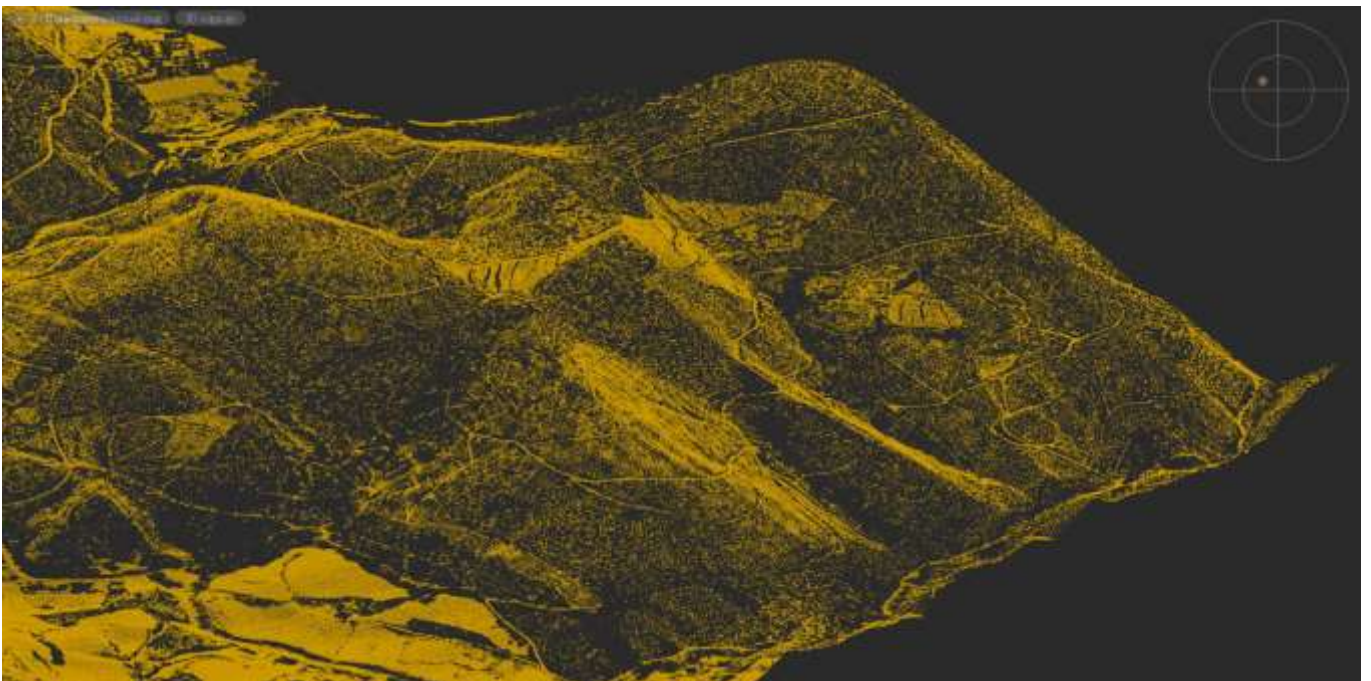
### ***Classification of Vegetation***

The command is enhanced with the functional for classification of points (vegetation) by height above ground level. Set **Yes** for the **Classify vegetation** option, additional options will appear.

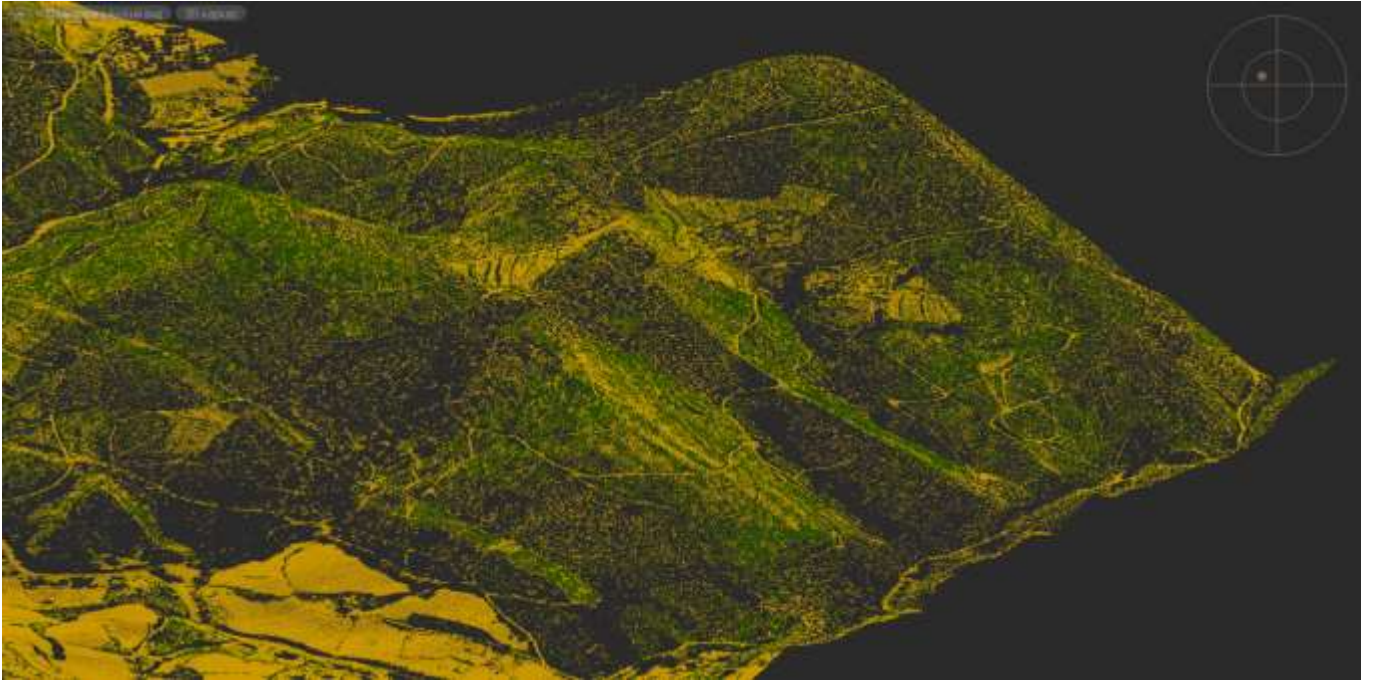


Classify vegetation	Yes
<b>Low vegetation</b>	—
Class	Low vegetation (code 3)
Height	0.3000
<b>Middle vegetation</b>	—
Class	Medium vegetation (code 4)
Height	2.0000
<b>High vegetation</b>	—
Class	High vegetation (code 5)
Height	100.0000

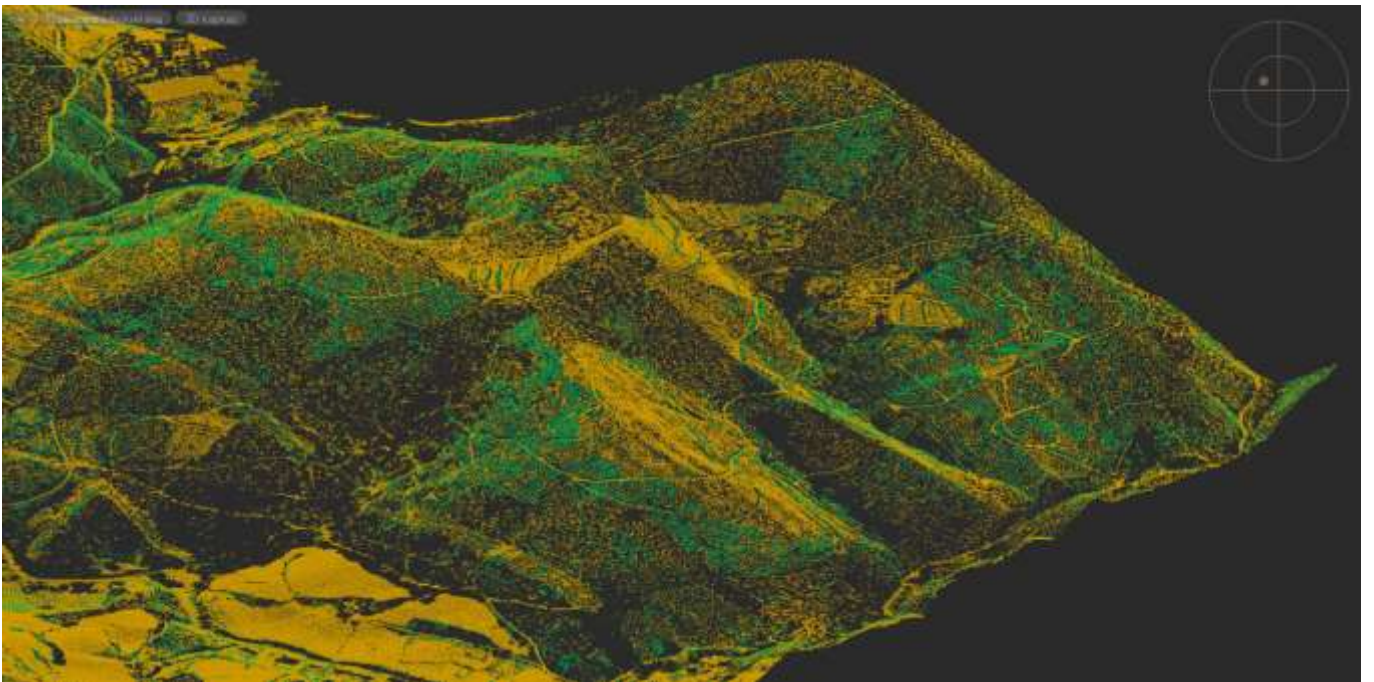
The command classifies points located in the given altitude range relative to the ground surface. Values are set in the units of drawing.



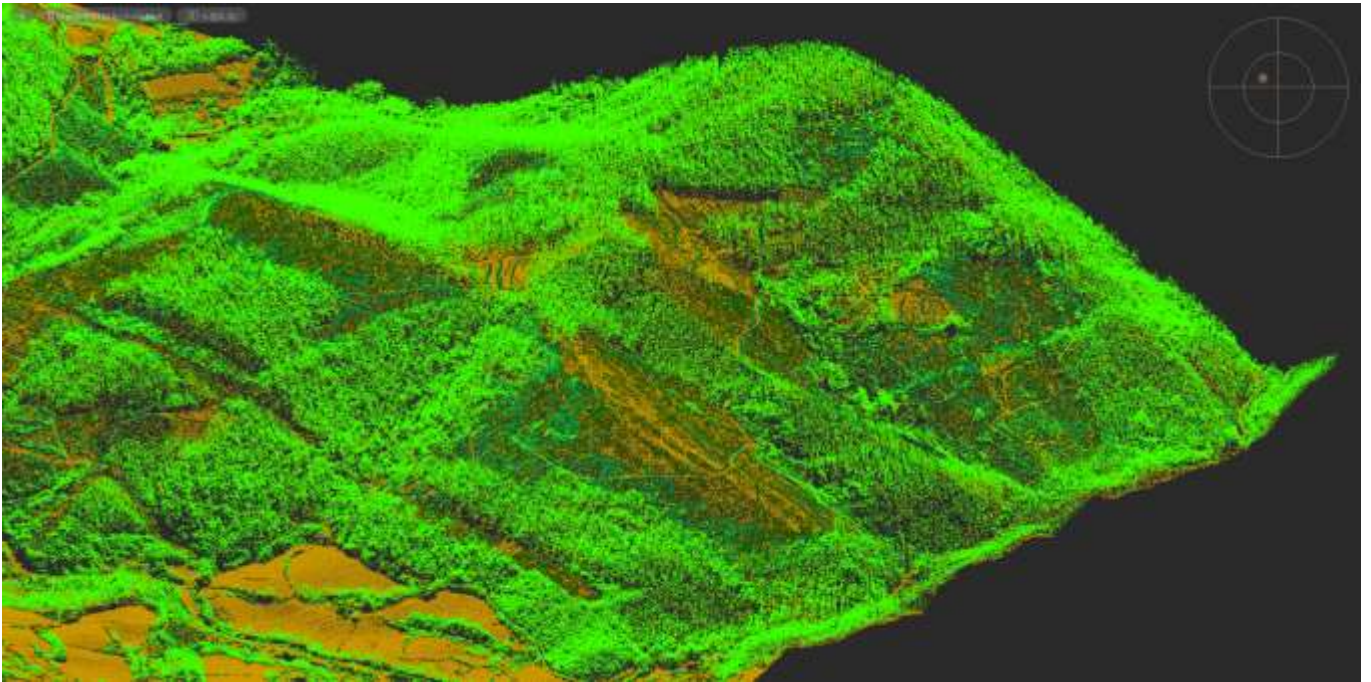
*Only ground surface points*



*Ground surface points with low vegetation points*



*Ground surface points with average vegetation points*





*Ground surface points, points of low, average and high vegetation*

This procedure can be used not only to classify vegetation, but also to classify any objects by height from the ground surface: low constructions, high-rise buildings, power lines, pipes, etc.


### ***Creating Triangulation Mesh of a Surface***

In addition to ground surface classification, the command allows you to automatically construct its triangulation model as a mesh object. For this the value of **Save ground model** option should be **Yes**.

In contrast to the triangulation mesh of ground surface created by  **Create TIN** command, this network is created with optimized edges. In the future the created triangulation mesh can be optimized by  **Model Simplification** command.

### ***New Commands and Improvements***

#### ***“Vegetation by Height” Command***

The functional of objects classifying by height above the surface is also put into a separate command  **Vegetation by Height**.

The command has parameters identical to those described above. As the surface relative to which the classification will be made, it is necessary to specify the existing

triangulation mesh, for example, the one built earlier by the command



**Classification of ground** or



**Create TIN**.

## ***Filtration Based on Source Classes in the “Create TIN” Command***



The command **Create TIN** (NPC\_2D\_TRIANGULATION) got new options that allow you to select particular point classes for creation of triangulation mesh based on them. The options are similar to ground classification command



**Classification of ground** (NPC\_CLS\_GROUND).

## ***Importing Clouds with WGS84 Coordinate System***

The possibility to import point clouds from LAS format with WGS84 coordinate system is added. Previously clouds with WGS84 coordinate system were imported with distortion.

## ***Changing Class for Visible Points Only***



A new command **Classification by class** (NPC\_CLS\_CLASS) changes class for visible points only. Just cut a part of cloud containing required points with one of the **Clip** or **Section** commands and reclassify only visible points by specifying their source class.


## ***Coloring Clouds by “Compare Point Clouds” Command***



Now the command **Compare Point Clouds** colors the cloud according to the normal direction (outside/inside) when comparing a cloud with triangulation ground surface. The cloud above TIN surface is colored in the spectrum from green to red, and below – from green to blue. It allows easily determine the position of points relative to the mesh (ground surface).

In addition, the coloring of clouds was fixed when resetting clip after comparing their trimmed parts.

## ***Changes and Corrections***

- Fixed errors of graphics (widget) displaying when comparing clouds.
- Correcting work of the command  **Fit Cylinder** (NPC\_FIT\_CYLINDER).
- Fixed errors of comparison algorithm in video card.
- Updated icons of cloud points classification commands.

## ***Platform Features***

nanoCAD 3DScan 2.0 based on 20.1 version of nanoCAD platform. Thus, nanoCAD 3DScan 2.0 gets all features of nanoCAD Plus, since 8.5 version.

### ***New Features of nanoCAD 10.0 Platform***

Key features of nanoCAD 10 platform are: new user interface style, compatibility with most DWG-CAD systems, speed and usability, checking drawings for standard compliance, new 2D-drawing and 3D-modeling functions, improved printing and much more.

#### ***Increased Performance***

nanoCAD visual graphic system that is responsible for rendering 2D and 3D-graphics, has been updated to support the Microsoft DirectX 10 technology. This accelerates the processing time of drawings with large amount of text objects, hatches, blocks and other complex primitives, and reduces the size of memory used.

#### ***DWG 2018***

nanoCAD 10 works directly with the latest version of DWG format. nanoCAD opens and saves drawing files without breaking their internal structure. This ensures the integrity of the built-in DWG format labels which can carry such characteristics as educational plot stamp, for example.

#### ***User Interface***

A new ribbon-style interface provides modern ways of interaction with application. Instant switch between classic and ribbon interface makes work very comfortable. The ribbon structure was initially designed taking into account the ergonomics of the designer, but it does not limit the user in the possibilities of customizing their workspaces. In addition, a powerful mechanism of managing the ribbon interface allows you to instantly switch between different pre-installed workspaces without restarting the program.

#### ***Interactive Tools in a Drawing Area***

Semi-transparent interactive tools placed in a drawing working area designed to manage views and visual styles.

The **Locator** element intents for convenient navigation in a three-dimensional space: rapid dynamic change of orthogonal, isometric, combined and free views. Its orange-dot marker shows current view orientation.

The **View Control** element at the top left of drawing workspace changes named view and visual style of a viewport.

## *Checking Drawings for Standard Compliance*

10 version of nanoCAD can check drawings for standards compliance and fix found violations.

There are a number of notification mechanisms that alerts user about violations of standards while opening, saving, printing drawing file. It is possible to fix any violation or stay it intact.

Information on each standard violation that has been found, but not fixed, is stored in a drawing. It is possible to disable alerts on such ignored objects in subsequent checks of the drawing.

Violations audit can be performed manually and automatically, for example during drawing export, save and print operations.

## *Partial Auto-save*

From now on, only the modified DWG data is sent to the file during background autosave, that significantly reduce the time of this operation even on rich documents.

## *Dynamic Blocks Extended Support*

Dynamic block processing was improved: nanoCAD completely supports property tables, insertion point management, dynamic blocks parameters correct processing. This ensures that most objects of this type are functioning correctly.

## *New Drawing Objects and Commands*

- In addition to its own leader objects, nanoCAD can create and edit multileader objects, manage multileader styles.
- Native DWG-tables creating and editing without conversion to nanoCAD-table is available. Managing DWG-table styles and conversion nanoCAD-table to DWG-table and vice versa.
- 3D-polyline creation and editing.
- A new **Region** command creates an object of a **region** type from 2D objects that form closed area.
- The **Stretch** command use pre-selected vertices that makes working with the command more convenient.

## *Transparency of a Layer*

The layers got the **Transparency** property. All objects with **By layer** transparency value will inherit the transparency from the layer that belongs to.

## ***Third-party Formats Import***

Now it is possible to import WMF and EMF formats, as well as DGN – 2D/3D-format used by the Bentley Systems MicroStation design program.

## ***Multiple Reworks***

The next stage of program optimization was performed in this version of nanoCAD. The common list of realized wishes, corrected errors and the taken into account notes exceeded seven hundred points. However, beginning with the 10th version, nanoCAD stops supporting Windows XP due to the need to speed up graphics and overall increase the stability of the program.

## ***New Features of nanoCAD 11.0 Platform***

### ***Incremental (Partial) Saving***

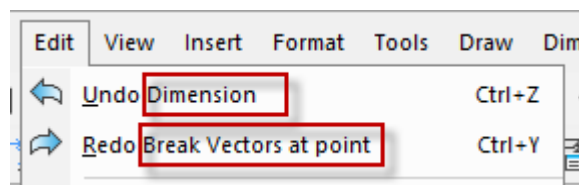
In addition to incremental (partial) autosaving implemented earlier, the incremental saving is added. This greatly accelerates saving of a large documents.

You can manage incremental saving modes in the **Save Documents** section of the **Options** dialog.

### ***Undo-Redo***

The new version has a number of improvements in undo-redo functionality.

#### ***Named Steps***



The names of the last operations (steps) are displayed in the **Undo** and **Redo** commands in the **Edit** menu (the classic interface).

#### ***Multiply Undo/Redo***

The flyout arrows are attached to the **Undo/Redo** buttons on the **Quick Access** toolbar to undo/redo several actions at once.

Please take a note that you cannot drop-down action history list in the command mode (when a command is running). In command mode these buttons perform internal undo/redo operations, if exists.

## Optimization Performance

### Selection Set

Performance is improved for comparing old and new selection sets. The speed is significantly increased for operation of selecting among other objects in existing selection set.

### Text Objects

The mechanism of graphical representation of text objects is optimized. As a result, image regeneration speed is increased and the amount of used memory is reduced.

### Blocks

Loading and regeneration a file with a large number of blocks was significantly accelerated.

### Complex Objects Insertion

The process of inserting complex objects into the model space was optimized, cursor freezing was fixed.

### Frozen Layers Optimization

Increased performance of loading and working in a document with frozen layers.

### Ultra-remote Objects Optimization

The problem of displaying distortion of a drawing elements at super large distances has gone into the past. You no longer need to regenerate the drawing when approaching objects that are far from the current location.

## OLE-objects

New commands for working with OLE-objects are implemented.



- The **Insert OLE-object** (INSERTOBJ) command allows you to insert linked or embedded OLE objects from another document.
- The **Open OLE-object** (OLEOPEN) opens specified OLE-object for editing in its source application. A similar result can be obtained by double-clicking the OLE-object in a drawing.
- The **Update Data Links** (OLELINKS) command opens the **Links** dialog to update and modify links of OLE-objects.
- The **Update All Links** (OLEUPDATEALLLINKS) command allows you to update all links for OLE objects in the document at the same time.



## ***Scripts (.scr)***

A new way of automation is added - by using scripts. A Command Script is a text file with a .scr extension that contains a set of instructions to run in the nanoCAD command line. Each line of the file contains a command call (with parameters) or a link to another script.

The following script processing commands are implemented:

- SCRIPT – loads the script file and executes the instructions described in this file sequentially;
- SCRIPTCALL – modification of the SCRIPT command to run the nested script;
- RESUME – if a script is suspended while the program is active, you can use RESUME to continue the script;
- RSCRIPT – repeats the last script.

## ***Digital Signatures***

The custom files signing (.dwg, pdf, zip, ...) with a digital signature is available.

Two types of signing were implemented:

- creating an internal signature when saving a file;
- creating an external signature for any file.

Commands for internal signature creating:

- DIGITALSIGN – allows you to choose a digital certificate for digitally signing a file and sets up signing while saving a file;
- SAVE - creates an internal signature during file saving;
- SIGVALIDATE - validates the digital signature of the file.

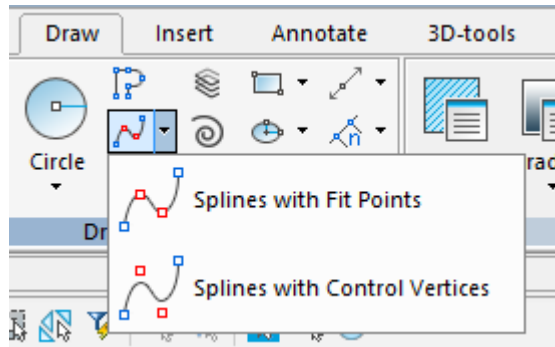
Commands for external signature creating:

- SIGNCERTIFICATECONTROL – allows you to choose a digital certificate for digitally signing a file;
- SIGNFILE - creates a file with external signature ("FileName.FileExtension.sig") for any specified file;
- VALIDATESIGN - validates the external signature of the specified file.

## ***Splines***

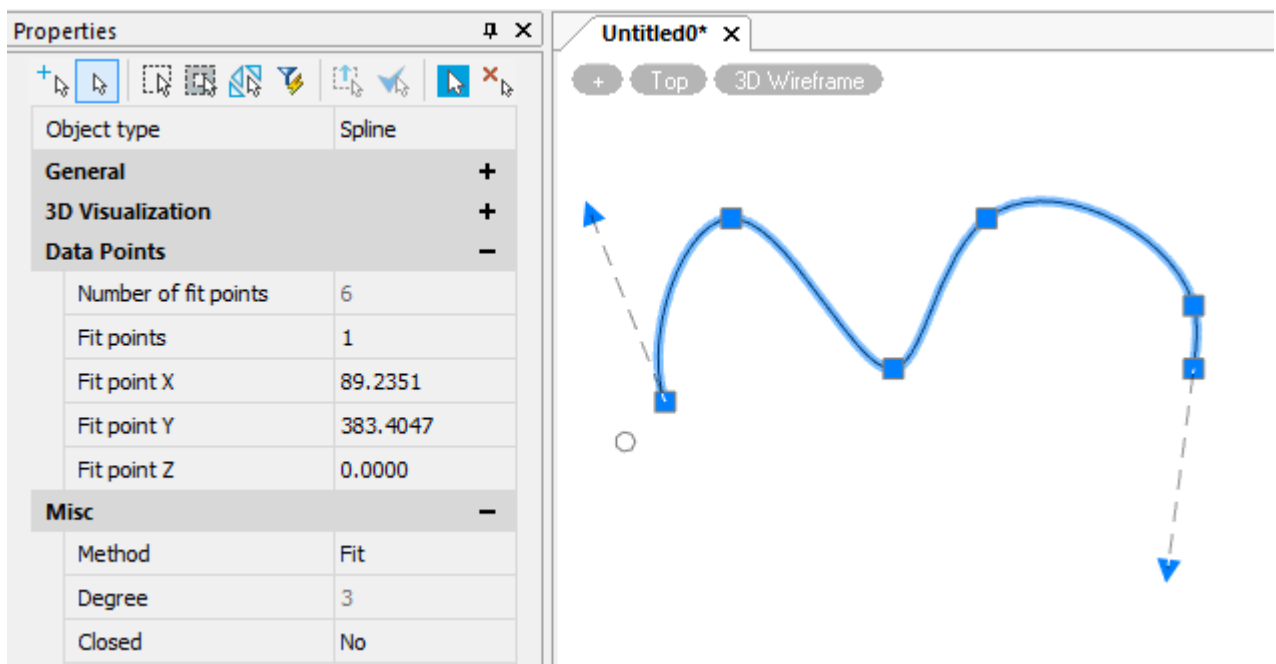
### ***New Spline Creation Method***

The second method of spline creation was implemented – by specifying control points.



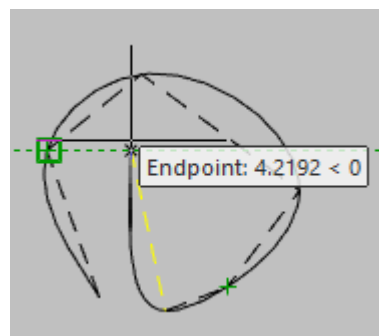
### ***New Grips***

To edit splines in the fit point mode, a new type of grips is implemented to change the tangent vectors direction.



### ***Rubber-band Line***

During spline creating an auxiliary rubber-bend line is displayed on the screen from one fit point (or control point) to another. You can also snap to this temporary dotted line.



### ***Snap to the First Point of a Spline***

Now you can snap to spline start point.

### *The “Simplify Spline” Command*

The SIMPLIFYSPLINE command optimizes the spline by changing its approximation accuracy and the maximum number of its points.

### *The “Export Layout” Command*

The EXPORTLAYOUT command allows you to export all the visible objects of the current layout to the model space of the new drawing.

### *The “Change Space” Command*

The CHSPACE command moves the selected objects from the model space (in a layout viewport) to the paper space, as well as from the paper space to into the layout viewport.

## *Dimensions*

### *The “Oblique Dimension” Command*

The **Oblique Dimension** (DIMOBLIQUE) command changes the oblique angle of one or more previously created dimension objects.

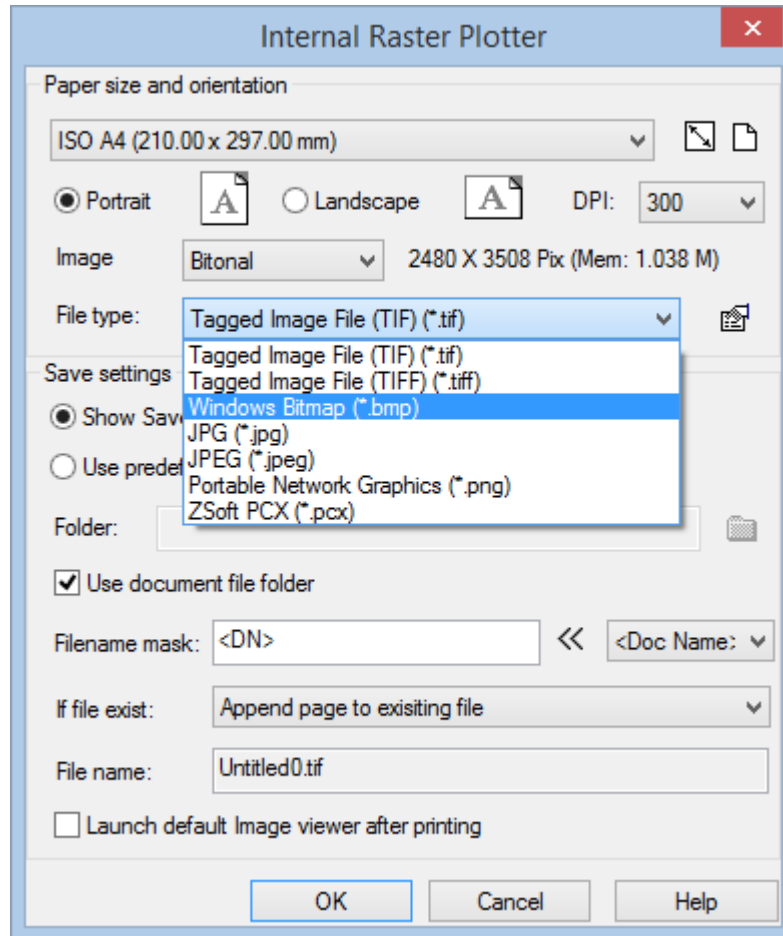
### *The “Angle Ordinate Dimension” Command*

The **Angle Ordinate Dimension** (DIMAORD) command creates an angular ordinate dimensions.

## *Printing*

### *Internal Raster Plotter*

The internal raster plotter is available for exporting data in raster files of supported formats:



### ***Internal DWF and DWFx Plotters***

The internal DWF and DWFx plotters are available.

### ***Isometric Views Printing***

Improved printing of isometric views when specifying the plot area with a rectangular frame.

### ***Verification of third-party print settings***

Verification of print settings made in third-party programs is performed. When these settings are detected, the **Print** command displays a message.


### ***Improved Print Preview***

It is now possible to specify the number of copies to print in the **Print Preview** dialog. The option is not available when you print to a file.

In the preview window:

- Increased display speed;
- Improved image quality;
- Eliminated artifacts when zooming, panning and resizing the preview window and the **Print** dialog.

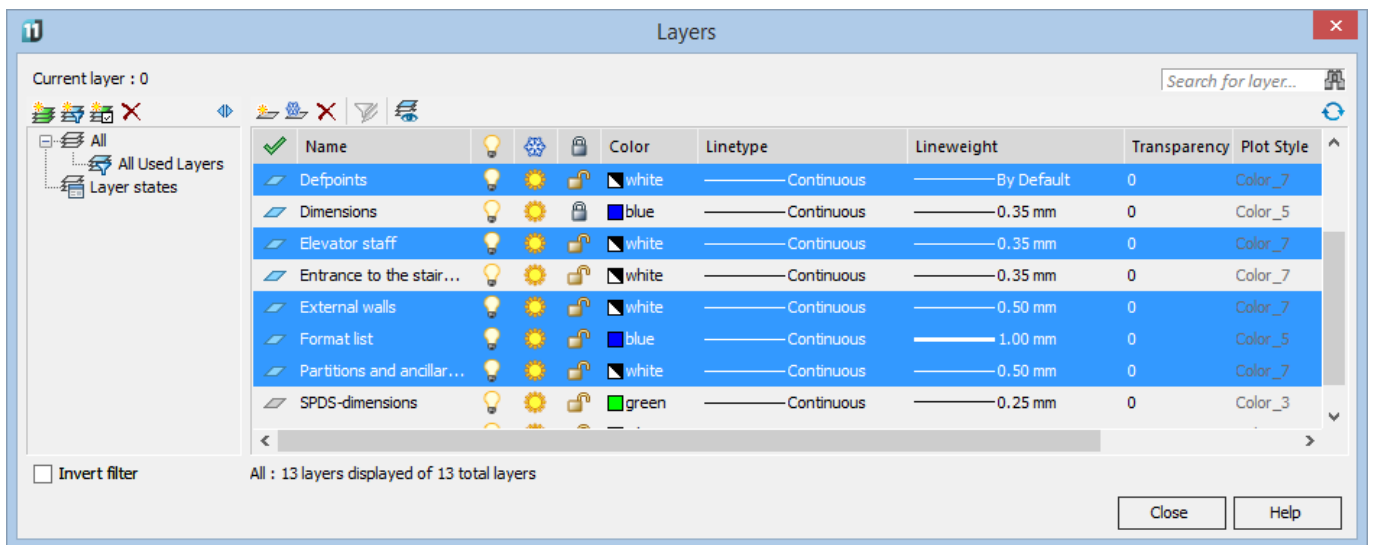
## Layers

The  “Previous State of Layers” command (LayerP) allows you to consistently return to the previous state of a layers.

### *SHIFT and CTRL Using*

The logic of selecting a group of layers with the SHIFT and CTRL keys in the **Layers** dialog was changed. Now the selection logic corresponds to standard logic implemented in Windows:

- use CTRL+click to add any individual layer in a selection set;
- use SHIFT+click to add group of layers in a selection set.



### *List of Layers Displaying Speed*

The display speed of the list of layers in the **Layers** dialog is significantly increased. Especially if there are hundreds of layers in the drawing.

## Check Standards

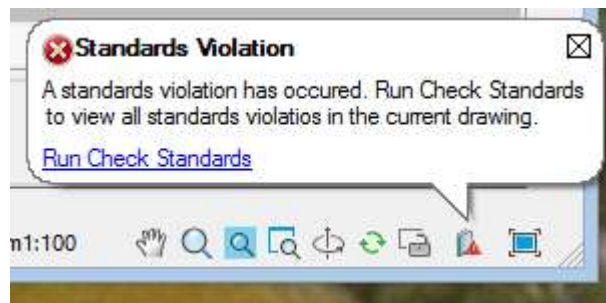
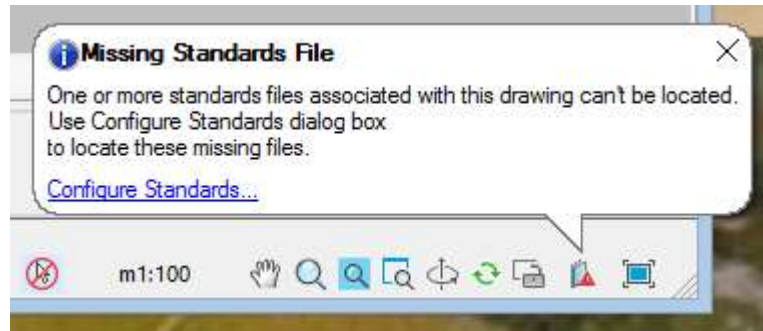
### *The “Standards Settings” Command*

The STANDARDSSETTINGS command is added to open the **Standards Settings** dialog.

### *Status Bar*

At now on, the button for checking CAD standards  is displayed in the status bar only if the standards file has been connected to the document.

A hyperlink is added in the balloons of the standards check button to open an appropriate dialog box.

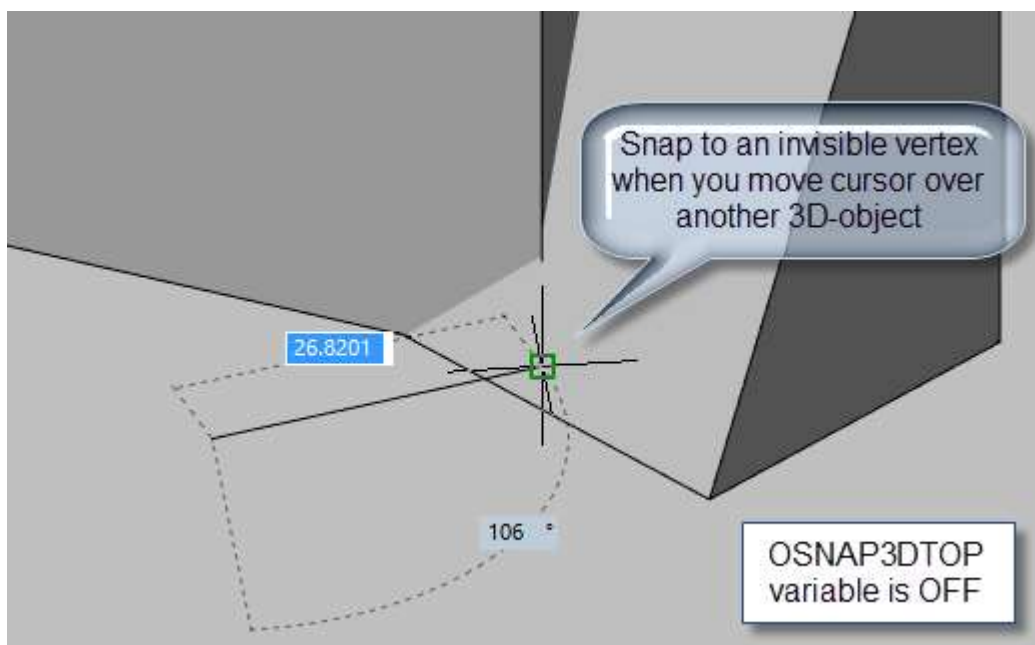


## ***Improved Snap Logic in 3D Space***

The snap behavior in the three-dimensional space was changed.

If a visual style with shaded faces is set and the OSNAP3DTOP variable is turned ON, then you can snap only to points of 3D-object behind the cursor.

If the wireframe visual style is set, or the OSNAP3DTOP variable is disabled (= OFF), snap will be performed to points of all objects in a drawing:



## ***The “Add Selected” Command Improvement***

The  ADDSELECTED command can now create multileaders and dwg-tables.

## The PELLIPSE Variable

New PELLIPSE variable determines the type of ellipse being created:

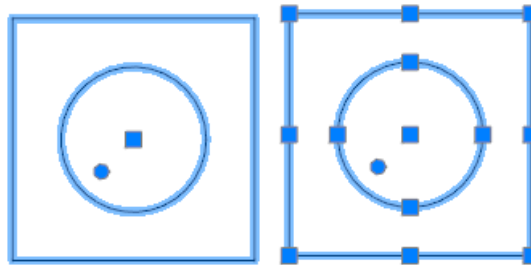
- if the value is 0, an ellipse object is created;
- when the value is 1, a 2D polyline is created instead of an ellipse object.

## Editing of the Region with Its Grips

Now you can edit the region contour with its grips.

2 modes of editing are available:

- the mode of moving the region by the central grip;
- mode of region contour editing with its grips



Modes are switched with a round grip.

## Clearing Invisible Proxy Objects from a Document

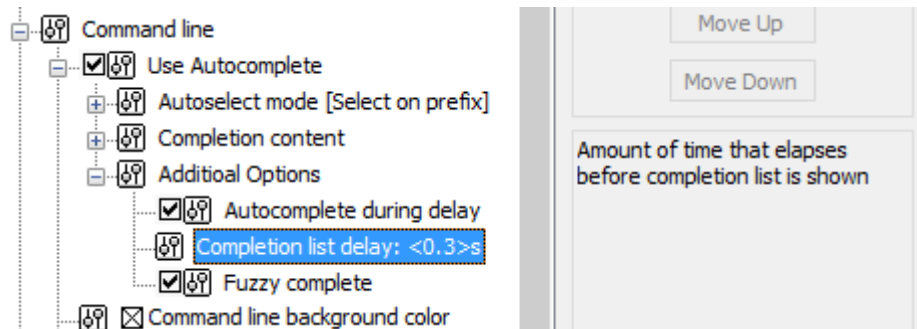
The RMPROXY command get the possibility to delete proxy objects that do not have a graphical representation.

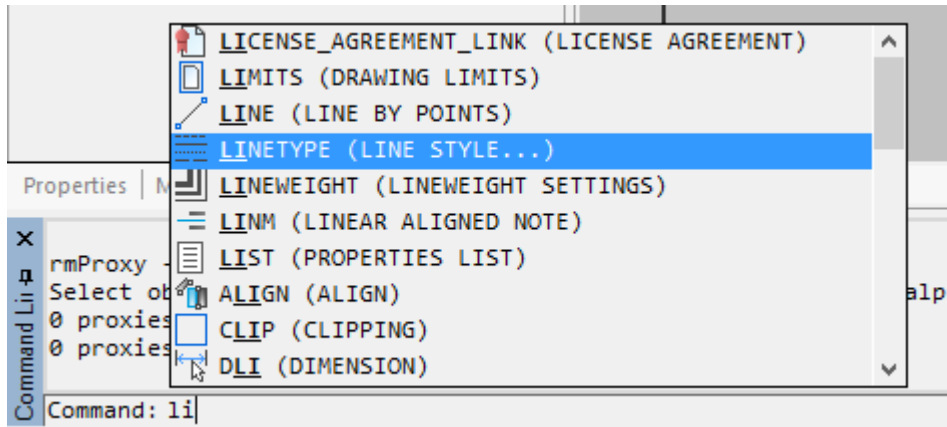
```
Command: RMPROXY
rmProxy - rmProxy
Select objects or [?/Drawing/Nongraphicalproxies]:
```

## Options

### Delays in Displaying the Command Completion List

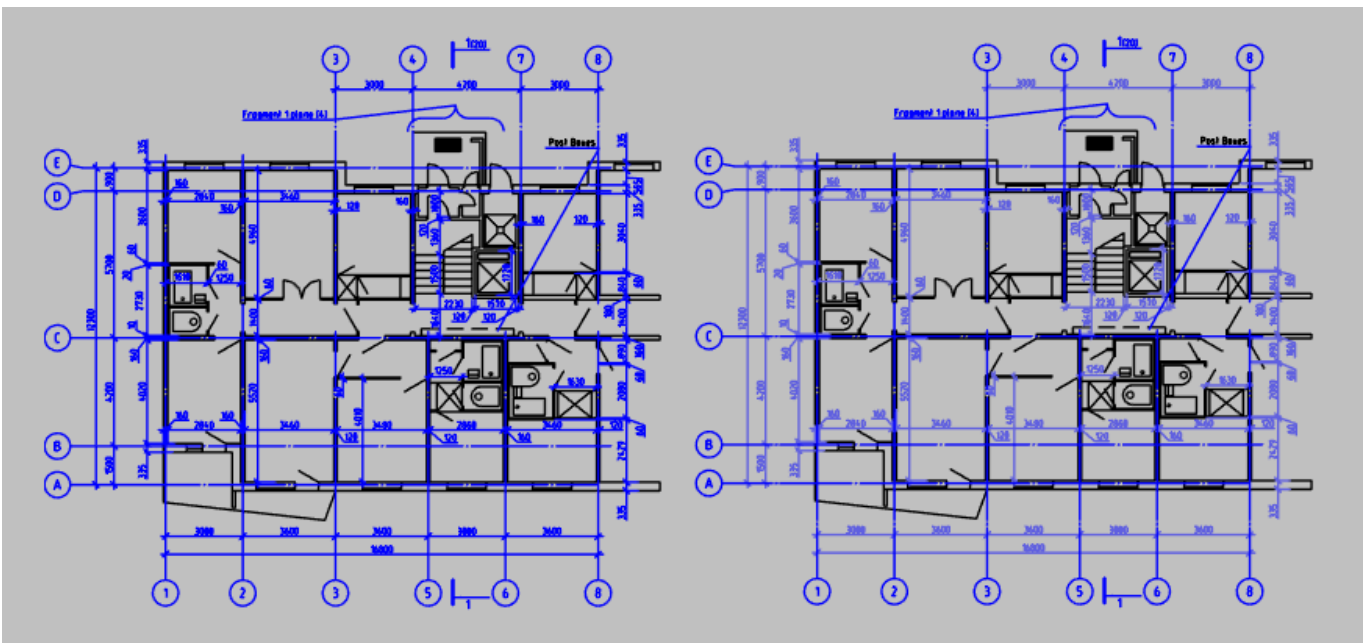
Amount of time that elapses before command line completion list is shown can be specified in the **Options** dialog: **Command line** > **Use Autocomplete Additional Options** > **Completion list delay**.



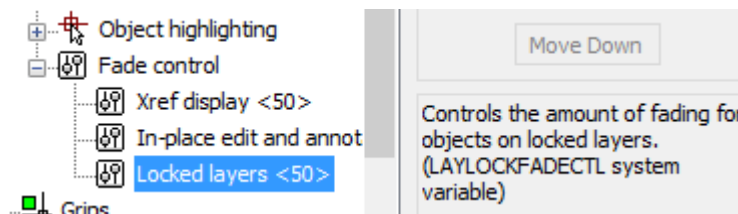


### The Value of Objects Fading on Locked Layers

The value of fading for objects on locked layers is defined by the LAYLOCKFADECTL variable.



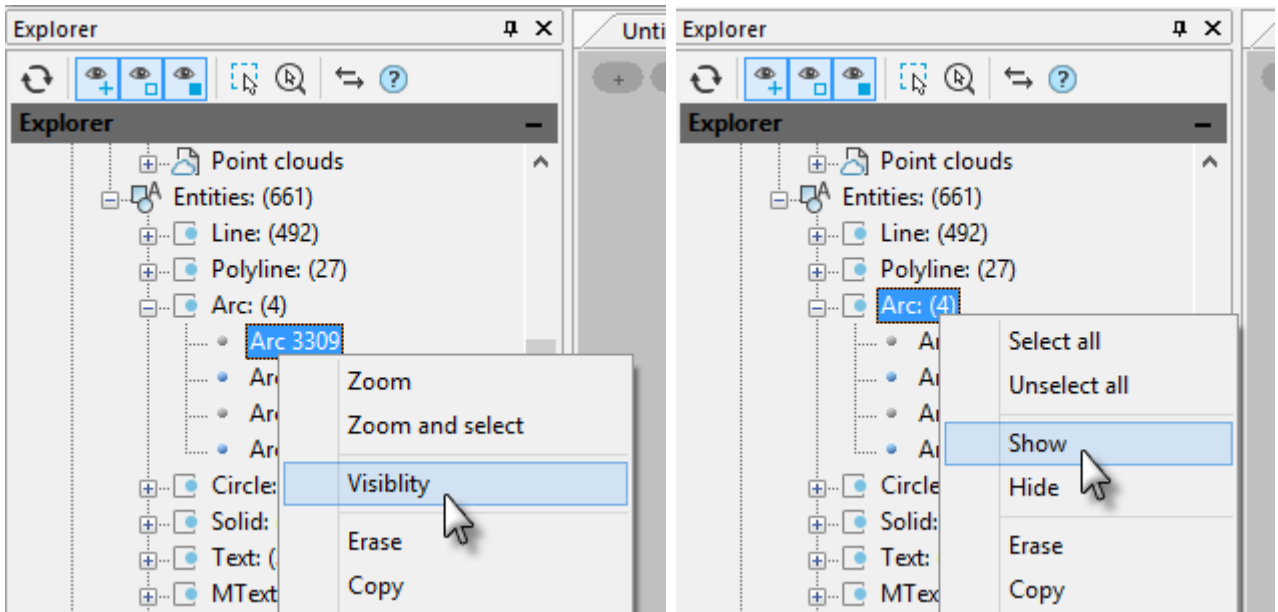
This value can also be changed in the **Options** dialog (**Select > Shading objects > Locked layers** item).



### Displaying of "Invisible" Objects

In a new version of the program, it became possible to make visible "hidden" objects – the objects with the "invisible" attribute. By selecting the appropriate option in the context menu of the **Drawing Manager**, you can make visible a single "invisible" object or a group of "invisible" objects:

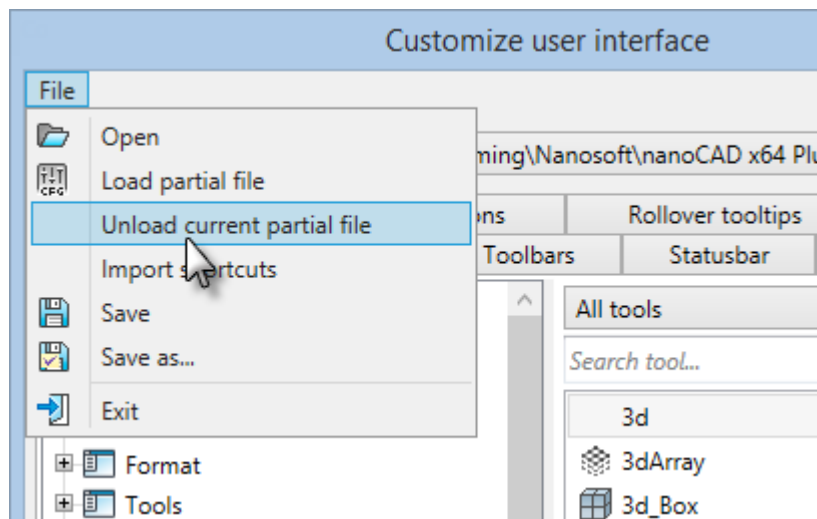




## User Interface Customization

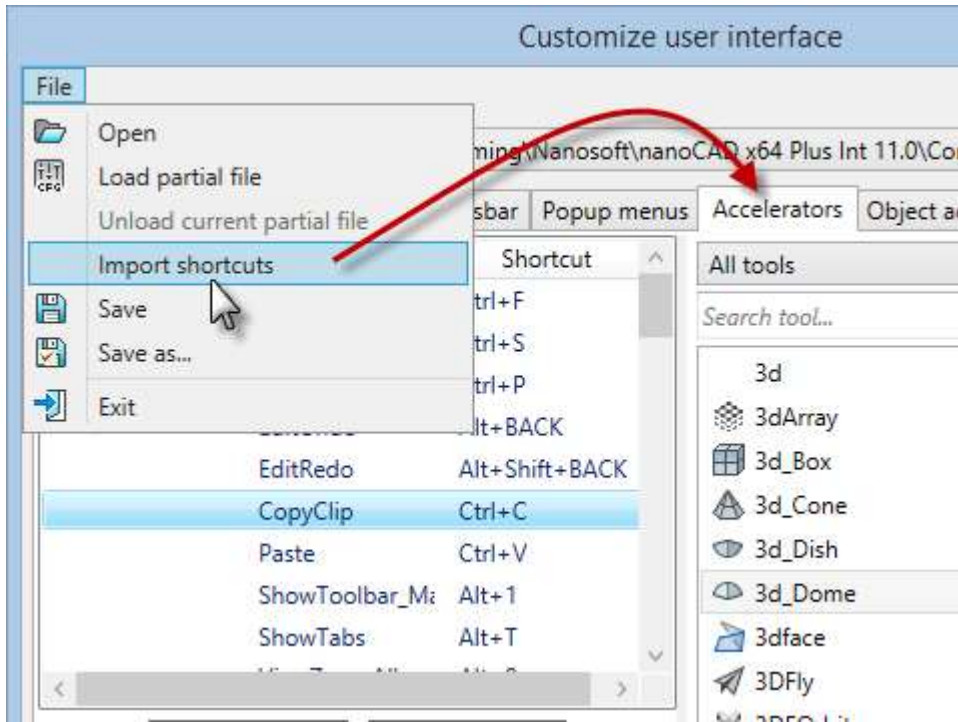
### Partial CFG-file Unloading

In the  **Customize User Interface** dialog, you can unload the current partial configuration file.



### Import Shortcuts

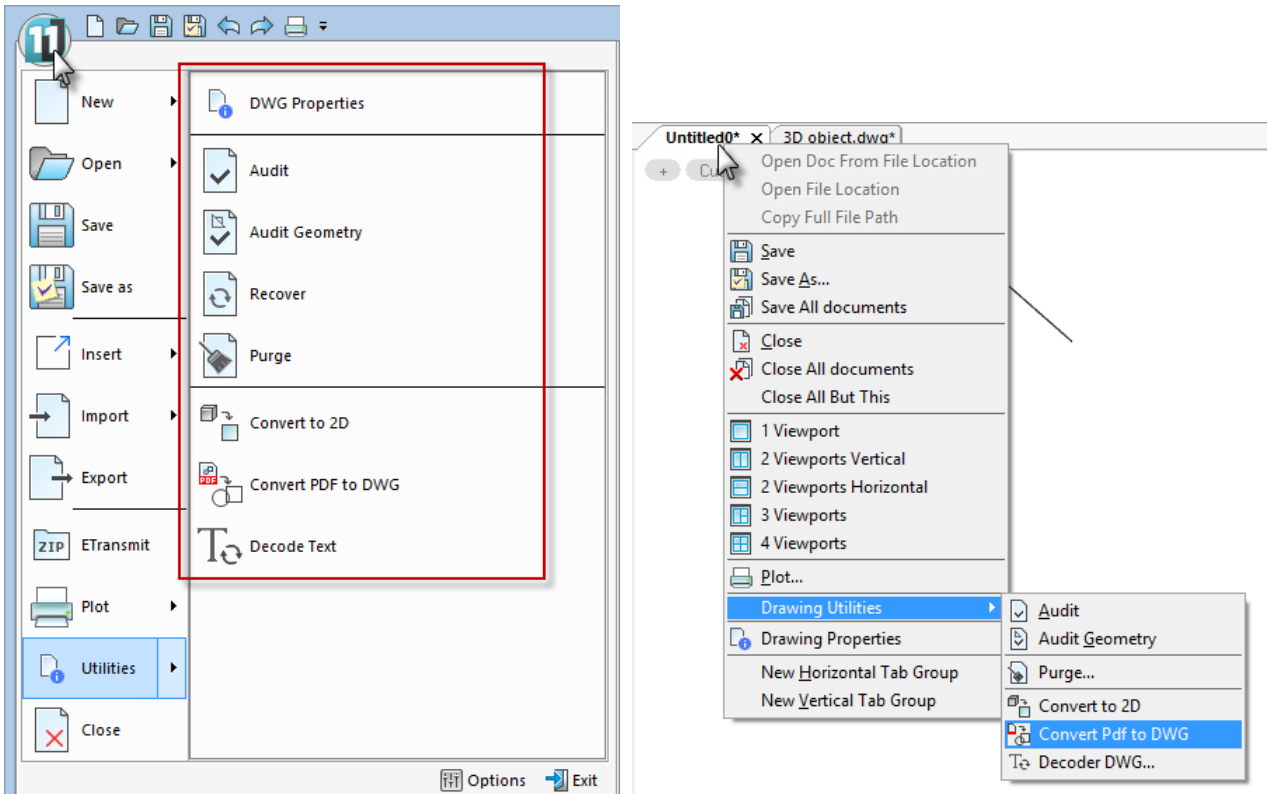
In the **Customize User Interface** dialog, the ability to import only keyboard shortcuts from .cuix and .cfg files has been added.



## User Interface Changes

### Utilities in nanoCAD Menu

The command for displaying the properties of a drawing, as well as the utilities for auditing, checking and working with a document, is added to the application menu and the context menu of the document tab.



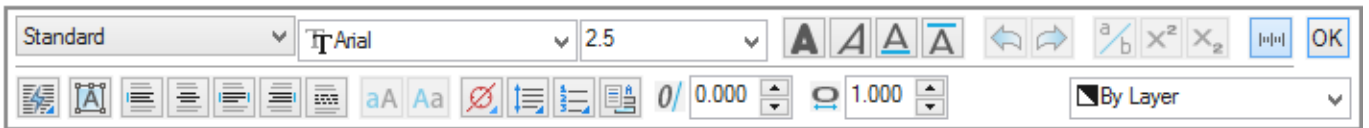
### Multiline Attributes Editing Panel

The ATTIPE system variable is used to change the **Text Format** panel that appears to create/edit multiline attributes.

- If ATTIPE = 0, the short variant of formatting panel is displayed:



- If ATTIPE = 1, the complete panel is used:



### Other Changes

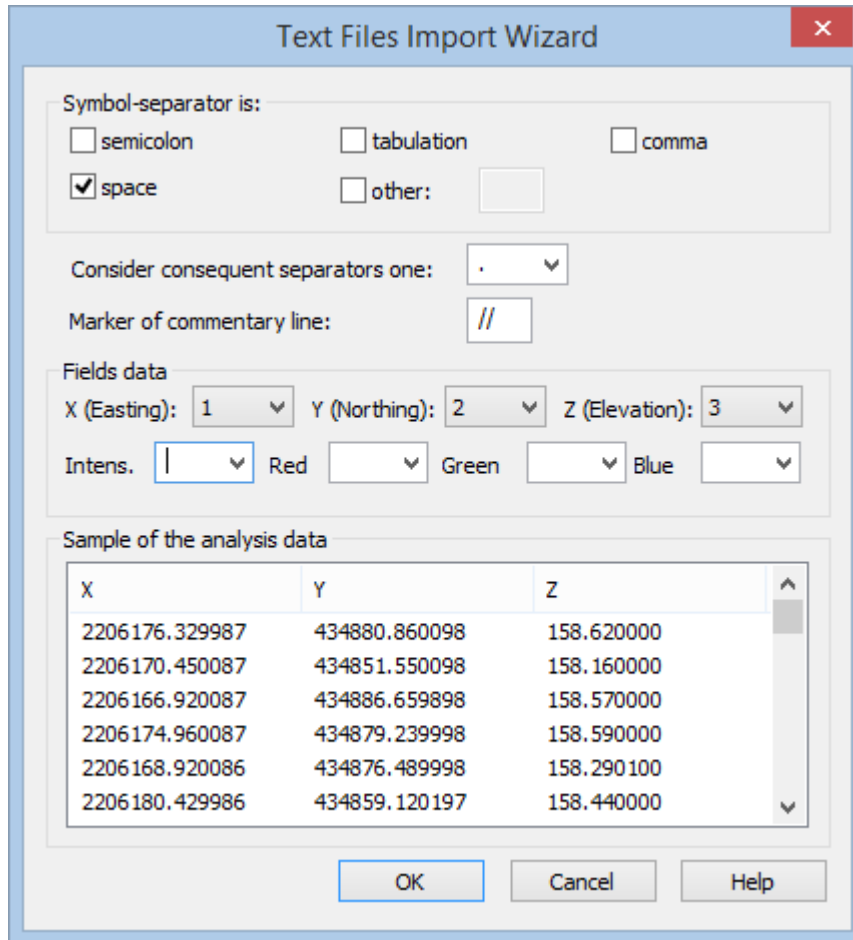
- The CTRL+W key combination is used to display or hide the **Select Objects** dialog;
- Dynamic input is available when creating a rectangle by 2 points;
- A new toolbar **Tools**:



- A large design work is continued in a new version of nanoCAD that improved the graphical interface of the program: the next package of icons was reworked; a number of improvements were made to the **Graphite** and the **Light** application visual styles.

### Text Files Import Wizard for Point Clouds

A special import wizard dialog is displayed when importing point clouds from text formats. It allows you to correctly interpret the point clouds data.



## Export

Export to **DWF**, **DWFX**, **STL** and **3D PDF** file formats is available.

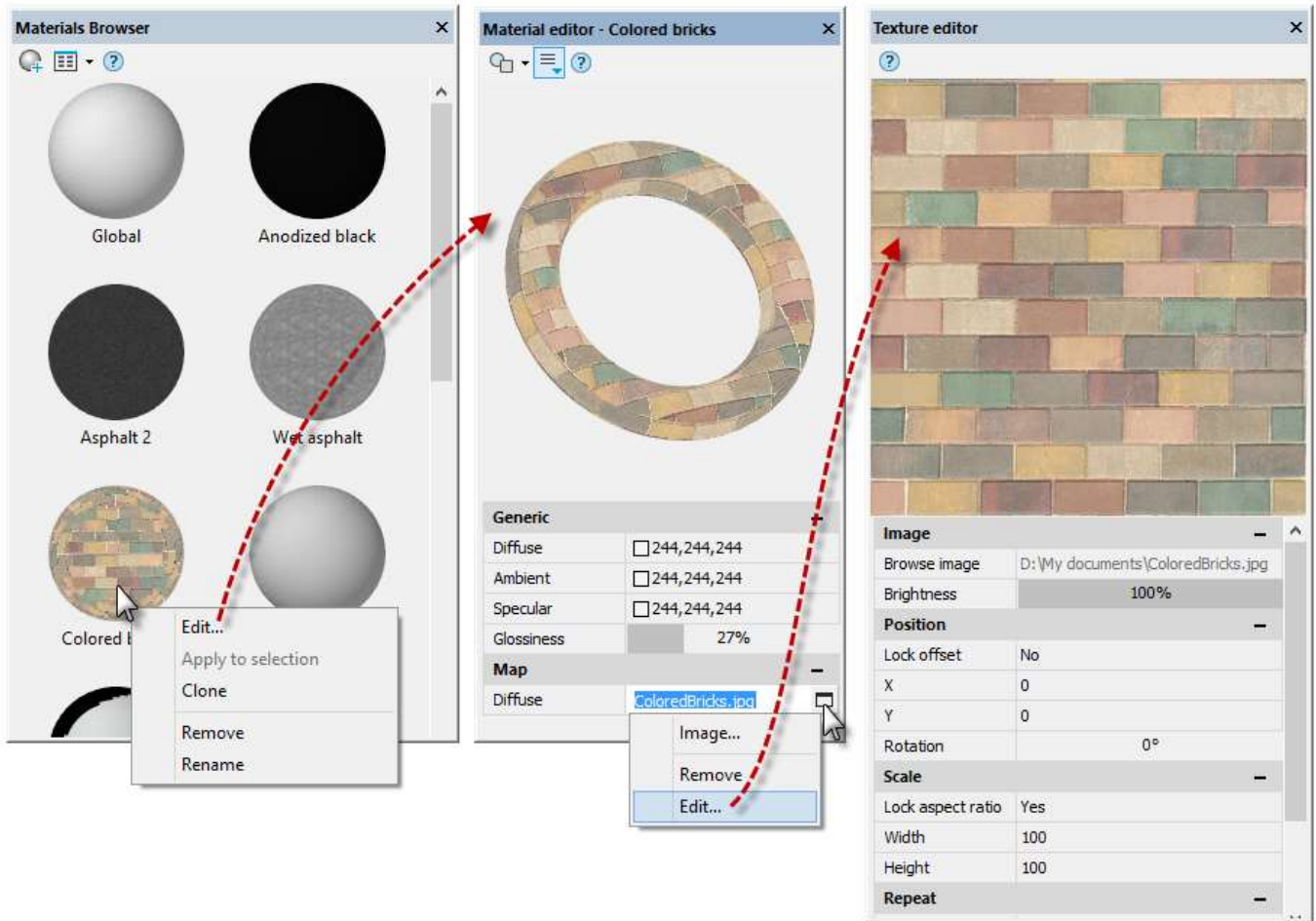


The **Export Point Cloud** command (NPC\_EXPORT) can export point clouds in a PLY (Polygon File Format) format. This one was developed to store 3D scanner data. The format allows you to store many properties of the object (color, transparency, normals, texture coordinates, etc.).

## Material Editor

The possibilities of editing materials have been extended:

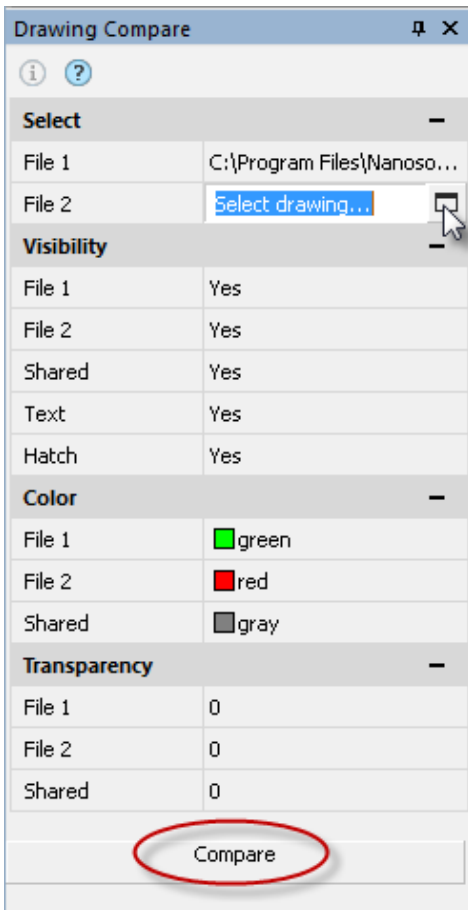
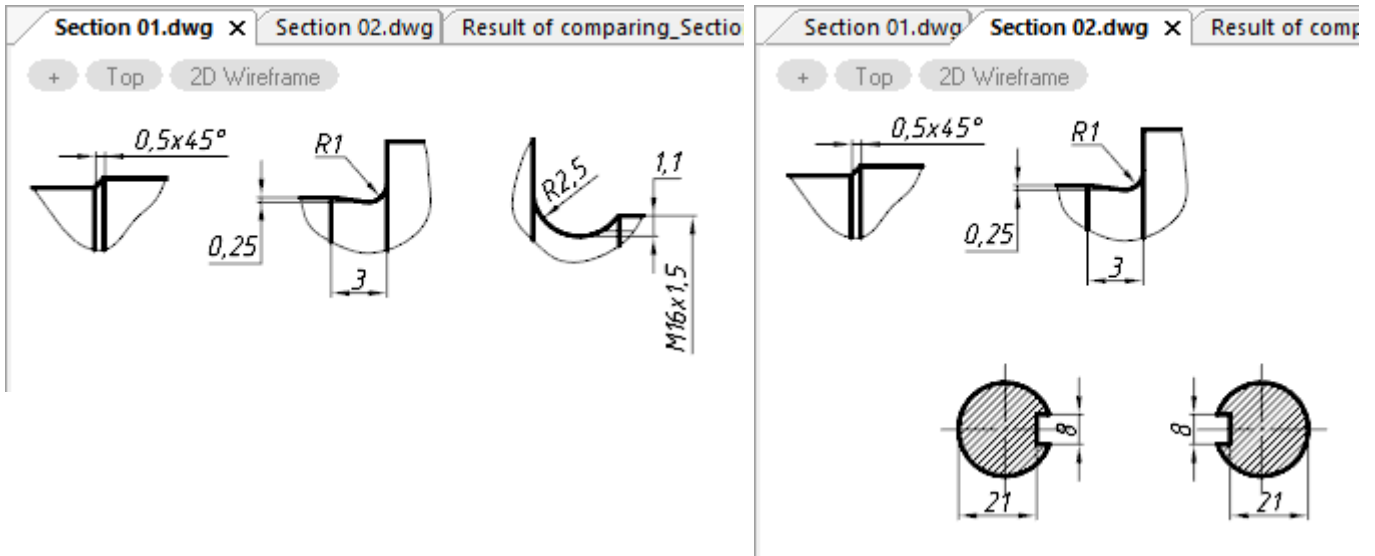
- the functionality was separated into three functional bars: **Material Browser**, **Material Editor** and **Texture Editor**;
- editing of material texture parameters is available.




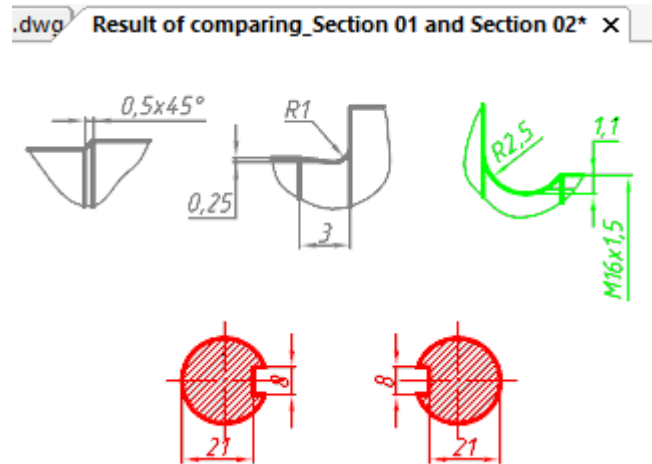
## *New Features of nanoCAD 20.0 Platform*

### *Drawing Comparison*


File comparison feature allows you to find the differences between the contents of the model space in two drawings: detect objects that have been modified, added, or removed in the compared drawings.

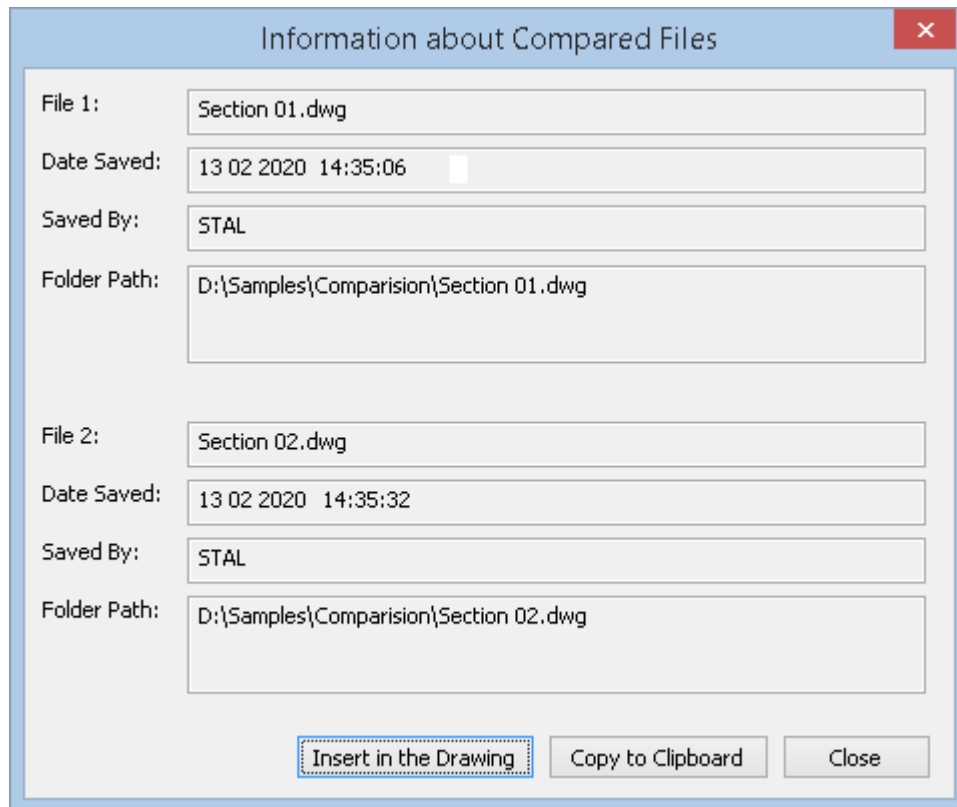


The  **Compare Files** (DWGCOMPARE) command (the **Functional Bars** section on the **Options** tab) displays a functional panel with settings and a launch button.



The feature creates a new dwg file that displays similar and different objects of the compared files, coloring them in several colors. In addition, you can switch off the displaying of unnecessary objects or set a transparency value for them.

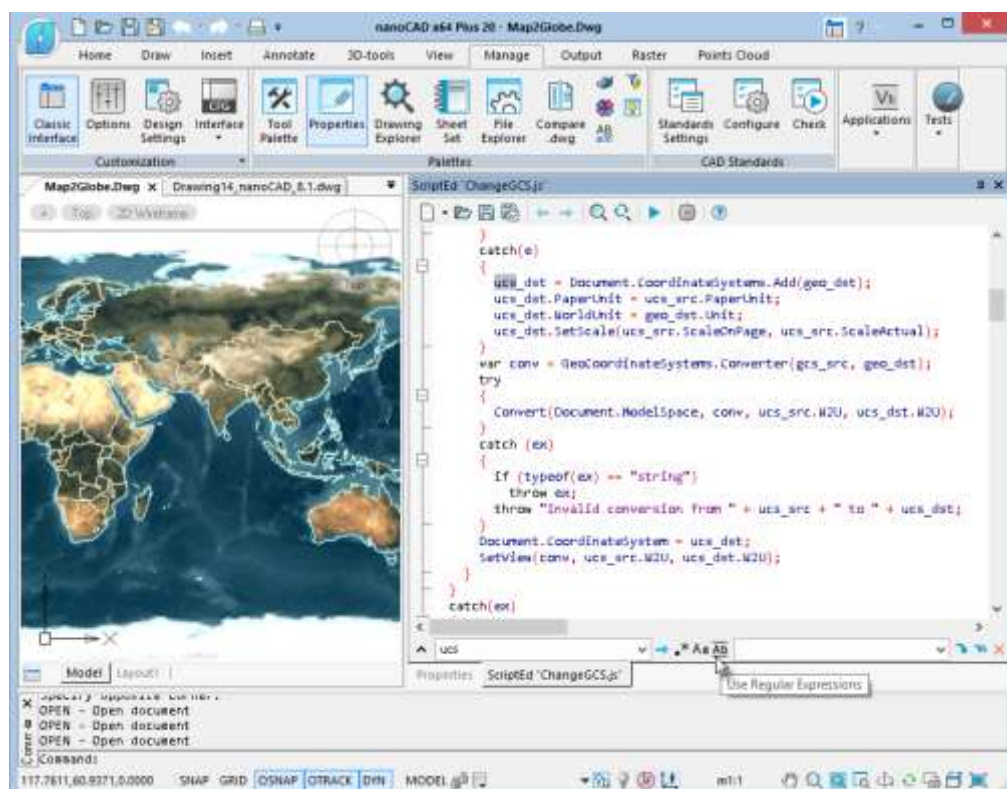
Click the  icon to open a dialog with information about the compared files:



## Built-in Script Editor




The built-in **Script Editor** (SCRIPTED) allows you to create, edit and run script files of various formats: ActiveX (.JS, .VBS), LISP (.LSP), DCL, SCR.

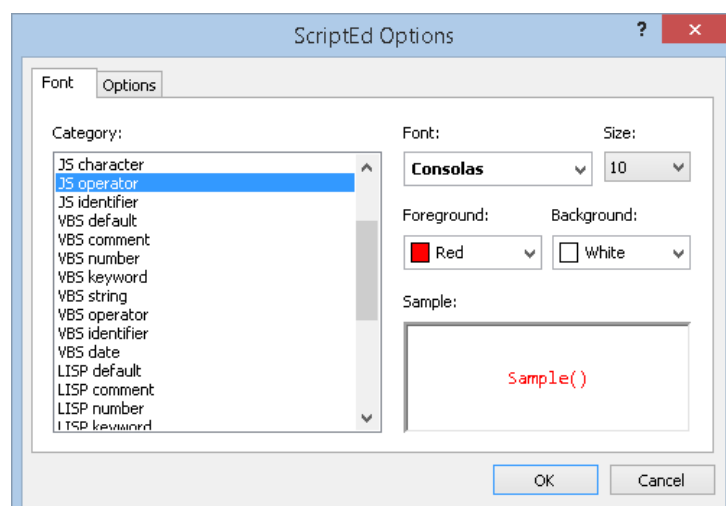


You can use the following keyboard shortcuts in the editor:

Action	Shortcut
Select All	Ctrl+A
Copy	Ctrl+C, Ctrl+Ins
Insert	Ctrl+V, Shift+Ins
Cut	Ctrl+X, Shift+Del
Undo	Ctrl+Z, Alt+BackSpace
Redo	Ctrl+Y
Go to String	Ctrl+G
Find	Ctrl+F
Find Next	F3
Find Selected	Ctrl+F3
Replace	Ctrl+H
Open	Ctrl+O
Save	Ctrl+S
Save As	Ctrl+Alt+S
Properties	Ctrl+P
Run	Ctrl+R
Find the Matching Pair Bracket	Ctrl+{


For LISP scripts, an executing of the selected part of the code is implemented. To do this, select the necessary LISP code in the editor and click the  **Run script** button with the **CTRL** key pressed.

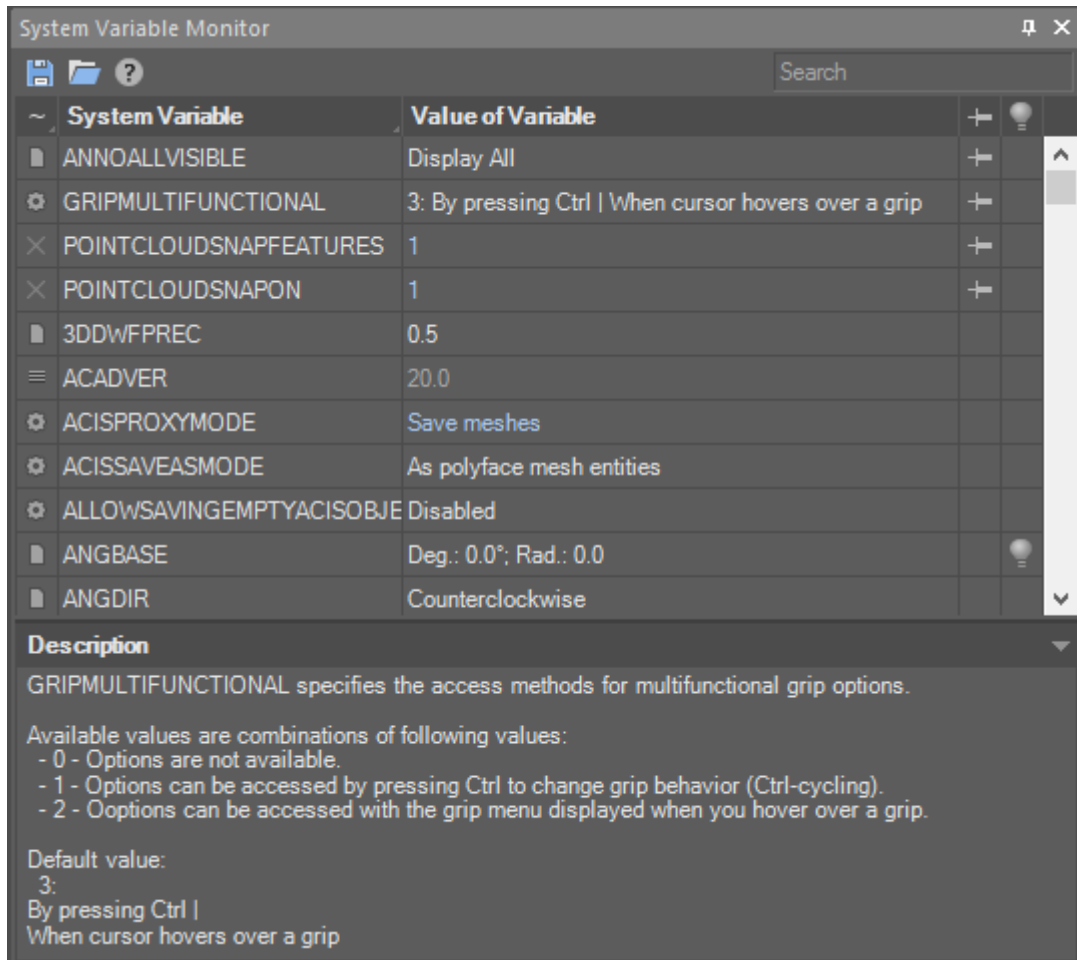
Editor's settings allow you to adjust the display and formatting of individual language elements.









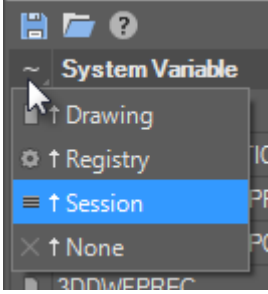
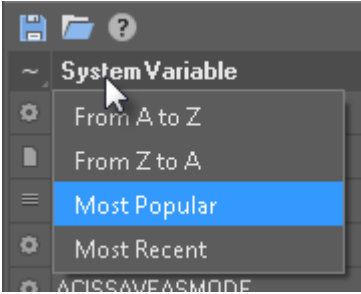
## System Variables Monitor

The  System variables monitor (SYSVARMONITOR) displays system variables, their brief description and values, allowing you to edit, sort, search them and track changes in real time.



Element	Action
	Save variables to a file and load from the file. It is possible to save both all and only pinned variables.
<input type="text" value="Search"/>	Search for variables by name.

Column	Value
	Storage location: <ul style="list-style-type: none"> <li> - the variable is stored in the registry. Always retains its value.</li> <li> - the variable is stored in the document. Its value changes only for a specific document.</li> </ul>

	<ul style="list-style-type: none"> <li>☰ - the value of the variable is saved only for the current session. After closing the program, the value is reset.</li> <li>✕ - no storage place. The value is stored until the document is closed.</li> </ul> <p>Sorting is performed by clicking on the column heading or by selecting an item in the menu, which opens by a long click on the heading.</p> 
<p><b>System Variable</b></p>	<p>The name of the system variable.</p>  <p>Sorting is performed by clicking on the column heading or by selecting an item in the menu, which opens by a long click on the heading.</p> <p>In addition to the direct and reverse alphabetical order, the variables can be sorted by frequency of use (<b>More popular</b>) and time of change (<b>Most recently</b>) during an active session of the program.</p>
<p><b>Value of Variable</b></p>	<p>The value of the system variable, which can be entered or selected from a list of fixed variants. Gray (muted) typeface indicates variables with constant values. If the value of a variable differs from its initial value, it is displayed in blue font. In most cases, the initial value is given in the variable description at the bottom section of the panel.</p>
<p>+</p>	<p>Click in this column opposite the variable to fix the variable at the top of the list, regardless of the sorting method. The variable moves to the top of the list, the + icon appears in front of it.</p>
<p>💡</p>	<p>Click in this column opposite the variable to start tracking the change of the variable. Turned off lightbulb 🚫💡 will appear, meaning that the variable has not yet been changed since the start of its tracking. If the value of the variable is changed (by a command or by a user), the bulb will "light up": 💡.</p>

## Undo/Redo Evolution

### Undo after Saving

It became possible to undo the operations performed after saving a document. In previous versions of the program, saving the document clears the list of operations of the **Undo** command.

Use the checkbox **Undo command settings > Clear after save** in the **Options** dialog.

### Preview for Undo/Redo

A preview of result of undoing (and redoing) operations was implemented. When you move cursor to a command item in the undo (/redo) list, a preview of the resulting scene is displayed in the drawing area, which will appear after undoing all operations, up to the specified command.

Undo/redo preview can be disabled with the checkbox **Undo command settings > Generate preview** in the **Options** dialog.

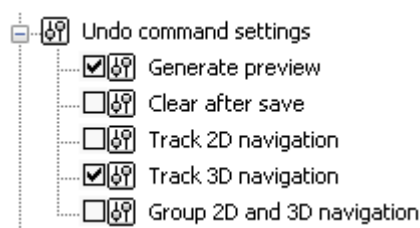
### Undo for Navigation Commands

In this version you can include the following actions in the undo / redo list:

- 2D navigation commands: panning and zooming operations, including those performed with the mouse.
- 3D navigation commands: Orbit commands (including performed in transparent mode with the mouse), **3D Walk**, **3D Fly**, Locator using.

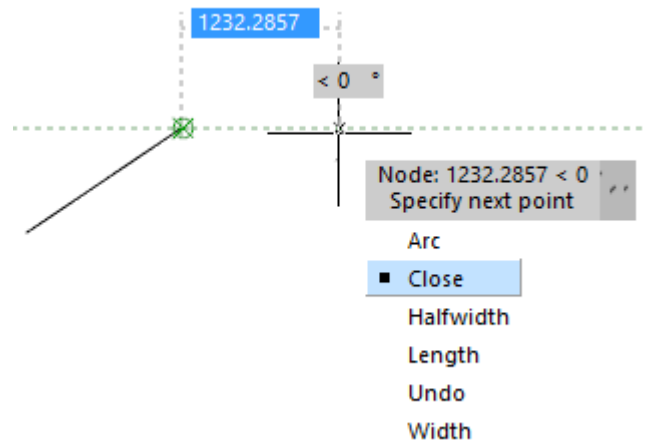
You can also group sequentially performed 3D and 2D navigation operations into one action to undo.

Configure these actions in the **Undo command settings** section of the **Options** dialog.



## Dynamic Input

Implemented dynamic input mechanism.

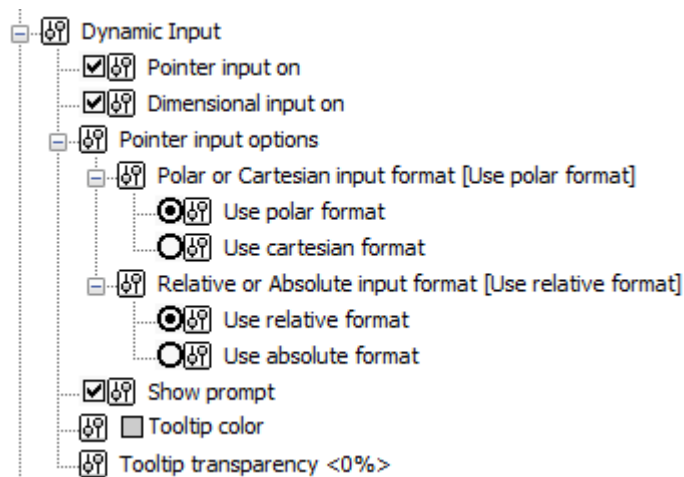


Dynamic input displays the command interface near the cursor in the drawing area. At the right time, current prompts, command requests, input fields, lists of selectable keywords, dimensions of edited geometry are displayed next to the cursor.

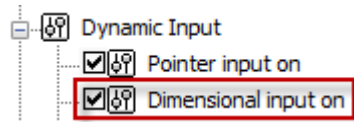
Dynamic input consists of three functional parts:

- Pointer Input;
- Dimension Input;
- Dynamic Prompts.

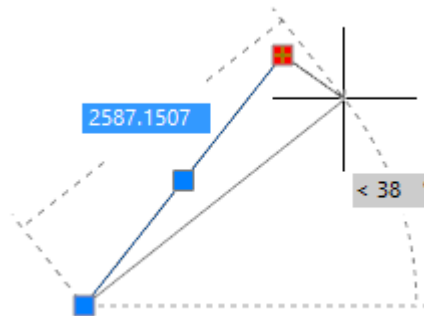
You can adjust them or switch them off in the **Dynamic Input** section of the **Options** dialog.



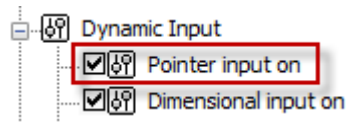
### Dimension Input



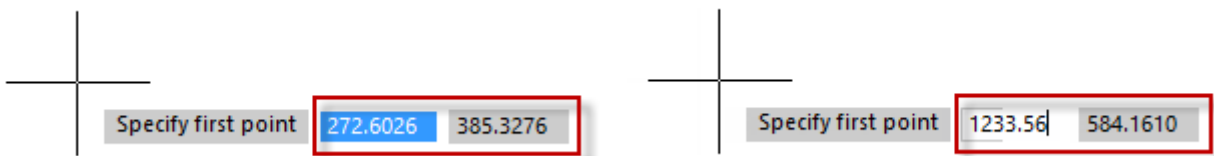
When you specify next point for a linear or an arc segment or a circle, the linear and angular dimensions are displayed on the screen relative to the previous point. These dimensions look like a dashed rubber line with a numerical field. As the cursor moves in the drawing area, dynamically displayed dimensions are constantly updated to reflect current values. By toggling the **TAB** key, switch to the required dimension value for editing it from the keyboard.



### Pointer Input



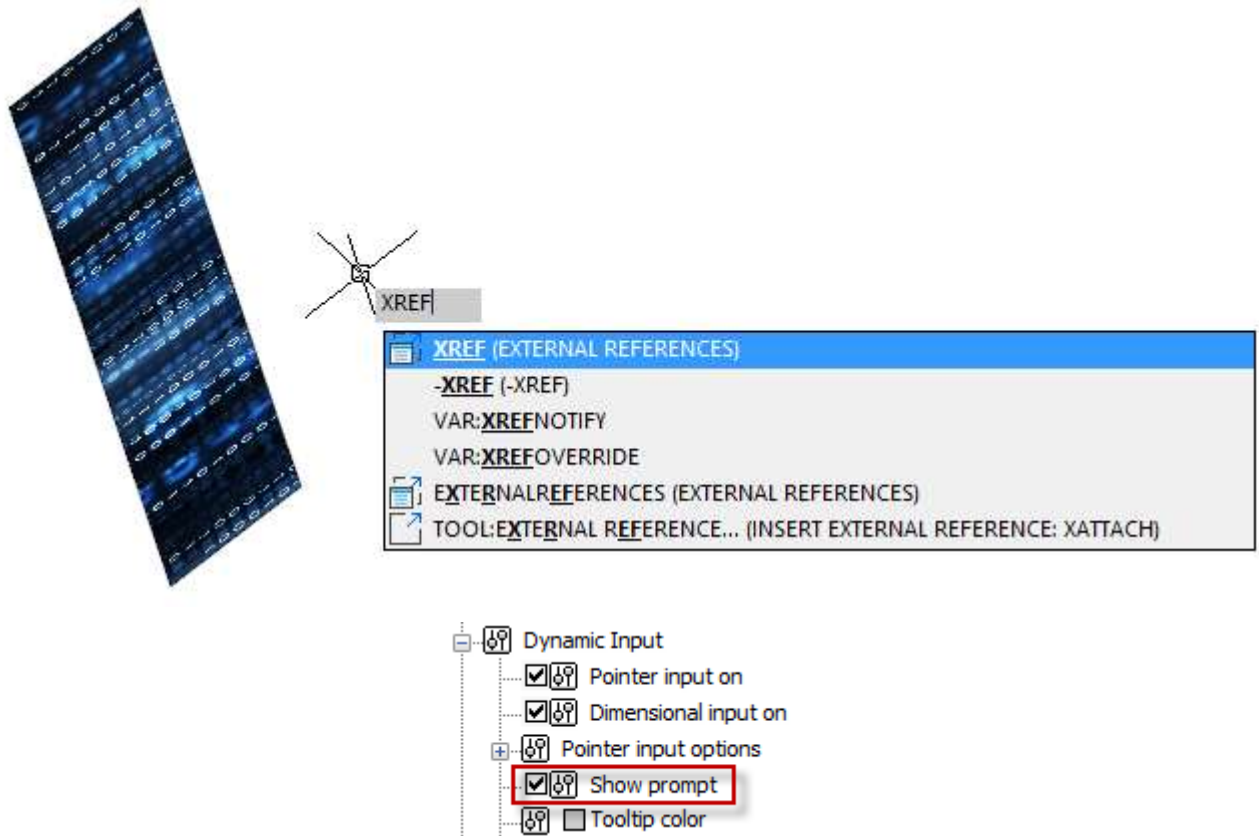
When a point is requested by a command, you can enter its coordinates in the tooltip box near the crosshair, where the current cursor coordinates are shown.



By toggling the **TAB** key, switch to necessary value to edit it from the keyboard. Press **ENTER** when done.

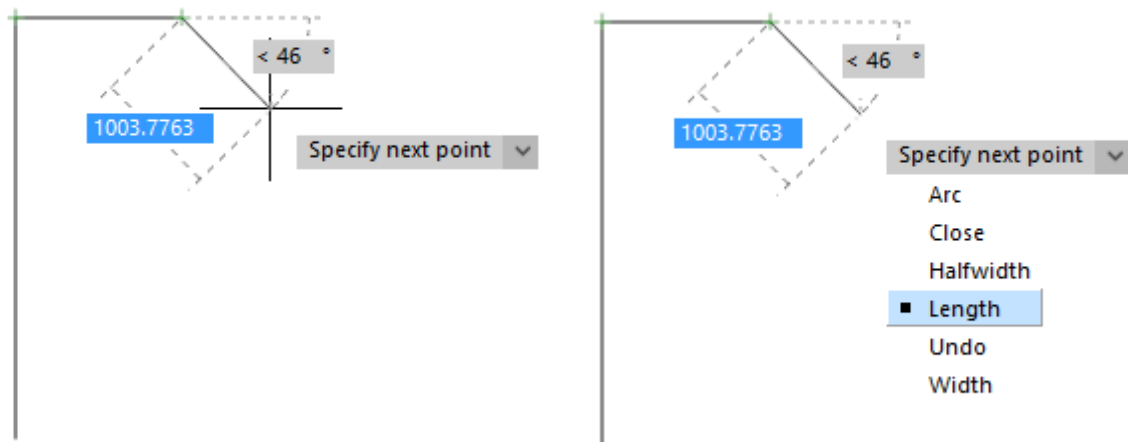


## Dynamic Prompts



Dynamic prompts provide an alternative way to enter command parameter values. In the tooltip field near the cursor, requests and prompts of the active command appear, necessary to complete the command. Data can be entered in the dynamic prompt fields instead of the command line. You can enter command keywords and requested numeric values.


The list of available values and options can be expanded by pressing the **DOWN ARROW** key on the keyboard. Then choose the desired value with an **ARROW** keys and press **ENTER**.

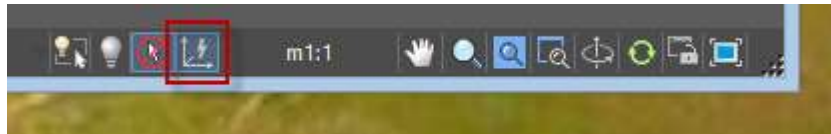


Dynamic input can be disabled by using a button **DYN** in the status bar or by pressing **F12**.

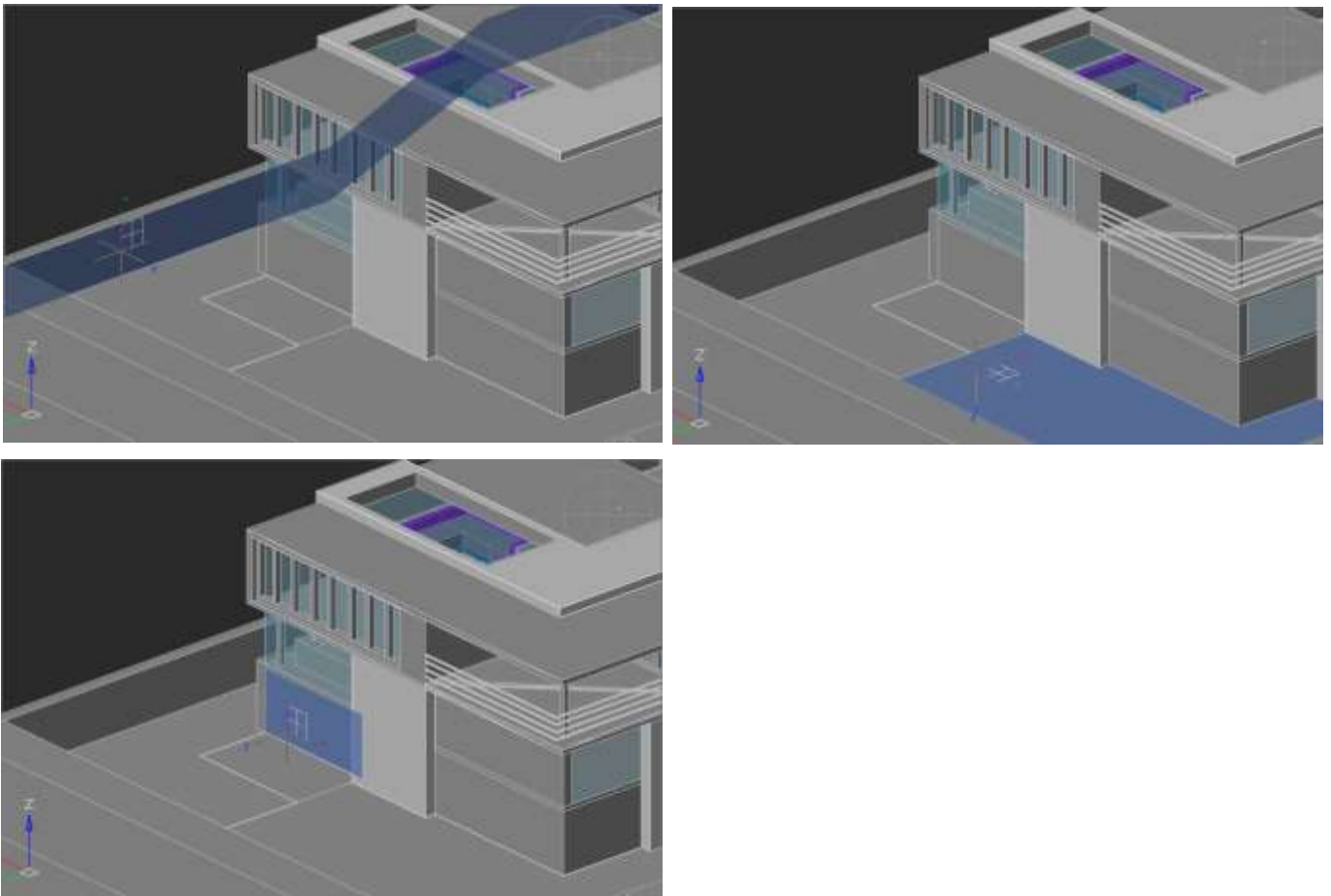
## Dynamic UCS

From now on, when you create an object and move the cursor over a flat segment, the UCS temporarily aligns with it. This makes it possible to immediately draw in the plane of the highlighted face without additional execution of change-UCS commands.

This feature is available in the **Dynamic UCS** mode. It is enabled by default, but can be disabled by the UCSDETECT variable or by a button in the status bar: .

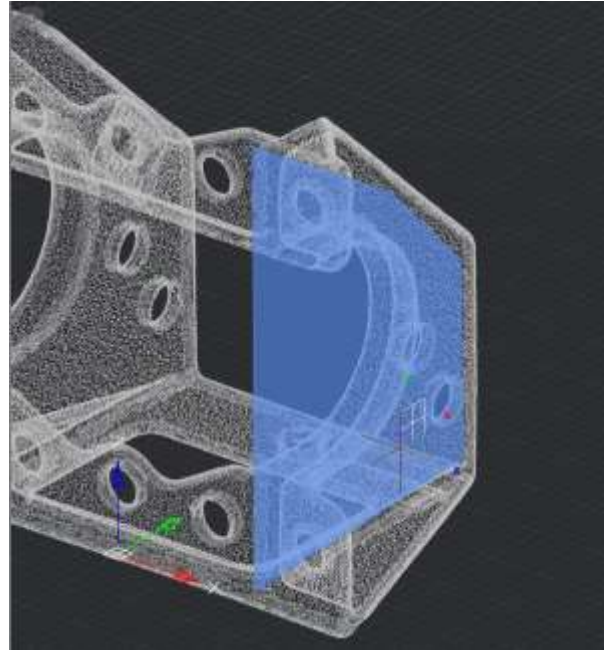


In the **Dynamic UCS** mode, UCS changes its orientation when you move cursor over a face of a 3D solid or a flat segment of a point cloud.



The Dynamic UCS works on point clouds with surfaces, previously recognized by

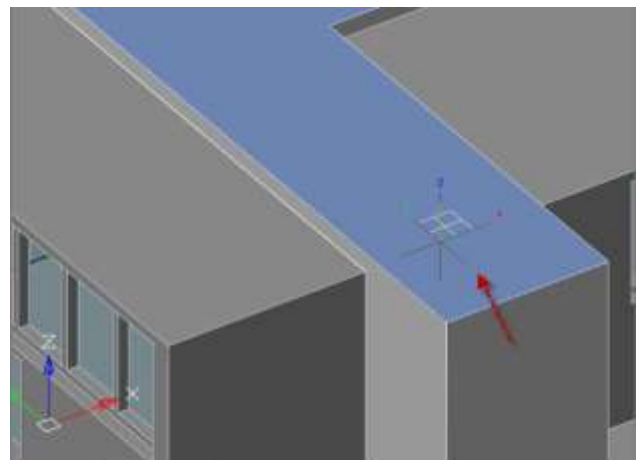
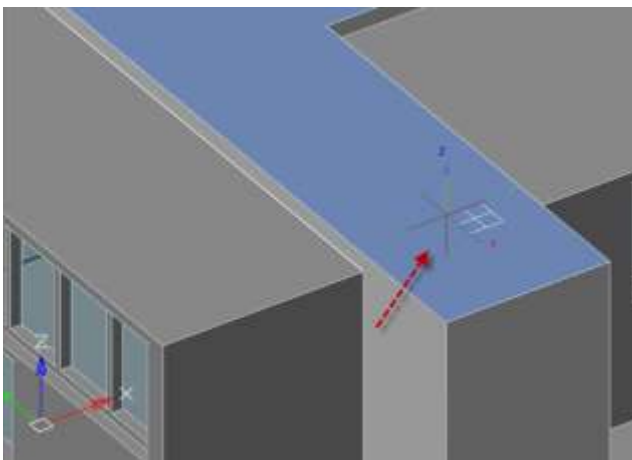
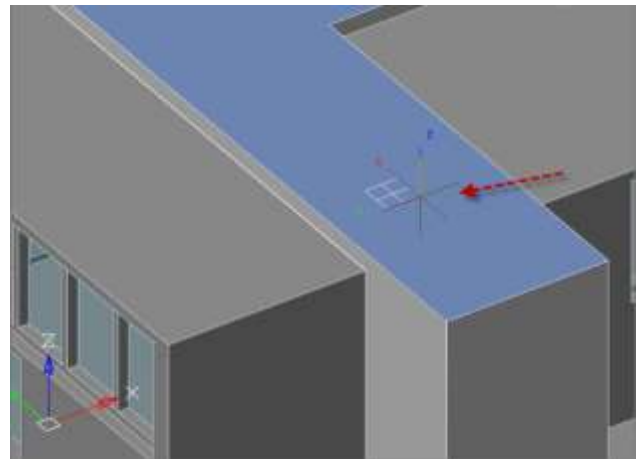
the  **Search Planes in Point Clouds** command.



The direction of the UCS axes depends on the edge that the cursor crossed when moving to the face.

So, the X axis is set parallel to the crossed edge in the direction from the initial vertex of the crossed edge. The Y axis perpendicular to the X axis and is directed toward the inner part of the face. Z axis is set so that the right coordinate system is obtained.

When you move cursor toward the same face through another edge, the orientation of the UCS axes also changes.



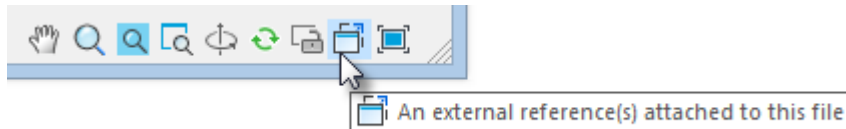
At the first mouse click, the UCS is fixed and the first vertex of the created object is set. Then you can continue to draw in the selected plane. Upon exiting the object creation command, the UCS is reset to its original orientation.



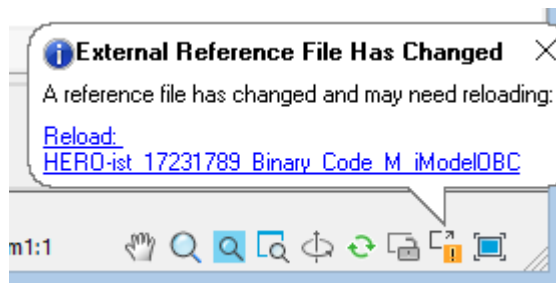
## External References Notification

While working in a document, changes in the files of external references made by external programs are tracked. Changes in the files of external references (xrefs), underlays, and raster images are tracked.

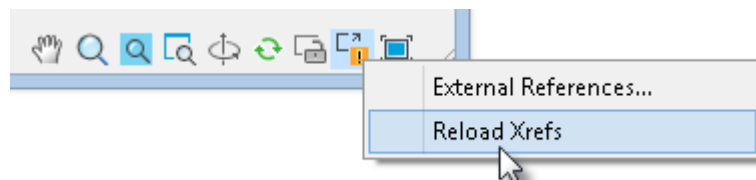
The status bar displays an indicator button of the status of external links in the active document.



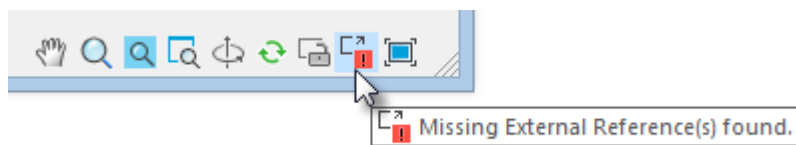
If an external link file was changed in a third-party application while working in the drawing, a warning message will appear and the indicator will change its status.



Right-clicking on the button opens a context menu from where you can call the **External Links** dialog box or update the external links of the drawing.



If the external link file was not found at all, the message appears, and the indicator button turns red.



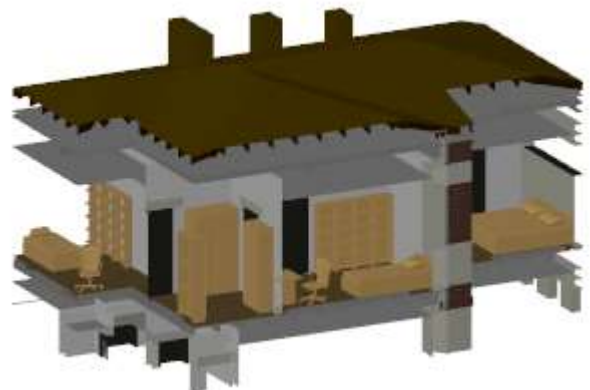
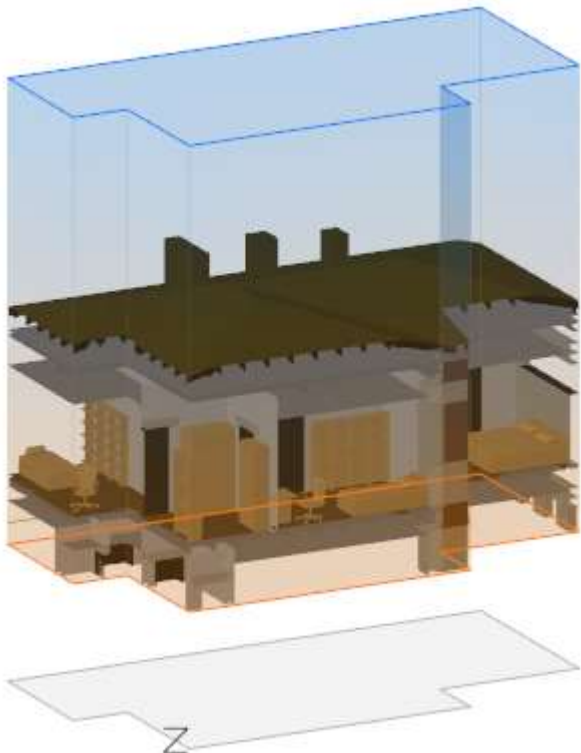
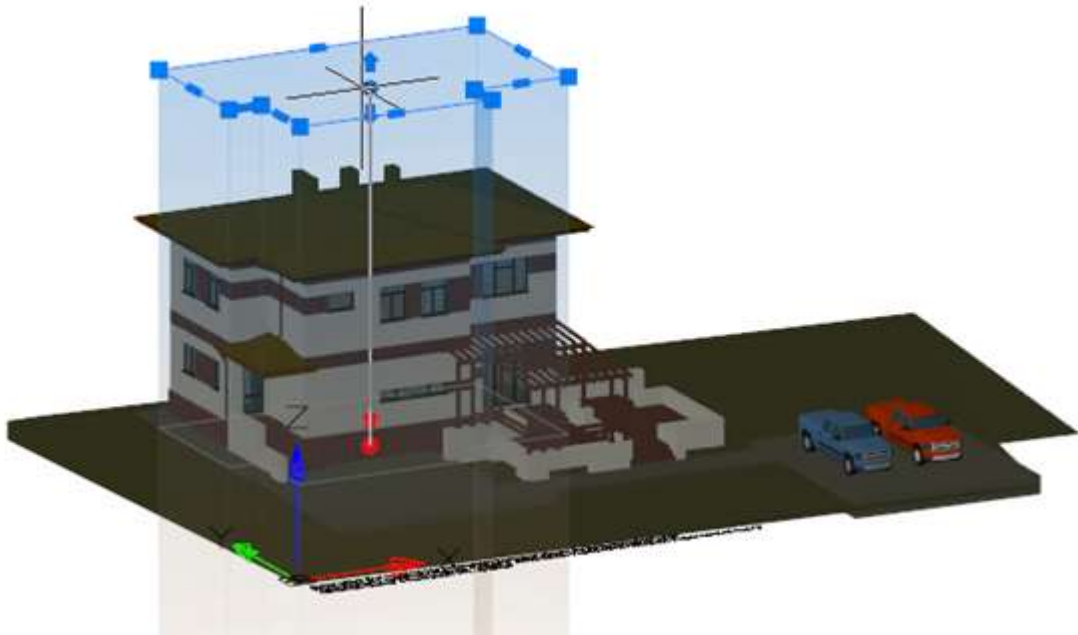
The XREFNOTIFY variable allows you to disable notification or change the type of notifications:

- 0 - disable notification of changes of external link files.
- 1 - display notifications in the dialog boxes.
- 2 - display notifications as pop-up balloons near the status bar.

The XNOTIFYTIME variable sets the time interval at which external link files are checked (in minutes).

## ***Bounding Prism – Viewport 3D Clip***

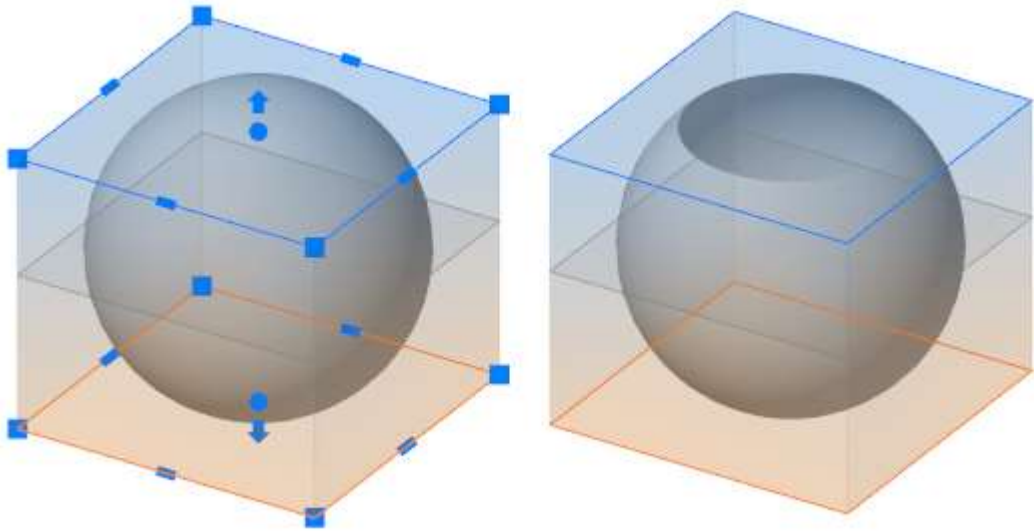
For each model space viewport, you can create a bounding prism – an area in a three-dimensional space, beyond which all the graphics are not displayed.



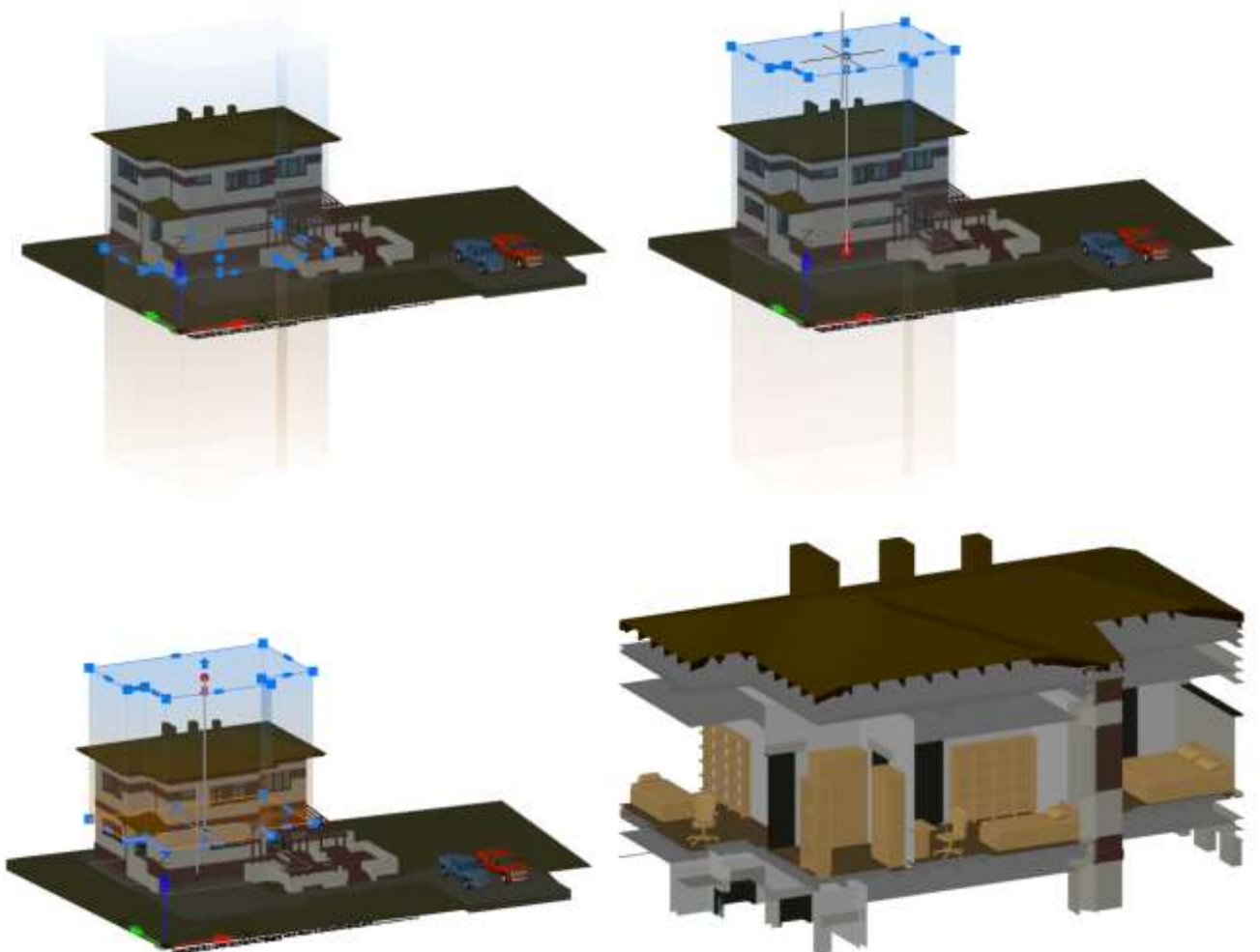
The area is defined as a rectangular or polygonal prism with grips. The prism can be:

1. created by specifying a contour on the screen,
2. created on the basis of an existing polyline,
3. based on a bounding box of selected objects.

The prism above the XY plane of the UCS is blue, below it is orange. The grey outline of the prism is displayed in the XY plane.



Drag grips to place the upper and the lower prism cutoff planes. Otherwise the upper and lower displaying area will be unlimited.



For each model space viewport, its own bounding prism should be created.



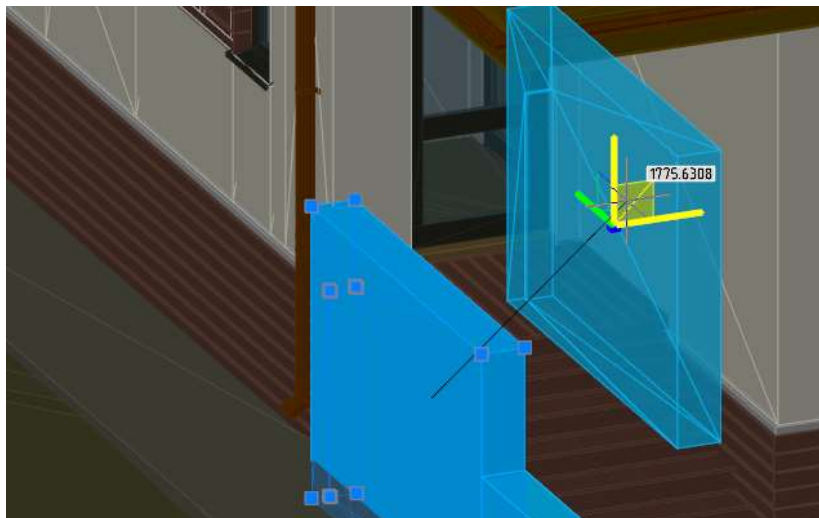
An existing prism can be changed, deleted or disabled/enabled. If the prism is disabled, graphics outside the prism become visible. Disabled prism object is still present in the model space and can be switched on later.



To create and modify bounding prism use the **Bounding Prism** (MCLIP) command (the **Viewport Tools** section of the **View** tab).

### *Semitransparency of Edited 3D-Objects*

Now, objects edited with the **Move**, **3DMove**, **Rotate**, **3DRotate**, **Scale**, **Stretch**, **Mirror**, editing by grips, drag&drop commands become semitransparent during editing for better positioning in space.



Disabled by the EDITTRANSPARENCY variable.

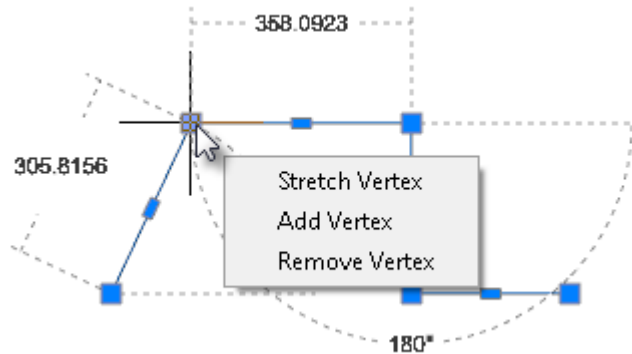
## Other Improvements

### Viewport Visual Style in Extended Named View

In this version, the Extended Named View keeps not only the section of the point cloud, but also the visual style of the viewport.

### Popup Menu for Multifunctional Grips

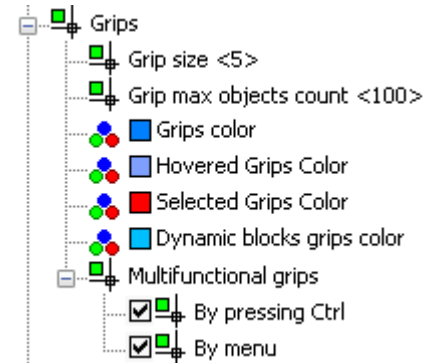
A pop-up menu with options for multi-function grips appears when you place cursor at an object grip.



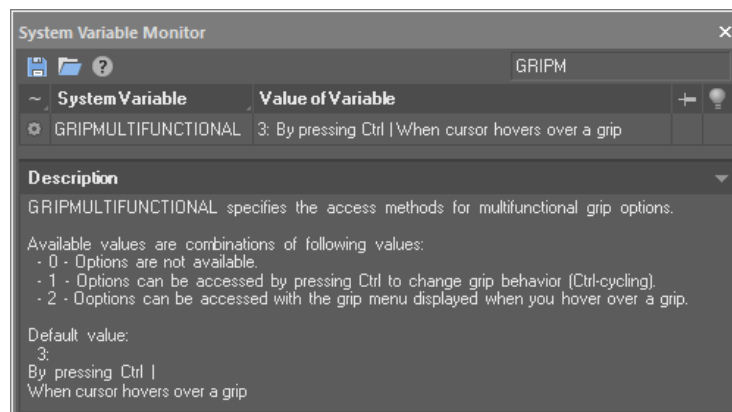
In the new version, the parameters of multifunction grips are available not only when the **CTRL** key is pressed repeatedly, they also appear in the pop-up menu when you place cursor on the grip.

The pop-up menu is implemented for grips of a line, arc, polyline, 3D polyline, hatch, multileader and spline object.

You can configure the way to access grip options in the **Handles > Multifunctional grips** section of the **Options** dialog.



These settings are accessed from scripts through the GRIPMULTIFUNCTIONAL variable.



### Greyscale Print Preview

When printing in greyscale, the preview window displays greyscale image also.

### **Coordinate Filters**

Coordinate filters are entered in response to a point request and define the axes along which coordinates will be set. You can enter the following filters: **.x**, **.y**, **.z**, **.xy**, **.xz** or **.yz**.

Coordinate filters allow you to input the coordinates of a point for each axis separately, specifying coordinates first along one axis, then along another.

Coordinate filters are useful when the value along one axis is determined by one characteristic point of the object, and along the other axis by another point. When using coordinate filters, it is possible to extract one coordinate value at a time from selected characteristic point of an object by using a snap.

For example, you need to enter the coordinates of a point, located in the center of a drawn rectangular window.

In response to a point request:

```
Specify point:
```

1. Enter filter: **.x**

The request will change its appearance. Now not all coordinates are requested, but only the value along the X axis:

```
Specify point: >> X
```

2. With the **Midpoint** snap activated, specify the horizontal edge of the window. This will define the coordinate of the point along the X axis.

Now the coordinates for the remaining Y and Z axes are requested:

```
Specify point: >> YZ
```

3. We are satisfied with the mode of cleaving coordinates from a point simultaneously along two specified axes. Therefore, it is not necessary to enter any other coordinate filters. With the **Midpoint** snap activated, select the vertical edge of the window. This will define the coordinates of the point along the Z and Y axes.

### **Improved Display of TrueType Fonts**

The display of TrueType font text was optimized, especially with its small size.

### **The Startup Screen**

The Startup Screen completely redesigned.

It became an interactive multi-functional window with a set of tabs, which allows you to create new documents, open existing ones, display and sort the list of previously opened documents, display their thumbnails, show a list of available applications with a description, links to training resources for each application, a list of new program features with a brief description.

The **AutoClose** checkbox automatically closes the startup screen on the next application start, if no manipulations were made in this window during startup.

### The “About” Window

The **About** window completely redesigned. It contains links to license file, trademarks and displays version number, build number, license status, server name and registration information.

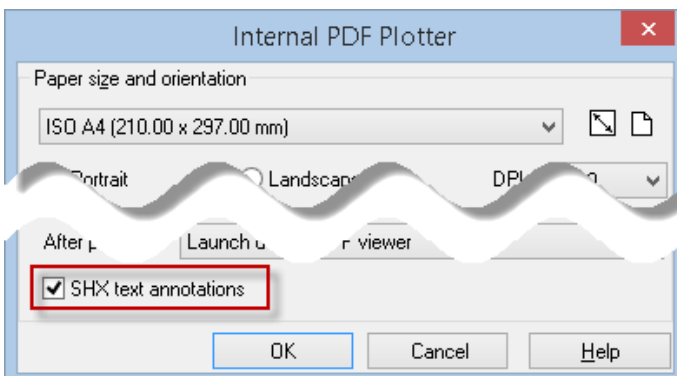
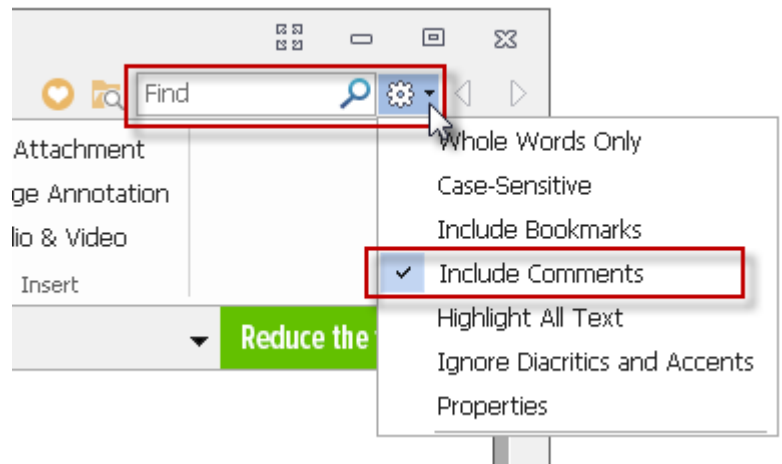


### Comments When Printing SHX Text to PDF

When printing to PDF, it is possible to convert SHX texts to PDF comments.

Such comments will be created at the locations of text objects with the SHX font. Comments contain the full text of these objects. This allows you to search the contents of SHX text in a PDF document.

To search for text in PDF comments, you need to activate the **Include comments** option in the Search settings of PDF viewer.



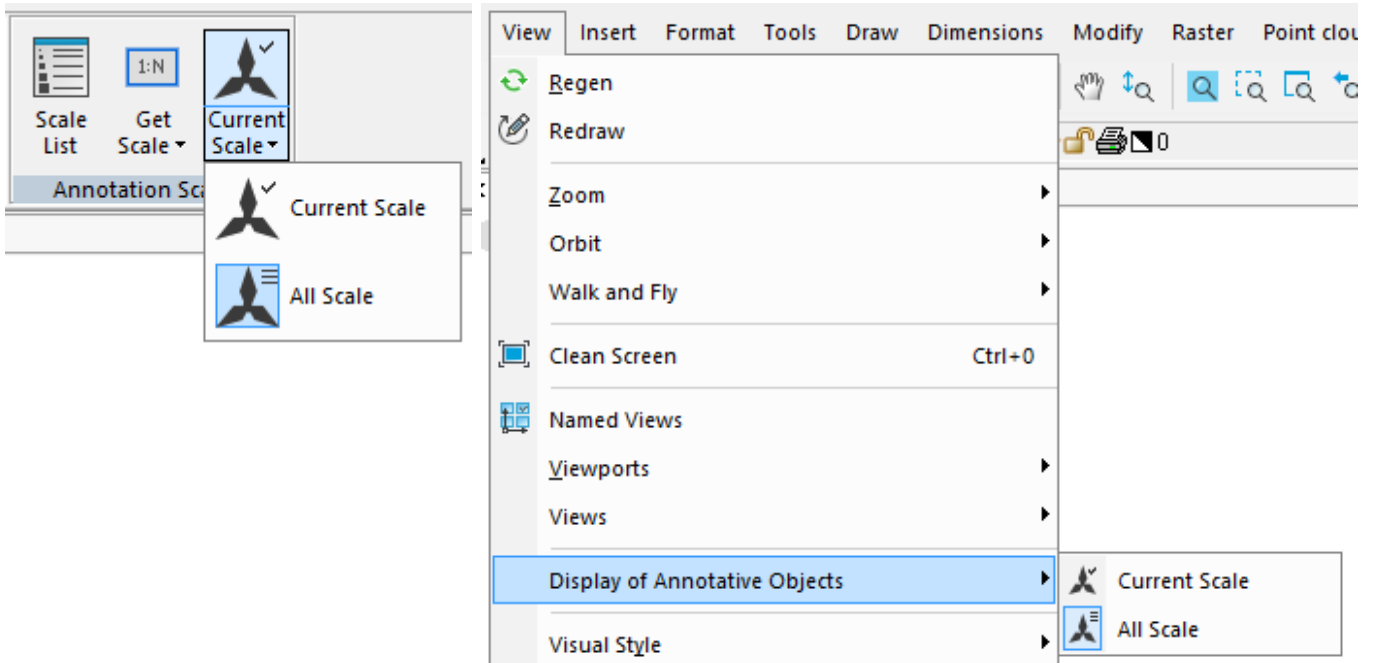
Comment creation can be disabled by the PDFSHX variable or with the **SHX text annotations** check box in the PDF printer's options.

### Pan in Active Paper space Viewport

You can drag mouse through all paper space when panning in active but locked paper space viewport.

### Annotative Objects

Commands for displaying annotative objects created in third-party CAD programs are added to the ribbon and in the View menu of the classic interface.



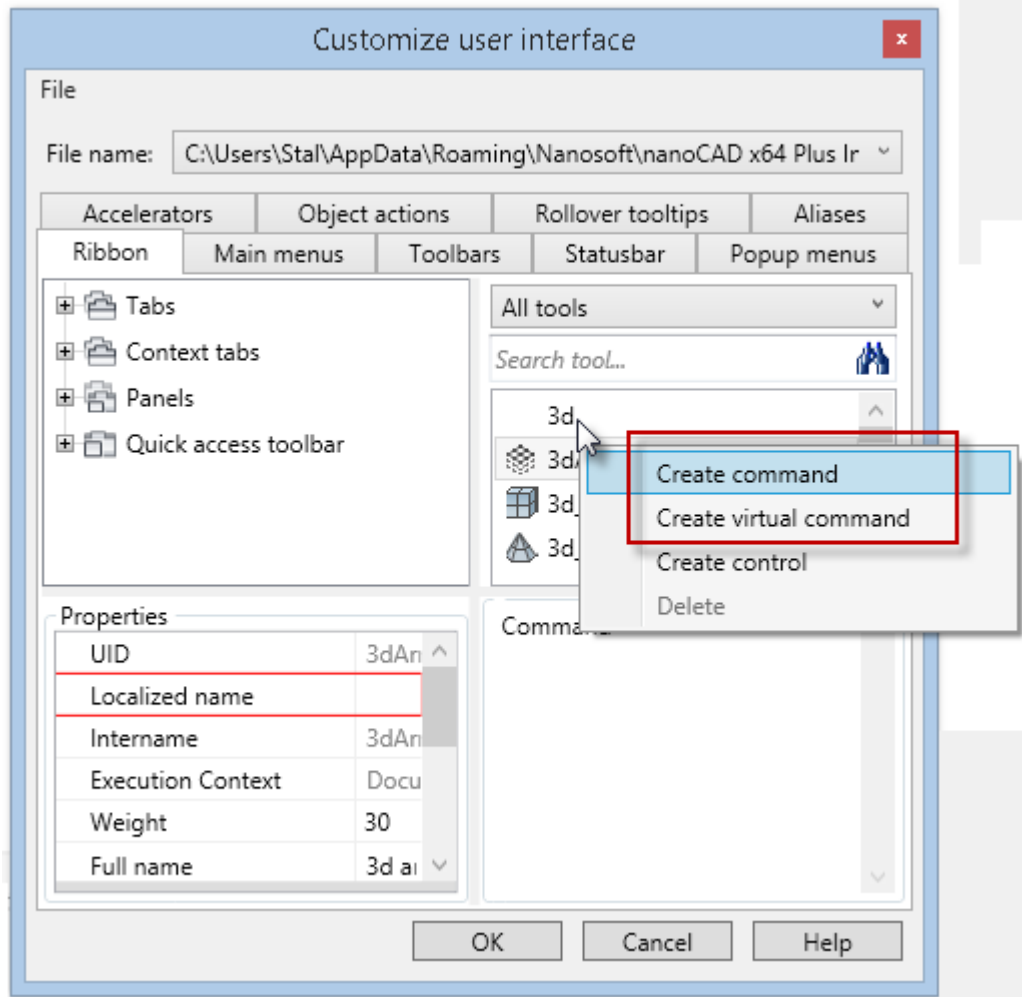
The current value is stored in the ANNOALLVISIBLE system variable.



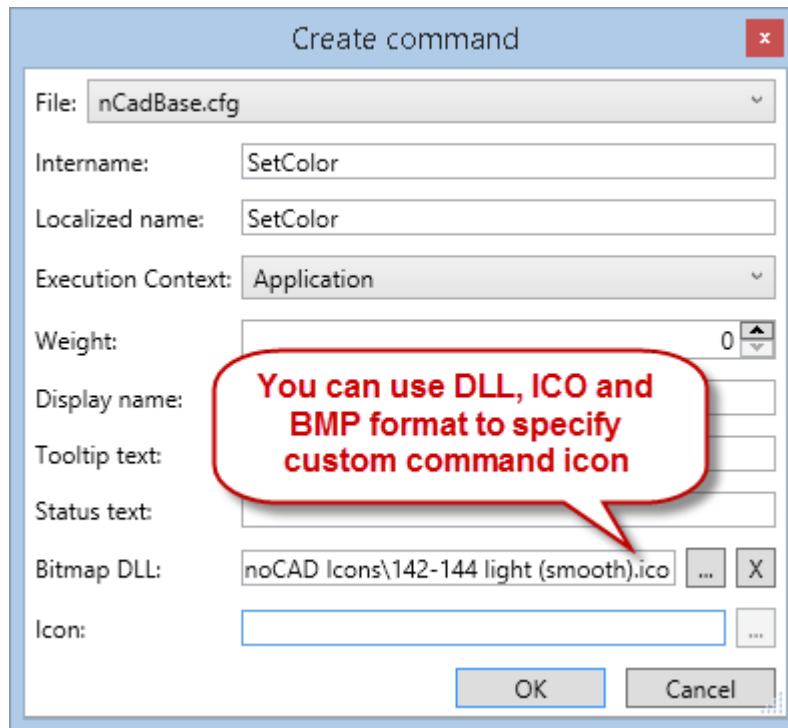
## New in Custom Command Creating



The **Customize Interface** command on the ribbon tab **Manage > Customization > Interface** opens the **Customize User Interface** dialog box where you can create new custom and virtual commands.



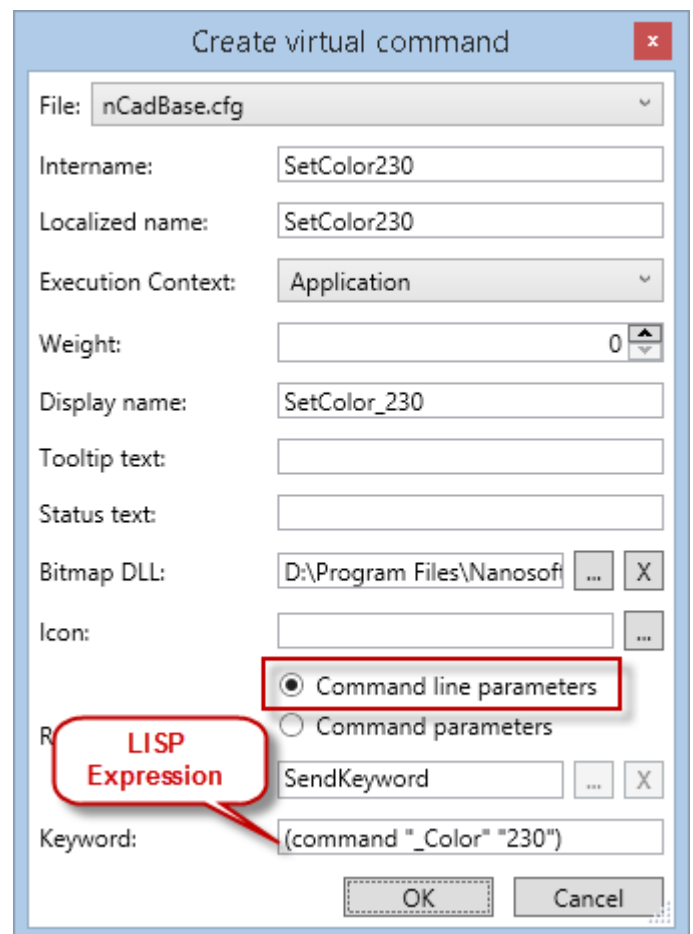
In nanoCAD 20 you can assign icons in ICO and BMP formats for custom and virtual command buttons.



The standard technique for creating Lisp command involves: creating an LSP file, including it in the startup list of the **Load/Unload Applications** dialog, and only then, creating a new command in the **Customize User Interface** dialog.

Now, the LISP expression can be written directly when creating a virtual command, without creating and downloading a separate LISP-file. To do this, select the **Command line parameters** radio-button and enter the LISP expression in the **Keyword** field.

Using these commands, you can create custom buttons in the ribbon, panels, menus, status bar.



It also became possible to create buttons for custom script commands.