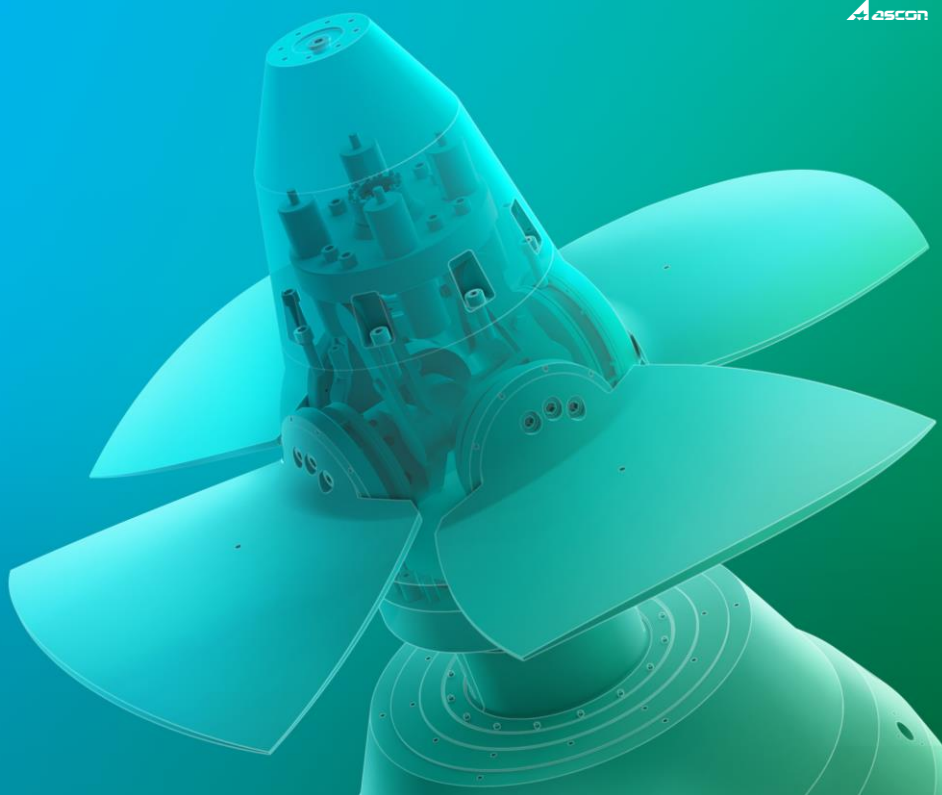
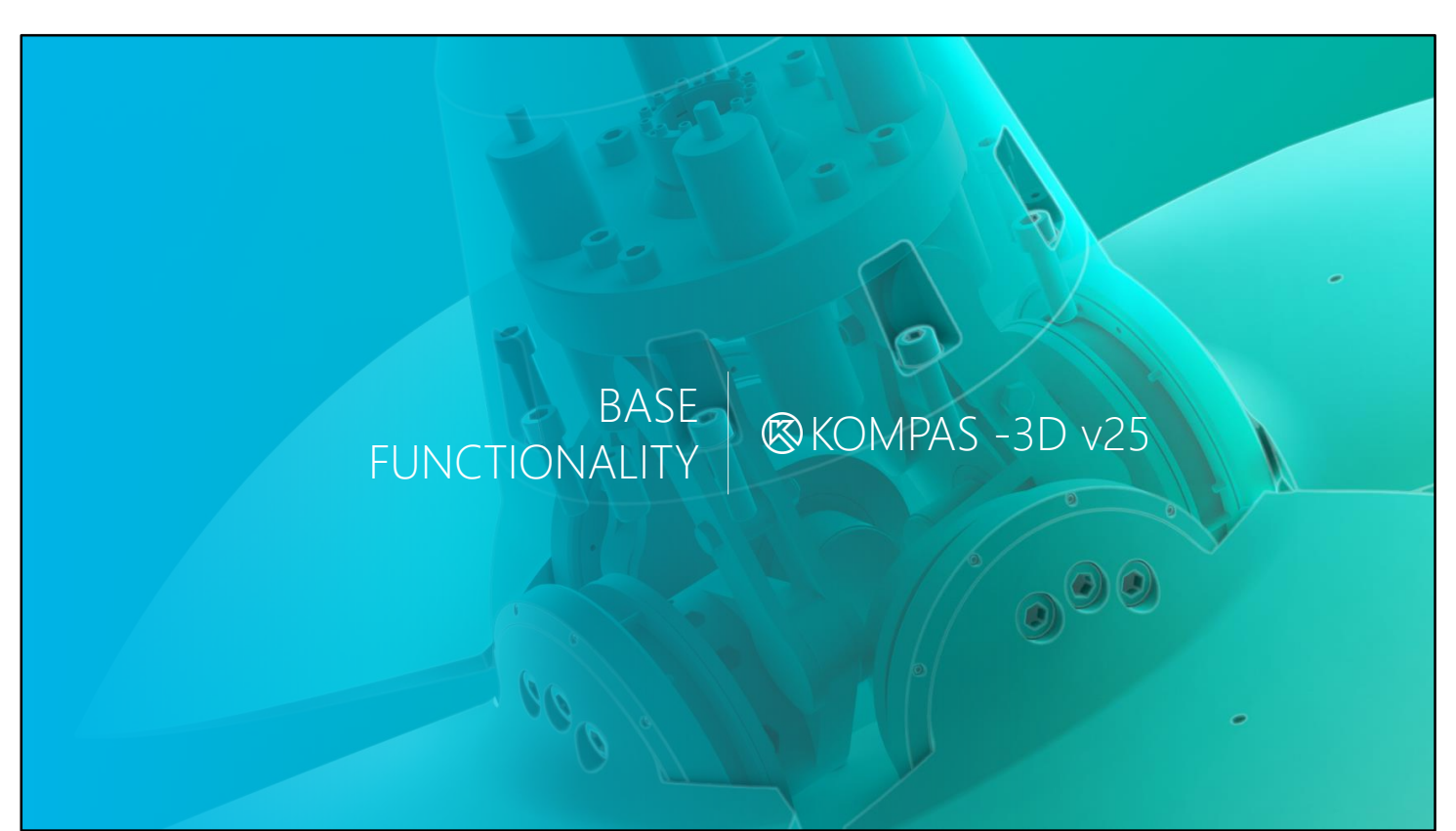


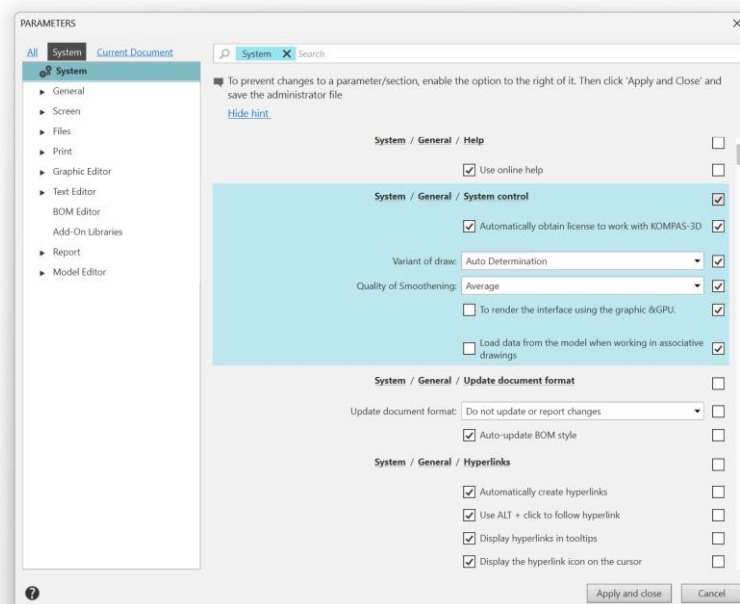
© KOMPAS-3D v25





BASE
FUNCTIONALITY

© KOMPAS -3D v25



Admin Settings

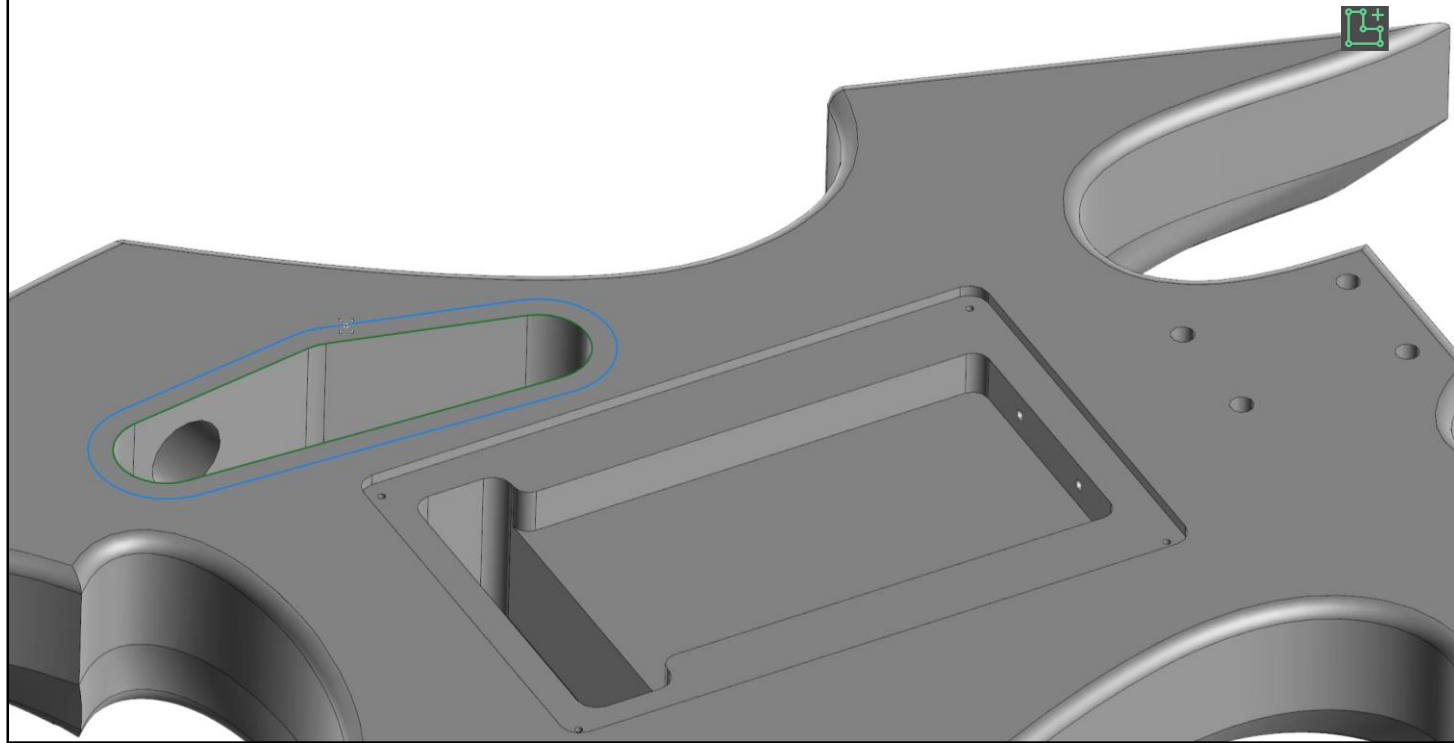
A new useful feature for KOMPAS-3D admins. If you launch KOMPAS-3D as an admin and open the “Parameters” dialog, you’ll see changes in system settings. In Admin mode, you can make parameters unavailable for editing in User mode by checking them. Unchecked settings will remain editable by users. The admin should save the file with the settings, specify the path to the file at users’ workstations. Once that's done, the system settings will take effect.

GENERAL CHANGES

1. Disabling the list of recent documents on the Start page
2. Quick reset of system parameters
3. Quick setup of technical notes
4. Placement of the Help system on a web server without Internet access

- 1) We've added the ability to disable saving a list of recent documents and turn off its display on the Start page.
- 2) Now, you can reset system parameters in the System section of the Parameters dialog.
- 3) We've added the ability to quickly access technical notes settings in the document
- 4) The KOMPAS-3D Help system can be placed on a web server without installing it on every workstation and without Internet access.

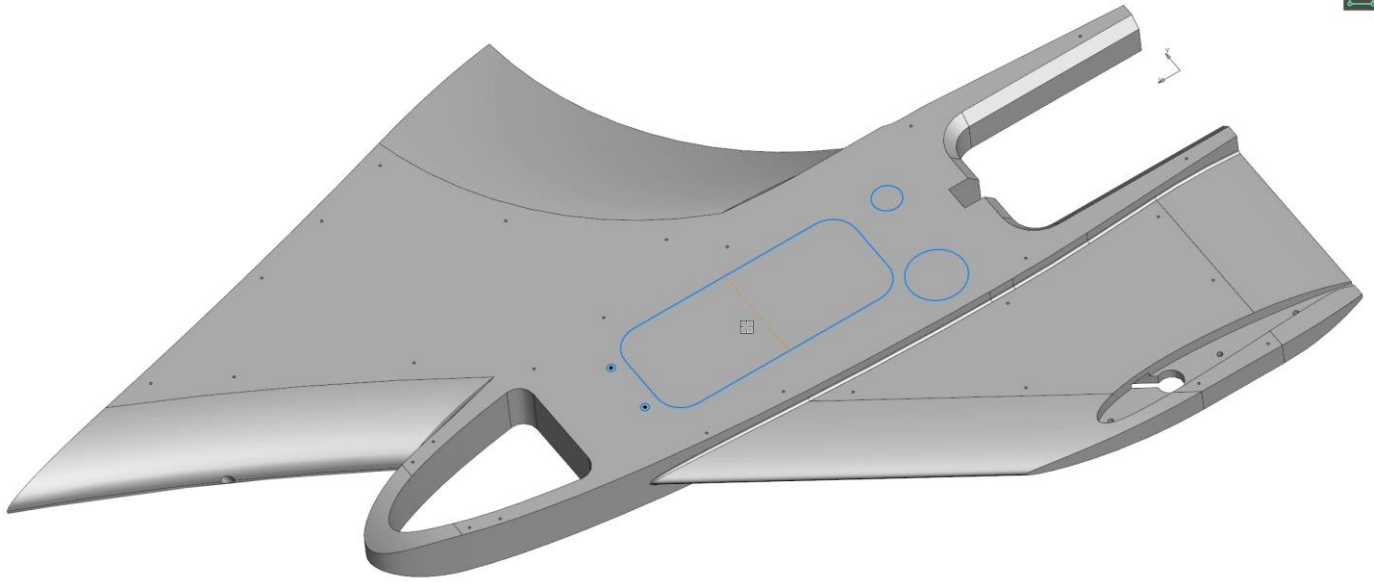
Dynamic highlighting of objects under the cursor



[Useful changes in Sketch mode]

Objects are now highlighted when an object snap is triggered, even if no command is active.

We've added a delay setting. You can set the delay, in milliseconds, that determines how long the cursor must hover over an object before it's highlighted.



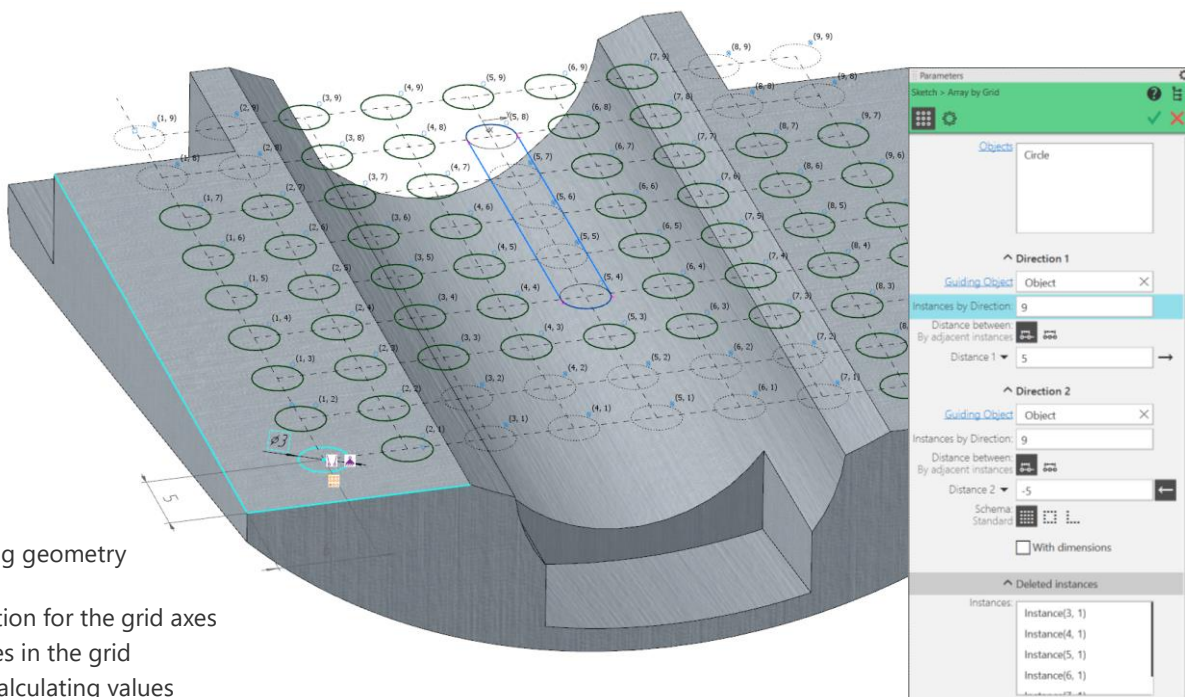
Symmetry of objects

The “Symmetry of objects” command allows you to apply the “Symmetry of objects” constraint to selected objects. You need to select a symmetry axis and two objects.

The objects can be individual elements (segments, circles, and so on) or entire complex objects (rectangles, slots, and so on).

The selected objects should be the same type, but there are some exceptions. For example, you can make a segment and an axis symmetrical, or a circle and an arc. As a result of the command, the objects are placed symmetrically relative to the axis and get the same size.

The same constraint is applied when you use the “Mirror” command.



Arrays of graphic objects

- Trimming/splitting geometry of instances
- Selecting a direction for the grid axes
- Deleting instances in the grid
- Expressions for calculating values of array parameters

We've refined commands for creating arrays and copying graphic objects.

- The "Edit array" command has been added to the context panel that appears when you select an array or part of it.
- Now, when you trim a segment, a circle or an arc that is part of an array instance, this instance will not be excluded from the array. If you split the instance geometry, the array will contain one of the resulting parts of the split geometry.

For the "Array by Grid" command:

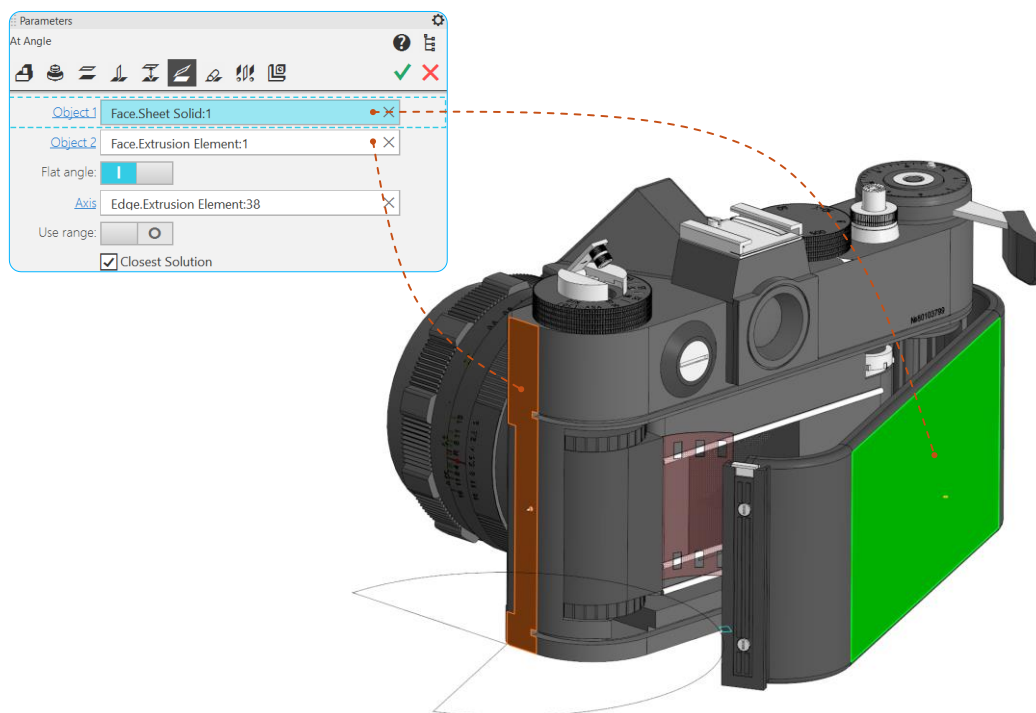
- You can select a straight object to define a direction for the grid axes. In Sketch mode, you can use 3D-model objects as the straight objects. The array keeps its connection with the selected objects and is rebuilt when the objects change position
- We've added the ability to select the result: deleting instances inside the grid, or placing instances only along the grid axes
- Expressions for calculating array parameter values can now be entered and displayed in the fields of the Parameters panel.

SKETCH MODE

1. A zero-length segment is not an error
2. Variable display in dimensions can be controlled with F11
3. The panel for entering reference dimension values has been disabled
4. An axis of the coordinate system can now be selected for the "Mirror" command

What's new in Sketch mode:

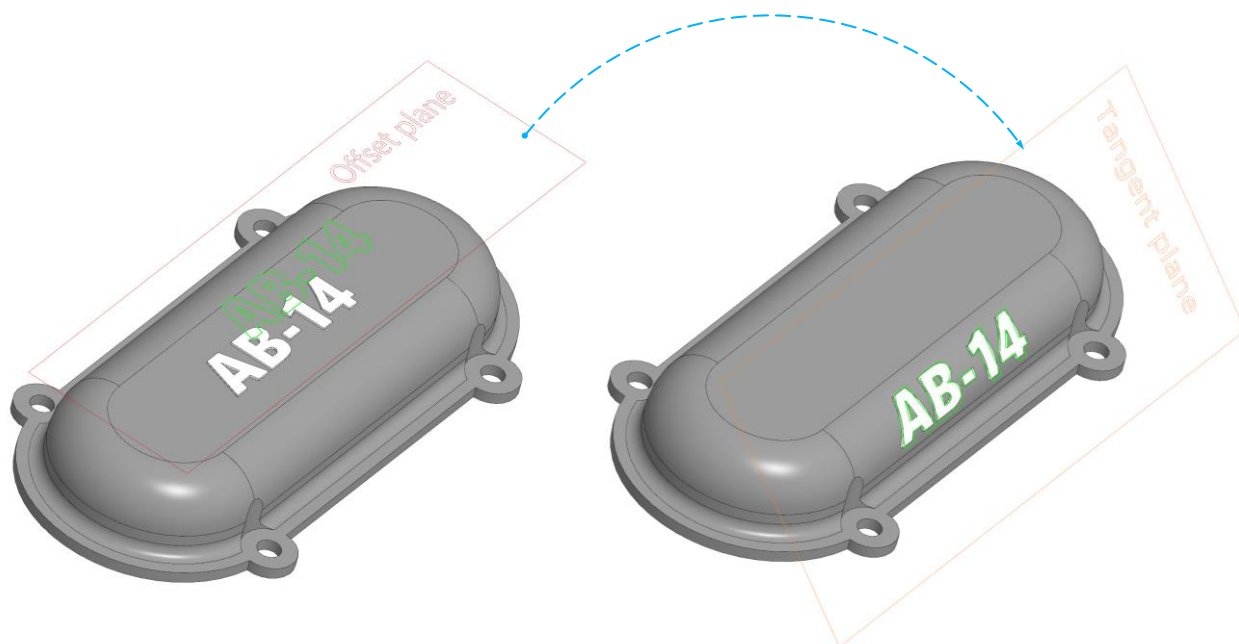
- 1) The system no longer treats zero-length segments as an error. Such segments can appear, for example, as a result of editing parametric geometry.
- 2) While working in parametric mode, you can enable or disable the display of variables in dimensions using the F11 key.
- 3) Now, if you add an associative dimension whose value is unavailable for editing – for example, a reference dimension - the dialog for entering a value doesn't appear.
- 4) You can select an axis of the coordinate system as a symmetry axis while working with the "Mirror" command.



Highlighting of selected objects

[3D modelling updates to improve the user experience]

If you select an object in a one-object field while working with a command, the selected object will be highlighted in the graphic area. In the previous version of KOMPAS-3D, this only worked for multi-object fields.



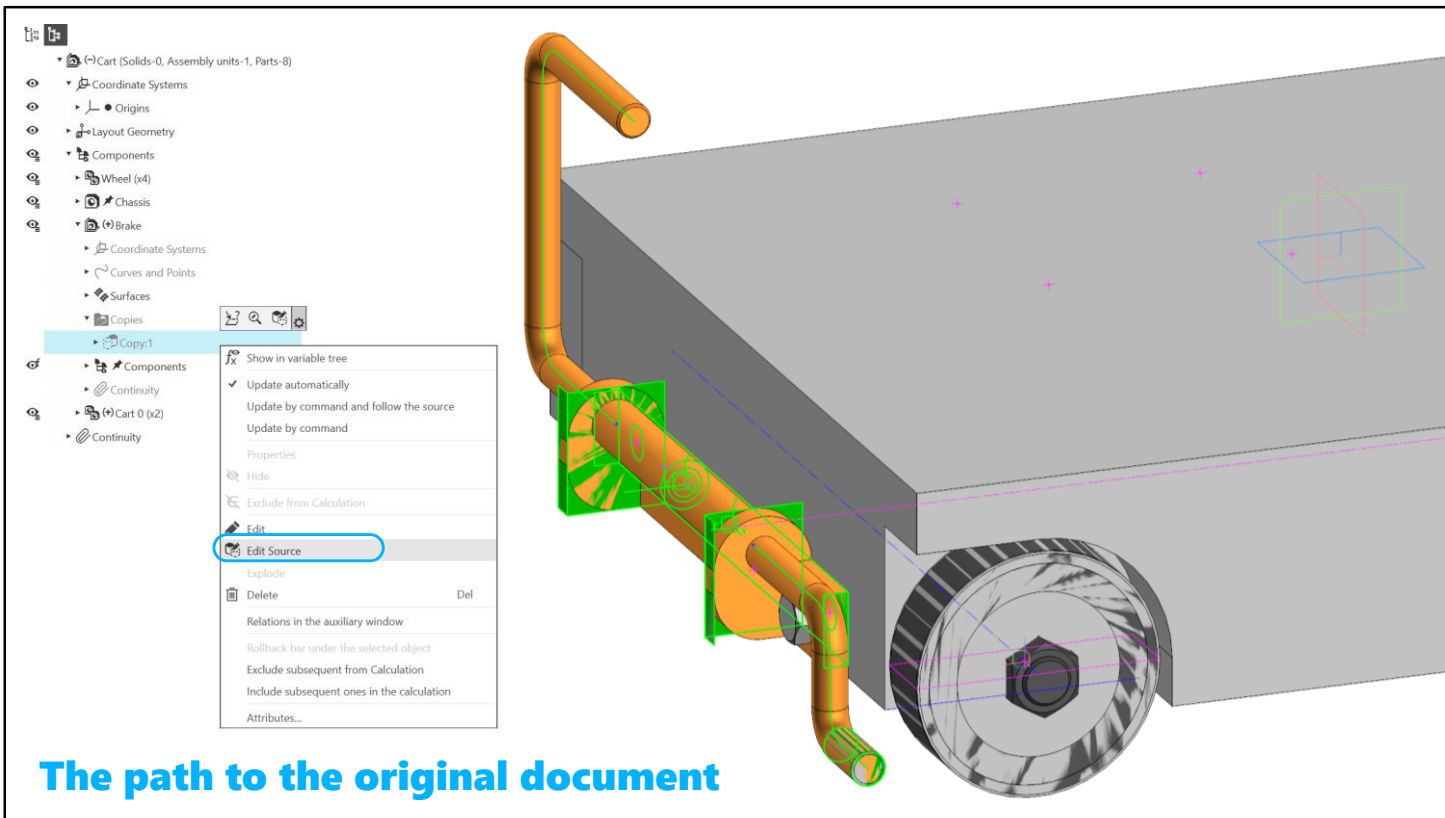
Changing the construction method of planes and axes

In KOMPAS-3D version 25, you can change the construction method while editing a construction plane.

In the Parameters panel, the same buttons are available when editing a plane as when creating a new one. Click the required button and select the reference objects.

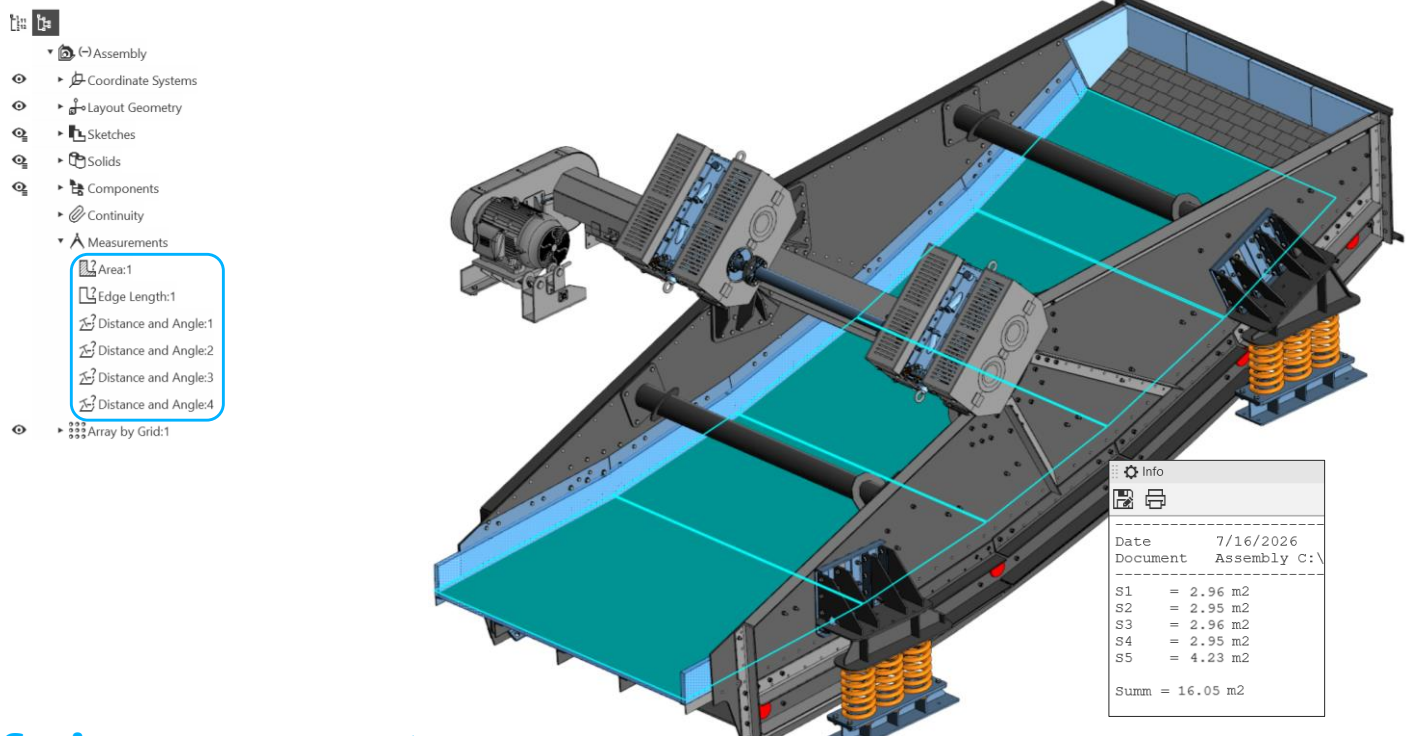
The links between the plane and its dependent are preserved after this edit. The plane properties are also retained.

The same functionality is available for construction axes.



We've added the ability to view the path to the original document – the source of copied objects – and open it directly.

You can also open the document that serves as the source of a context connection.

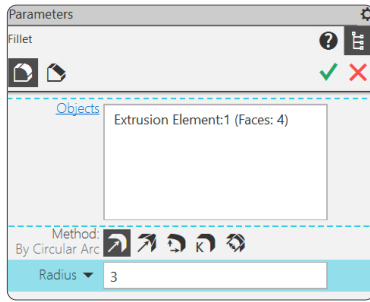


Saving measurements in the tree

We've refined the "Distance and Angle", "Edge Length", and "Area" commands.

- Measurement objects created with these commands can now be edited. You can select a different object to measure.
- The standard **Create Object** button has replaced the **Create Measurement** button in these commands.

Selecting groups of edges and faces based on a condition



Conditions of face selection

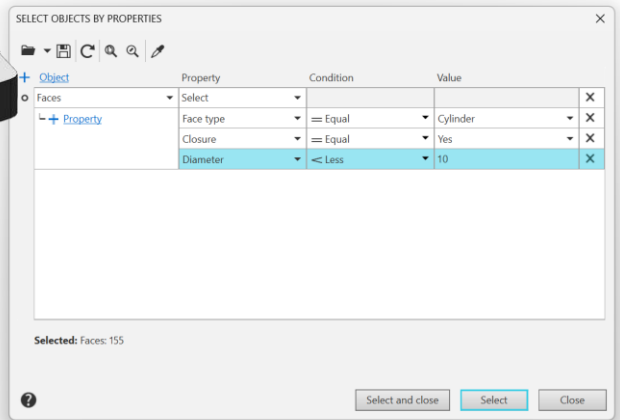
- ☐ Single face
- ☐ Adjacent to selected face
- ☐ Face and adjacent faces
- ☐ Tangent faces
- ☐ Tangent fillet faces
- ☒ Operation faces
- ☐ Solid/Surface faces

[Modeling functionality updates]

When you start a command, you need to select reference objects. Now, you can automatically select groups of edges or faces that meet specific criteria. For example, you can select an entire chain of edges by clicking a single edge in the chain, or select one face to automatically select all faces created by the same operation.

Additionally, we've refined the selection filters:

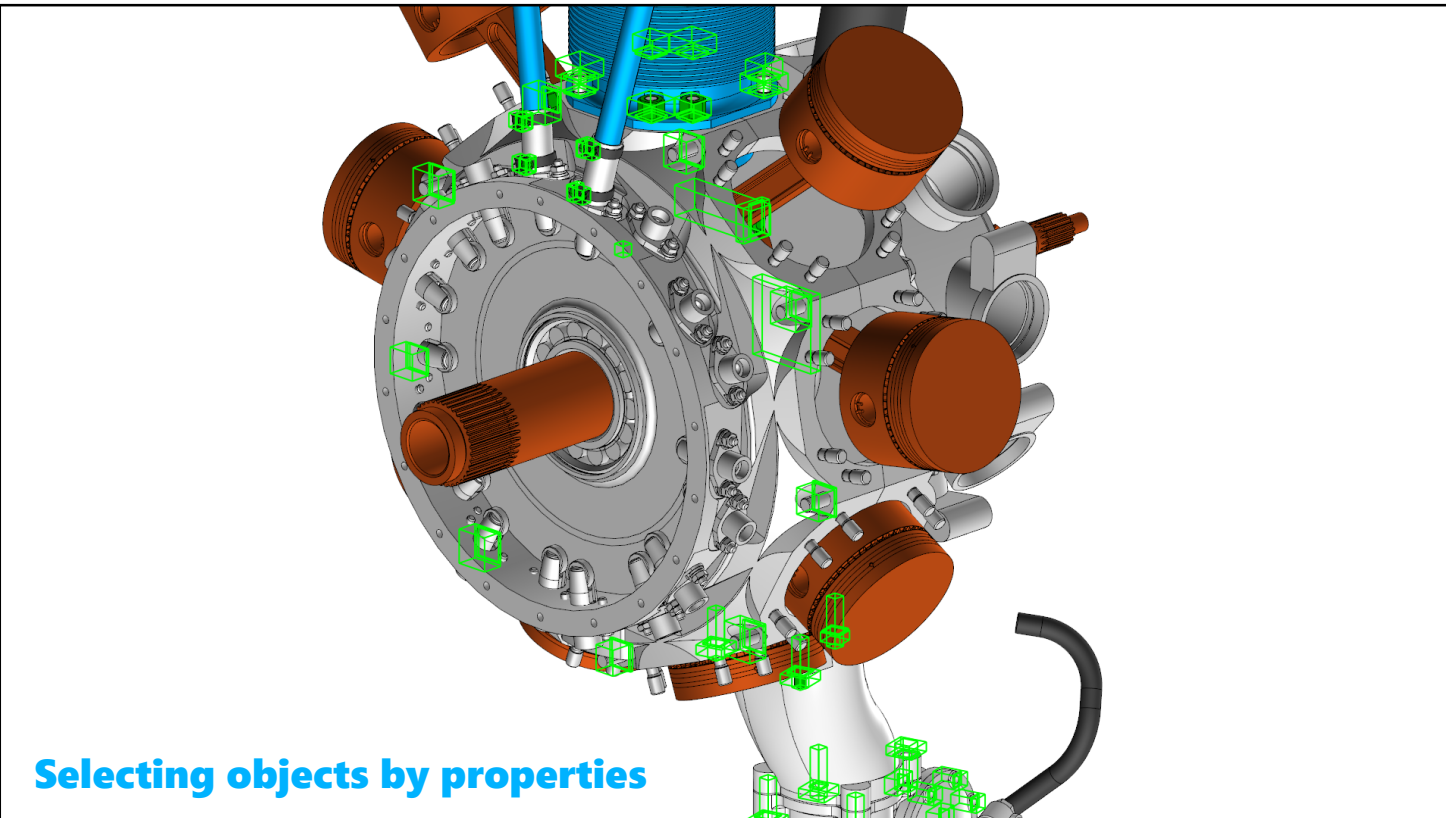
- You can create your own custom filter sets for each command.
- Each command displays only the filters applicable to that command.
- A separate filter set is available when no command is active.
- All custom filter sets are saved until the document is closed.
- If at least one filter is active, a special icon appears in the upper-right corner of the graphic area. Click it to disable all filters in the current filter set.



Selecting objects by properties

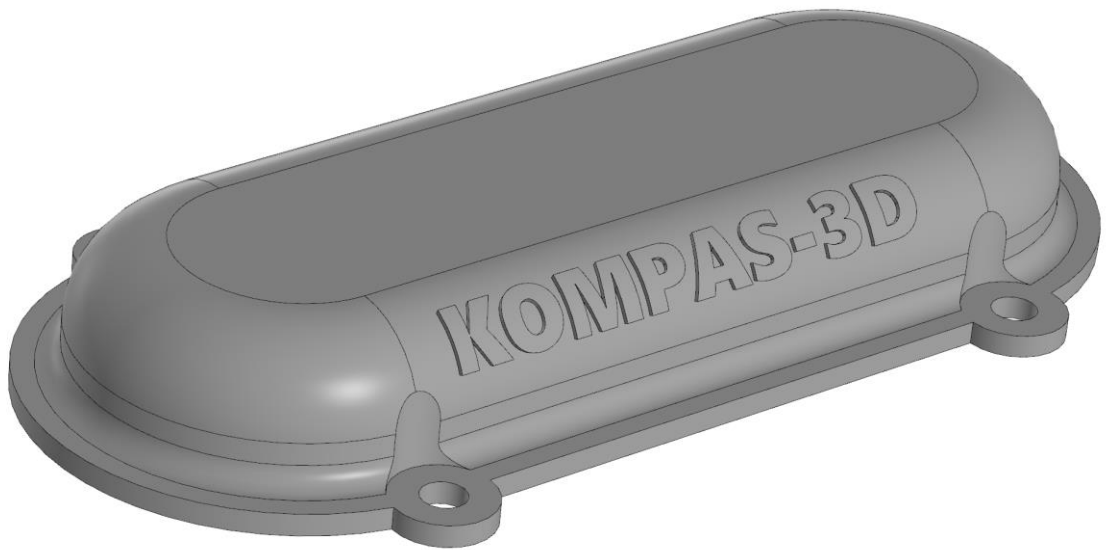
We've added the "By properties" command to the "Select" menu. It allows you to select objects of different types based on their properties. For example, you can find assembly components with specified overall dimensions, all curves located on a specified layer, or operations of a specified type.

You can define one or more selection conditions. If you want the selected objects to satisfy all of the specified conditions, place them in the same condition block. If you place the conditions in different blocks, objects that satisfy at least one of the conditions will be selected.

A 3D CAD model of a complex mechanical assembly, possibly a pump or engine component. The model features a central grey circular flange with a large orange shaft protruding from the center. Surrounding this flange are various orange and grey components, including what appear to be pistons or valves. Numerous small fasteners, such as bolts and screws, are visible throughout the assembly. These fasteners are highlighted with green wireframe boxes, indicating they have been selected for a specific operation. The background is plain white.

Selecting objects by properties

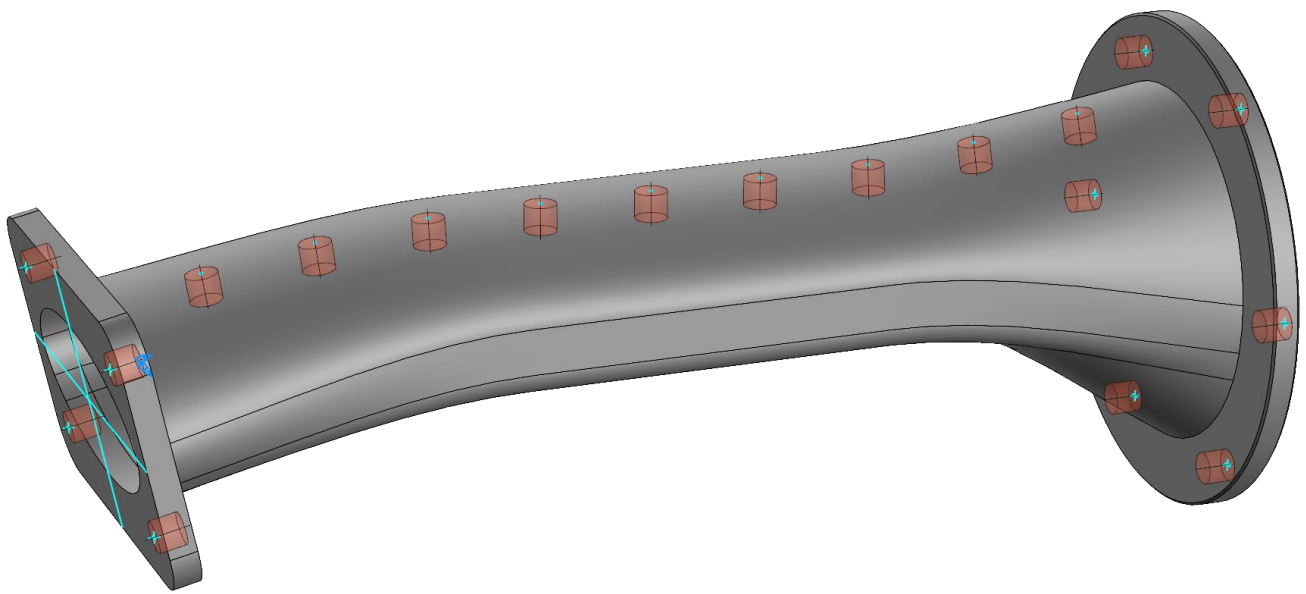
In this slide, all fasteners were selected using the “By properties” command.



- Emboss
- Deboss

Relief

The new “Relief” command allows you to create relief patterns on faces. It projects selected closed contours onto the faces and then adds or removes material within the projected contours.

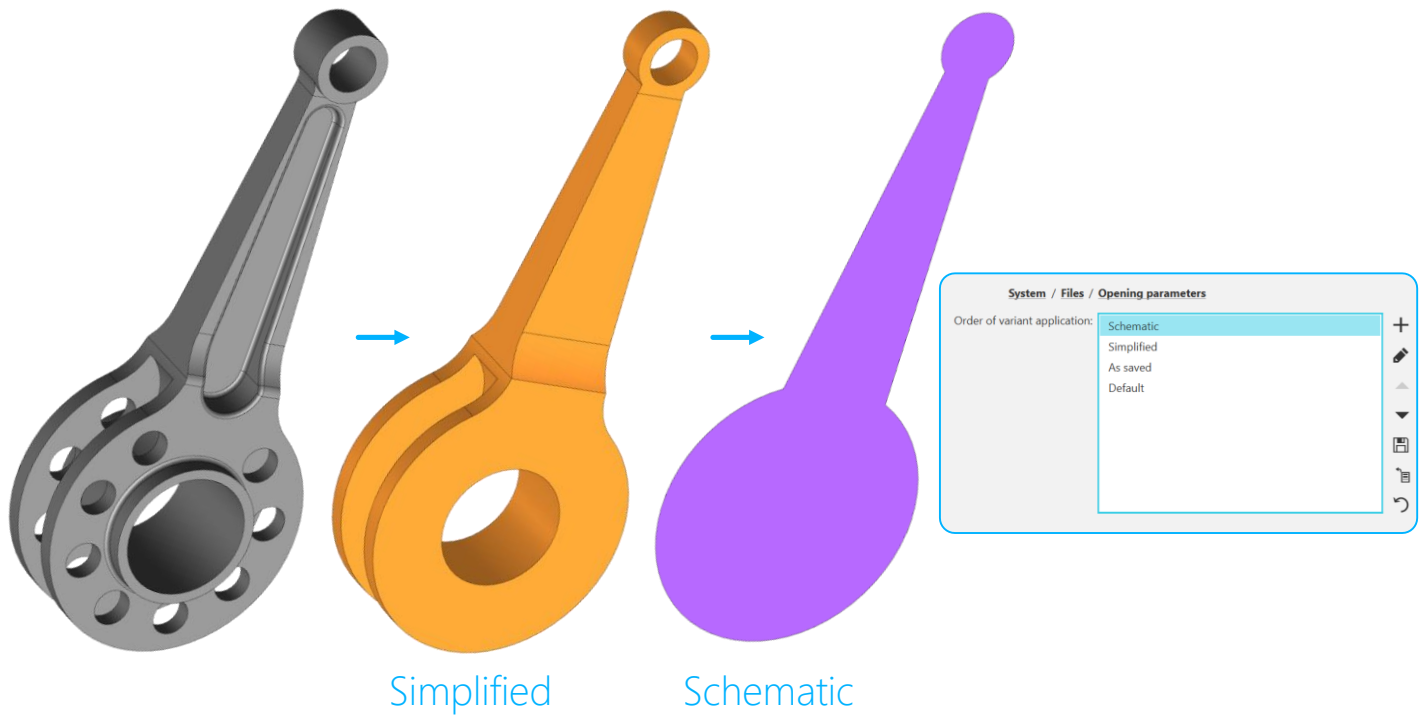


Multiple holes at once

Hole creation commands now allow you to create multiple holes at once. The hole positions are defined by user-selected points. These can be individual points or groups of points, such as point arrays or sketches containing points.

Holes are created normal to the surface through each selected point. If the points lie on a surface, the command uses that surface to determine the hole direction. Otherwise, you must select a surface manually. You can also select a linear object to define the same direction for all holes.

Detailing variants



You can create special model variants called detailing variants. For example, the rod shown on this slide has three detailing variants:

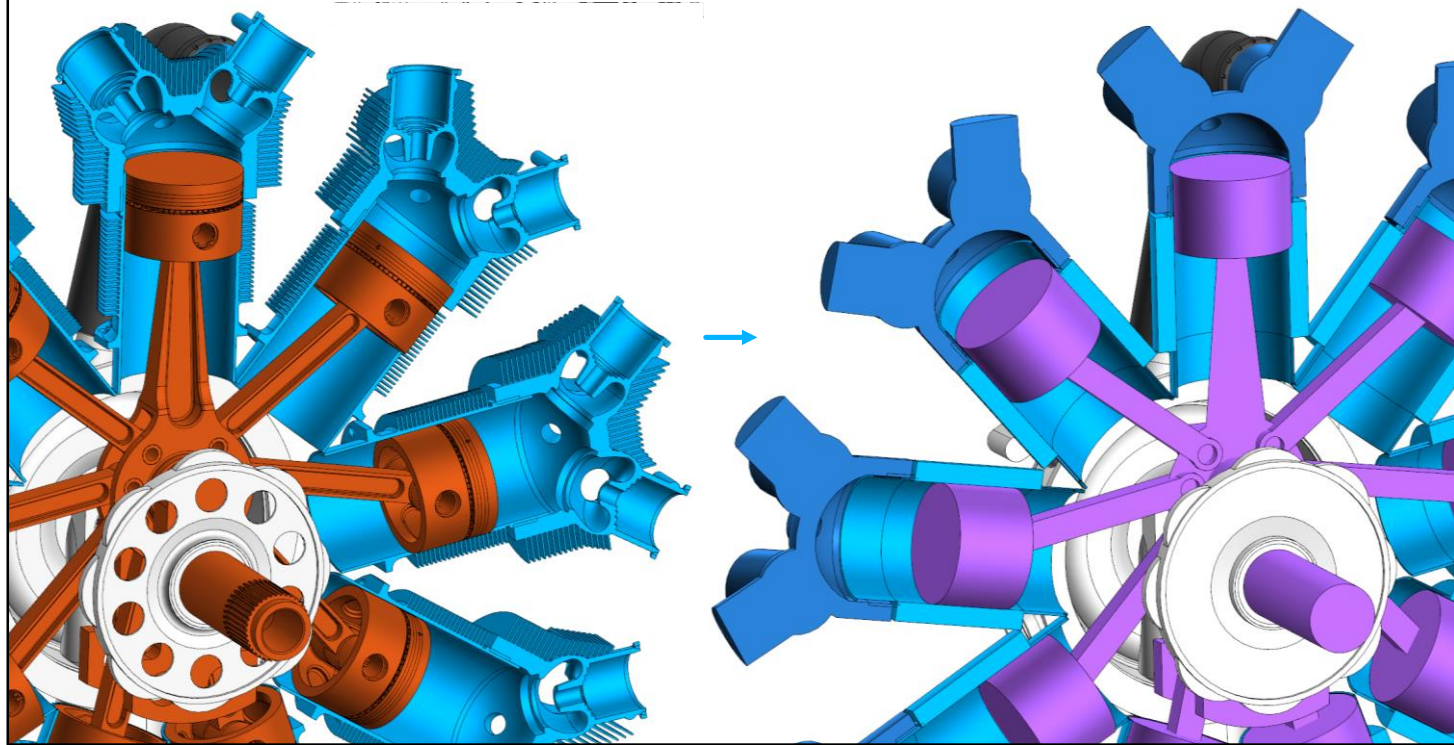
- Default – the complete model with all the details.
- Simplified – only the main shape and overall dimensions are retained.
- Schematic – only the outline of the rod is preserved.

You can create as many detailing variants as needed.

The feature is especially useful when working with large assemblies. Using detailing variants can reduce memory usage and improve overall system performance.

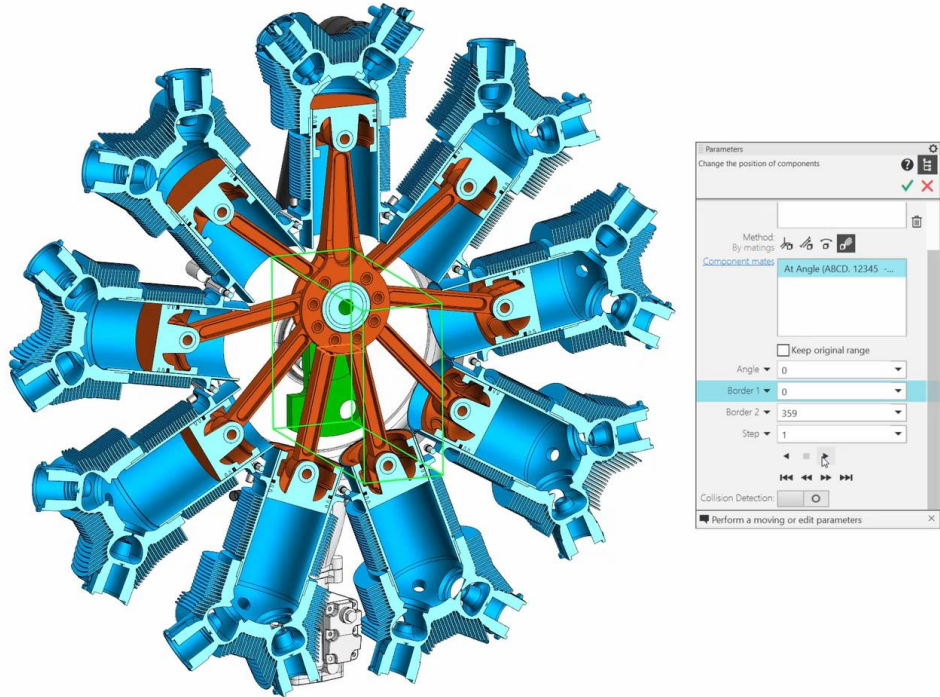
Before opening an assembly, you can specify the order in which detailing variants are applied using a new system setting. In the example shown, the Schematic variant has the highest priority, so the schematic variants of the components are loaded first when the assembly is opened. This can significantly reduce assembly loading time.

Detailing variants



On the left, the assembly is displayed with full detail; on the right, with simplified geometry.

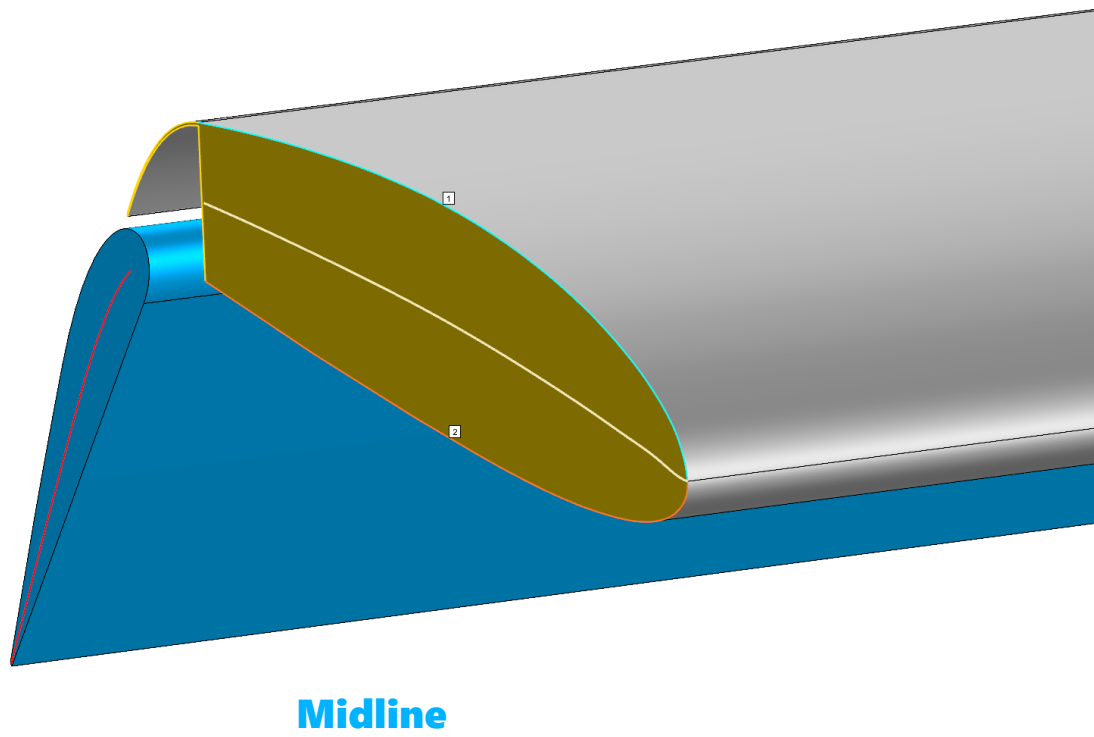
Changing the position of the component



We've added the "Changing position of component" command. It allows you to reposition components in several ways:

- Specify the position and orientation relative to a selected coordinate system
- Move a component in a specified direction by a defined distance or rotate it about a selected axis
- Move a component using the "At Distance" and "At Angle" mates. This method allows you to animate the component's movement.

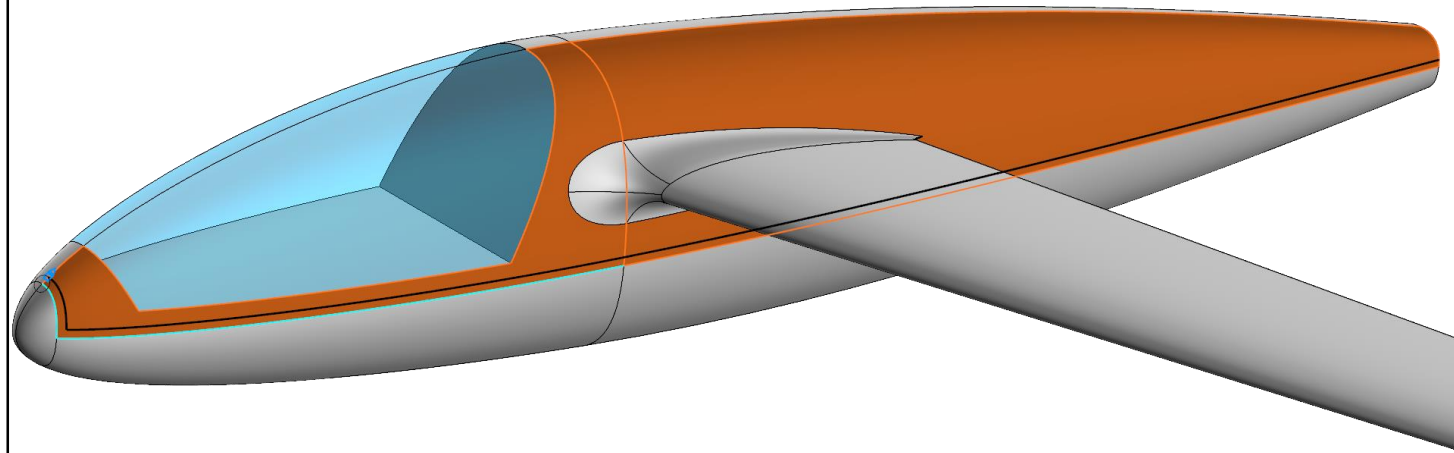
You can also select multiple components and reposition them simultaneously.



[Surface modelling]

We’ve added the “Midline” command. It allows you to create a curve whose every point is equidistant from two selected planar curves.

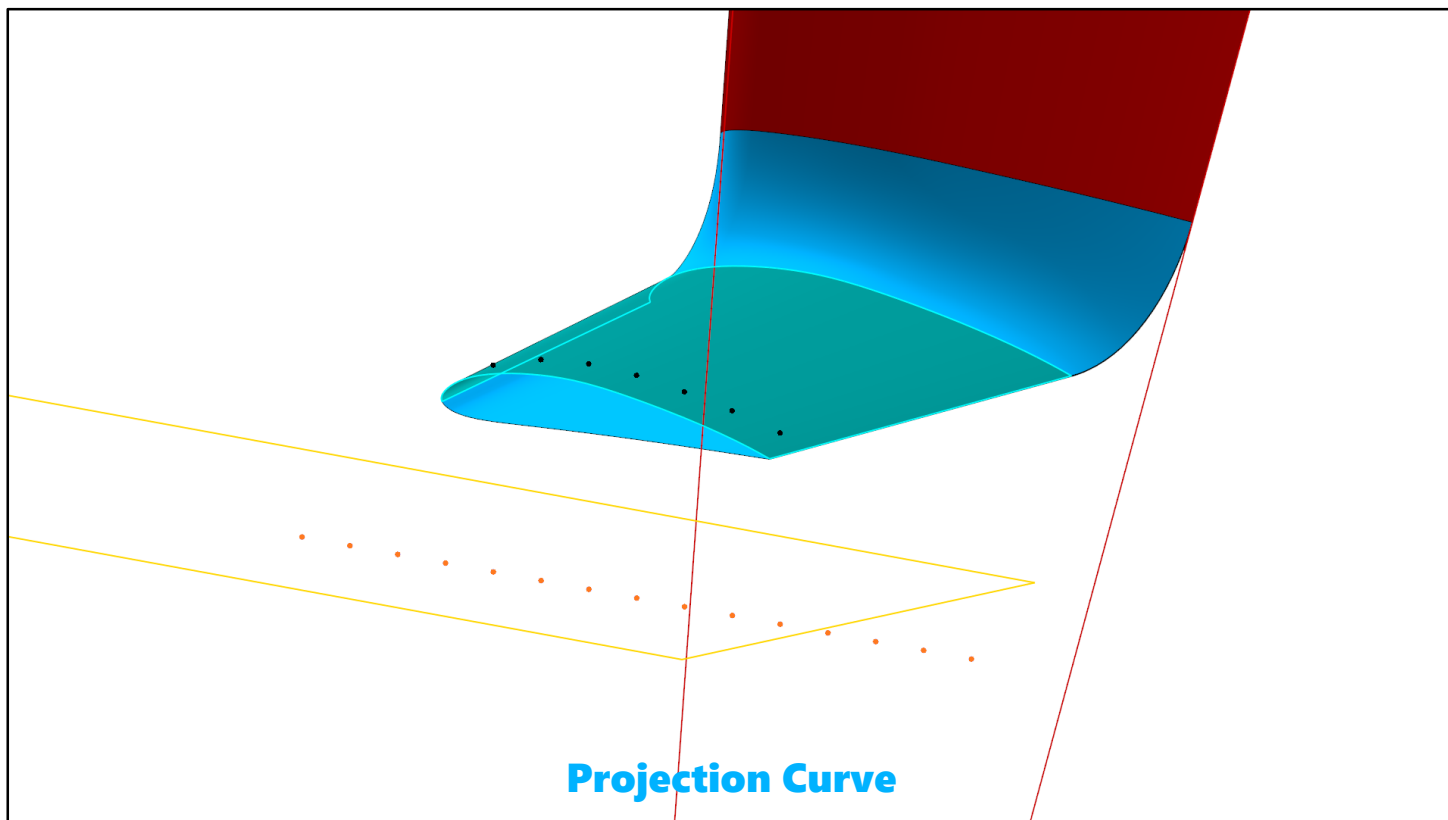
To use this command, select two curves and the plane onto which they will be projected. The midline is then created on this selected plane.



Equidistant to curve

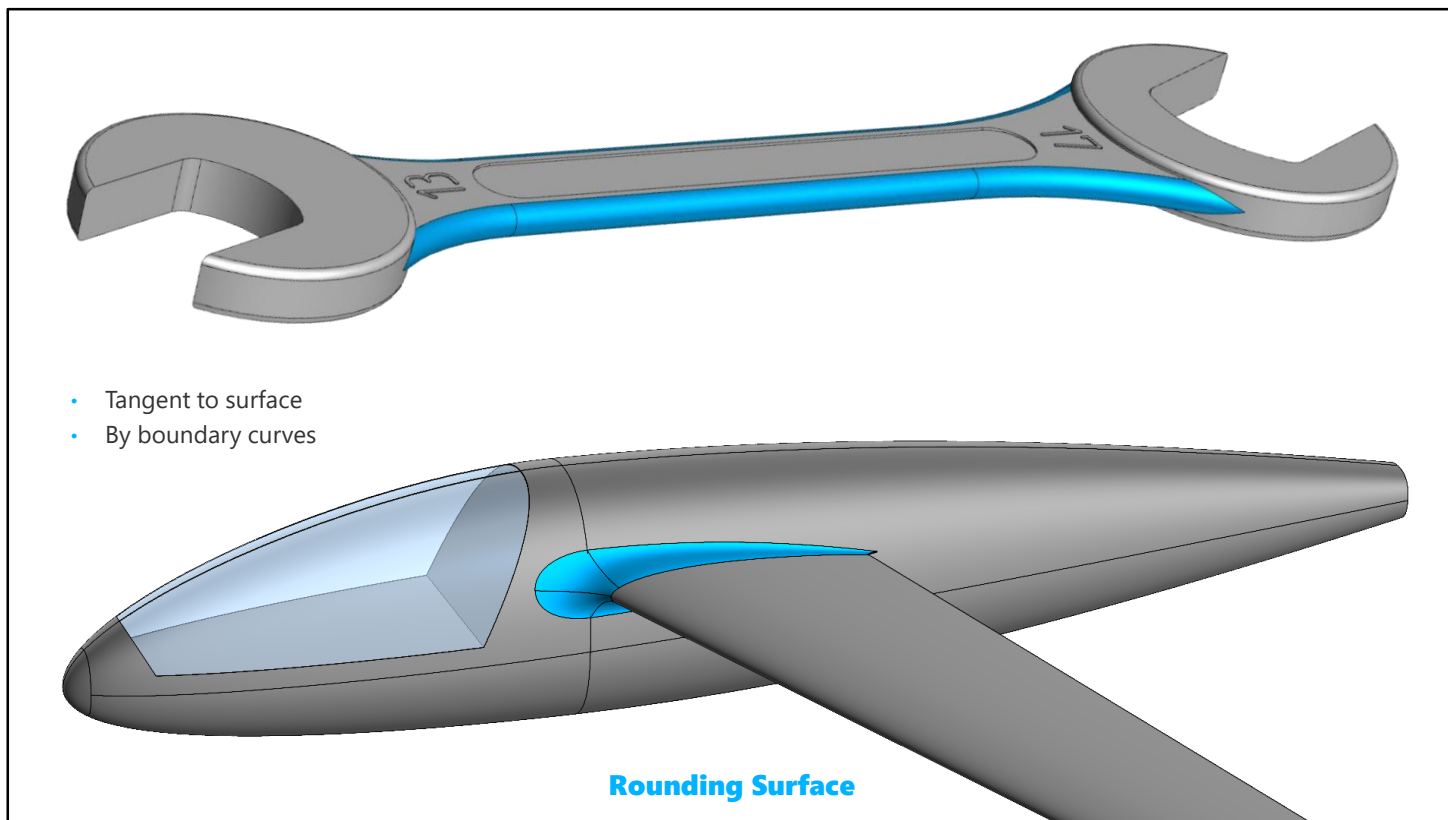
We've refined the "Equidistant to curve" command. When using the "Along surface" offset method, you can now select multiple faces belonging to the same solid or surface.

We've also added the ability to trim the resulting curve to the face boundaries or extend it to the face boundaries.



We've refined the "Projection Curve" command.

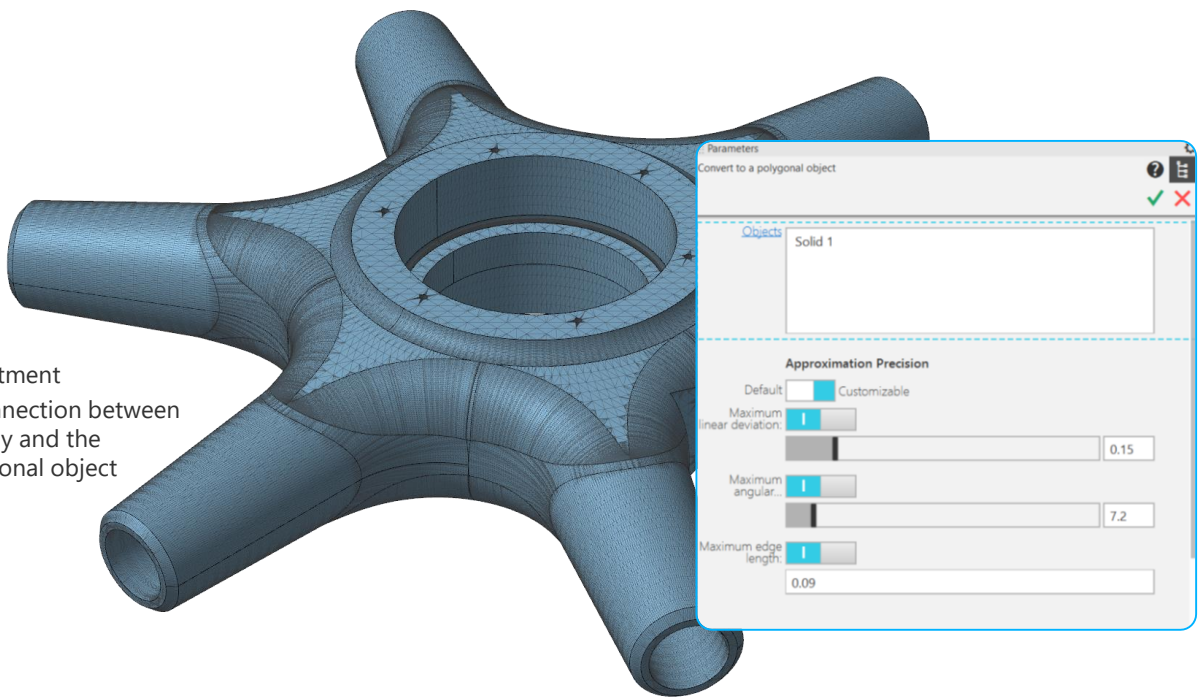
- The command now supports points and point groups. You can also project curves and points simultaneously.
- When using the "By Direction" projection type, you can enable the creation of curve segments parallel to the projection direction. This helps prevent breaks in the resulting curve when projection onto the selected faces is impossible – for example, on planar faces that are parallel to the projection direction.



We've refined the "Rounding Surface" command by adding new ways for defining the surface width:

- "Tangent to surface" creates a rounding surface that is tangent to both the selected surface and the base surface.
- "By boundary curves" creates a rounding surface whose boundaries coincide with the selected curves on the base surfaces.

We've also added a new method for controlling how the surface parameters vary along the surface path: "By values at points". When you select this method, specify points along the surface path. Then, define the parameters values at those points and how the parameters change between them.



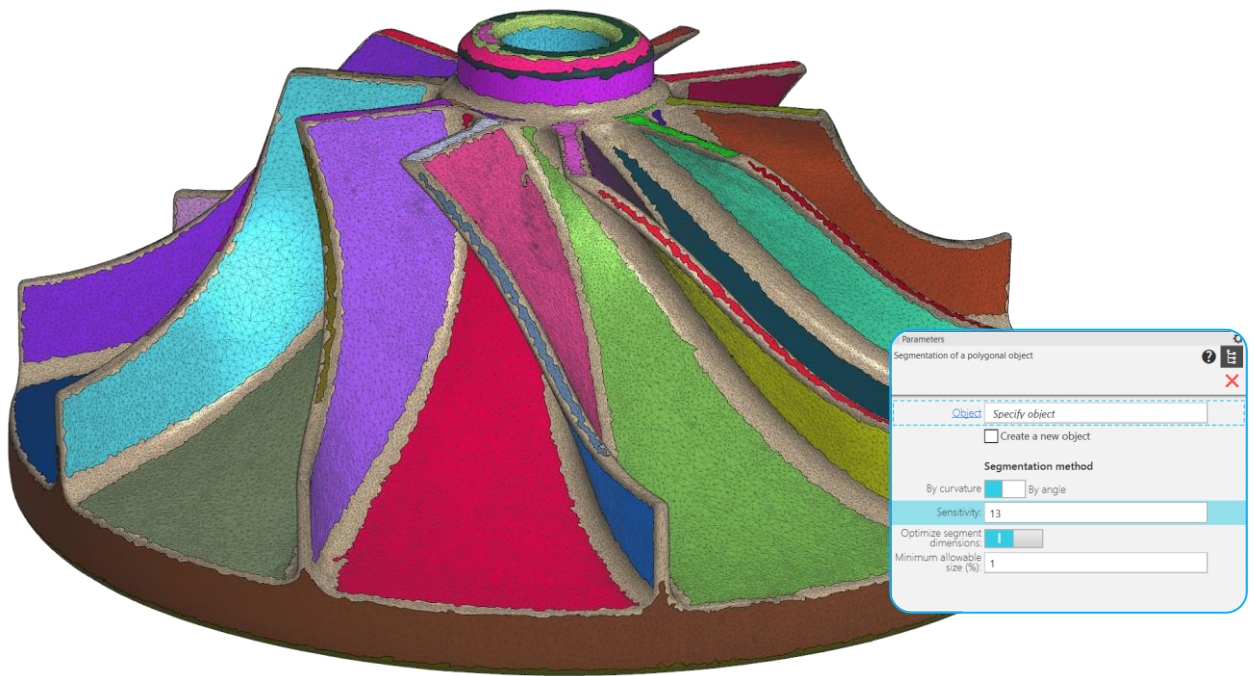
- Precision adjustment
- Associative connection between an original body and the resulting polygonal object

Converting a model into a polygonal object

[Polygonal objects and reverse engineering functionality]

We've refined the "Convert to a polygonal object" command.

- we've added a new option for controlling the conversion accuracy. You can specify the maximum linear deviation, angular deviation, and edge length of the resulting polygonal object.
- Polygonal objects remain linked to their original solids and surfaces. If the original objects are modified, the polygonal objects are updated automatically. You can break this link at any time if needed.
- As long as the link to the original body is preserved, you can edit the command parameters.



Segmentation of a polygonal object

[Key new feature for reverse engineering]

We've added a new command – “Segmentation of a polygonal object”. It divides a polygonal object into segments.

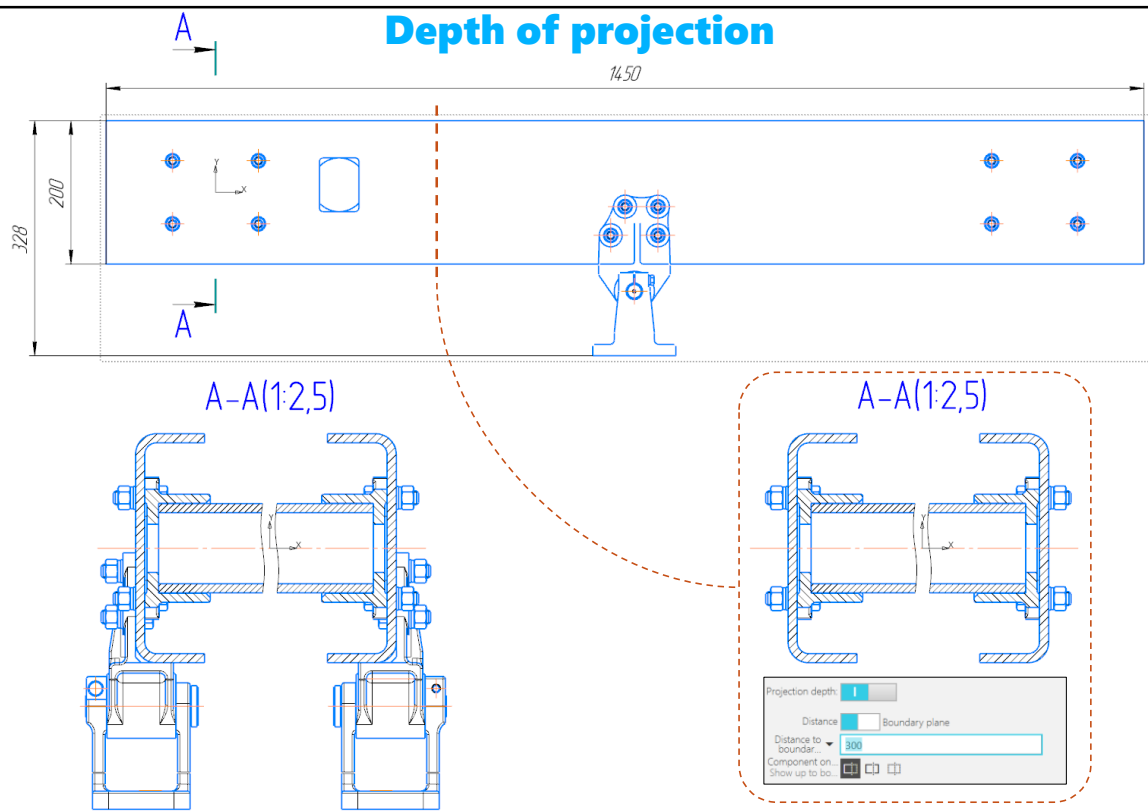
A segment is a group of connected triangles with similar curvature at their vertices and similar angles between adjacent triangles.

The command allows you identify areas of a polygonal object whose shape is close to a plane, cylinder, sphere, cone, or torus.

3D MODELS

1. Updating operations with algorithms of the new KOMPAS-3D version
2. Model Family: inserting new columns into the existing family table
3. Changing the direction of the construction plane normal
4. Surface Trimming: tolerance for the accuracy of joining cutting objects
5. Copying a cosmetic thread while copying the geometry
6. Network Surface: the "Keep direction" option for connecting surfaces
7. Setting a default layer for parts and standard parts
8. Access to the parameters of the model layers in Sketch mode
9. Material is not assigned to new assembly models by default

1. We've added the "Update version of selected features" command. It allows you to recreate operations created in previous versions using the algorithms of the current KOMPAS-3D version.
2. We've refined the "Model family" command. You can now add new columns to an existing table.
3. You can now change the direction of a construction plane's normal while creating or editing the plane
4. The "Surface trimming" command now includes an option for specifying the tolerance used when joining curves or surfaces that are part of the cutting object. This allows multiple objects to be joined into a single continuous cutting object.
5. When copying geometry, you can also copy cosmetic threads created on the copied body's faces.
6. We've refined the "Network Surface" command. The "Keep direction" option is now available when no V-direction curves are specified, provided that at least one mating condition is selected.
7. We've added the ability to specify default display parameters and the default layer for solids, surfaces, and assembly components.
8. In Sketch mode, the "Layers" panel now displays both the Sketch layer tree and the Model layer tree. In the Model layer tree, you can modify layer parameters.
9. The default value of the "Material" property for new assembly models has been changed. By default, no material is assigned.

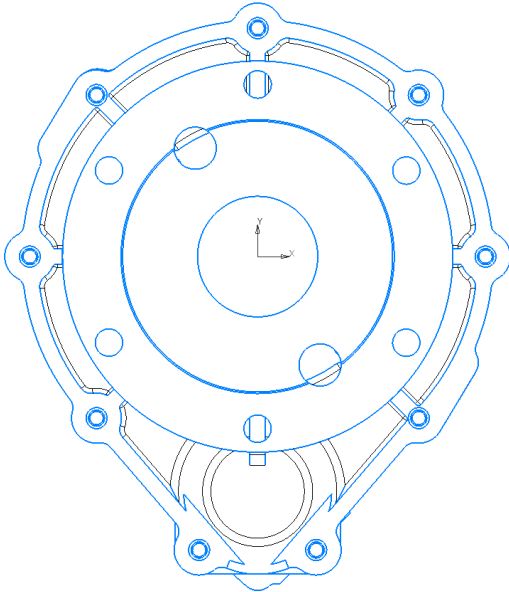


[Drawing]

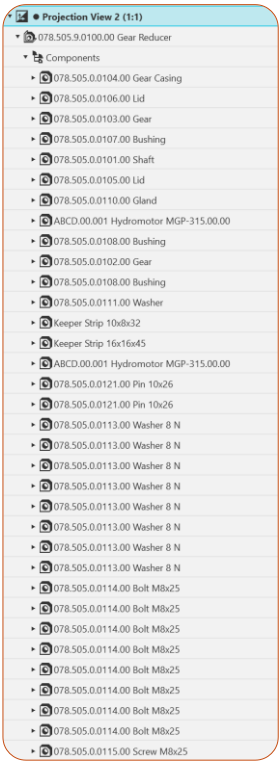
In KOMPAS-3D version 25, you can limit the area of a model projected into a drawing view. This area is defined by two planes within the drawing view. The planes are parallel to the view's projection plane. The first plane passes through the point closest to the viewer, while the second plane is located behind it.

For a section view, the first plane coincides with the cutting plane.

The distance between the two planes is defined by the user. You can enter the value directly from the keyboard or define the position of the second plane in another projection view.



Hiding invisible objects in the drawing tree



In the object list displayed in the drawing tree for an associative view, all model components are shown by default. However, not all of them are visible in the drawing view.

You can now hide invisible components in the drawing tree, making it easier to work with drawings of large assemblies.

DRAWINGS

1. Entering vertical text into the Title block cells
2. Surface finish symbol: enclosing a sign in parentheses, adding an asterisk
3. Work with macro elements: excluding objects, adding objects to the macro element while inserting them from the clipboard
4. Creating references to assembly component properties in the drawing
5. Linking model components with drawing objects

1. In the new KOMPAS-3D version, you can enter vertical text in title block cells.
2. You can now enclose a surface finish symbol in parentheses and add an asterisk after it
3. We've added several new features for macro elements:
 - A new command allows you to exclude objects from a macro element
 - Object can now be added to a macro element when they're pasted from the clipboard
4. In associative drawings, you can create references to the properties of assembly components.
5. You can now link assembly components with drawing objects, such as balloons and geometry.



MORE |  KOMPAS -3D v25

PERFORMANCE



1. Switching between current version/variant
2. Generating family instances
3. Creating a rounding surface
4. Working with complex geometry

Performance improvements (Faster operation, reduced memory usage, and smaller file sizes).

- We've made switching between the current version and a variant of an assembly component faster.
- Generating family instances is now faster
- We've significantly improved the performance of the **Rounding Surface** command. The greatest improvement can be seen when the two base surfaces contain a large number of faces..
- We've optimized the data structure for models with complex surfaces and a large number of dependent objects, such as points or holes. As a result, memory usage while working with these models has been reduced, and the files now require less disk space..

IMPORT | EXPORT

1. Import DXF/DWG using C3D functionality
2. Export documents to *.SVG
3. Import models from proprietary formats: text-reading settings
4. Import assemblies from *.STEP: selecting a component data source

1. Importing and exporting DXF and DWG documents is now performed using a proprietary component.
2. We've added the ability to save graphic documents in SVG format.
3. When importing models from proprietary formats, you can now configure how text in dimensions, symbols, and other objects is imported.
4. When importing assemblies from the STEP format, you can now choose the source of component data. You can import component attributes either from the assembly or from the component files.

© KOMPAS-3D v25

