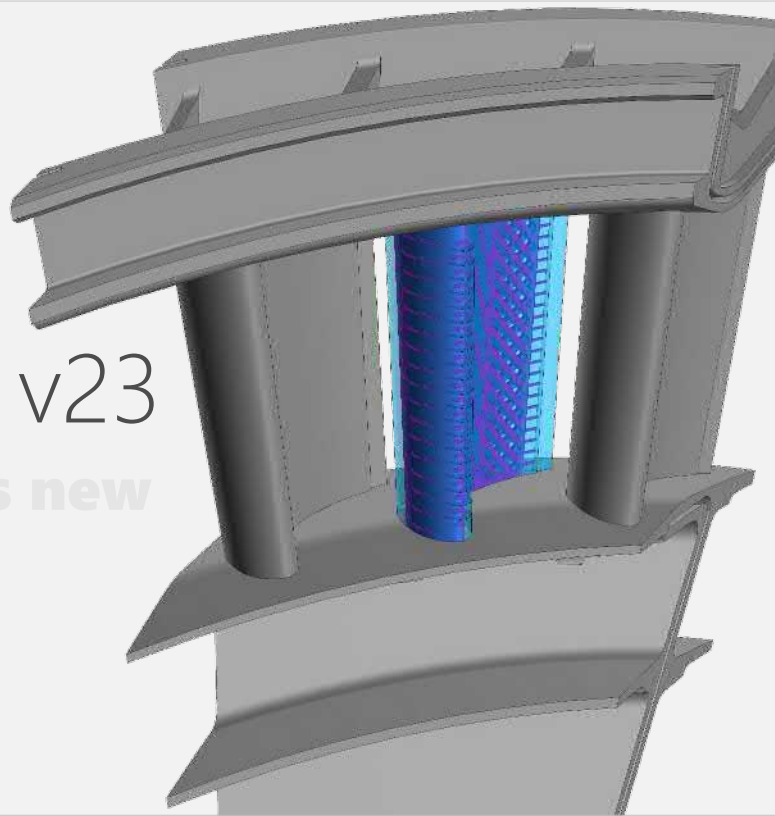




KOMPAS-3D v23

What's new





BASIC FUNCTIONALITY

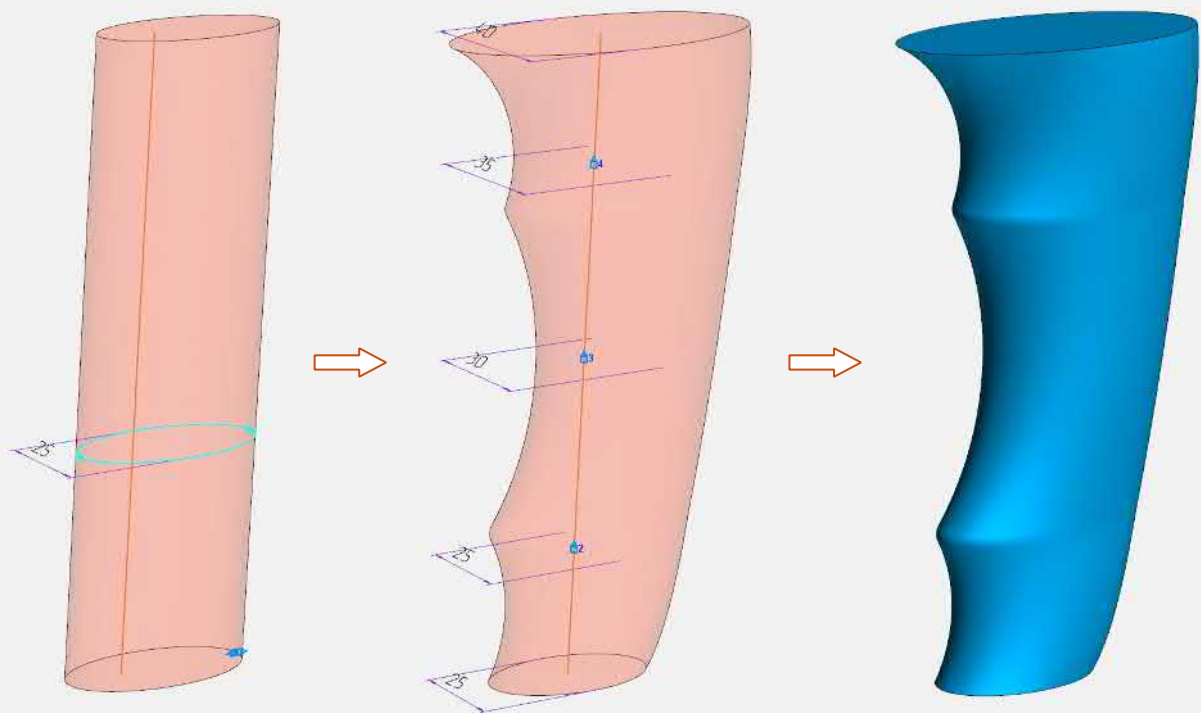
Let's start by changing the basic capabilities when working with models, drawings, and specifications.



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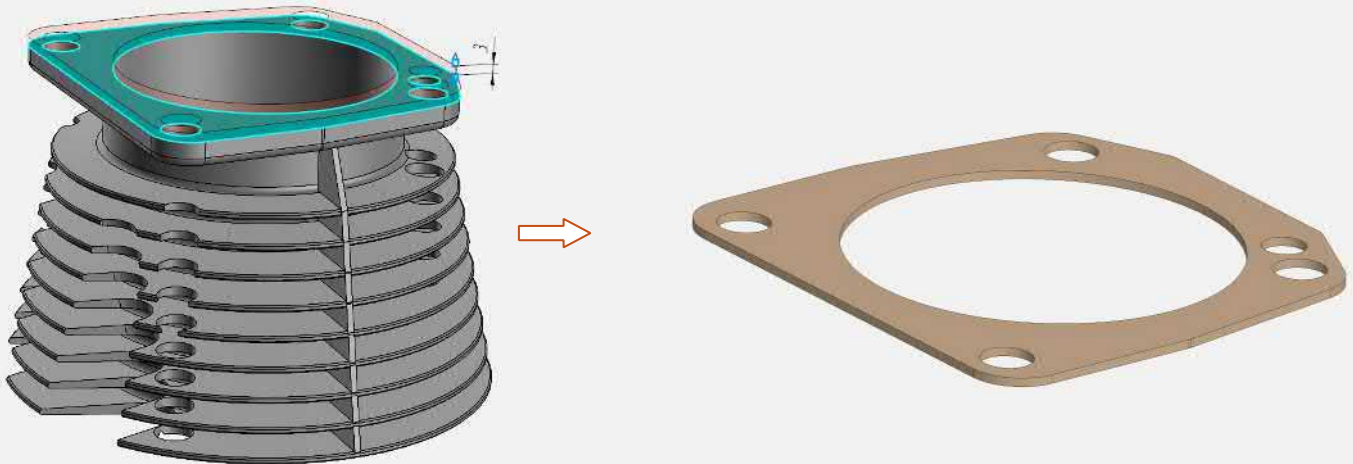
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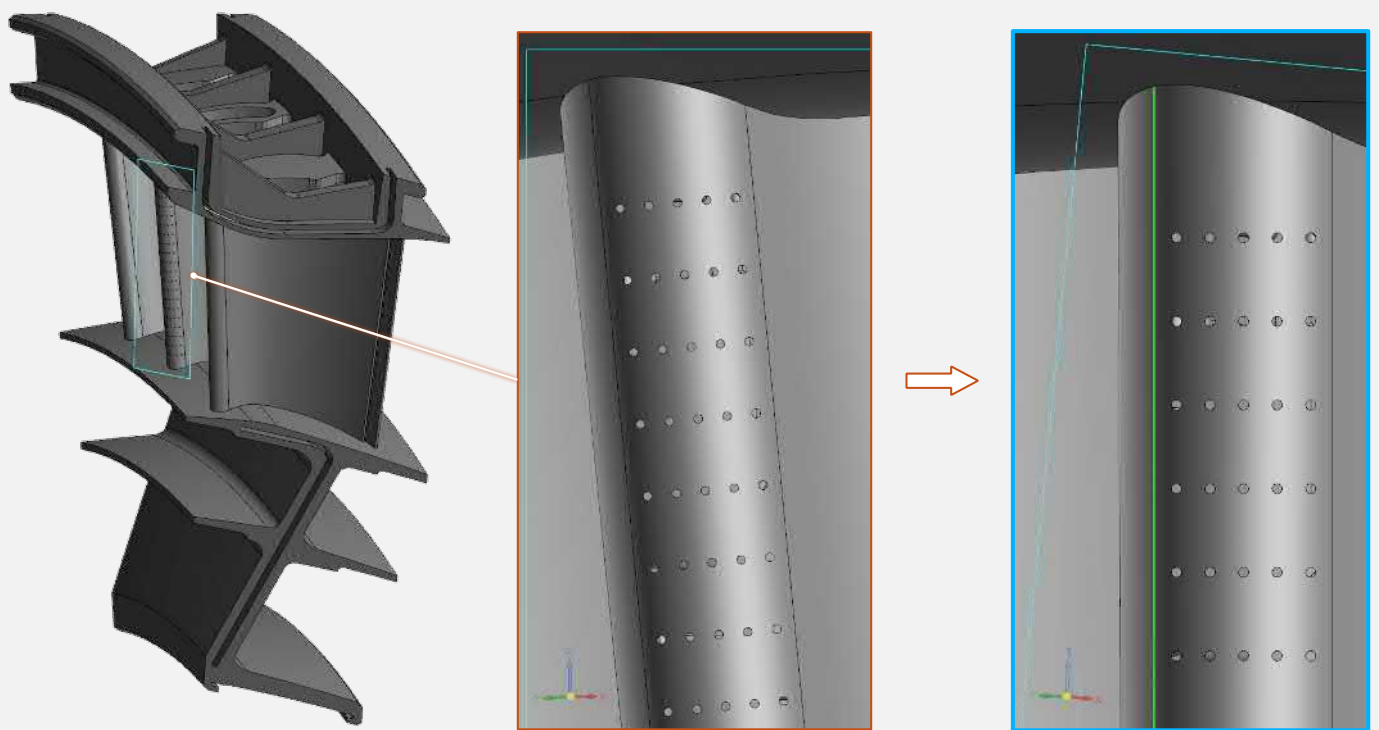
Control of the shape of the section of the Element along the path

You can change the shape of the section as it moves along the trajectory. The shape of the section can be controlled, for example, by changing variables in the sketch. Using the example of such a handle, it can be seen that the initial data were: the trajectory in the form of a straight line and the cross section itself with a size of 25 mm (at any point of the trajectory). Then, being in the "Element by path" command, without interrupting the command, we build the required number of points on the trajectory, and at each point we change the value of the variable. Thus, we get the result much faster and easier, because we do not need to create separate planes for sketches and the sketches themselves.



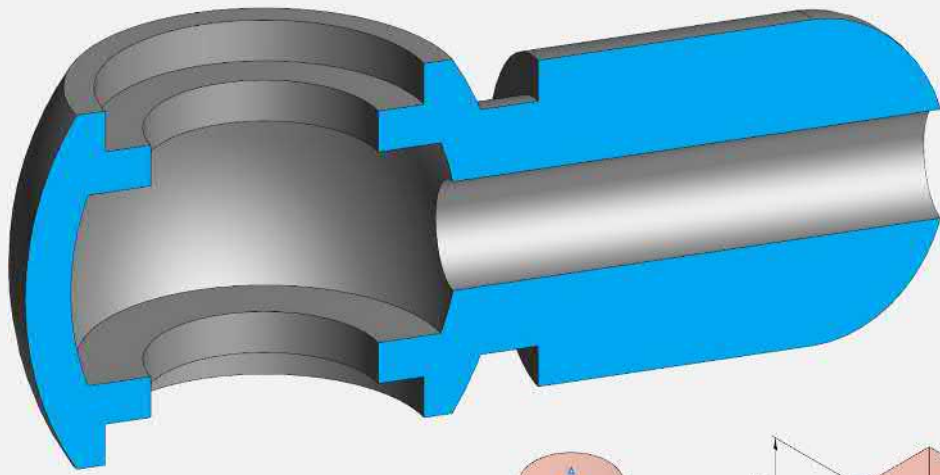
Delete solid/face

The new command is Delete solid/face. For example, it can be used to remove the results of intermediate constructions that are no longer needed. The command also allows you to remove unnecessary bodies or surfaces from the imported component. Example – creating a gasket on the surface of the cylinder. The cylinder is an imported component with no history of modelling. Next, we add thickness and remove the excess cylinder body. At the same time, the connection with the source is preserved, because the cylinder is not actually removed, but, as it were, excluded from the calculation.

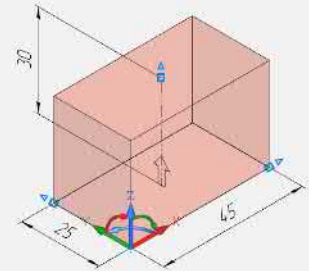
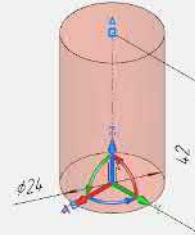
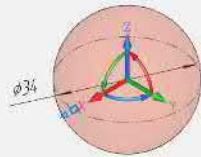


Normally with alignment

The new command changes the orientation of the model Normally with alignment. The command sets the orientation in two steps: first, the specified plane becomes parallel to the screen (as in the Normal command), and then a straight line is indicated, relative to which the model is "rotated" horizontally or vertically. The final orientation can be saved as a custom orientation and accessed if necessary.

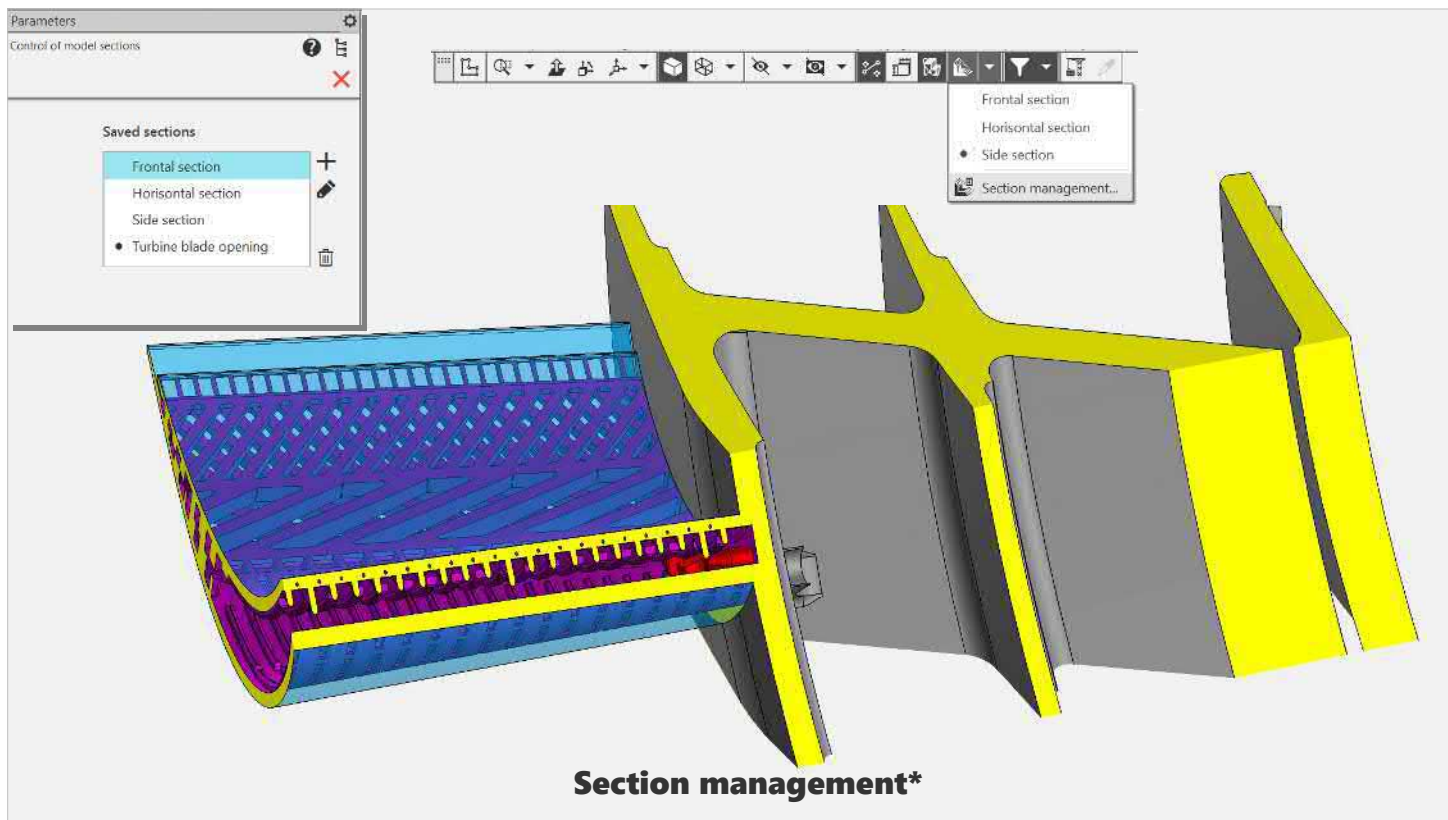


-  Box by a point and three dimensions
-  Box by two points and height
-  Cylinder
-  Sphere



Elementary bodies

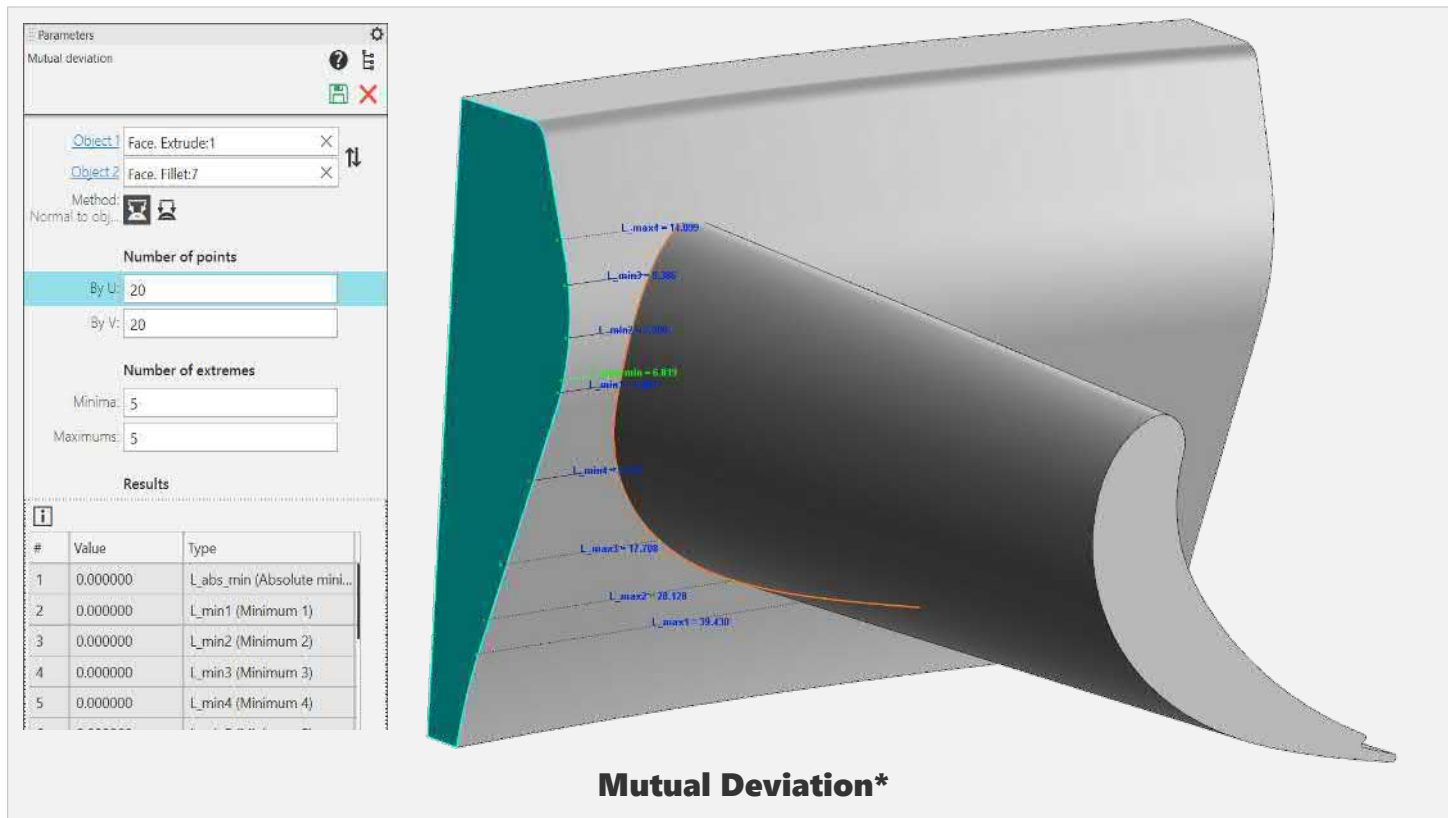
The so-called **Elementary bodies**. It is now possible to quickly build standard bodies according to specified parameters without source data (sketches, axes, guides, etc.). In version v23, the most popular bodies are implemented – a parallelepiped, a cylinder and a sphere. This Fitting model was obtained using such elementary bodies, which can be seen in the Model Tree. In the future, we plan to add new types of bodies.



The convenience of working with model sections has been improved:

- the button on the Quick Access Toolbar contains a list of sections and a command to start the control process;
- the control itself is now performed not in the dialog, but in the process on the Parameters Panel.

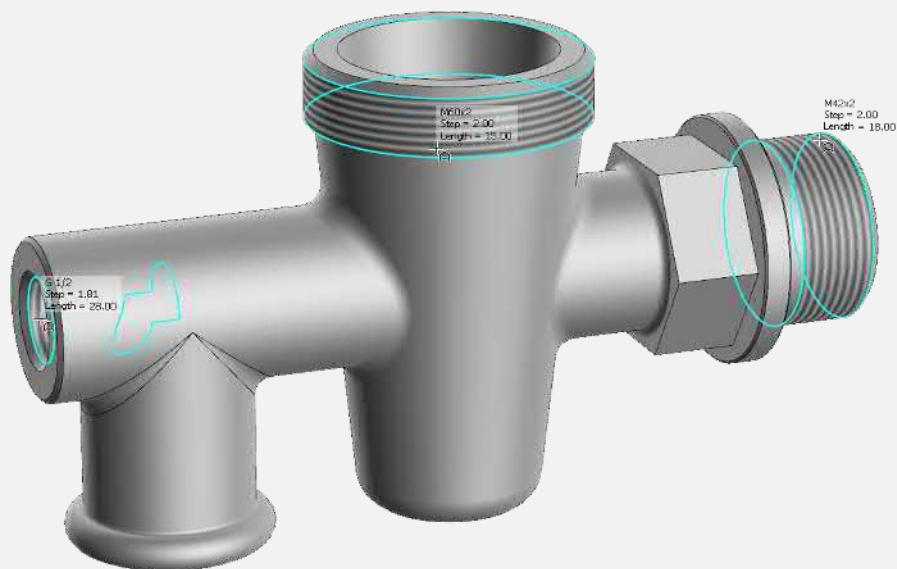
The asterisk (*) in the name of the new product means that the team is NOT new, but has been improved/refined.



A number of changes in the model diagnostic commands.

The capabilities of the Mutual Deviation command have been expanded:

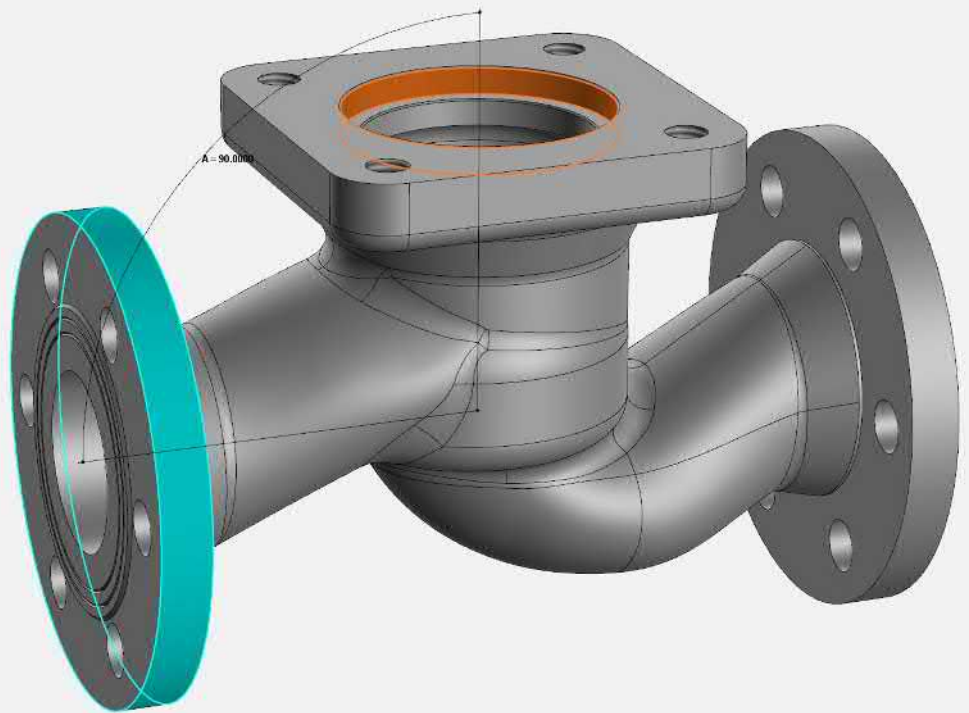
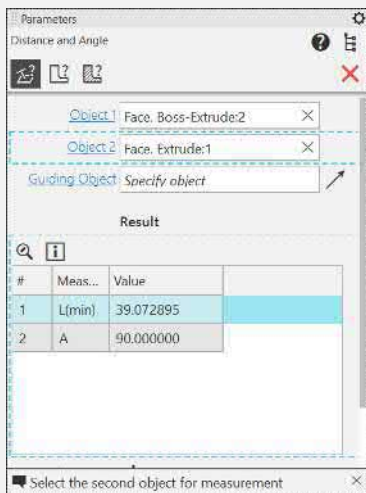
- the distance between points can be measured; the absolute minimum has appeared;
- the measurement results are displayed in a table in the Parameters Panel and in the graphical area;
- the measurement parameters are now stored in the Model Tree, i.e. you can update the measurements after making changes to the geometry.



Object info*

The ability to display thread parameters has been added to the **Object info** command.

Hover the cursor over the conditional image of the thread – we will get the necessary information on the thread in the Graphic area, and when clicked – in the information window.

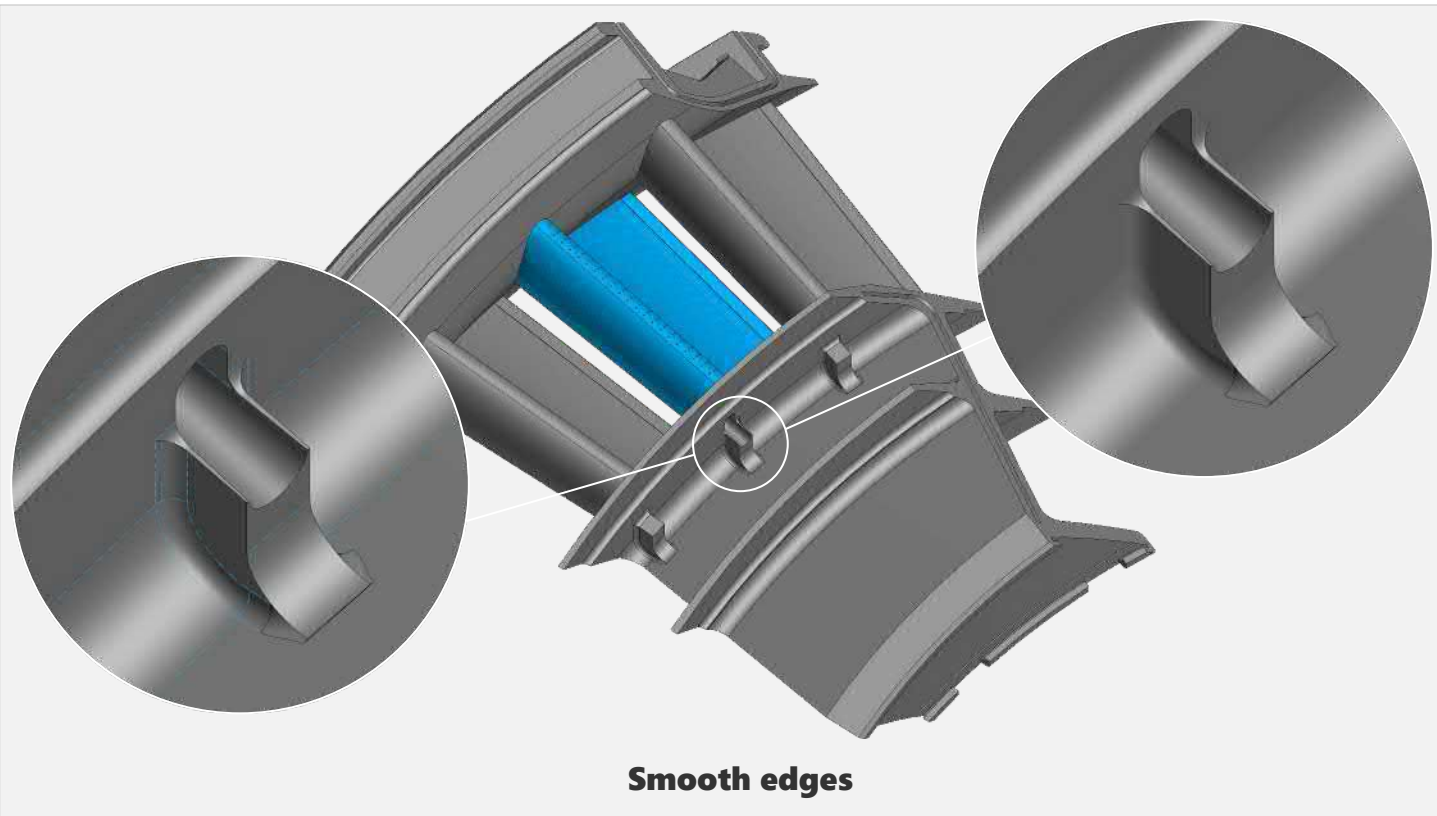


Distance and angle*

In the measurement command "Distance and angle":

- the measurement results are now displayed in the Parameters Panel;
- the calculated angle is now displayed in the Graphics area;

And in general, the result is more informative in the Graphics Area (in terms of "how exactly it was obtained").



Another interesting change that can also be used as a diagnostic tool is the display of **Smooth edges**. This refers to the edges between such faces (or surfaces) that meet each other tangentially (i.e. smoothly merge into one another).

Which edges are considered "smooth" is determined by the allowable angle between adjacent faces (it is adjustable from 0 to 5 degrees).

You can customize the style and color of smooth edges (the example is shown on the left) or disable their display in the model (on the right).

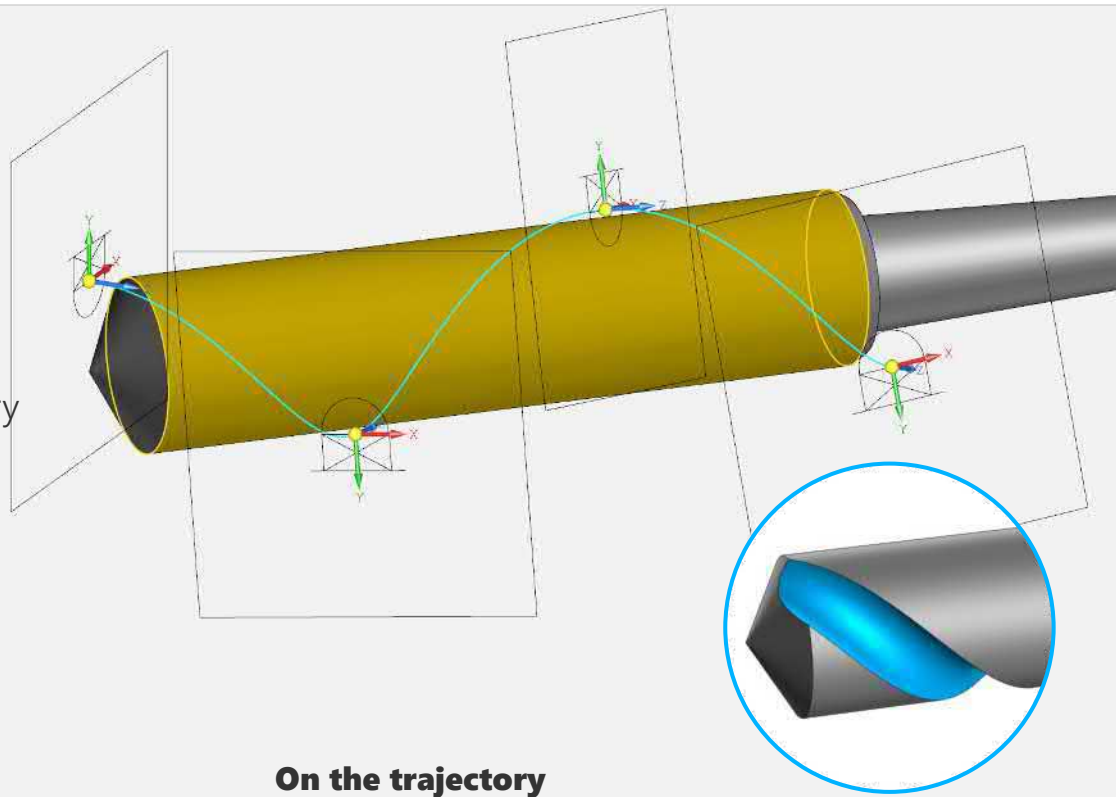
Thus, you can visually track collisions in constructions.



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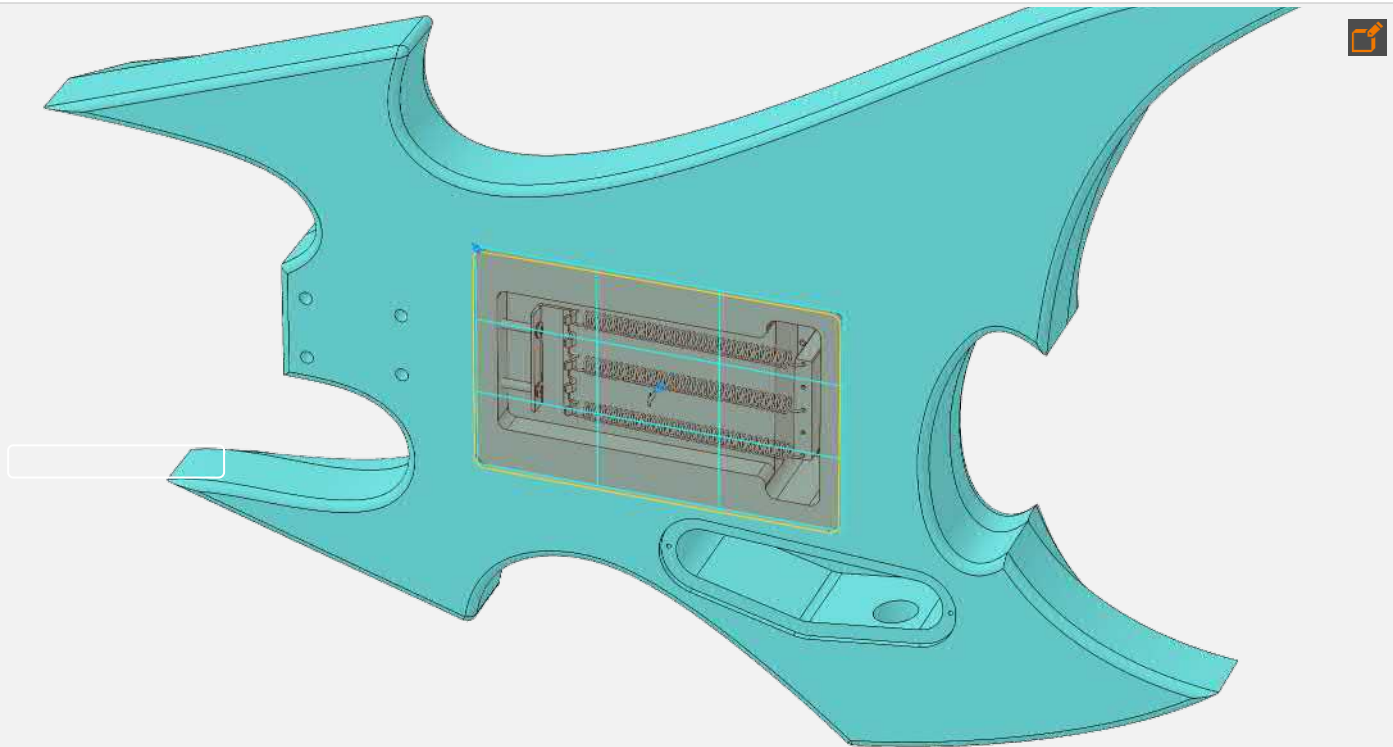
- ✓ On the plane
- ✓ On the trajectory
- ✓ Fixed



On the trajectory

There are three ways to place the sketch in the "Sketch Placement" command:

- on the specified plane or flat face (was earlier);
- fixed (was earlier) — cancels the connection of the sketch with the reference objects and fixes its current position in space;
- "on the trajectory" method is shown on the slide — here the sketch is placed on a plane perpendicular to the specified curve at the selected point (the same point will be the origin of the sketch coordinates).

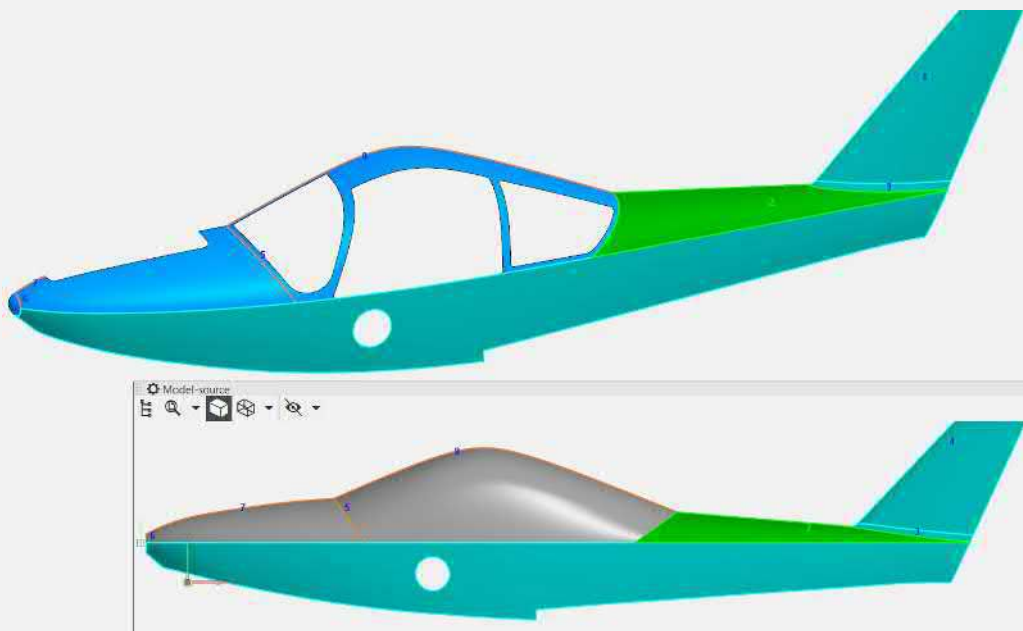
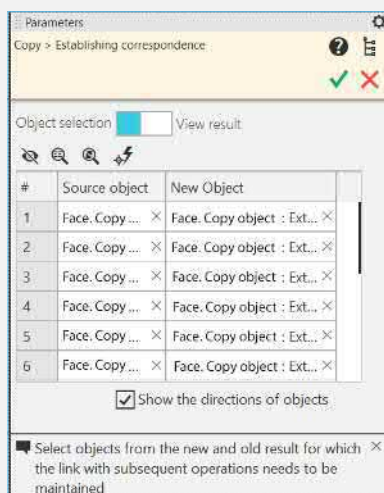


Operations with sketches of components in an assembly

The so-called "external" sketches. Previously, the sketch of the operation must necessarily belong to the same model in which the operation is performed. Now, if the operation is performed in an assembly, then you can use sketches of components. And vice versa. On this slide, we are in the editing mode of the "Back cover" component in the context of the head assembly. But as a contour for the lid, we take a sketch from another component – the "Electric guitar Deck".

Working with such "external" sketches has some limitations:

it is impossible to resize the sketch and start editing it inside the process where this sketch is involved, and it is also impossible to use separate areas of the sketch (you can only use the entire sketch).



Set a match when changing the source in a copy geometry

A key novelty when working with copying objects! Now you can replace the "old" copy source with the "new" one and specify the correspondences between them. What for? So that the model objects associated with the old source do not lose their reference objects. For example, the first copy of the Fuselage was without cutouts for window openings, and the second copy was with cutouts.

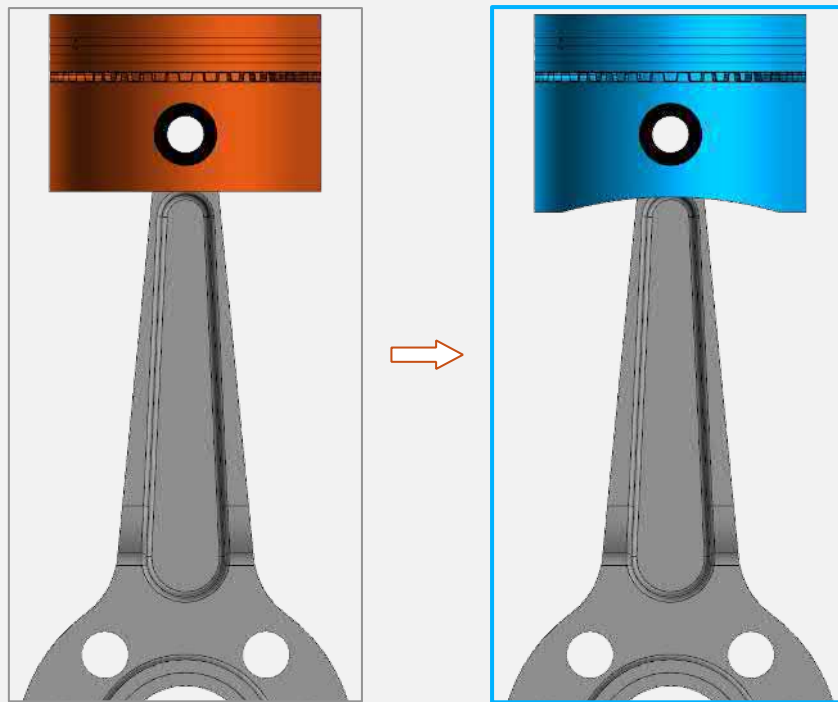
Or vice versa. The geometry has changed, so it is necessary to specify the correspondence between the old and new copies. This can be done directly in the Graphics Area by selecting a pair of matching objects. During the installation process, you can make a "preview" of the result to identify potential errors. This way, you can reduce the number of errors and reduce the time to finalize the model after editing.

The ability to set correspondences is also available when editing extrusion, rotation, trajectory, and cross-section operations.



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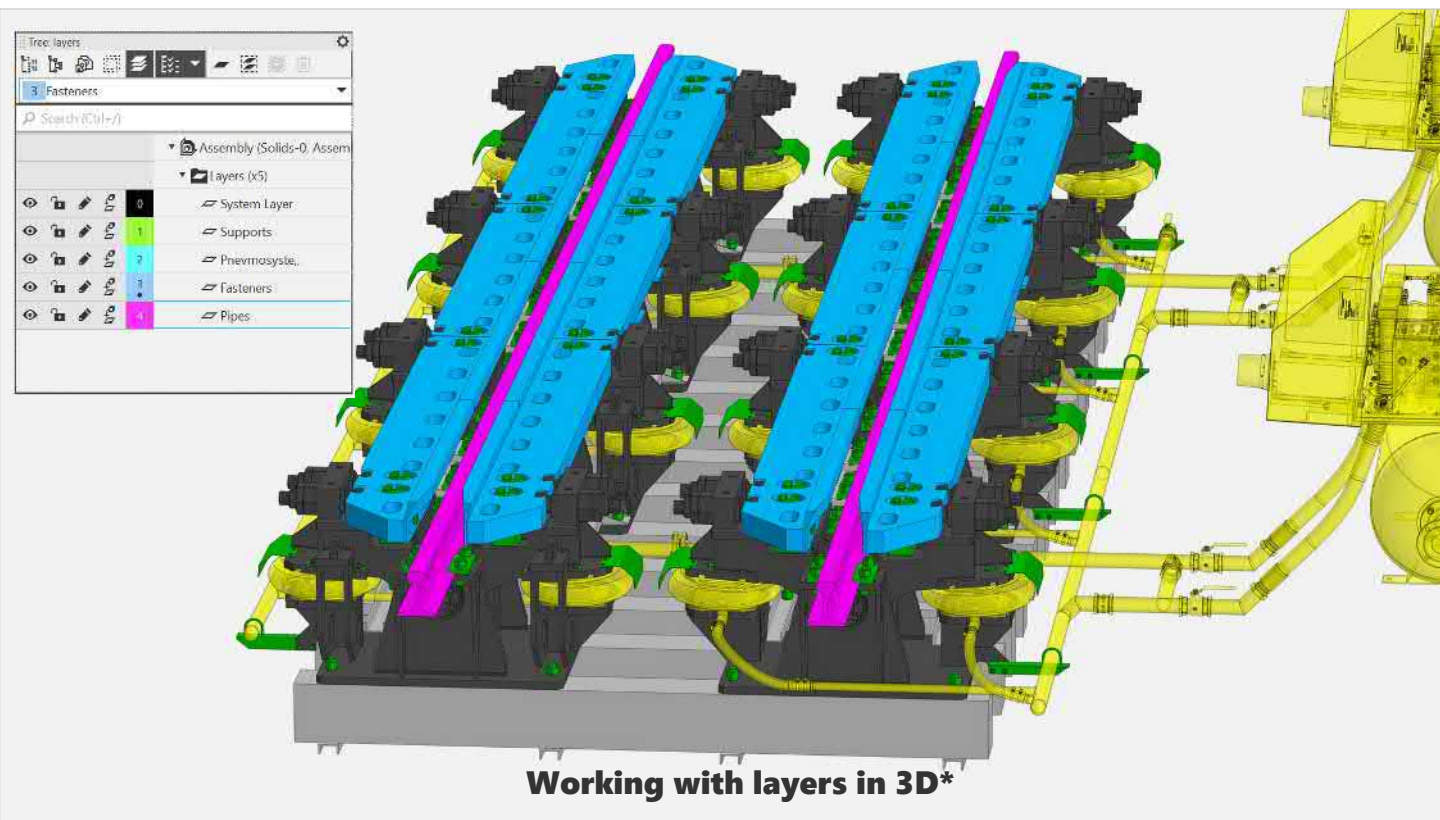




Replace components

The new "Replace components" command.

It allows you to set a new source file for one or more assembly components at once. Similarly to the previous slide, you can set up correspondences between the original and new elements. Due to this, after replacement, the operation of the interfaces will not be disrupted, and the operations performed in the assembly will not lose the objects on which they relied.



The work with layers has been improved:

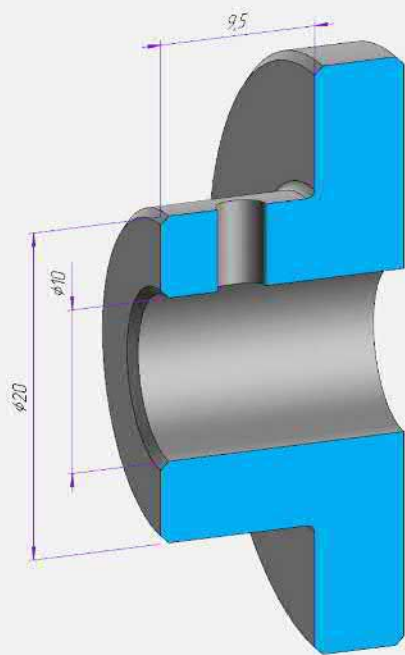
- A "Layer Tree" has appeared on the Tree Panel to replace the Document Manager.
- The layer can be active or background. You can work with objects on the active layer as usual, and objects in the background layer are not available for selection (except for diagnostic objects).
- When inserting a component into an assembly, it can be placed on source layers, then the component layers will be transferred to this assembly.

<https://best.ascon.ru/gallery/31083/> - Universal pneumatic retarder with a set of control equipment

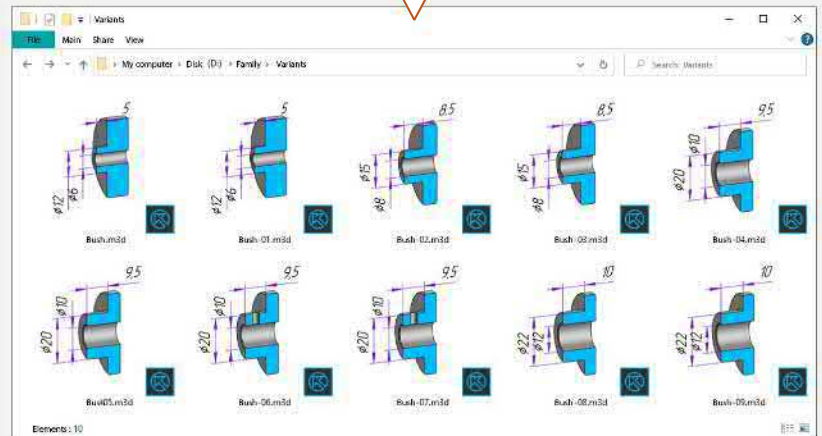


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Variant	Designation	Name	D	d	h	v19
Bush	ABCD.818222.033	Bush	12	6	5	1
Bush-01	ABCD.818222.033-01	Bush	12	6	5	1
Bush-02	ABCD.818222.033-02	Bush	15	8	8.5	1
Bush-03	ABCD.818222.033-03	Bush	15	8	8.5	1
Bush-04	ABCD.818222.033-04	Bush	20	10	9.5	1
Bush-05	ABCD.818222.033-05	Bush	20	10	9.5	1
Bush-06	ABCD.818222.033-06	Bush	20	10	9.5	0
Bush-07	ABCD.818222.033-07	Bush	20	10	9.5	0
Bush-08	ABCD.818222.033-08	Bush	22	12	10	1
Bush-09	ABCD.818222.033-09	Bush	22	12	10	1



Model Family

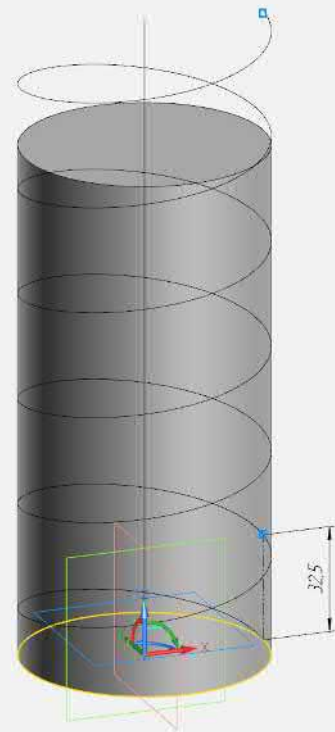
The ability to automatically create a set of new models based on a template model has been implemented. There is a "Bushing" model with several variables.

We create an electronic spreadsheet in Excel or OpenOffice, describe variable data for future models in rows – these are the values of variables and properties – and then the KOMPAS-3D will automatically form a "family" of models based on these data.

After that, the family files will be read-only.

You can edit them after changing the spreadsheet and rebuilding the family through the original model. By the way, when creating a family, you can run a table check for errors.

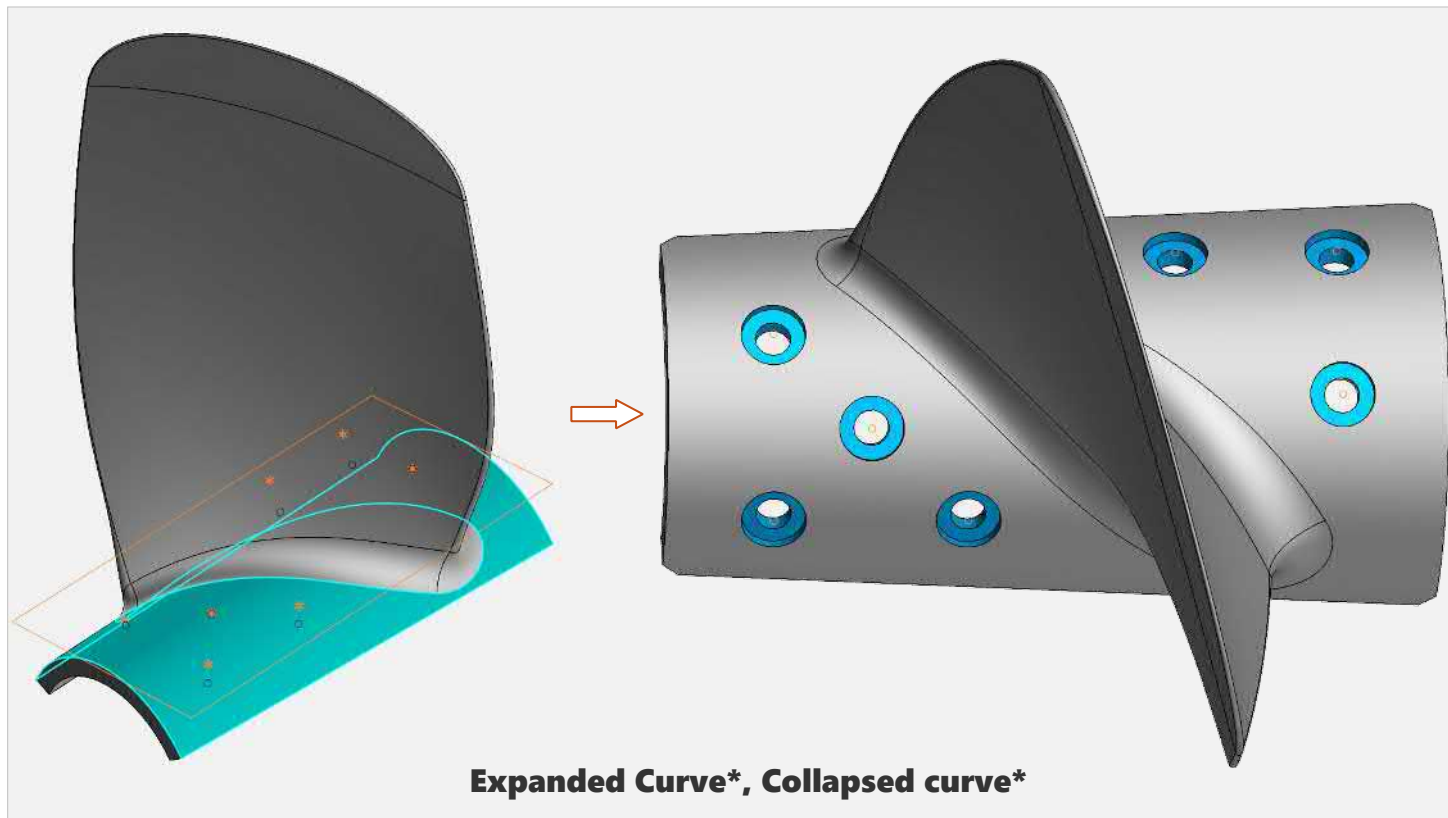
- ✓ New placement methods
- ✓ Creating an axis
- ✓ Choosing a coordinate system
- ✓ Orientation control by variables



Cylindrical spiral*, Conical spiral*

A number of improvements in the field of frame-surface modeling.

- The processes of building cylindrical and conical spirals have been redesigned, here are some: the placement of the spiral now resembles the process of building and placing LSCs;
- added variables that control the coordinates of the base point of the spiral;
- you can immediately create a spiral axis and select the SK, relative to which the spiral placement parameters are set.

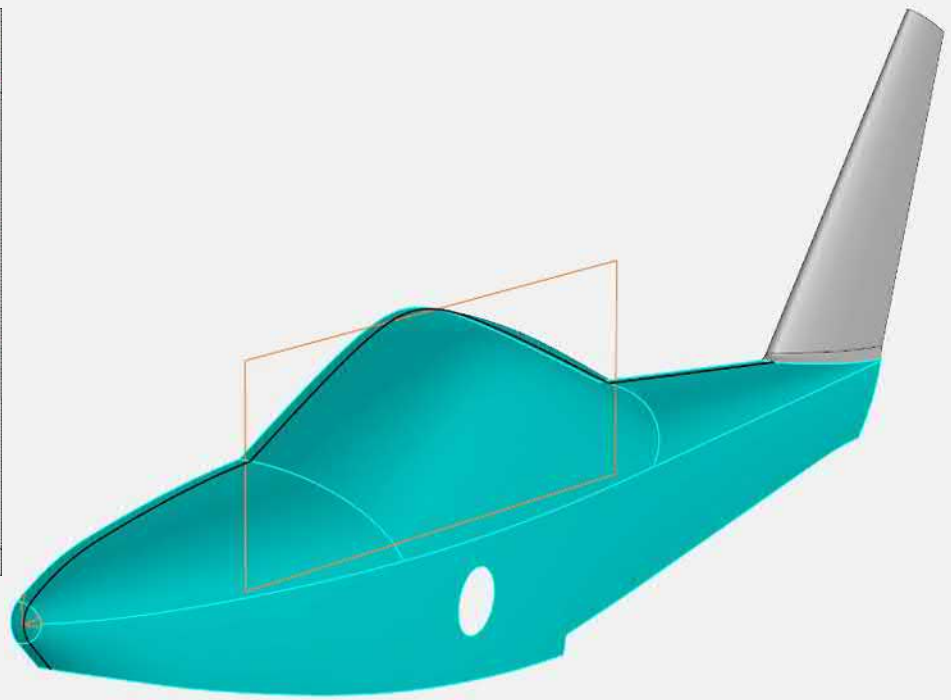
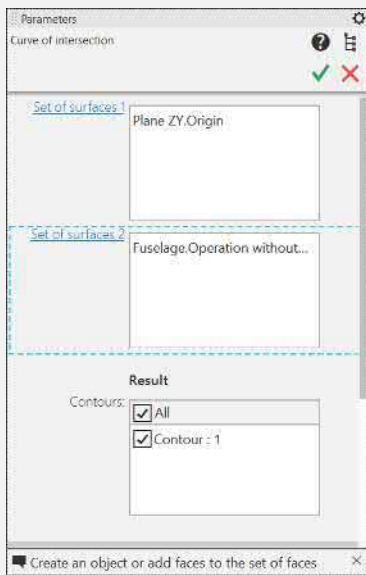


The novelties of the two teams that appeared in the previous version of KOMPAS-3D.

Now they allow you to work with dots as well.

- 1) In the Expanded Curve command, points from the surface will be transferred to the plane.
- 2) The command has a collapsed curve – on the contrary – from the plane to the surface.

This slide shows the removable propeller blade. To model the mounting holes, a conical surface was first deployed, points for the location of future holes were created on the resulting plane, and then points from the plane were rolled onto a conical surface.



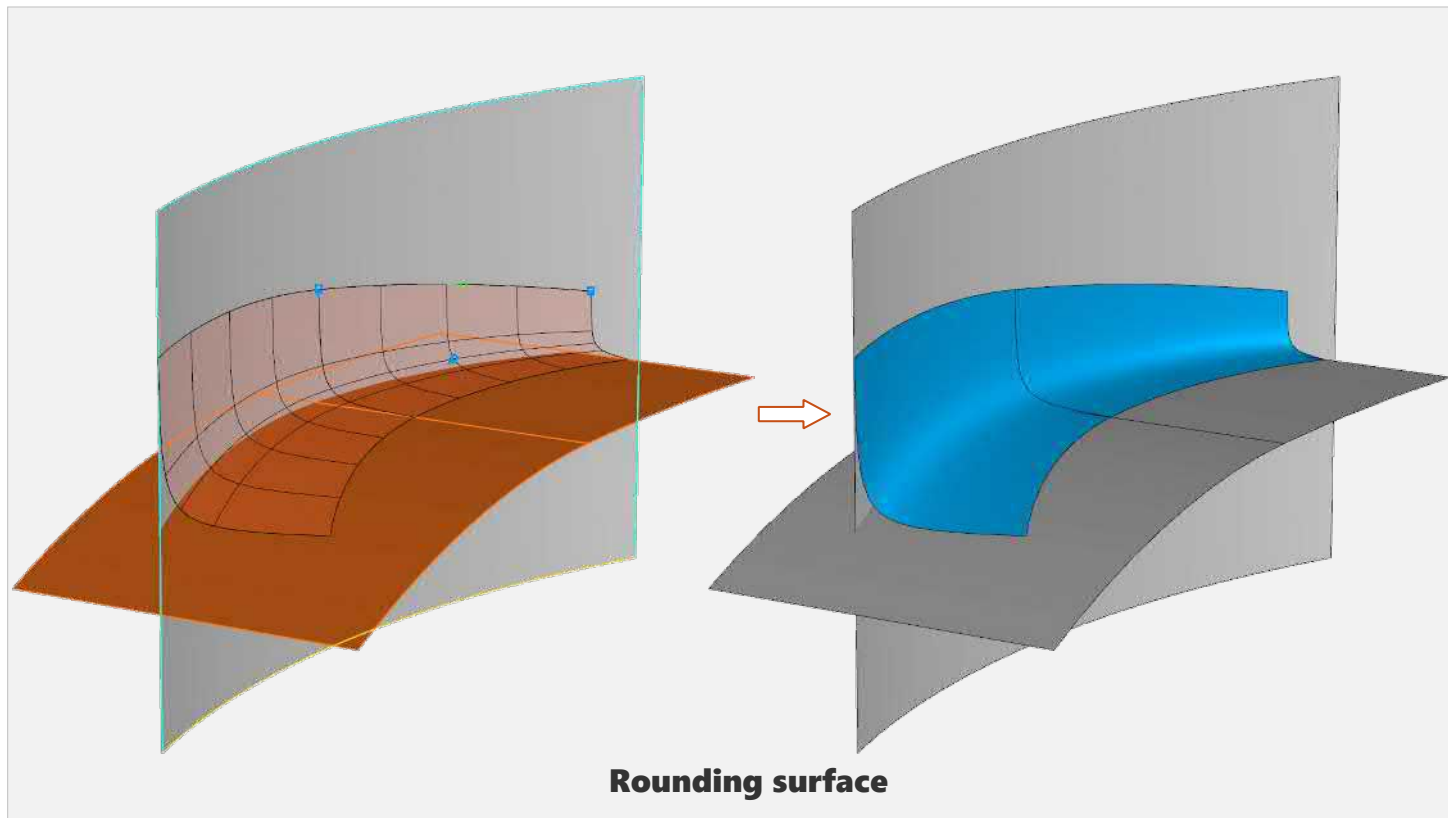
Curve of intersection*

The **Intersection Objects** command has been modified in a similar way.

The **Intersection Curve** command has been finalized: the object in it can be a surface or an entire body.

Previously, individual faces were entered in the list of objects and their disappearance could lead to an error.

Now the intersection curve is correctly reconstructed when the original surface is changed.



One of the most popular commands implemented according to user requests – **Rounding surface**.

This is the transition surface between the two specified surfaces, which may not have a common edge.

A variety of options are available to customize the shape of the **transition surface** and its **sides**.

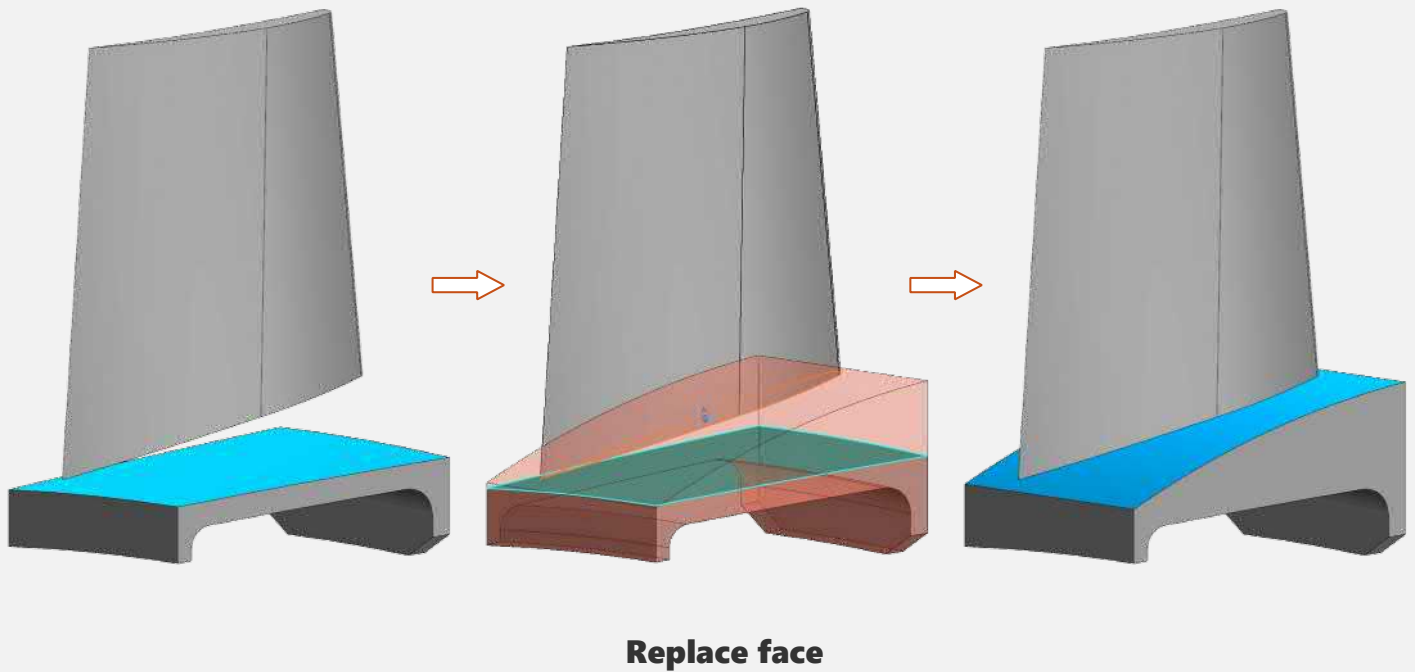


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Direct modelling



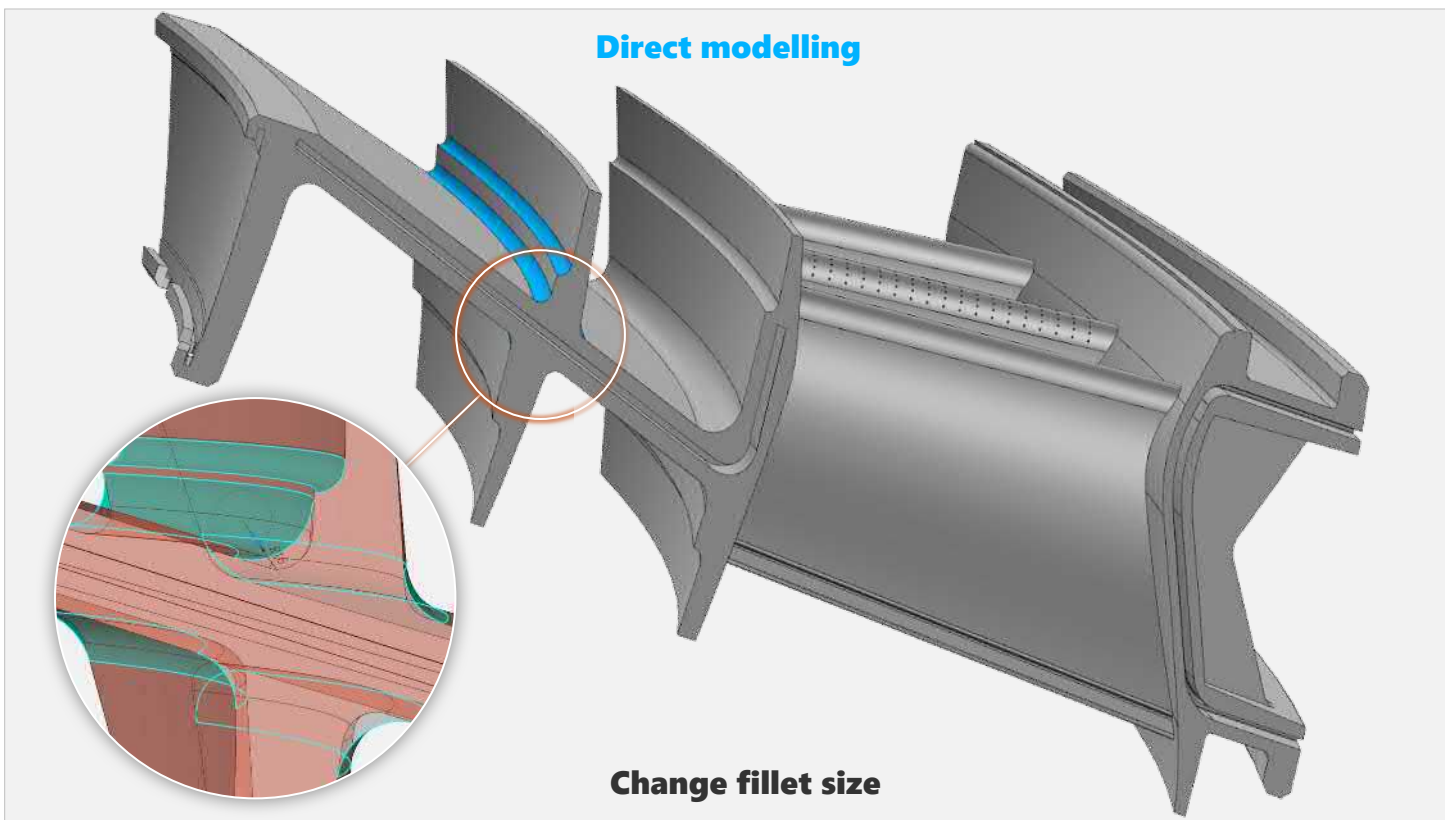
A number of novelties on the topic of **direct geometry modelling**.

These commands are mainly designed to modify imported models without a build history, but they can also be applied to models with a history.

The new command is to **Replace face**. It allows you to replace the selected face with a new face or plane.

For a new face, you can also set an offset relative to the specified one.

The remaining adjacent faces are extended or trimmed to preserve their integrity.

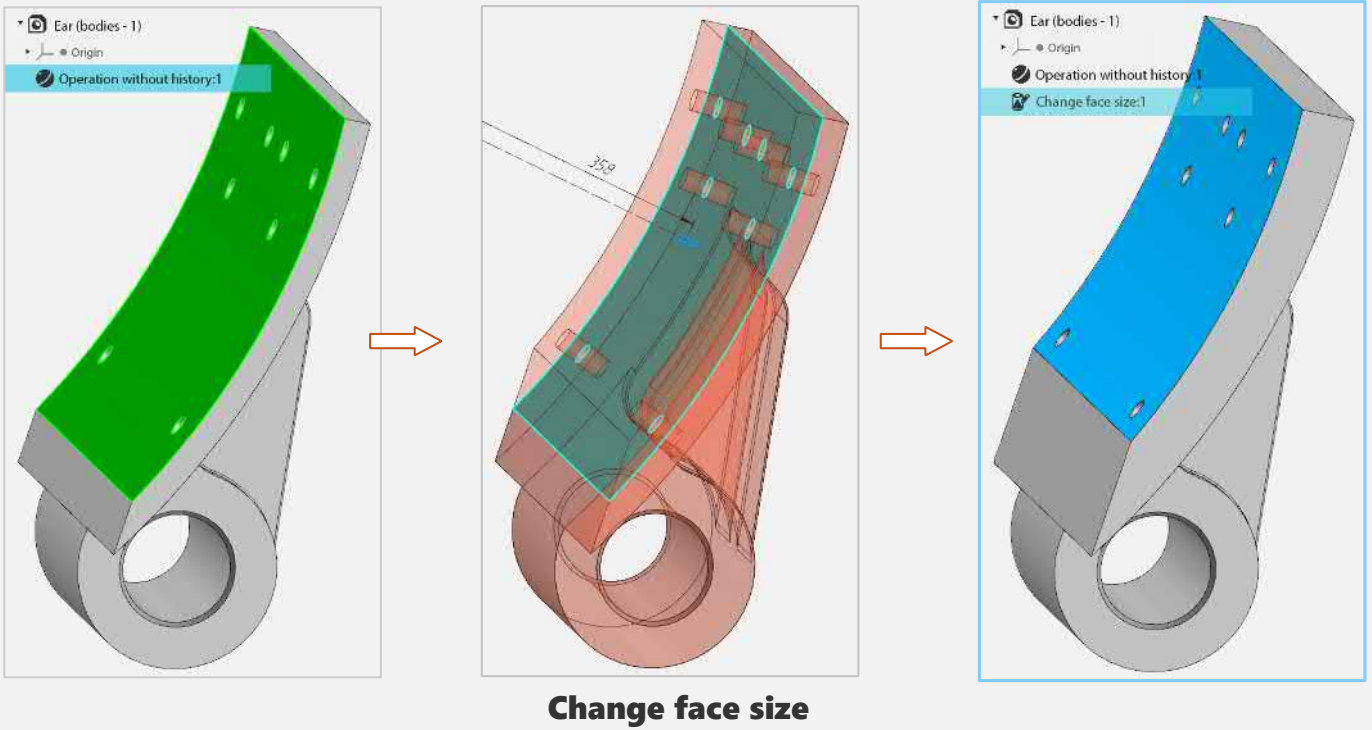


The **Change file size** command allows you to edit fillets with a constant radius.

After specifying the rounding, the current rounding value will be shown, which can be increased or decreased.

The presence of a phantom in the process of direct editing indicates that the new value is correct, i.e. the rounding can be changed.

Direct modelling

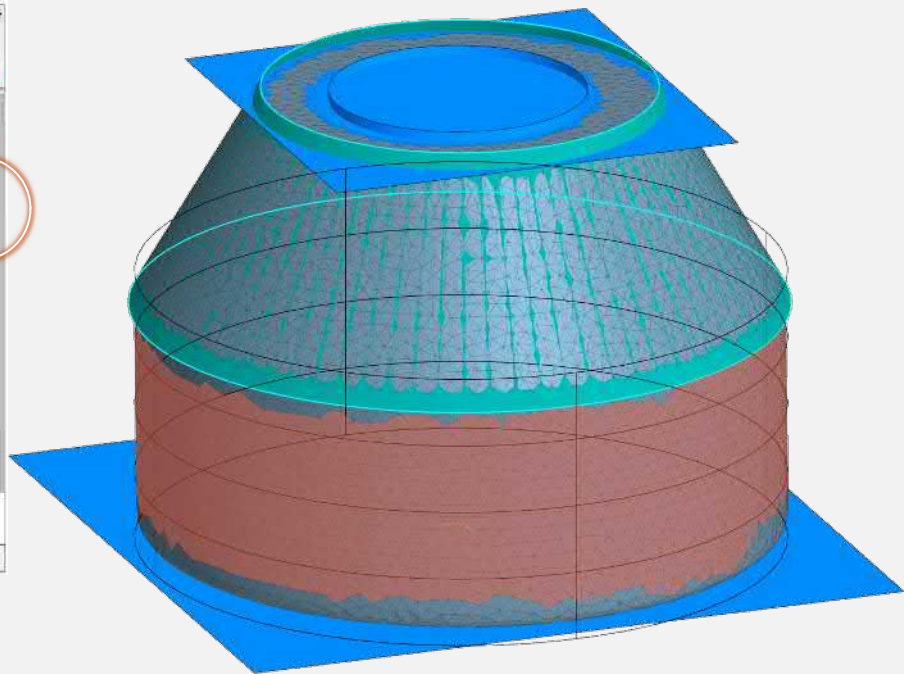


The **Change face size** command allows you to edit the diameter of a cylindrical or spherical face.

On the left, we see a detail without a history and select a cylindrical surface.

Then we reduce its diameter to get the necessary result.

Reverse engineering



Surface adjustment*

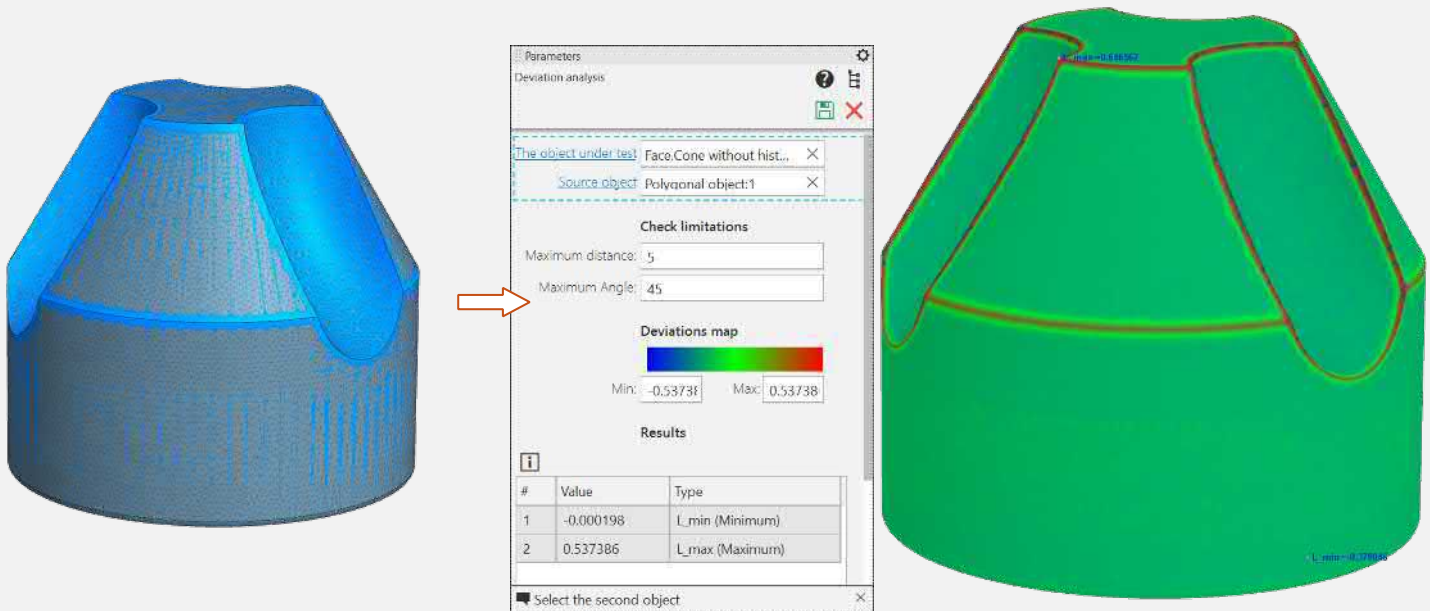
Next is the topic of **Reverse Engineering**. The topic is not new to us, we have been developing it for several versions.

For example, in v22 version, we have a tool for building regular-shaped surfaces in the likeness of a polygonal object.

The **Surface Adjustment** command significantly been upgraded in the v23 version of KOMPAS-3D.

- Firstly, it is possible to **Specify the guiding object** (for example, when creating a cylindrical surface, you can specify the plane to which the axis of the cylinder should be perpendicular);
- Secondly, it is now possible to change the numerical parameter, in this case, the value of the radius of the cylinder.

Reverse engineering



Deviation analysis

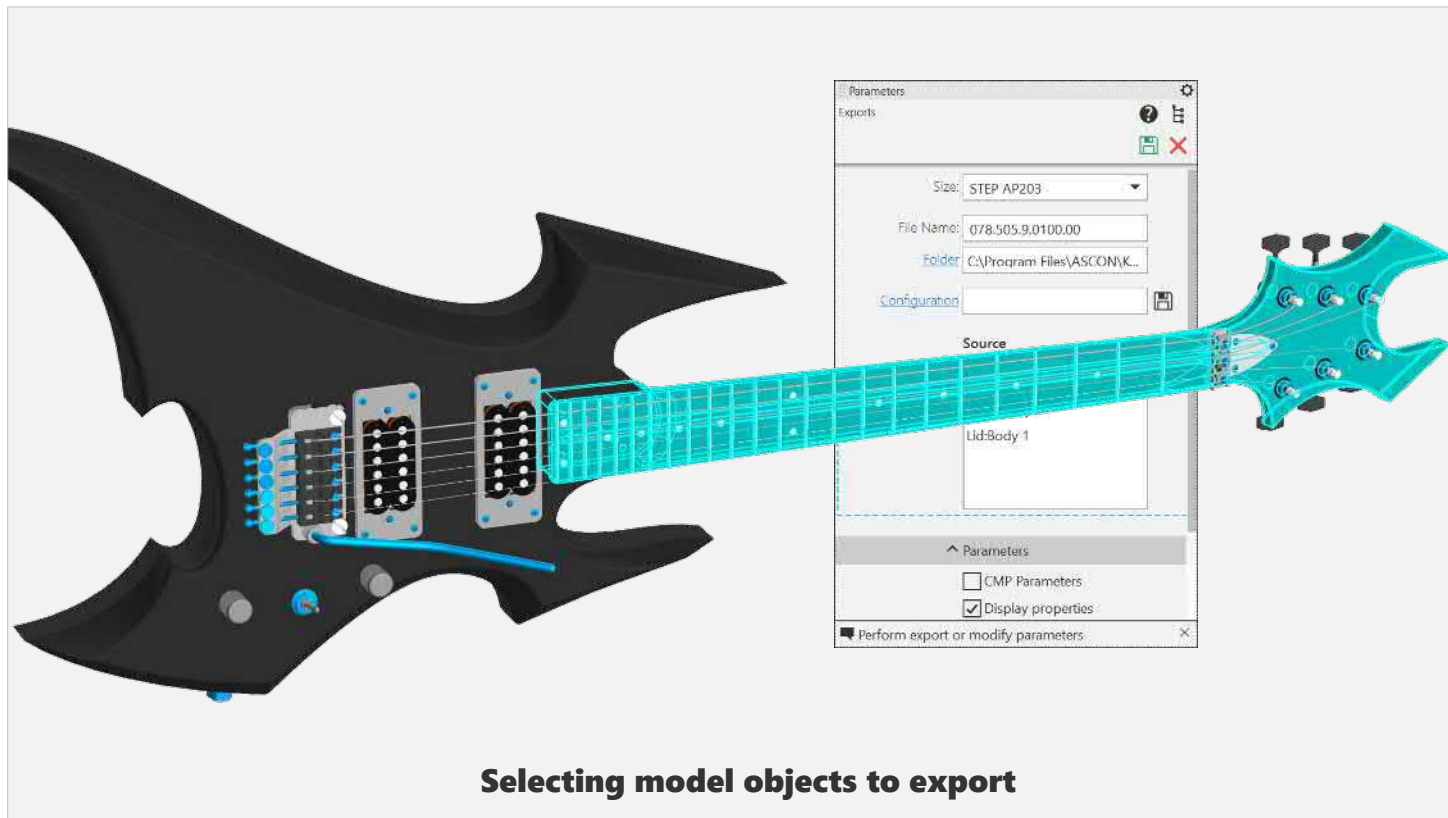
Another key innovation on the topic of reverse is the **Deviation Analysis** feature.

It is used to evaluate the deviation of a scanned or polygonal object from a body, surface, or face.

The result of the analysis is displayed as a color map.

The maximum and minimum deviations are also shown.

The color deviation map can be adjusted, for example, to get a more contrasting map.



Selecting model objects to export

The export of models has been redesigned and improved. Now it is an independent team with a number of changes.

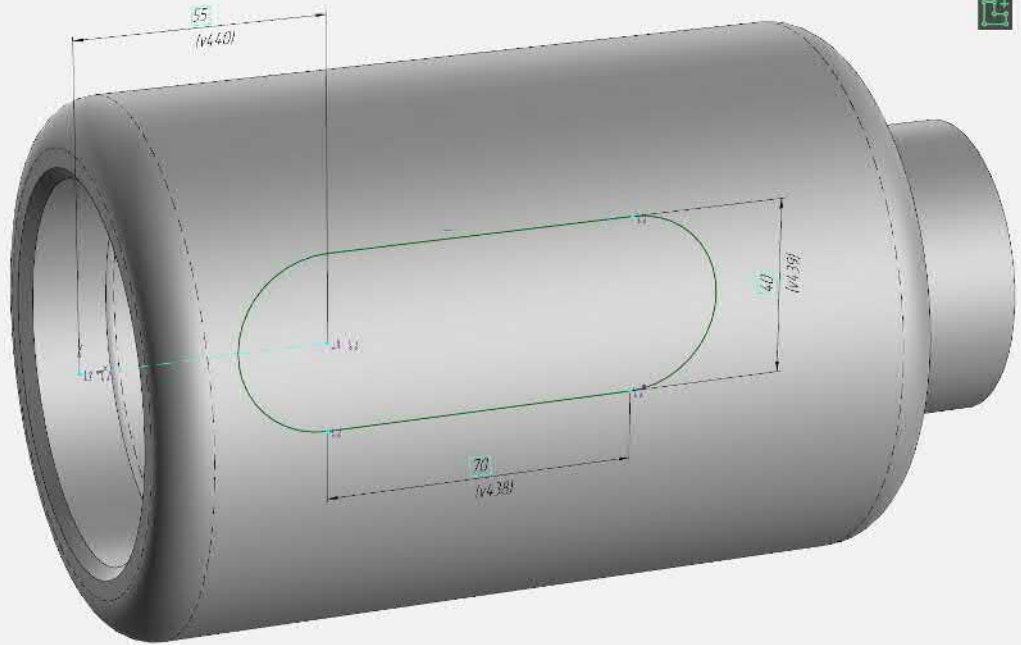
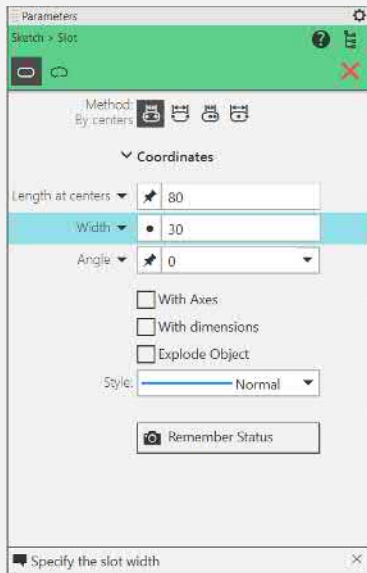
And here, perhaps the most useful thing is that you can specify objects in the model that should be exported. The **Save As** command is still available, but it is not possible to configure the export settings.

We are working in a new way.



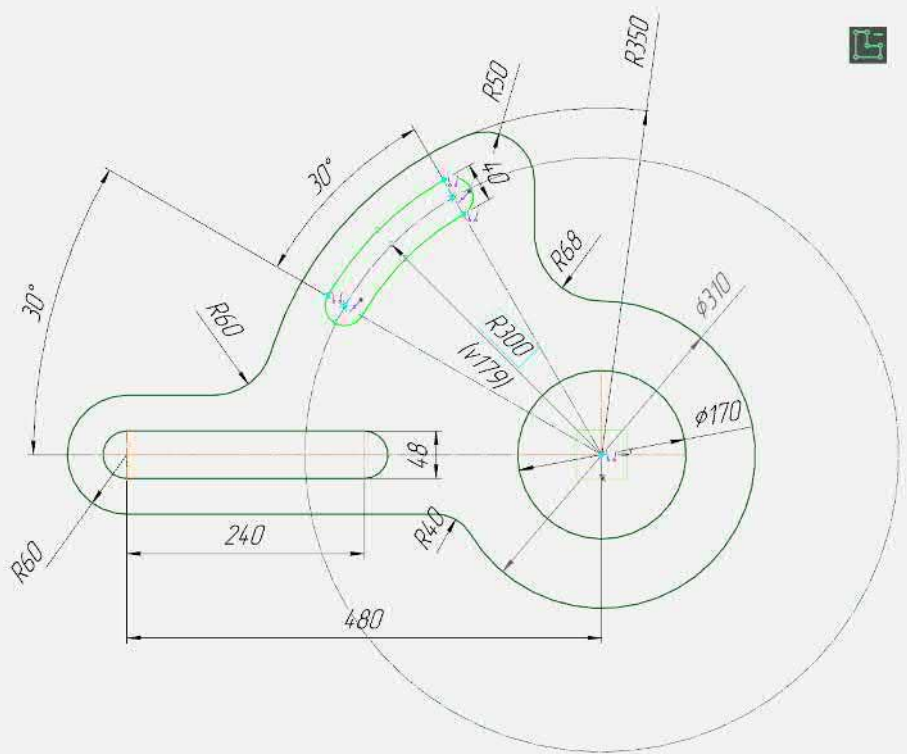
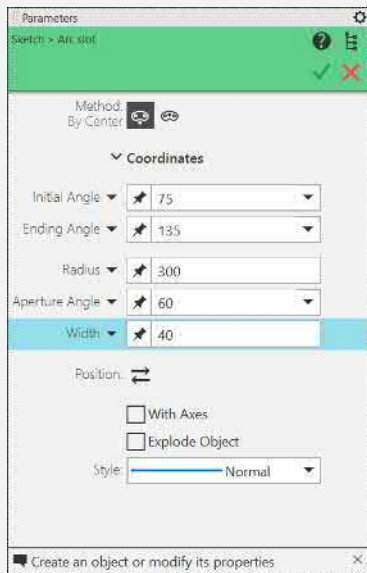
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Slot

We smoothly move on to the topic in 2D design. And here we see the most popular novelty in terms of user requests – the **Slot** creation commands. There are 4 ways to build such a straight **Slot**. There is an automatic alignment of its centerlines and dimensions.

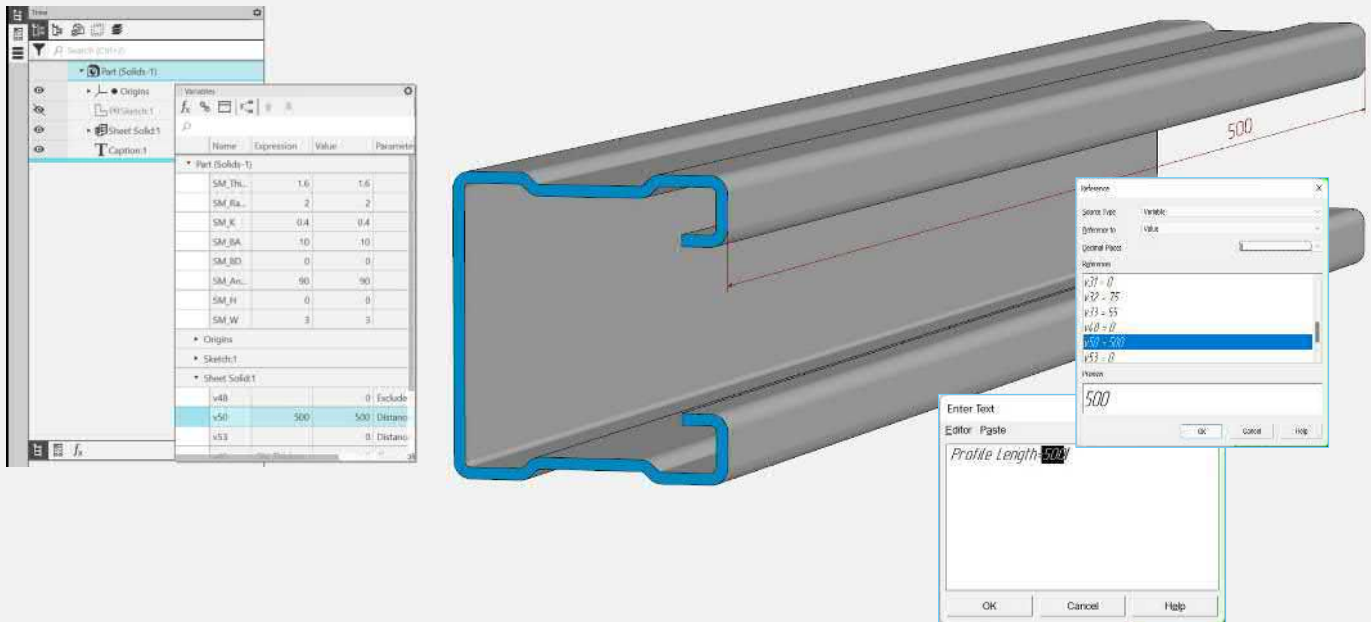


Arc slot

And, of course, the Arc slot!

It has two ways of building.

Both slots are available when working with graphic documents and in 3D models in Sketch mode.



Inserting a group of links into the value of the Name property

Another extremely popular novelty! The value of the "Name" property can now include a **link**, **plain text**, or **multiple links**.

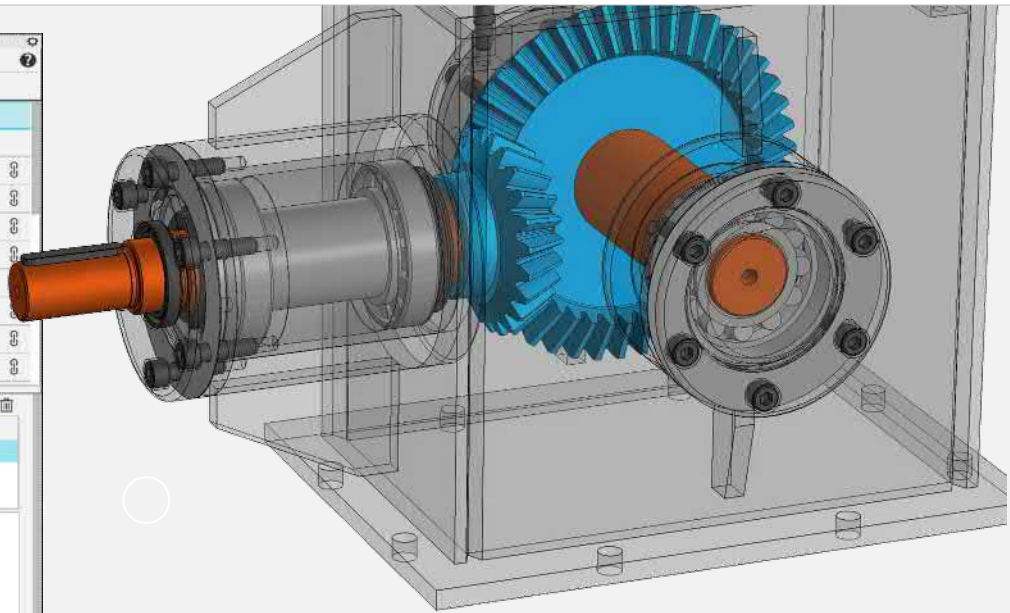
Previously, it was possible to use simple text or a single link – now there are no restrictions.

Creating multiple links in the **Name** may be required, for example, when entering information about a part without a drawing indicating the designation of the material, standard sizes, weight and other properties.

Of course, we are talking about associative references to the properties of the part.

Product BOM			
Name		Designation	
Gear Reducer		078.505.9.0100.00	
Parts (x58)			
Shaft		078.505.0.0101.00	
Gear		078.505.0.0102.00	
Gear		078.505.0.0103.00	
Gear Casl...		078.505.0.0104.00	
Lid		078.505.0.0105.00	
Lid		078.505.0.0106.00	
Bushing		078.505.0.0107.00	
Bushing (...)		078.505.0.0108.00	

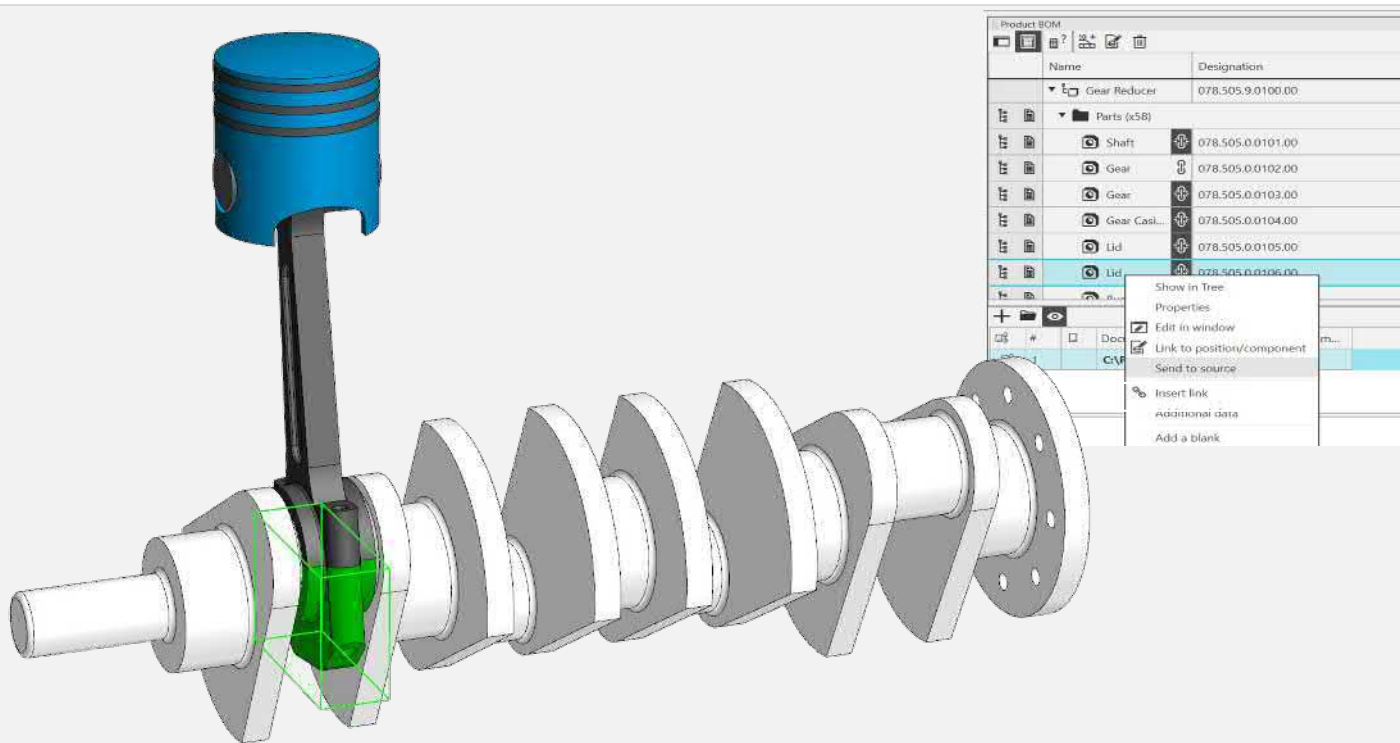
#	Document	Designation	Com...
1	A4	C:\Program File... 078.505.9.0100....	



Automatic transmission of BOM format to the assembly model or assembly drawing

The **Product BOM** panel has undergone a number of useful changes.

When connecting a BOM template to a component part, the format of this document (usually A4) is automatically transferred to this component part.

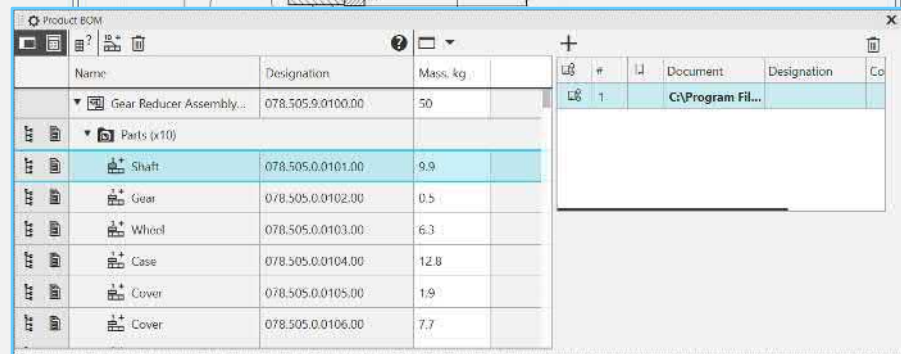


Transferring property values from the Panel to the Product BOM

It became possible to transfer the values of the Designation, Name and Note properties from the product to its component parts.

This can be done for all the components at once. That is, at the beginning of product design, you can not specify properties at all (especially since the Designations may often not be definitively defined), then assign properties on the Composition Panel and transfer them to the source – for example, to a 3D model.

- ✓ Viewing and editing document properties, views, macro elements, and model drawings
- ✓ Creating an assembly drawing without a visual representation



Product BOM Panel in the drawing

The Product BOM Panel is now available for working with the Drawing.

With its help, you can control the properties of the product and its account.

And, by the way, this Panel is now also available in Fragments, but there is less functionality there.



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Additionally

- ✓ View the list of installed modules and their versions
- ✓ Manual sorting of selected document templates
- ✓ Improved mechanism for the operation of geometric object constraints when editing a parameterized image
- ✓ Construction of angle release holes on the flat pattern
- ✓ Associative spline by objects
- ✓ Calculation of the Mass properties of the model Variants according to the basic design
- ✓ Manual input of values of axial and centrifugal moments of inertia
- ✓ Reading NX and SolidWorks models using the C3D core
- ✓ Speeding up model rendering

These are just the most interesting novelties of the basic functionality of the v23 version, but not all of them.

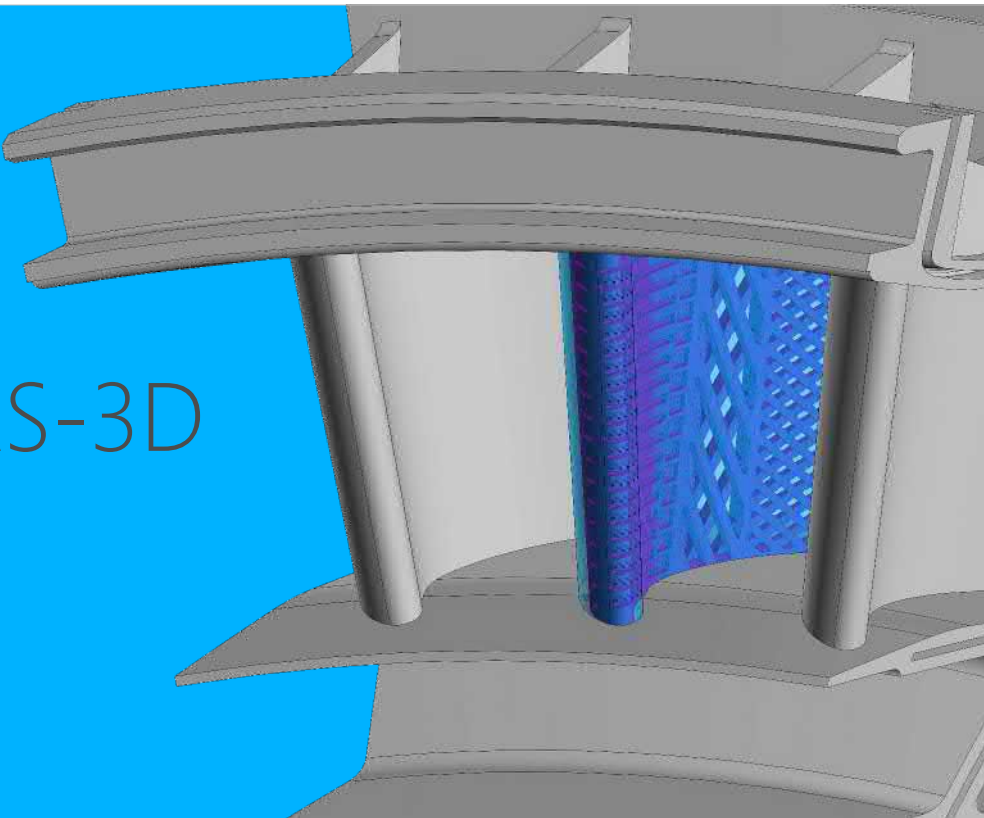
Take into account:

- The behavior of the parameterized image after editing is now preserved to a much greater extent. For example, restrictions imposed on the endpoints of objects are preserved if these points are not deleted as a result of editing (previously the restrictions disappeared).
- Thanks to the new component of our C3D core, additional tools for reading 3D models created in NX and SolidWorks systems have appeared.
- Phantom rendering has been accelerated when creating and editing surfaces, as well as model rendering with the "Grayscale with frame" and "Without invisible lines" display types.



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Thanks for your attention!



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