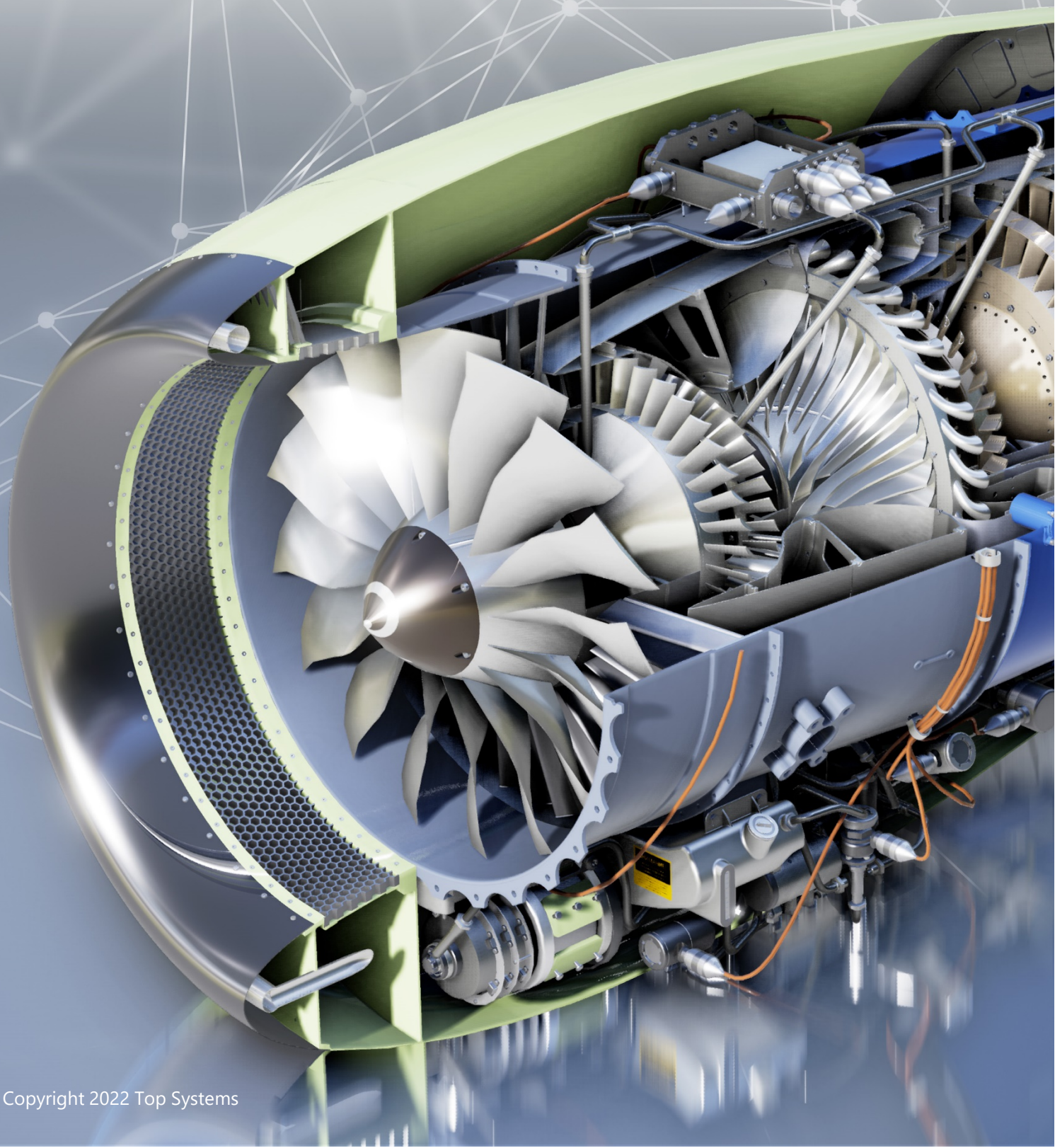


T·FLEX CAD 17 and Add-on Modules

What's New



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Welcome to T-FLEX CAD 17

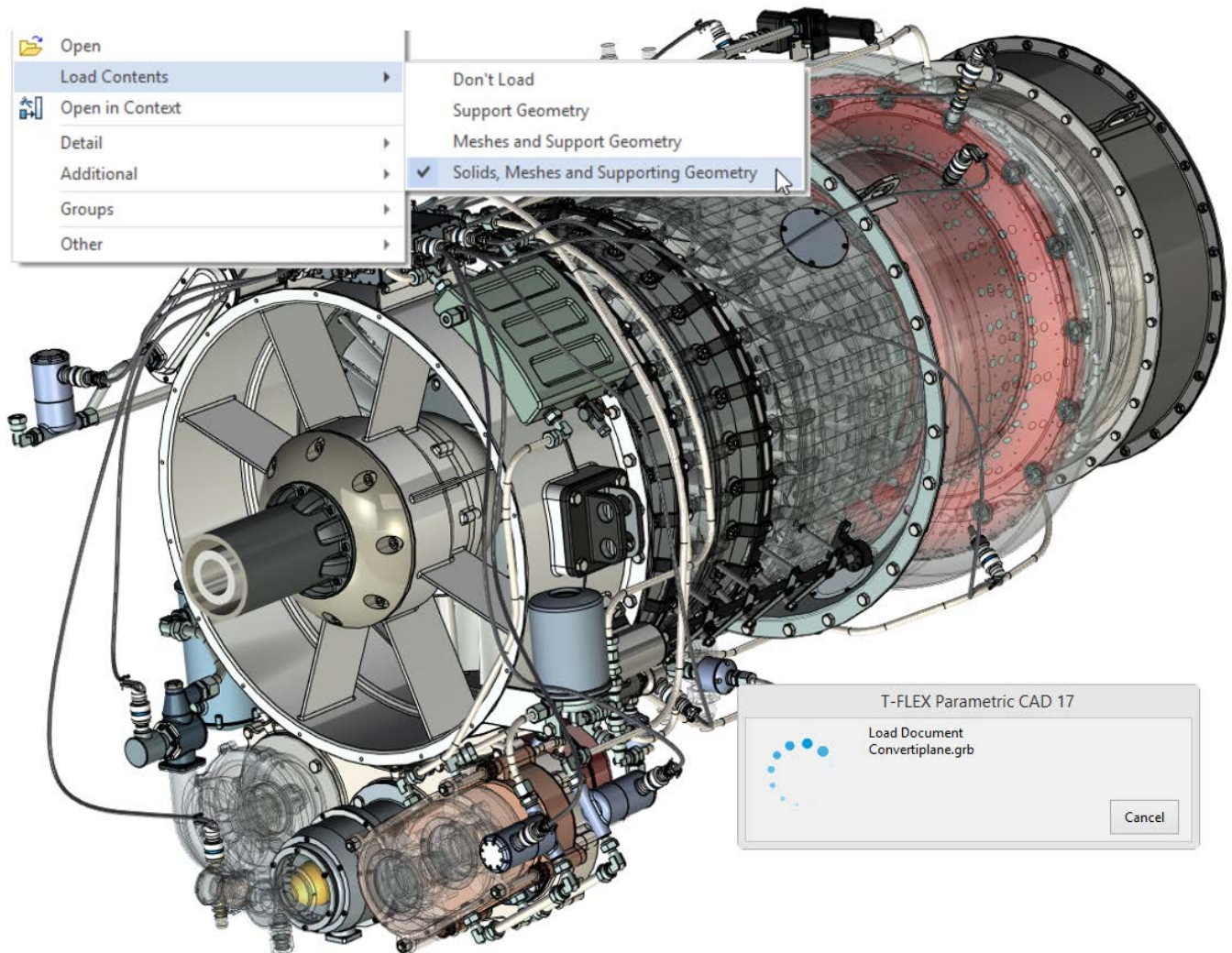
T-FLEX CAD 17 offers a large number of new functionality and improvements, many of which are implemented according to users' requests, including: a significant increase in performance when working with assemblies, teamwork using T-FLEX DOCs, a new mechanism for 3D design, new operations for surface modeling and working with 3D curves, a new mechanism for selecting objects, an improved interface, and much more.



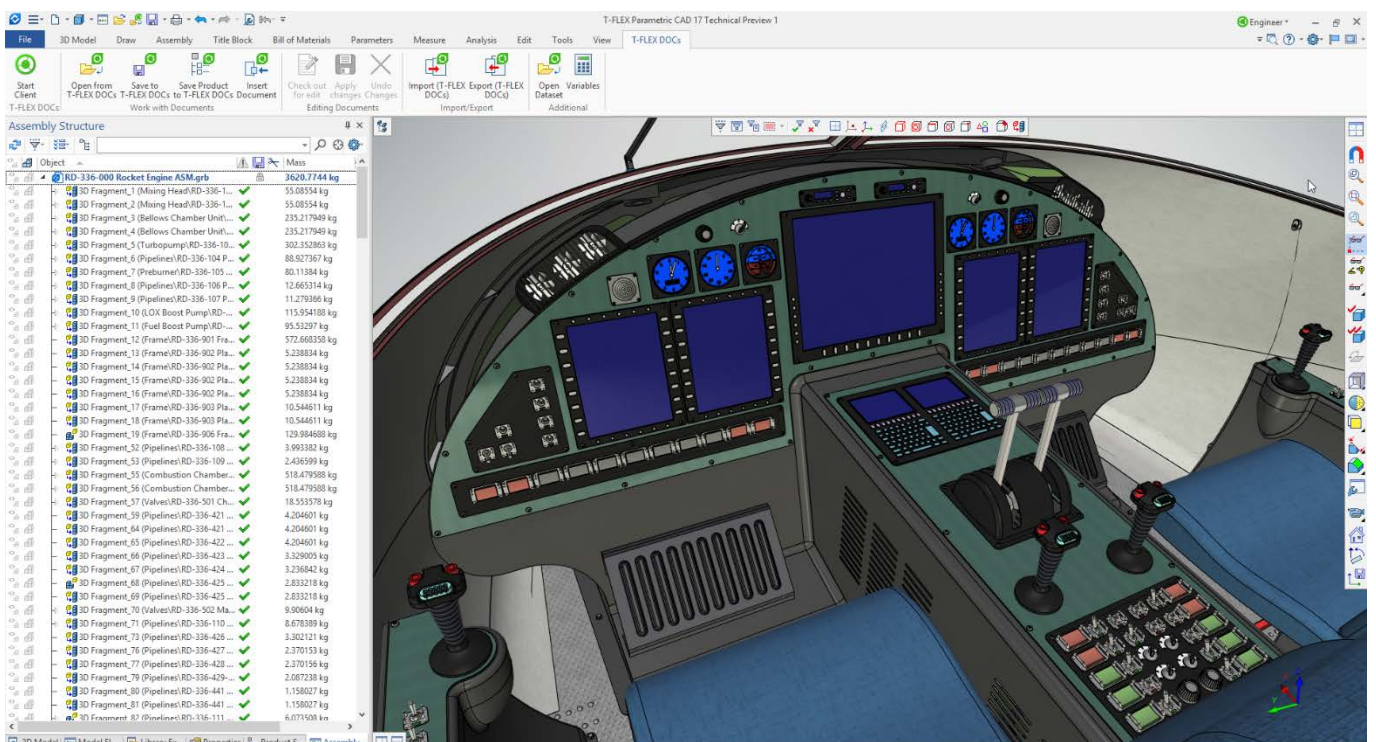
Teamwork on Large Projects

Our general direction is the designing of new tools for collective work on projects. This includes new integration modes with T-FLEX DOCs and a new mechanism for creating comments, as well as optimization of work with assemblies, and improvement of assembly management windows.

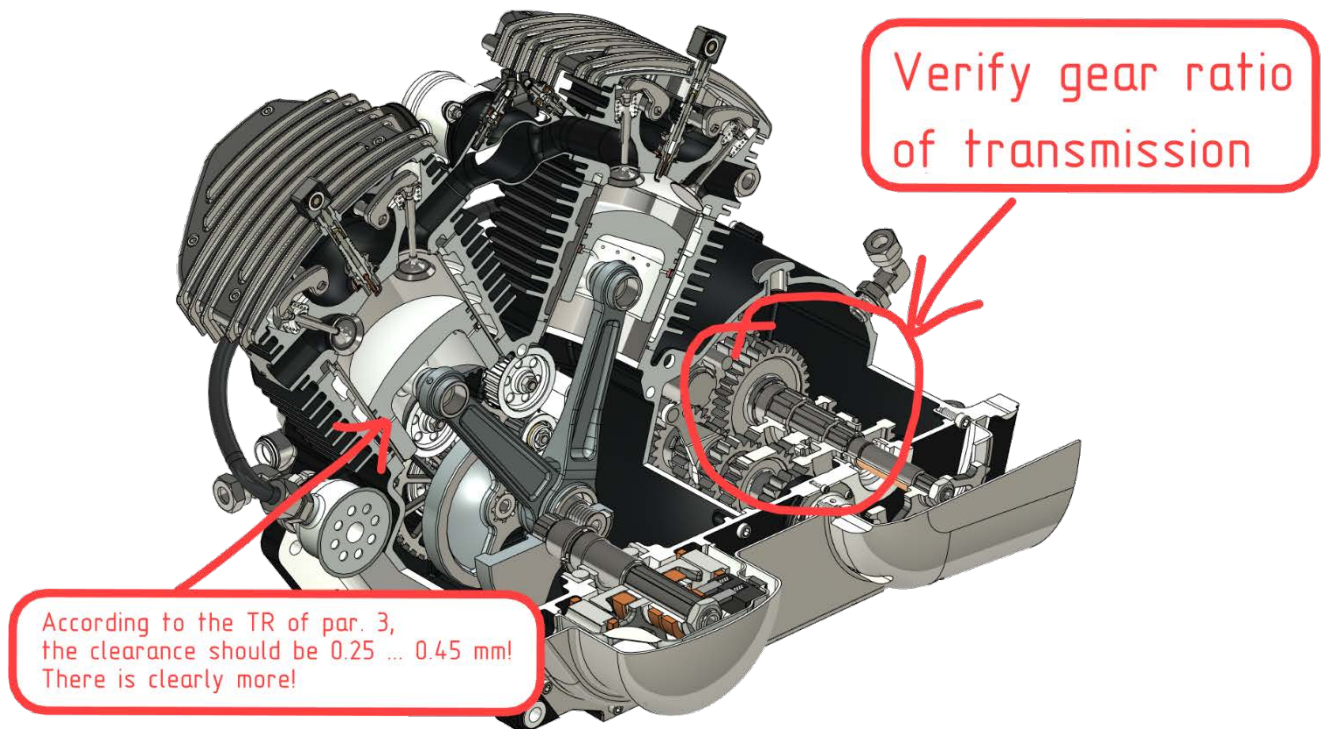
Working with large assemblies was improved: loading, editing, and recalculating assemblies is up to ten times faster thanks to new mechanisms for loading assemblies and reducing memory consumption. Now you can work with assemblies with unloaded geometry and mesh, including the insertion of such fragments. Data on grids and solid geometry is loaded by the user's request or automatically, if necessary, during designing.



New mechanisms for collective work under the control of T-FLEX DOC's appeared.



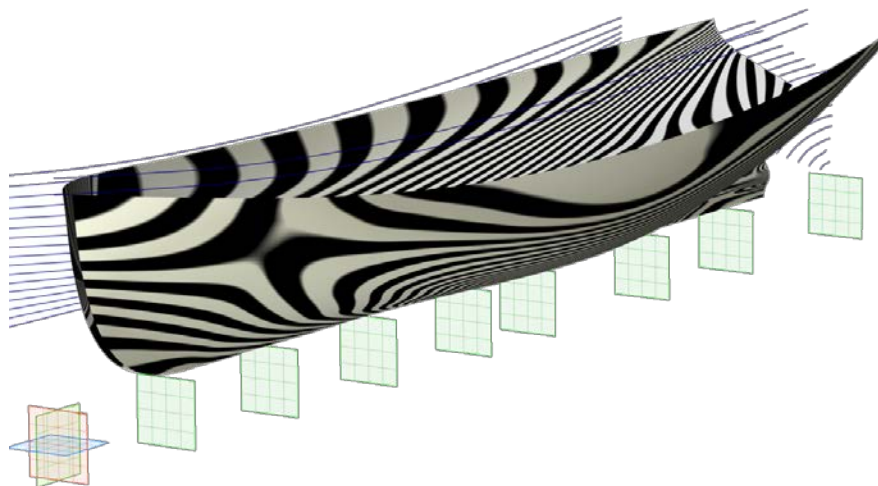
A new remarks mechanism was added for teamwork. The new **Remarks** window enables you to control 2D and 3D remarks - new object types. Remarks are stored in separate files or in the new dataset **Remarks** in T-FLEX DOCs. You can edit remarks in the same way that you edit fragments in the context of an assembly.



Quality control tools were added. Verification of models and drawings according to the established corporate rules was added. These tools are used by security services, standard control departments.

New Commands for 3D Modeling and Working with 3D Curves

The functionality of the system aimed at working with surfaces and wire geometry has been significantly developed. A new group of commands that enables you to receive bodies and surfaces based on existing bodies and surfaces has been designed: **Offset Body**, **Offset Surface**. The capabilities of the **3D Node**, **3D Path**, and **3D Profile** commands were significantly improved, which expanded the possibilities of working with wire geometry and made it more convenient. The existing surface modeling commands were updated: **Transition Surface** and **Law Extension**, **Bridge**, **Fill Hole** and new surface creation commands were implemented.



The edge blending command capabilities were expanded, additional tools for managing the interface of surfaces in the new **Transition Surface** command have been added, and the calculation of mass inertial characteristics now supports multithreading.

New Features of Dedicated Applications

Various specific areas of application were enhanced: springs, sheet metal, VR, electrical engineering, cutting, photorealism, dynamic analysis, libraries of standard elements.

User Interface

The improved interface now has additional features for managing objects. Two options for selection of objects were added: **Closed Curve** ("Lasso") and **Cutting Curve**. A new **Layers** window was added, the **Materials** and **3D Model** windows were improved. You can now customize transparency of the list under the cursor and time of the list appearance.

Engineering Analysis

Significant improvements were made in the functionality of T-FLEX Analysis. Improvements have affected the expansion of the capabilities of the commands for creating boundary conditions, stabilizing the calculation model for calculations with insufficient boundary conditions, and algorithms for calculating contacts. Algorithms for refining the calculation of stresses and strains on tetrahedral elements were added.

Production automation

In the new version of the T-FLEX CAM module, a lot of work has been done on the user interface, it has become even more convenient to work with the module. Many options have been added and the old ones have become much easier to work with. The window of the machining manager has been redesigned, the ability to form installations with different positions, configurations and set of rigs has been added, and much more.

Performance Improvements and System Optimization

It is a well-known fact that when working with large assemblies in CAD systems, one of the factors that reduce design efficiency is a significant decrease in system performance. This applies both to the time spent waiting for the assembly to load, and the time spent waiting to complete the operations with the assemblies: insertion, moving, recalculation of parts, etc.

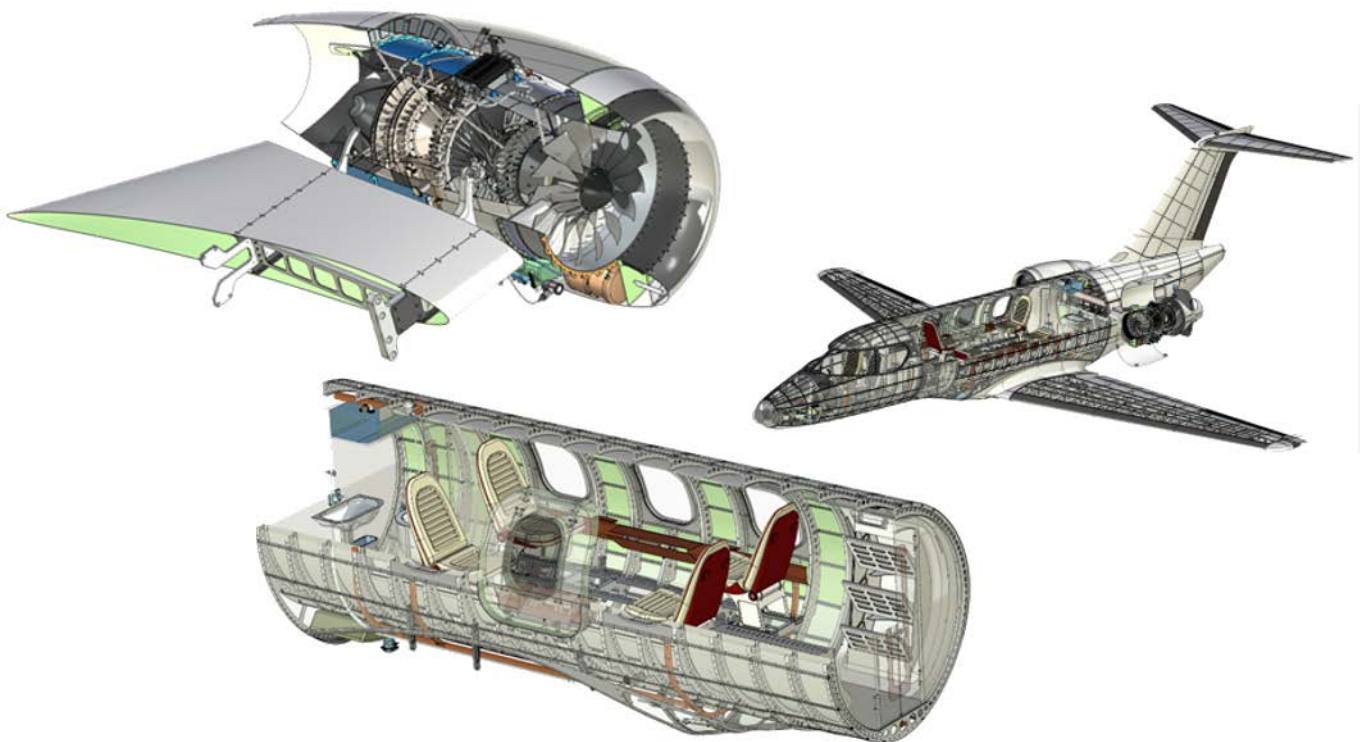
New approach to the loading of assembly fragments significantly improves performance in comparison with the previous versions of T-FLEX CAD and other CAD systems.

Performance

It is now faster to load all documents, including assemblies. Assemblies with the number of bodies in the order of several tens of thousands are loaded in 20-40 seconds. This is about ten times faster than it was before.

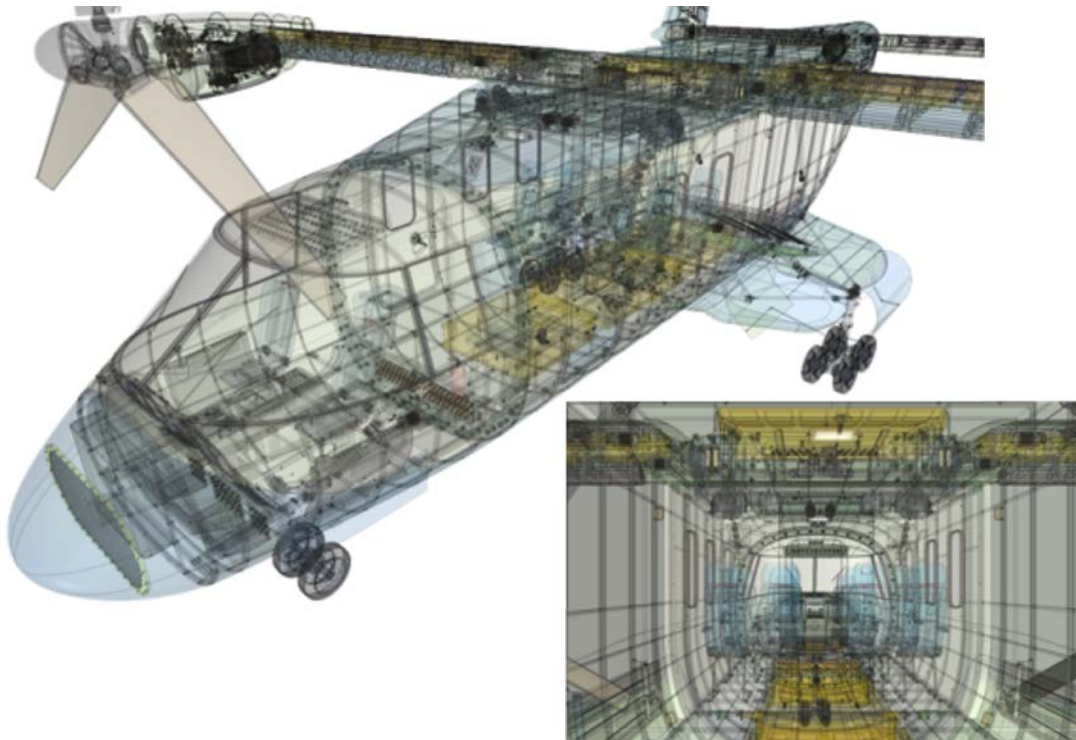
Aircraft model, 48372 bodies.

Version	Load time
T-FLEX CAD 16	5 min 15 sec
T-FLEX CAD 17	38 sec



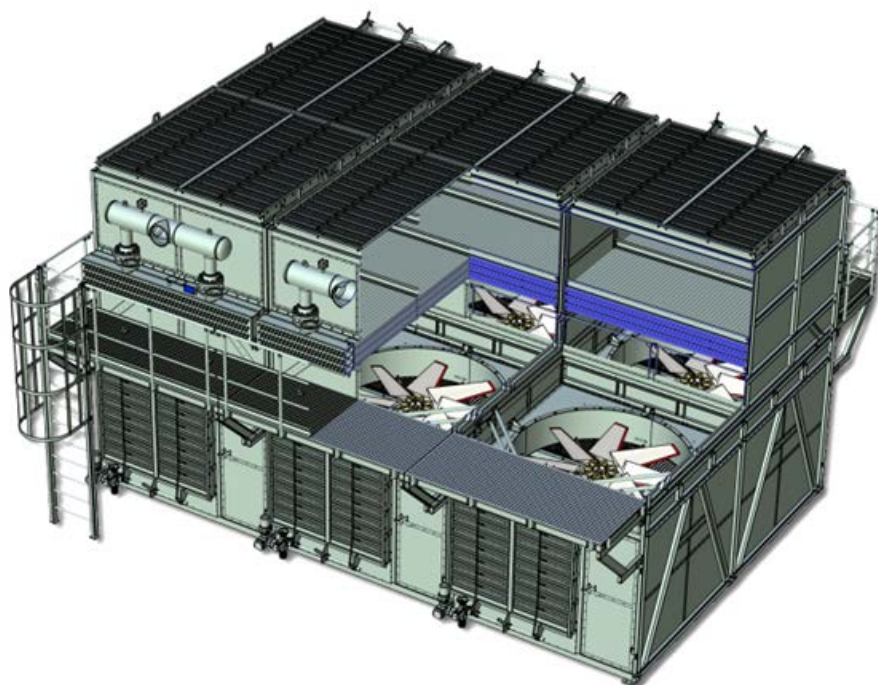
Heliplane model, 60687 bodies.

Version	Load time
T-FLEX CAD 16	1 min 30 sec
T-FLEX CAD 17	25 sec

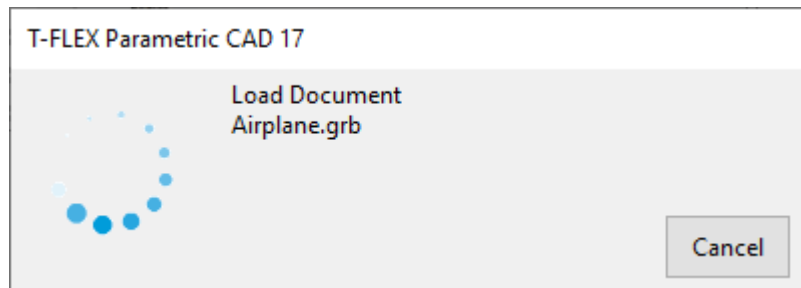


Air cooling unit model, 51395 bodies

Version	Load time
T-FLEX CAD 16	3 min 15 sec
T-FLEX CAD 17	18 sec



A standby window appears during model loading. If you need to interrupt the download, use the corresponding button.



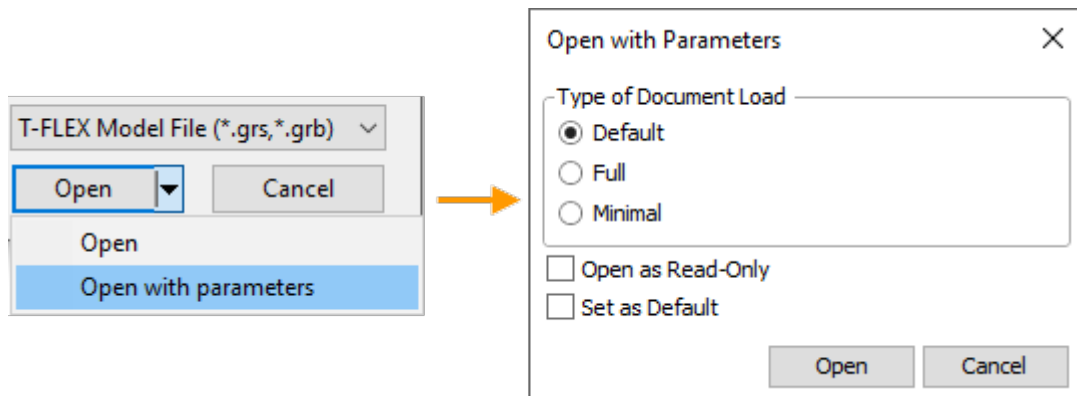
PC specifications on which the assemblies were opened: OS Windows 8.1, RAM 20 GB, SSD (!), Intel Core i5-2500 CPU 3.30 GHz, Video: NVIDIA GeForce GTX 1080.

- i** Note: The specified assemblies were opened locally. Opening over the network will depend on your network and hardware settings.

Assembly Loading Setting

Now you can configure the entire Assembly and its fragments loading.

When you open an assembly, you can choose how to load it:

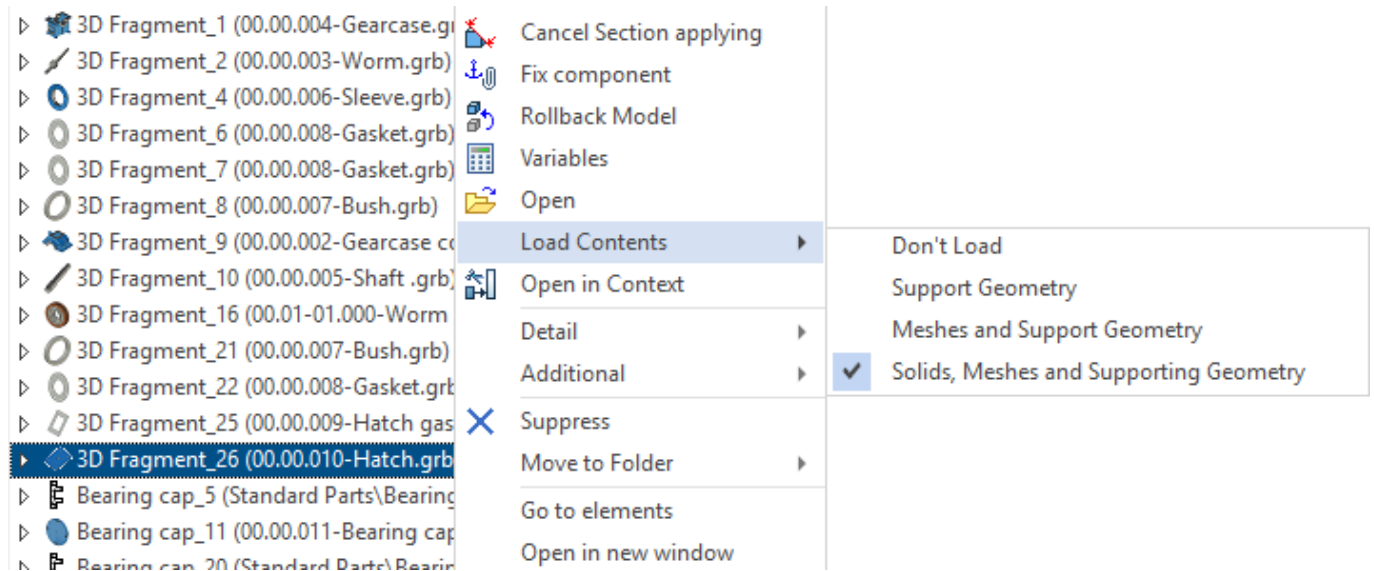


By default, the Assembly will be loaded without bodies, which accelerates the loading process. When you choose the full load, all the fragment bodies will be loaded. When you choose the minimal load, fragments are not loaded. The assembly tree with all the fragments is available for the user but the 3D scene will be empty.

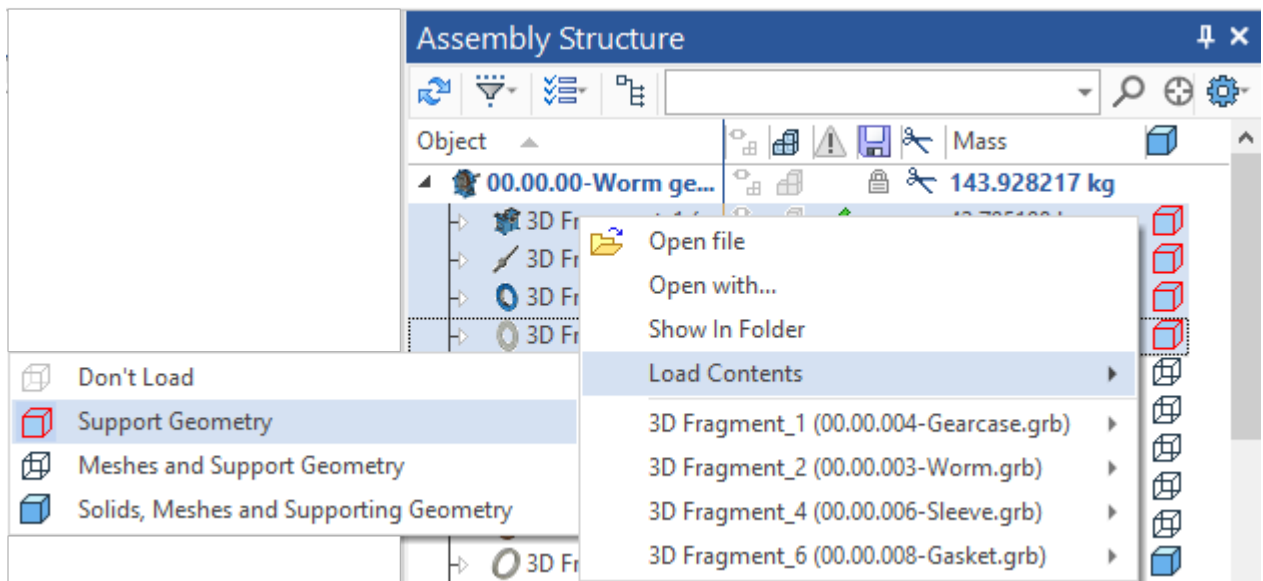
After the assembly is loaded you can set the loading method for each fragment. This enables you to both upload the necessary elements of the fragment and unload the extra ones. The following options are available:

- Solid, Meshes and Support Geometry;
- Meshes and Support Geometry;
- Support Geometry;
- Don't Load.

Each option you select affects performance. By default, the fragments have grids loaded, which allows you to see their visual display, and the reference geometry is loaded, which shows the external objects of the fragments. If an Assembly fragment is selected as an element of any modeling or measurement operation, bodies are automatically loaded for it. You can upload the bodies, grids, and supporting geometry of a fragment, or unload it using the context menu of the selected fragment or the group of fragments.



You can control the completeness of fragment loading either in the "3D Model" window or in the "Assembly Structure" window. In the assembly structure window, a special column "Load Contents" has been added. Special diagrams are used to show, what elements are loaded for each of the fragments.



Working With Assemblies

When developing this new version of T-FLEX CAD, much attention was paid to optimizing the work with assemblies. Now editing, recalculating, and loading assemblies are much faster, both in comparison with previous versions of T-FLEX CAD, and in comparison, with competitors. Our priority is convenience of working with large assemblies, according with general direction of system development as a tool for large teams working on large-scale projects.

Modifications of "Assembly Structure" Window

Now you can manage reference elements in the assembly structure through the panel of the "Assembly Structure" window.

Object	Geometry Source	Get Document Paramet...
Reference Face_1	Reference Elements.grb	Reference Elements.grb
Reference Face_2	Reference Elements.grb	Reference Elements.grb
Reference Face_3	Reference Elements.grb	Reference Elements.grb

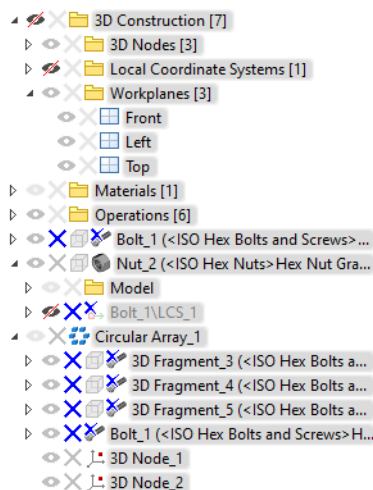
Open Geometry Source
 Show in Reference Source Document

New Modes for Copying Fragments

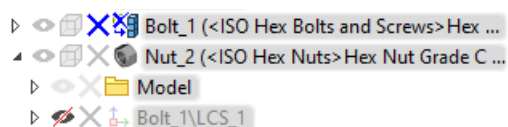
Associative copy of a fragment and **Array of fragments** with the ability to disable associative link with the source fragment were implemented. Multiple fragments or a group of fragments controlled by a single set of variables are positioned independently, or as an array.

Fragment Suppression

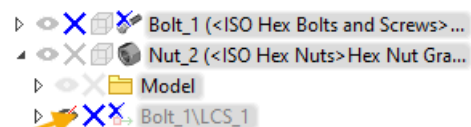
Fragment suppression functionality was enhanced.



When suppressing fragments or array/copy/symmetry of fragments in the 3D model tree, the suppression immediately applies to all reference elements, "raised" objects and created fragments.



T-FLEX CAD 16

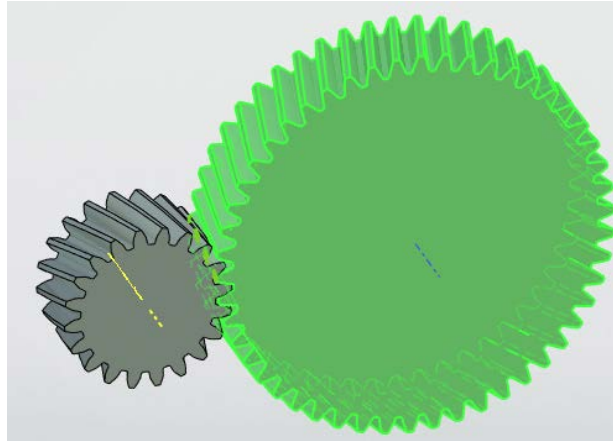


T-FLEX CAD 17

Mates

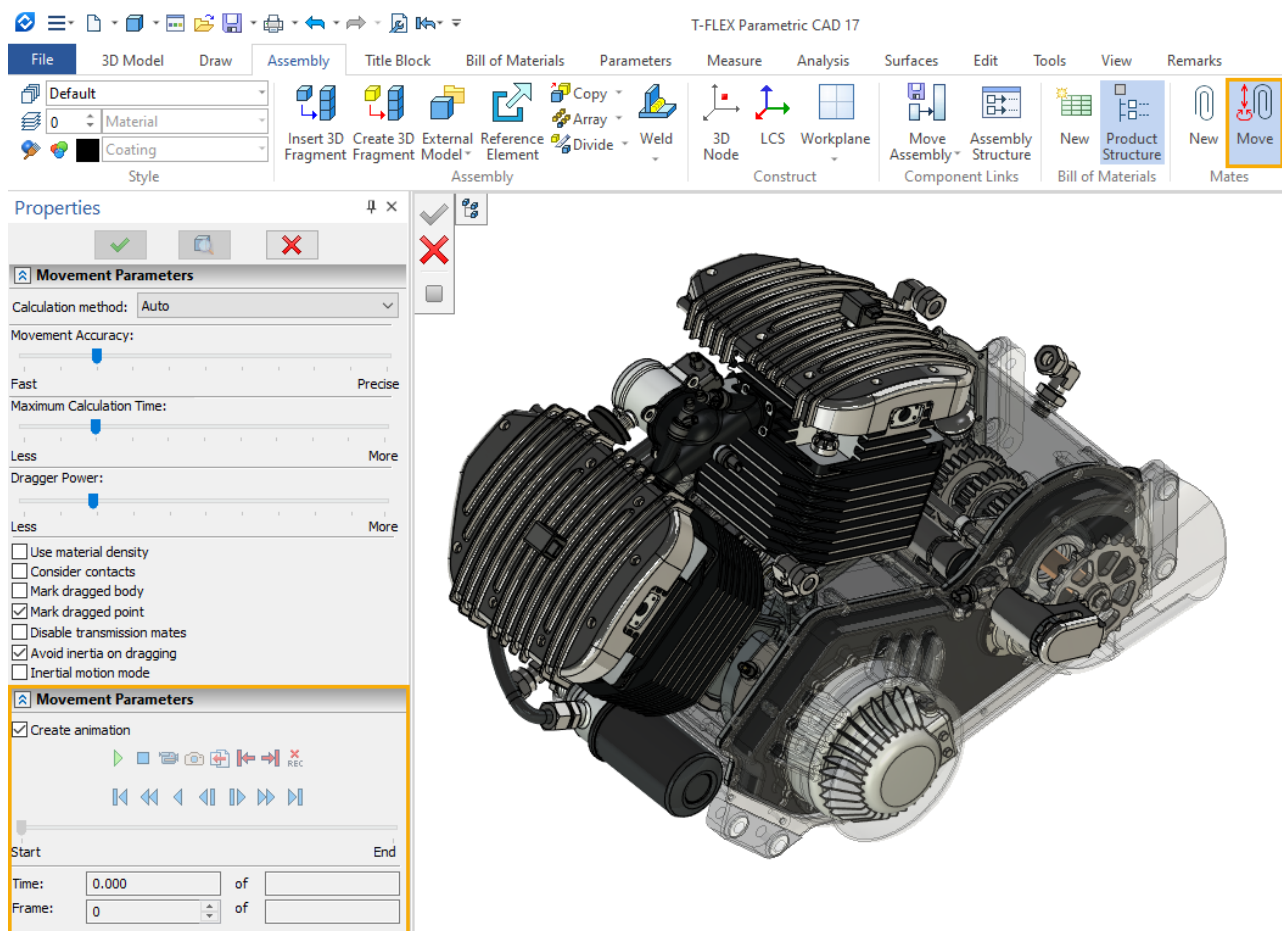
New Types of Mate

New variants of object couples were added to **Tangency** and **Distance** mates: «Curve-Axis» and «Surface-Axis». The new mates simplify positioning of the bodies with cylindrical surfaces and the mates are calculated faster than the ones between surfaces or curves.



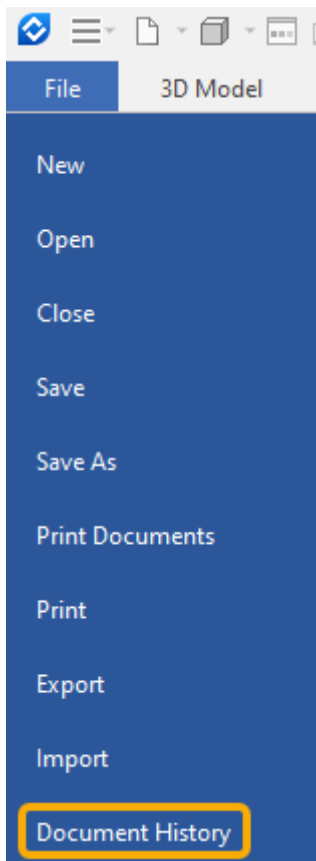
Record Animations for Moving Mated Elements

The ability to record animations has been added to the command for moving mated elements. Recorded animations can be saved to a video file.



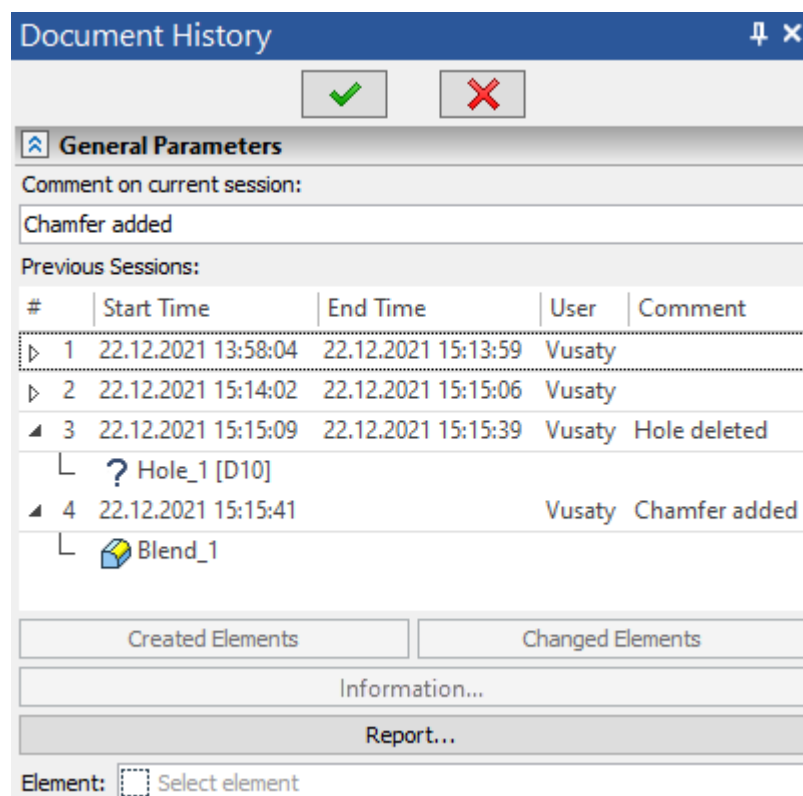
Document History



You can now enable the automatic tracking of document's changes history (**Document Parameters > Document > Save > Save > Save Document History**). It can be viewed using the **Document History** command, which is available in the **File** tab of the ribbon:

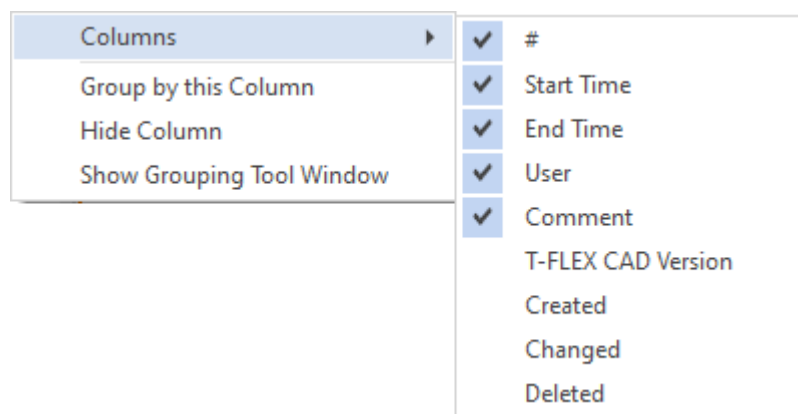


Command's parameters window contains the list of sessions, in which the current document was saved. By default, following information is shown for each session:

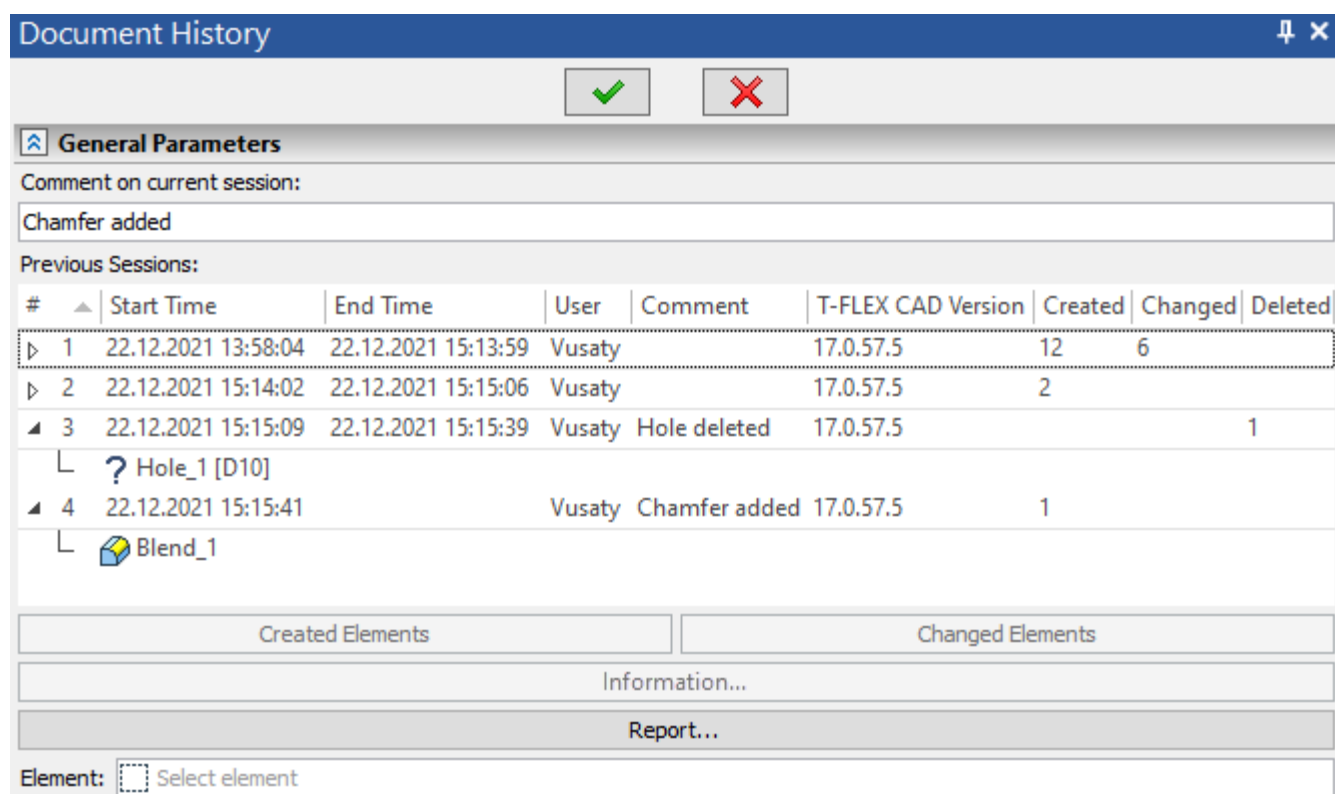
- **#**
Number of session in chronological order.
- **Start Time**
Date and time of opening the document.
- **End Time**
Date and time of a last save of the document before closing. Intermediate saves in the same session are not tracked.
- **User**
Windows or T-FLEX DOCs (when working in the integration mode) user-name of a user who saved the document.
- **Comment**
Comments can be added by typing text into the **Comment on current session** input box. The comment is saved in history upon saving the document. Adding, editing or deleting comments on finished sessions is not allowed.



Additional columns of the list of sessions can be enabled by clicking  on a header of any column and selecting  desired options in the contextual menu.



Additional columns include **T-FLEX CAD Version** and number of document's element, that were **Created**, **Changed** or **Deleted** in session.





The # column also allows to expand/collapse the list of elements created, changed or deleted in session. The list contains names of elements and icons of their types. Icons of deleted elements are replaced by the question mark.

Following buttons are available below the list of sessions:

- **Created Elements**
When a session is selected in the list, the button selects elements created during this session in 2D and 3D windows, if such elements were not deleted.
- **Changed Elements**
When a session is selected in the list, the button selects elements changed during this session in 2D and 3D windows, if such elements were not deleted.

Elements selected using these buttons remain selected upon exiting the **Document History** command.

- **Information...**
Calls the  **Information** command for elements, that have been selected using the **Created Elements** and **Changed Elements** buttons.
- **Report...**
Exports the document history into a .csv-file.

The **Element** input box is located in bottommost part of the command's parameters window. Any element can be selected in 2D or 3D window upon activating  this input box. A name of a selected element, icon indicating its type and in some cases a name of a parent element are displayed in the **Element** input box. Upon selecting an element, the list of sessions is filtered so, that it contains only sessions in which the selected element was created and edited.

Element can be selected in advance prior to calling the command. In this case, the list of sessions will be filtered at once upon calling the command.

If saving document history is disabled in document parameters, then the system suggests to enable it upon calling the **Document History** command.

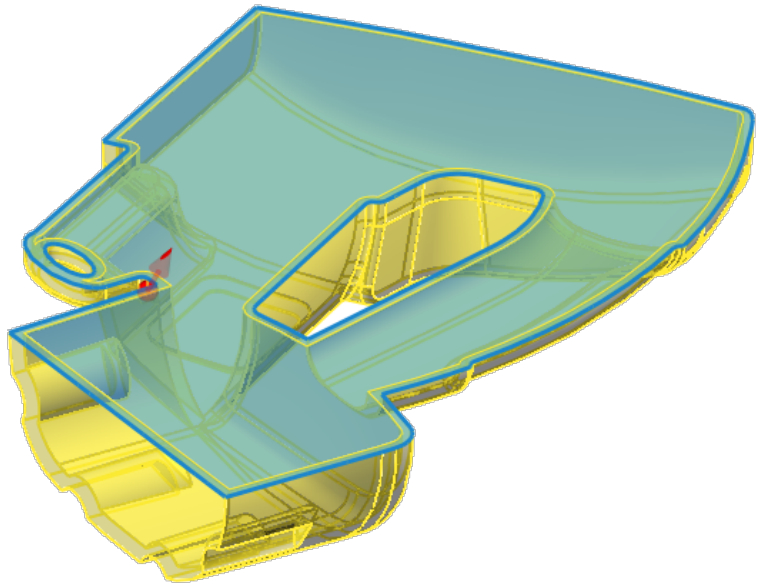
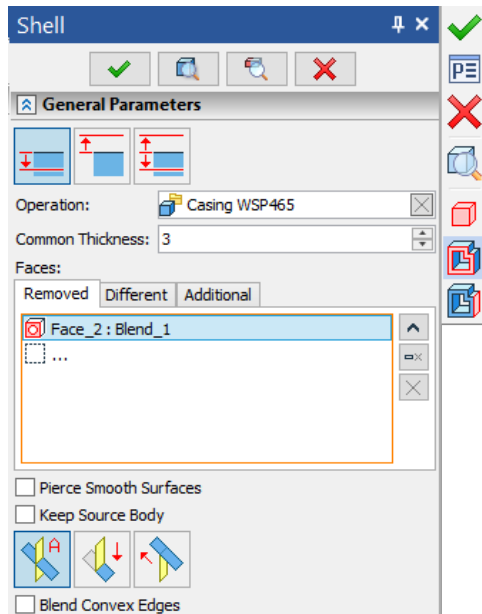
Saved document history can be deleted by disabling it in document parameters and saving the document.

3D Modeling

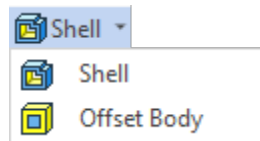
A lot of work has been done to unify the interface of existing operations for 3D modeling, to optimize and improve performance of their work. Several new operations were added.

Shell And Offset Body

The **Shell** operation has been divided into two operations: **Shell** and **Offset Body**. The **Shell** command dialog now looks like this:

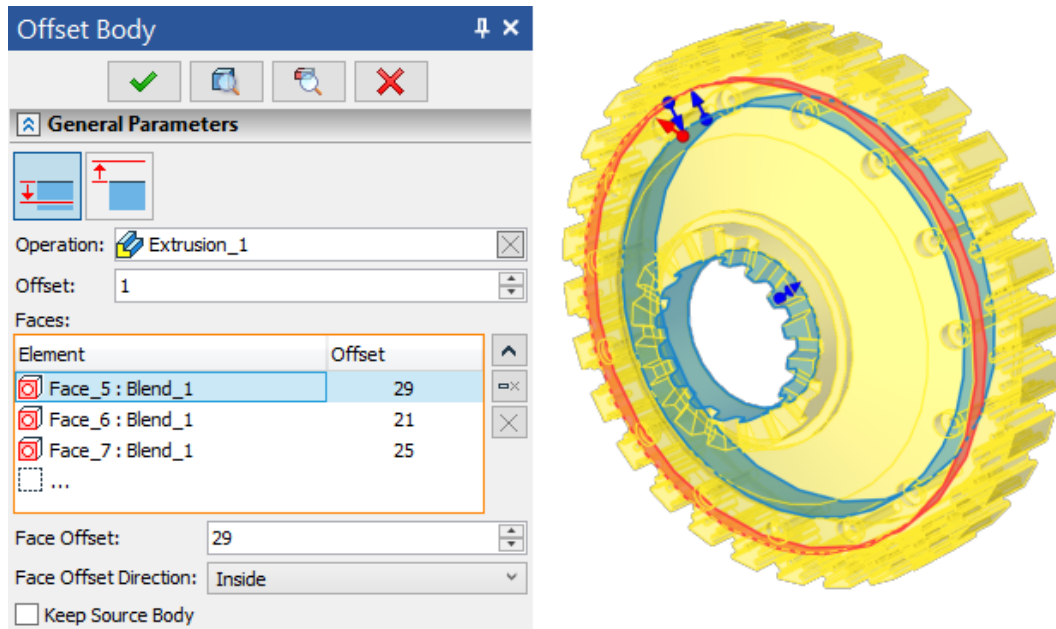


Thus, the **Create Offset Body** option in the **Shell** command is no longer required. You can call the new **Offset Body** command from the Ribbon from the same list as the **Shell** command.

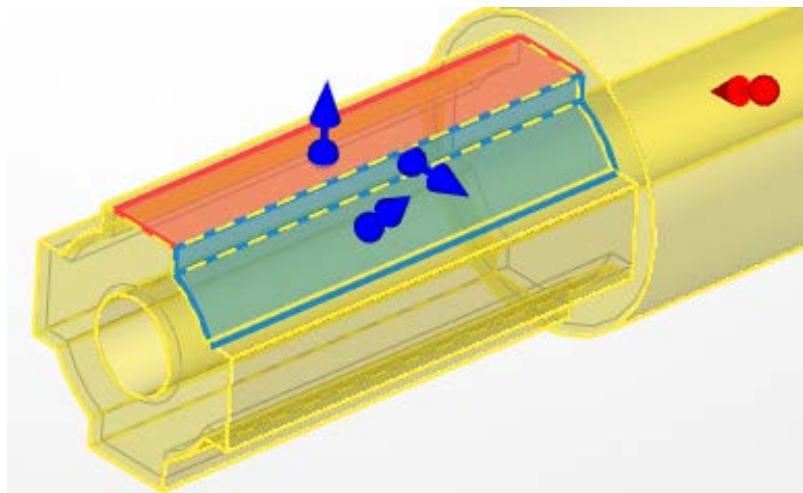


This command enables you to create a body whose selected faces will be offset equidistant from the faces of the original body. You can also keep the original body.

The **Offset Body** command has its own dialog box. The new type of dialog for parameters enables you to delete and add elements and edit the parameters of each element or group of elements separately.

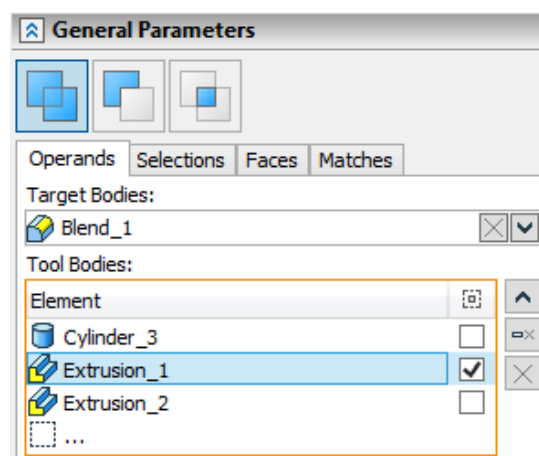


When creating a body, you must specify the amount and direction of the offset. The offset value of each face can be set in the dialog or using draggers in the 3D scene.



Boolean

The command interface was improved.

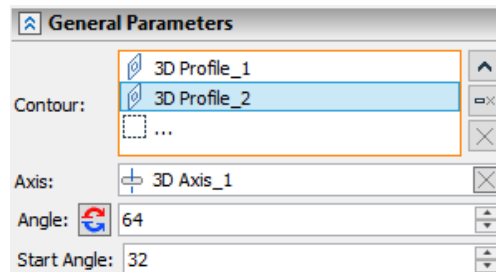


As for the operations and commands with an updated interface, you can change the size of lists of elements, search for elements by name, delete and add elements using special icons, and select the type of operation by switching the icons at the top of the dialog.

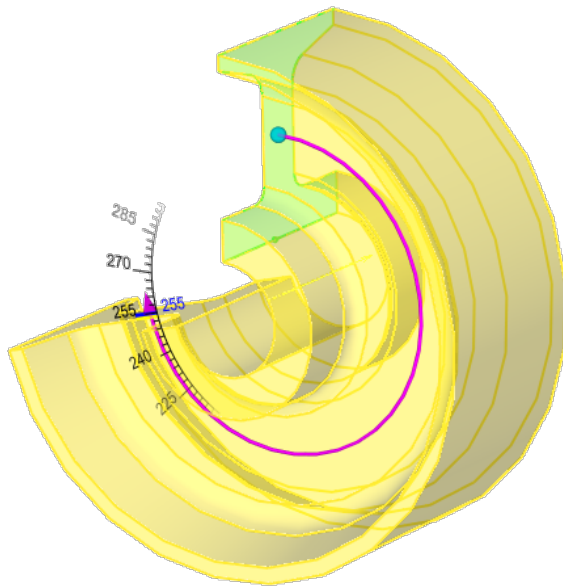
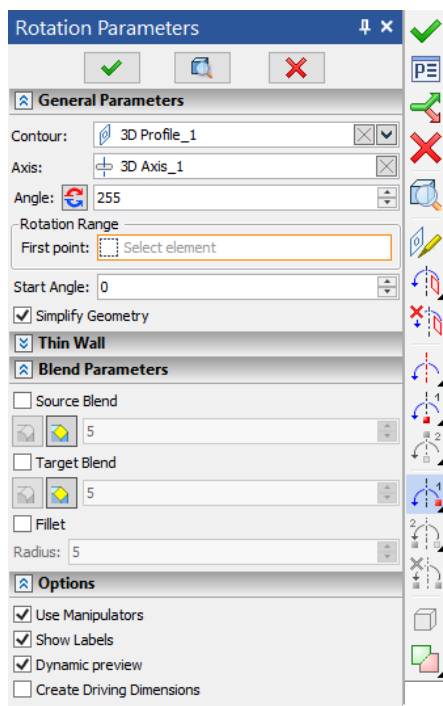
Rotation

The following improvements were made:

- the ability to control rotation angles using draggers;
- support for multiple paths selection;
- the command interface was improved.

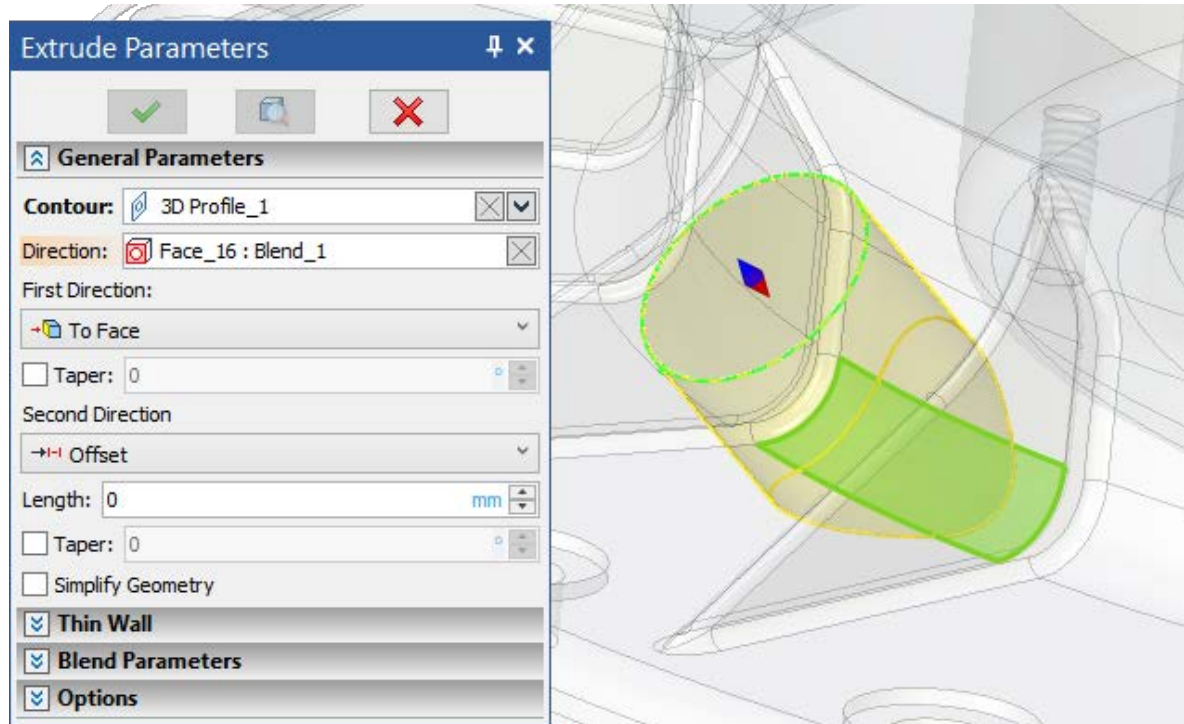


You can now manage the selected items: contours, axes, and points that define the rotation range.



Extrusion

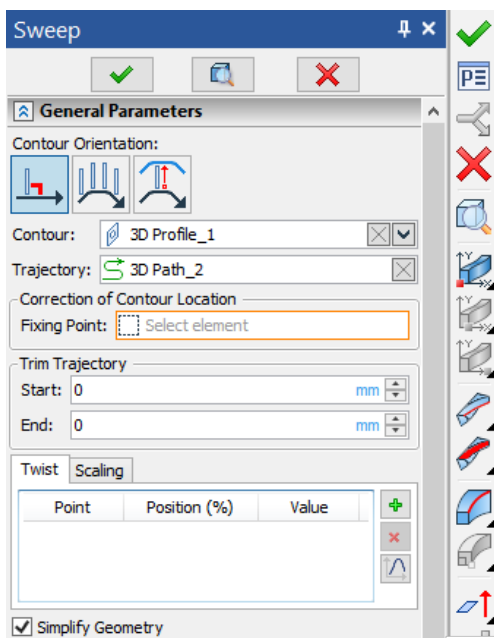
The operation interface has been updated, the ability to control the contours and direction in the operation parameters dialog has been added.



Added ability to select units and **Simplify Geometry** option.

Sweep

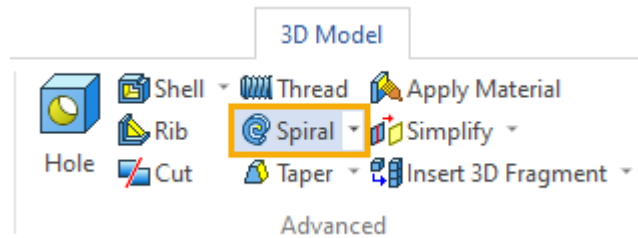
The command interface was improved: selected elements can be managed, the types of created operation are switched using icons.



The correction points are selected according to a filter. All the points that do not belong to the selected profile are filtered. The size of the body along the trajectory can now be adjusted using draggers.

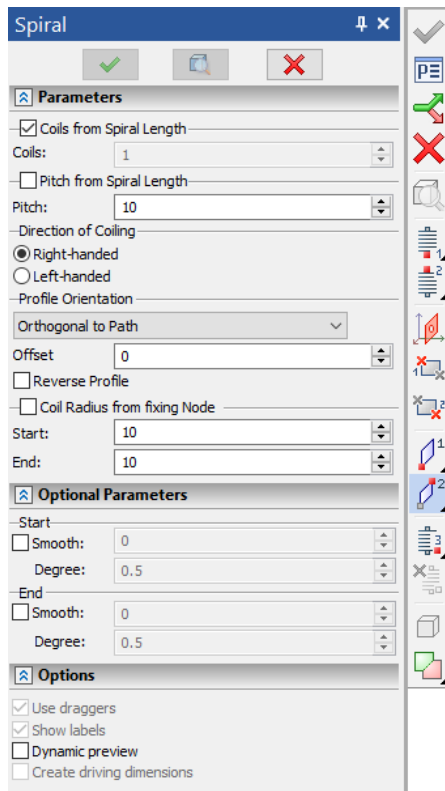
Spiral

The operation of creating a spiral has been updated.

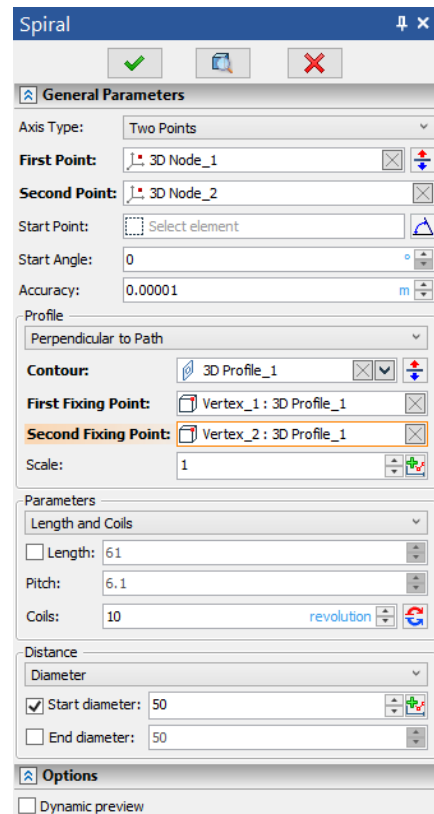


The operation works on the basis of a spiral 3D path (see [Helix 3D Path](#)), so the interface and basic features are identical.

Below are the operation parameters windows in T-FLEX CAD 16 and T-FLEX CAD 17, respectively:



T-FLEX CAD 16



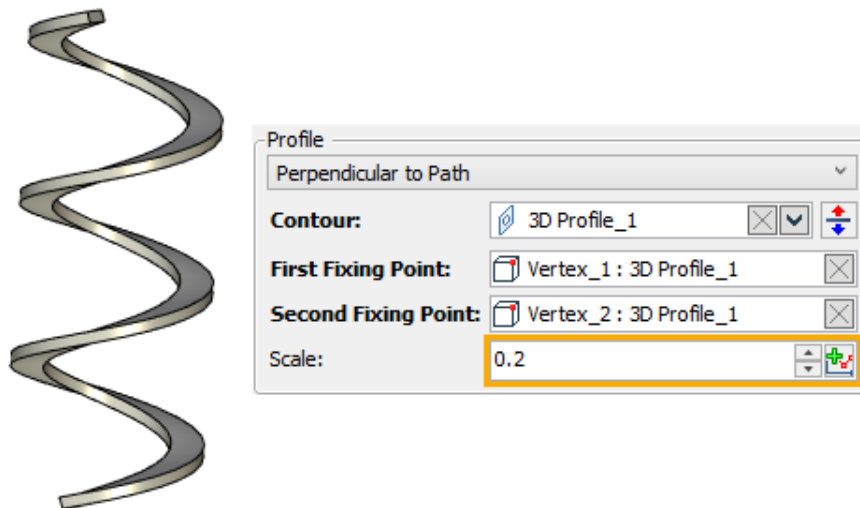
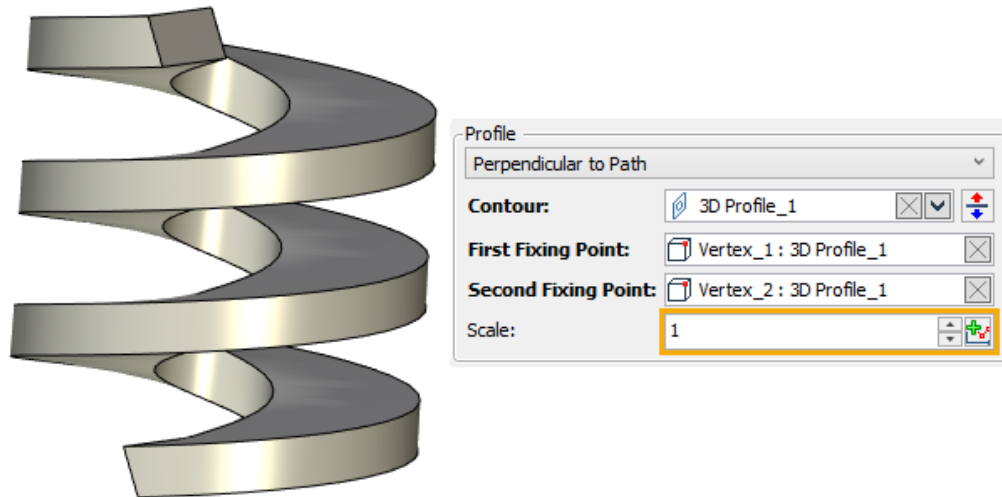
T-FLEX CAD 17

Updated the **Profile** group. You can select 3D profiles, paths, edges, or faces.

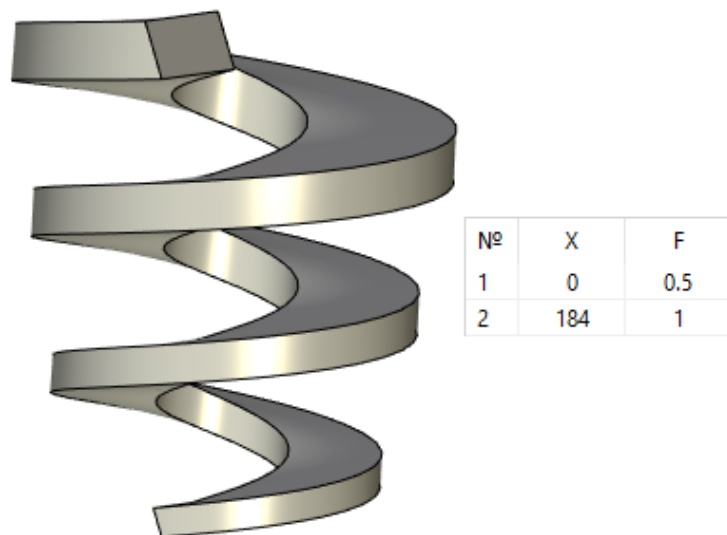
Multiple edge selection is available.

Fixing points define the orientation of the profile. The first point is aligned with the beginning of the spiral path.

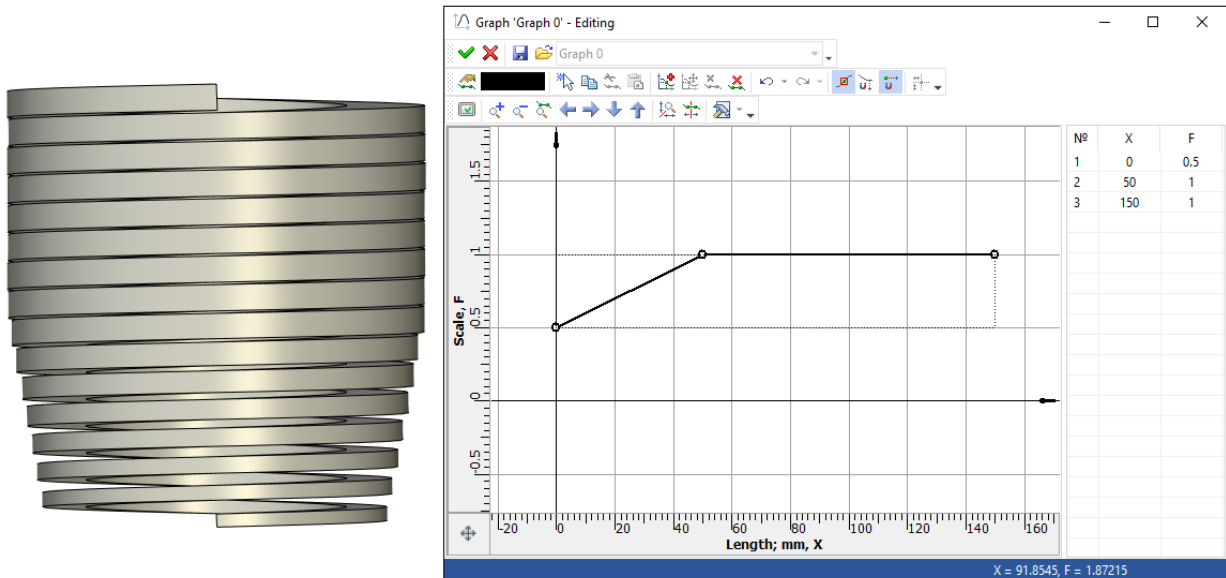
The **Scale** option is an analogue of smooth from version 16. The scale can be set as a constant. A value other than 1 resizes the spiral along its entire length.



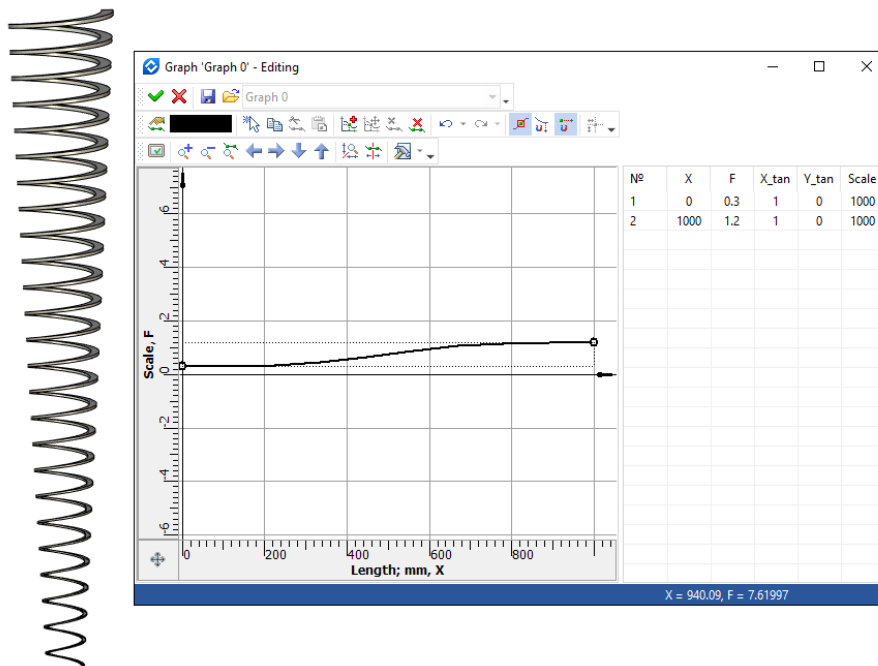
You can select graphs as a scale. A Polyline graph with two points scales linearly from the beginning to the end of the entire length:



A Polyline graph with more than 2 points changes the scale linearly:

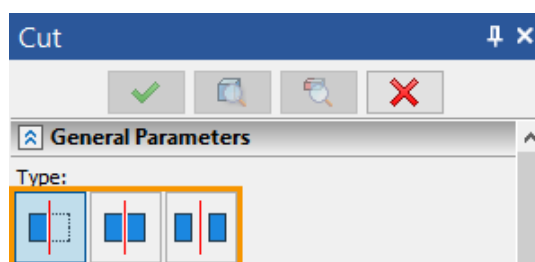


The **Cubic Spline** graph scales smoothly, according to the graph, along its entire length:



Cutting

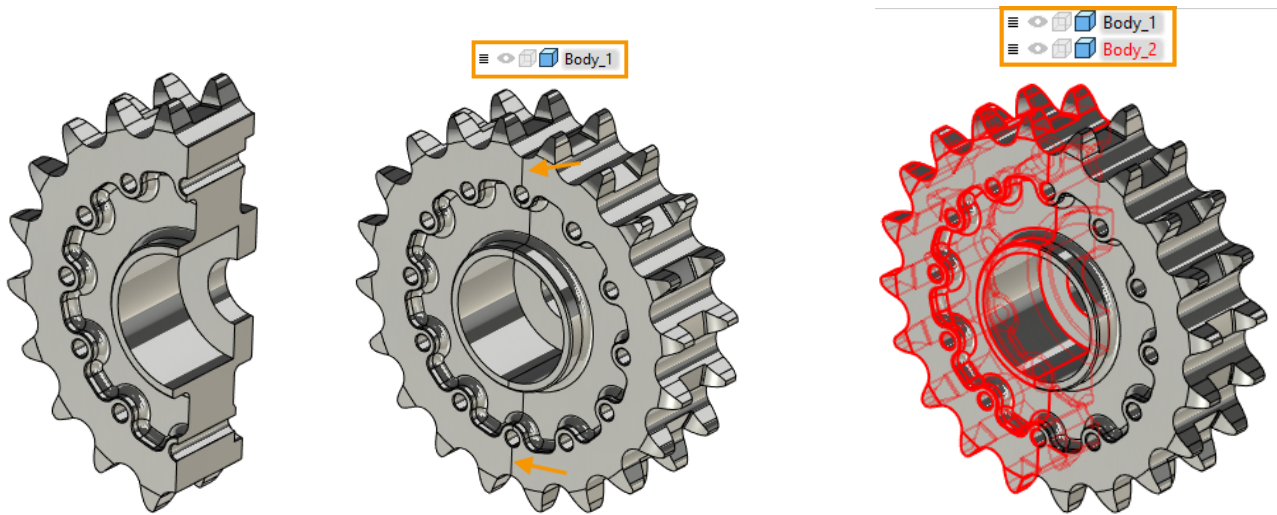
The operation has new dialog type. You can quickly choose between three types of cutting - **Cut**, **Split**, **Split with Separation**.



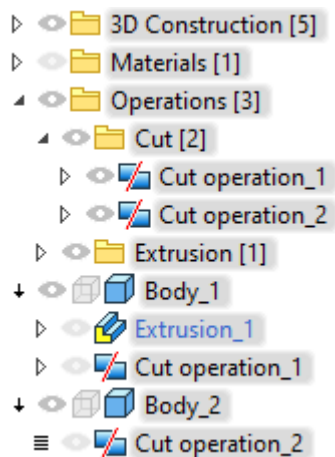
Cut - standard type of cutting used in previous versions of T-FLEX CAD.

Split - a new type of cutting in which the object is split by the selected type of secant elements, while remaining in the form of single resulting body.

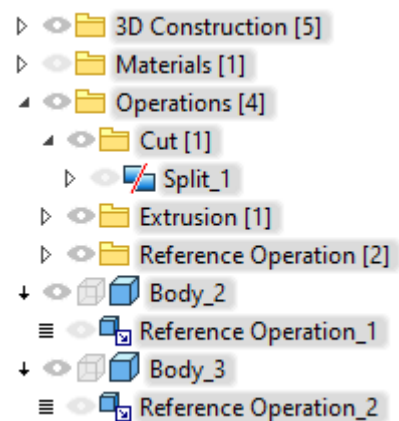
Split with Separation - new type of cutting, in which the object is also split by the selected type of secant elements, but at the same time it is separated into several different bodies.



A similar operation was in the 16th version of T-FLEX CAD, when operation resulted with creation of two cutting operations, independent of each other. Now a new split operation is created and its change will affect all applied bodies.



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If you select several cutting surfaces, the result of the operation will be many bodies.

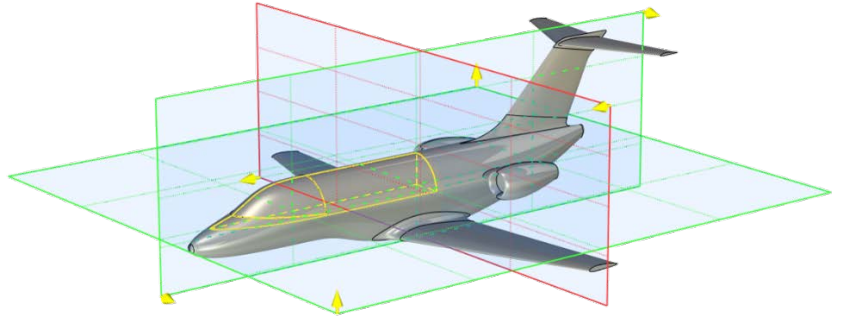
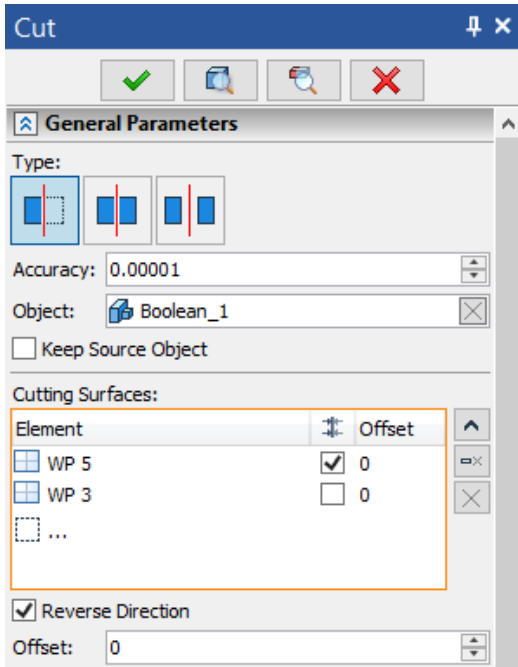
You can now quickly select options for cutting elements in the filter toolbar: **3D Sections, Workplanes, 3D Profiles/3D Paths**. By default, all variants of cutting elements are active.



This dialog enables you to specify an offset for each cutting workplane or surface. You can delete or add a cutting element of the selected type.

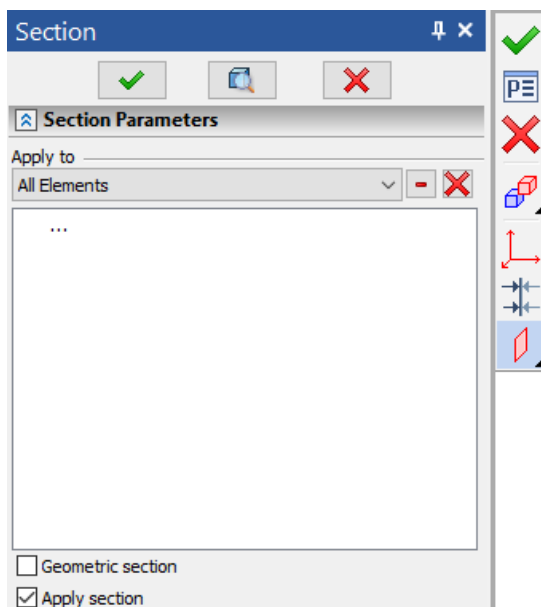
Added the ability to keep source object.

The **Accuracy** parameter was added. If the distance between the cutting elements is less than the accuracy, the cutting will be ignored.

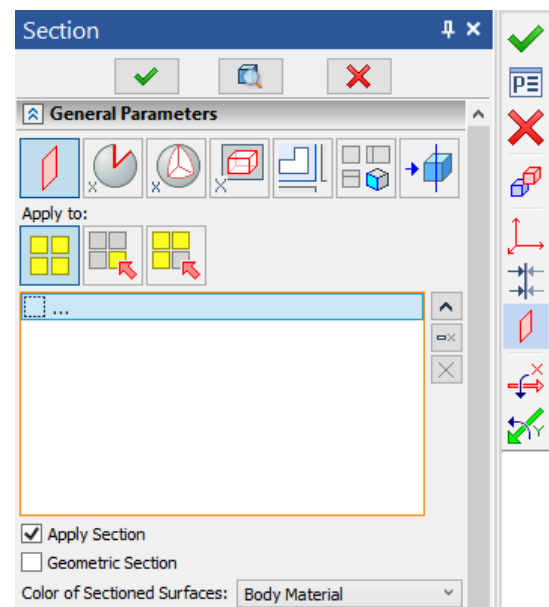


3D Section

The section creation command interface was updated.



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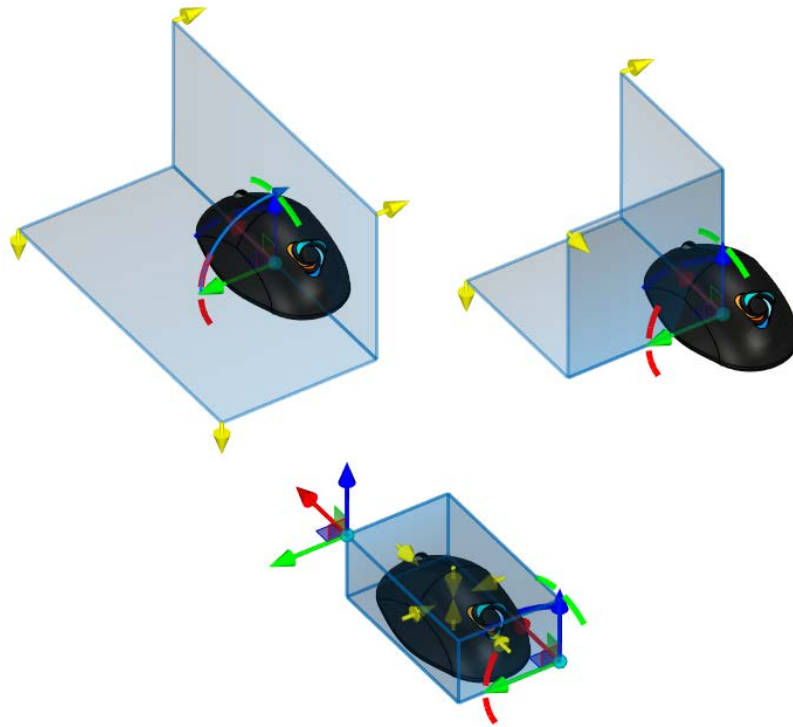


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The ability to use Body Material / Body Color / Section Color when applying a section has been moved to the dialog.

Buttons for quick system rotation around the X/Y/Z axes by 90 degrees have been added to the automenu.

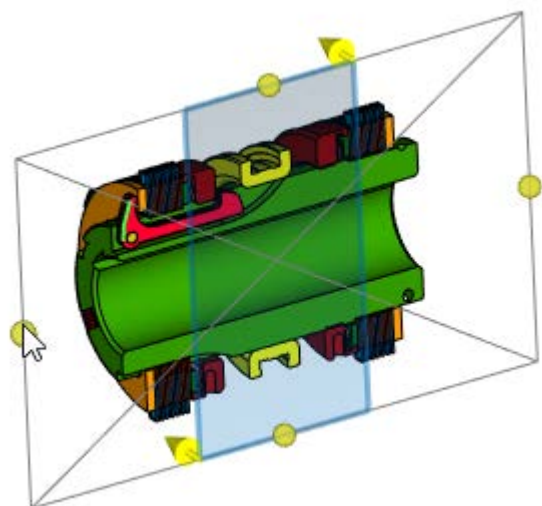
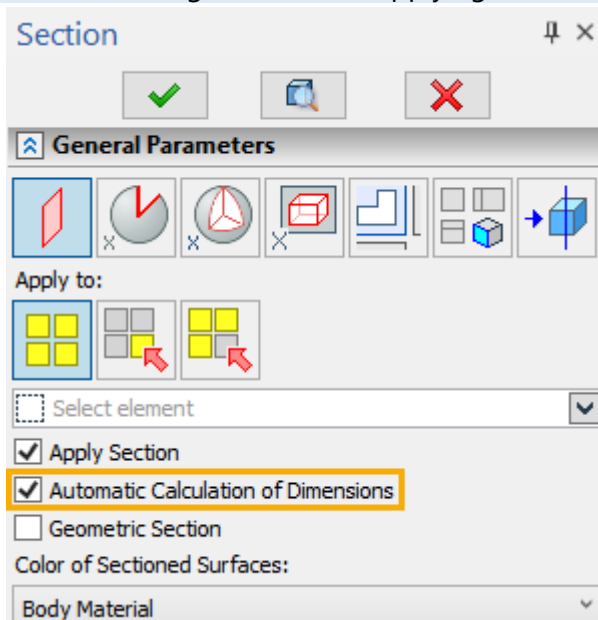
To create sections by plane angle, octant and parallelepiped, the ability to change the direction of the section using the yellow arrow manipulators or a special button in the automenu has been added.



Added yellow ball manipulators, which allow you to change dimensions of a cutting plane, when creating a section by plane.

By default, the **Automatic Calculation of Dimensions** checkbox is enabled in command's parameters window. In result, when you select a 3D object defining the cutting plane, the dimensions of the plane automatically coincide with dimensions of the selected object. If you move the cursor over a dimensions manipulator and press , the manipulator will start following the cursor along one of the axes of the section's LCS, until you press  again. The dimensions of the cutting plane change in accordance with manipulator's position. The dimensions change symmetrically in both directions along the selected axis. After using the manipulator, the **Automatic Calculation of Dimensions** checkbox gets disabled automatically.

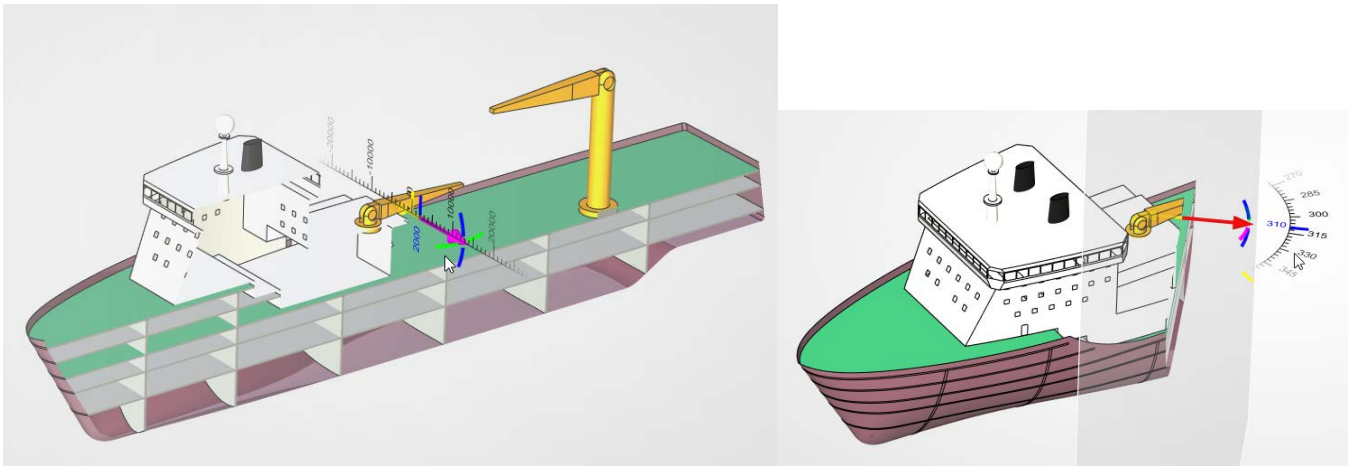
Dimensions of cutting plane are symbolic. They're used only for displaying the section in 3D scene, without affecting the result of applying the section. A model is cut by an infinite plane.



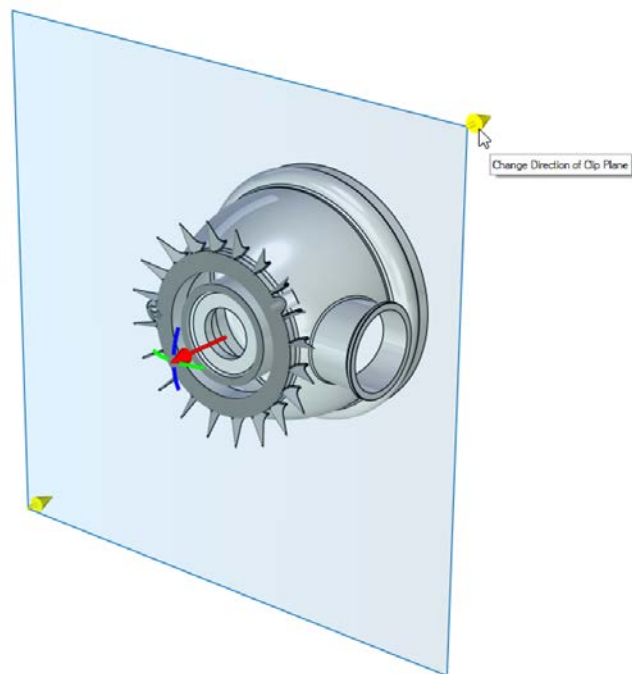
Clip Plane

Clip Plane command, which is used to quickly create simple sections by a plane on a 3D model, was improved.

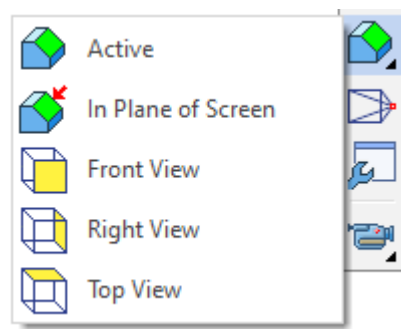
- Added standard manipulators for moving and rotating the clip plane with rulers and protractors;



- Added manipulator to change the direction of the clip plane;



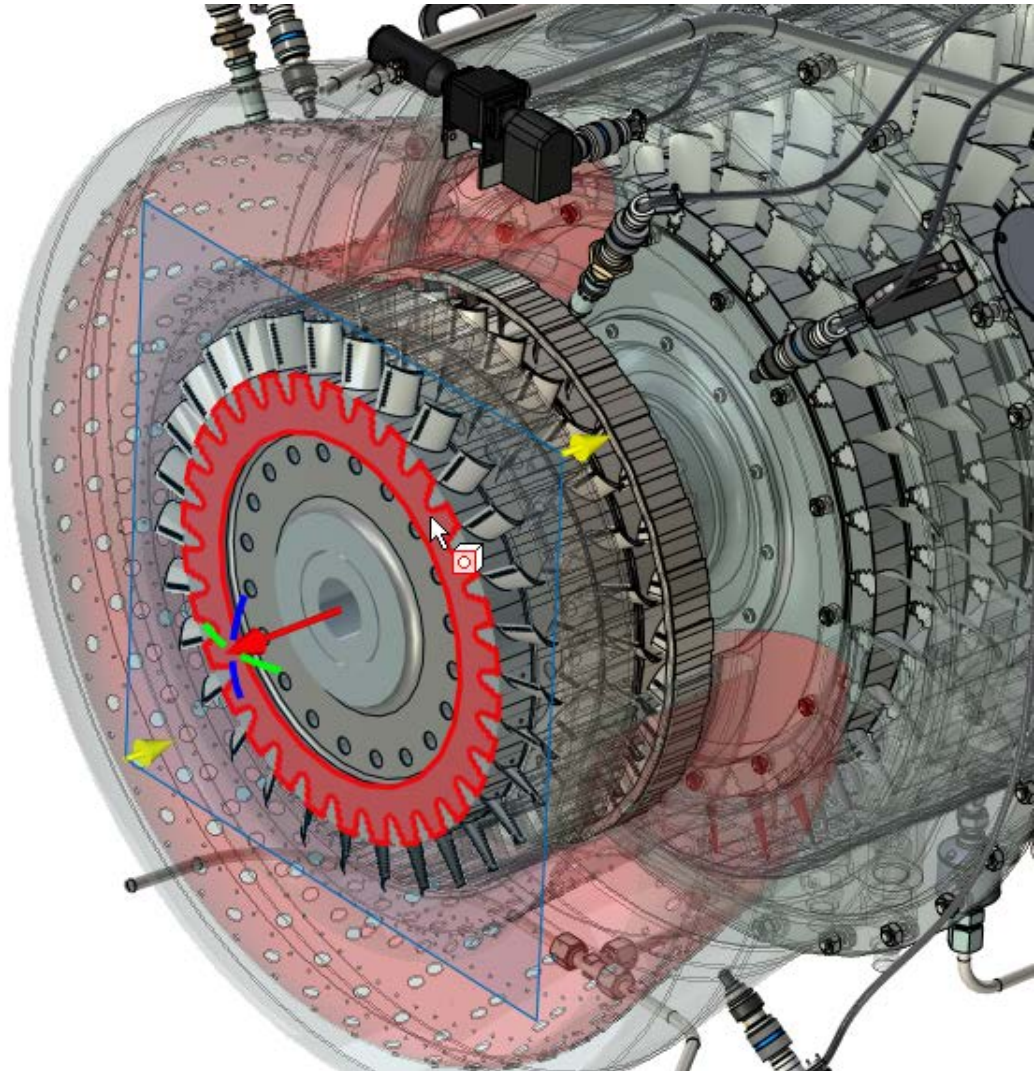
- Added standard views for quick dissection of the model - Front View, Right View, Top View. The section for the Back, Left, Bottom views can be obtained by quickly switching the direction change manipulator;



The command works in a special mode, which allows you to apply or cancel changes. In this mode, the clip plane is displayed and manipulators are available;



- The clip plane is constantly displayed while the clip plane creation mode is active. The command is excluded from the drop-down menu;

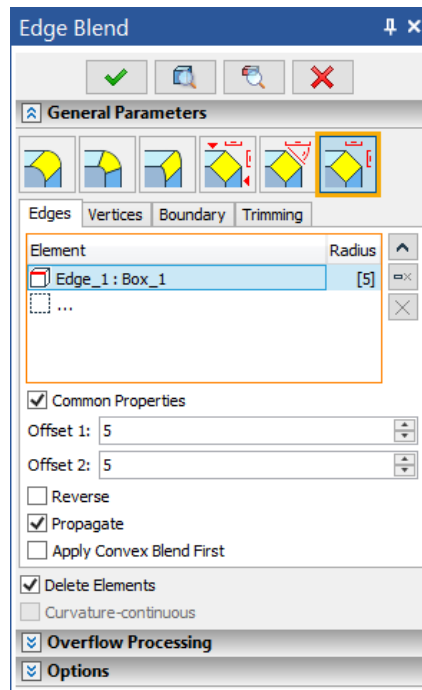


When the clip plane is activated, a manipulator and a secant plane appear in the scene, and a special mode is activated. The **Active** option allows you to enable/disable the previously set clip mode - i.e., go into editing it. The remaining options re-create the clip plane.

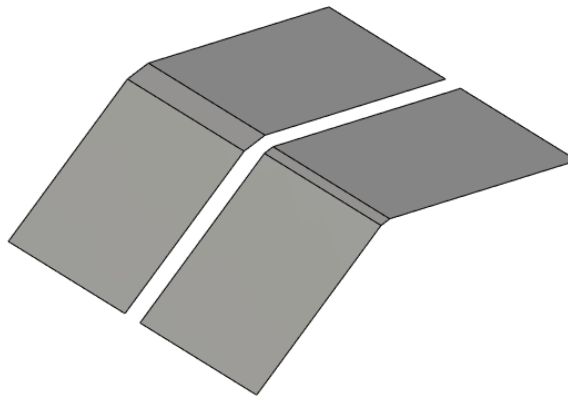
When saving the clip plane - exit the mode with confirmation of creating the clip plane - to move the clip plane, you can use **ctrl + shift + LMB** - as in previous versions of the system.

Blend

Added a new type of chamfering - **Chamfer (Offset along Faces)**.

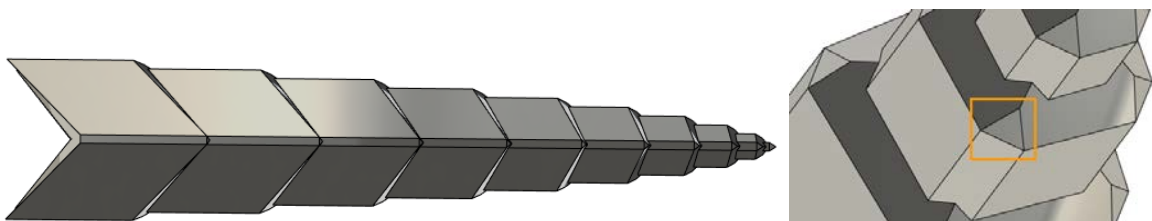


The chamfers of type **Offset along Face** (left) and **Offset** (right) with offset 1 and 2 parameters by 5 mm are shown below:



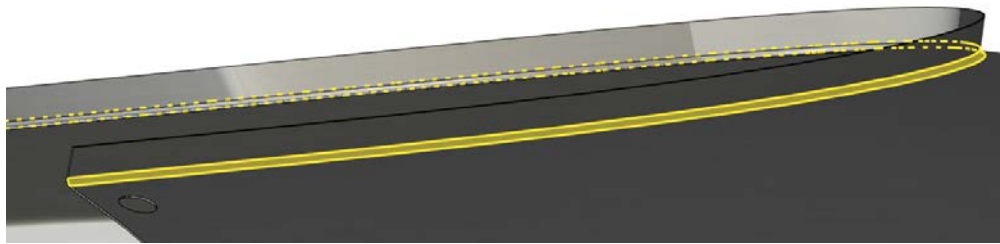
As you can see from the figure, in the new type of chamfer **Offsets along Face** are offset by 5 mm along the edges, while in the old type of **Offset** 5 mm values are converted to other values due to the angle between the edges other than 90 degrees.

Also, the ability to create a chamfer in the **Blend** command has been enhanced. Now the chamfer can be created at the intersection of several edges.

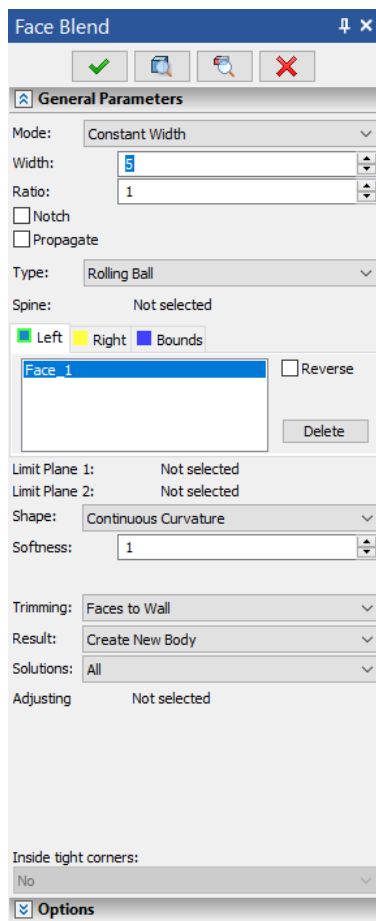


Face Blend

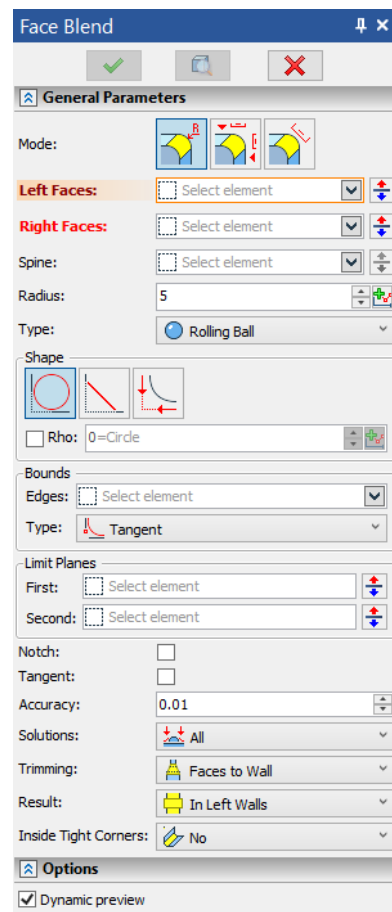
Serious improvements has been done in the **Face Blend** command.



The command dialog has been completely redesigned.

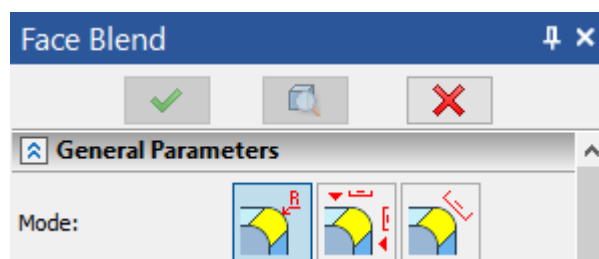


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Mode group of options has been reworked. Old options have been replaced with **Radius**, **Offset**, and **Width** values:



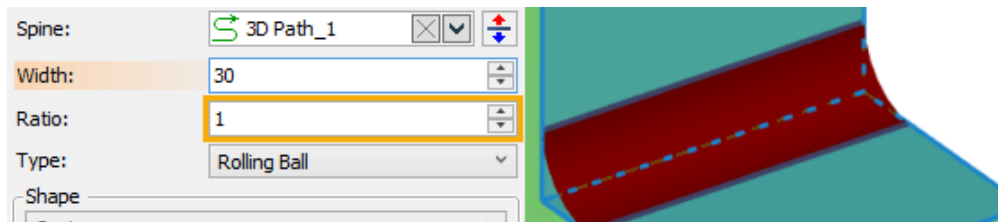
Radius mode is designed to create a symmetrical blend with a constant or variable radius. A variable value is indicated by using graphs.

In the **Offset** mode, you can control the offset of the two directions separately.

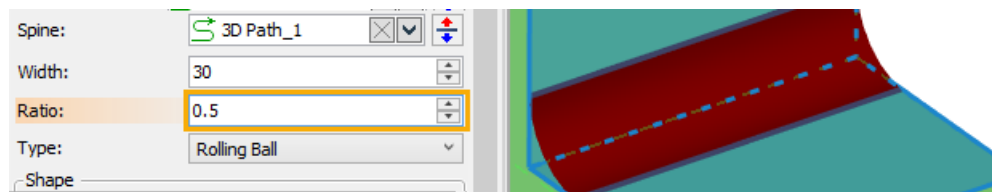


Width mode allows you to create a rounding with a constant width. The setting of the **Ratio** option is available.

If the ratio is 1, the rounding is symmetrical:

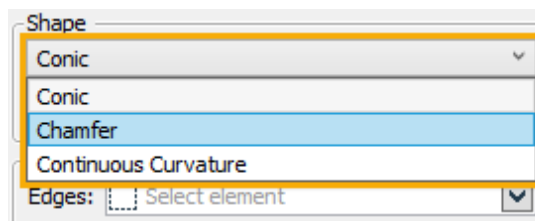


If the ratio less or greater than 1 - the rounding is asymmetrical:



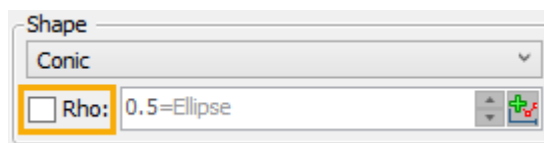
As a spine, you can now select paths, profiles, and edge sets.

A group of **Form** options has also been reworked. As before, three types of shapes are available: **Conical**, **Chamfer** and **Continuous Curvature**.



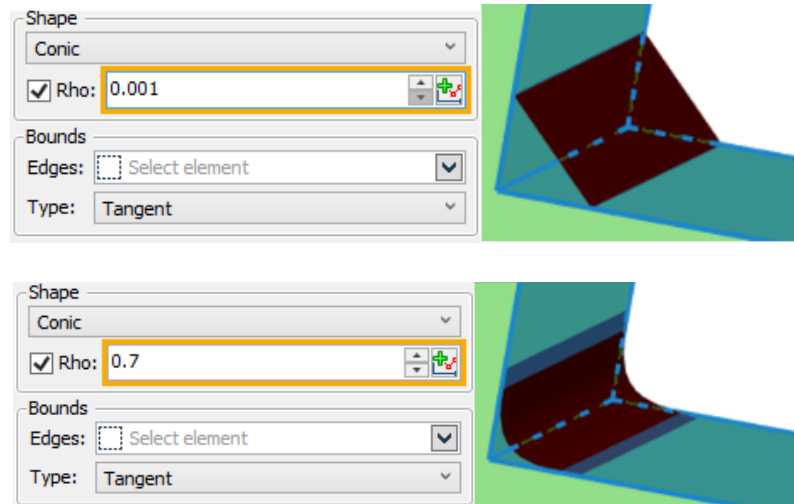
There are two possibilities for constructing a conical section.

The first is with the **Rho** option turned off:

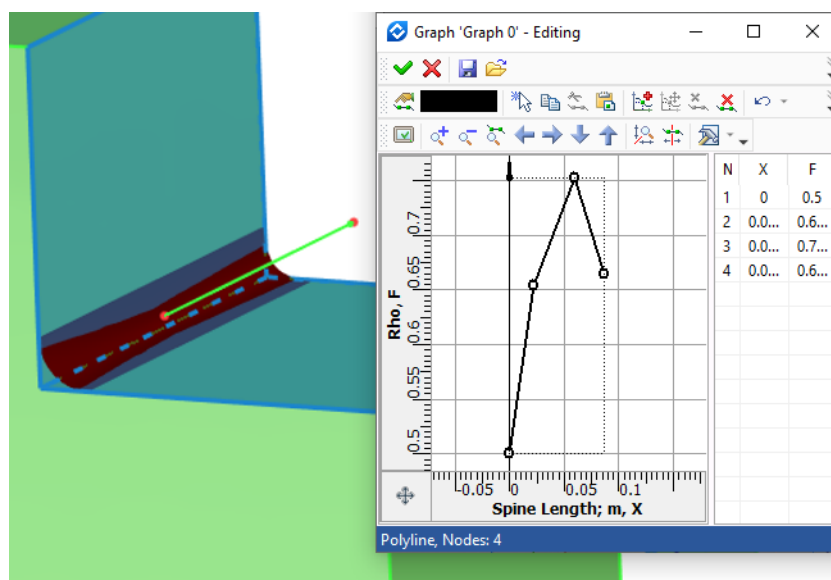


In this case, a circle will be drawn if the rounding is symmetrical, and if the rounding is asymmetrical - an ellipse.

The second possibility of creating a conical section is with the **Rho** option enabled. In this case, you will be able to set the value manually, or set the graph. The **Rho** value can be set from 0 to 1.

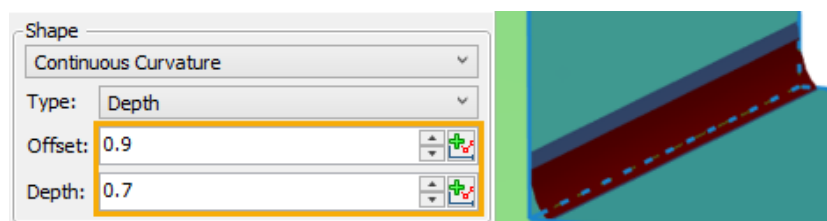


In the case of a graph, you will need to specify a guide.



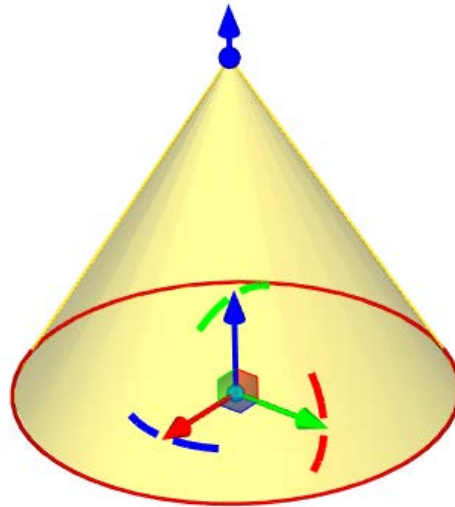
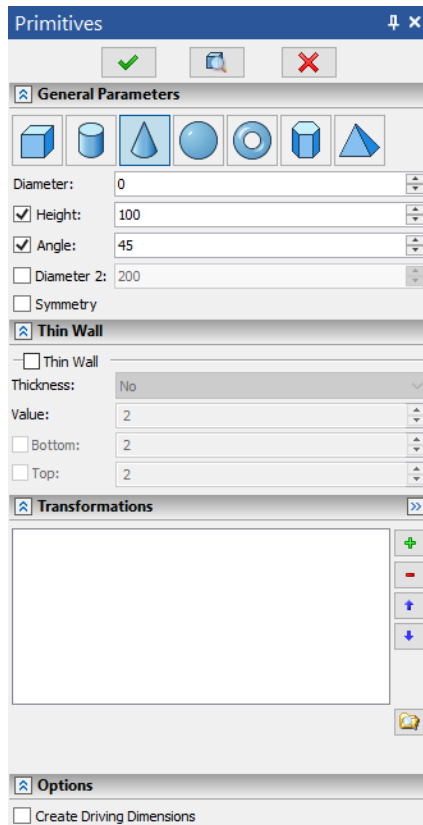
For the **Continuous Curvature** form, it became possible to select **Type**. Two types are available - **Softness** and **Depth**. Softness works similarly to the previous version of T-FLEX.

For the **Depth** type, it is possible to set the **Offset** and **Depth** values. For both options, you can set both a constant value and a value using the graph.



Primitive – Cone

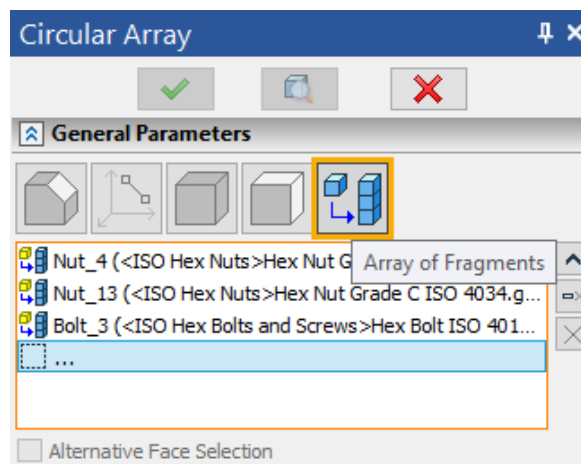
Added the ability to determine the set of initial geometric parameters of the cone by activating options opposite the required geometric parameters. Also added the ability to create a cone as a thin-walled element.



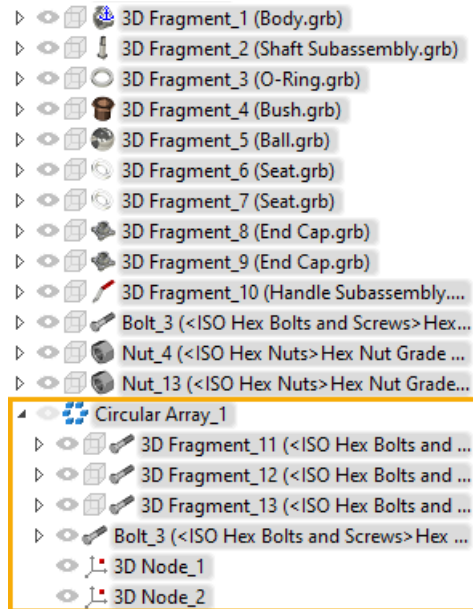
Array/Copy/Symmetry of Fragments

In arrays, copies, symmetries, a new type has added - the array/copy/symmetry of 3D fragments, which is intended to work with assemblies.

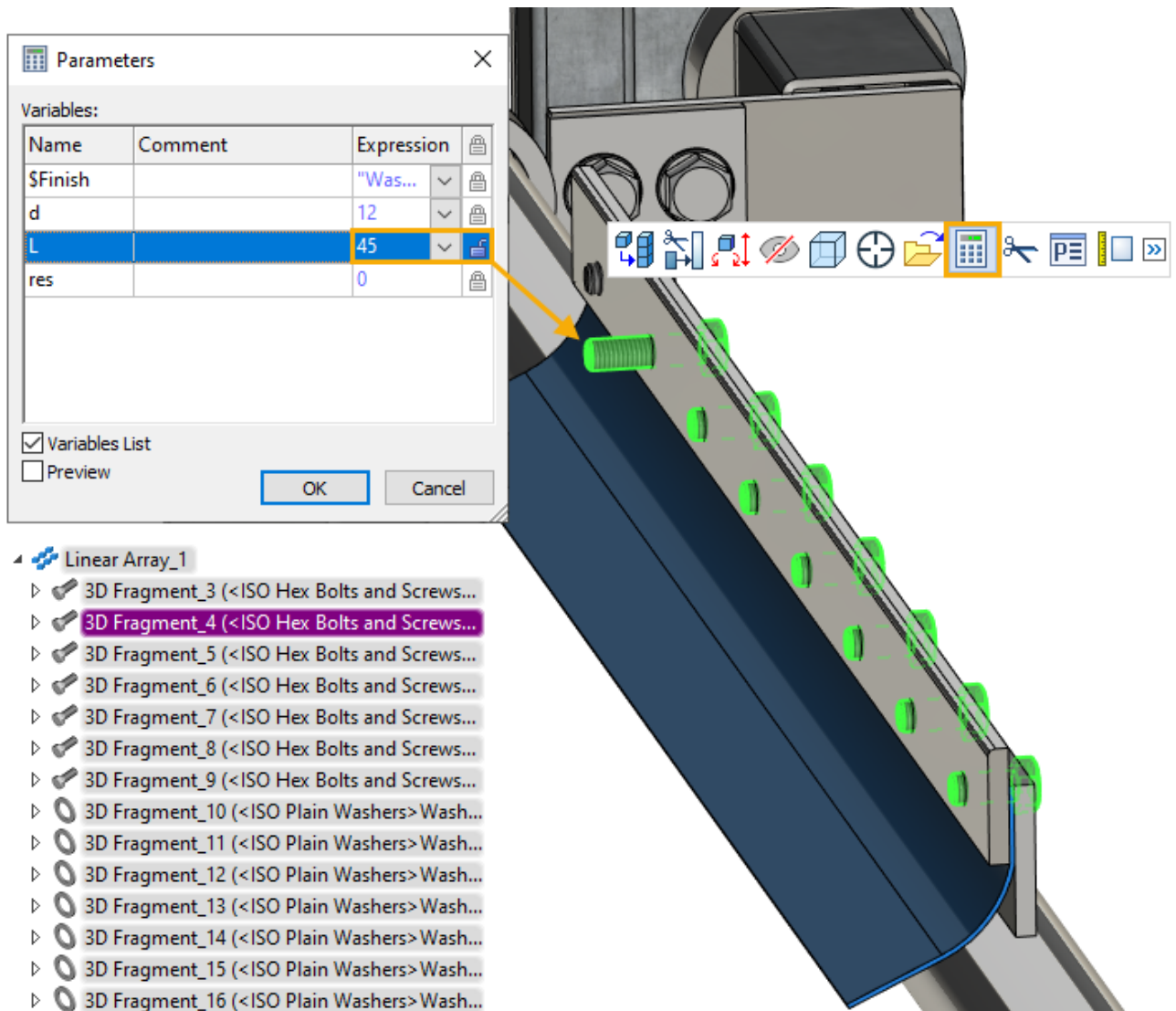
The new type allows you to select only 3D fragments as copy objects.



The operation gets into the model tree in the form of an item, inside which all the elements of the array/copy/symmetry are fully functional fragments. This allows you to apply, for example, transformations, exploded view scenario, and other operations to selected elements of arrays/ copies/symmetries.

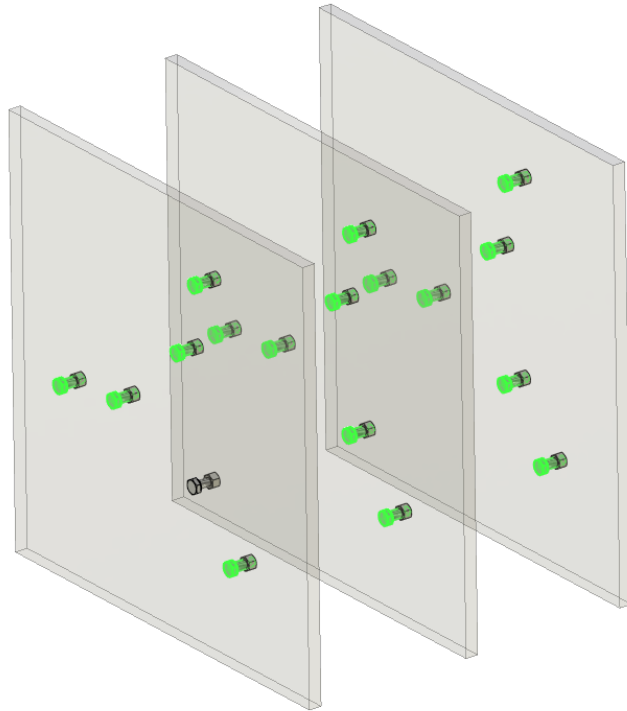
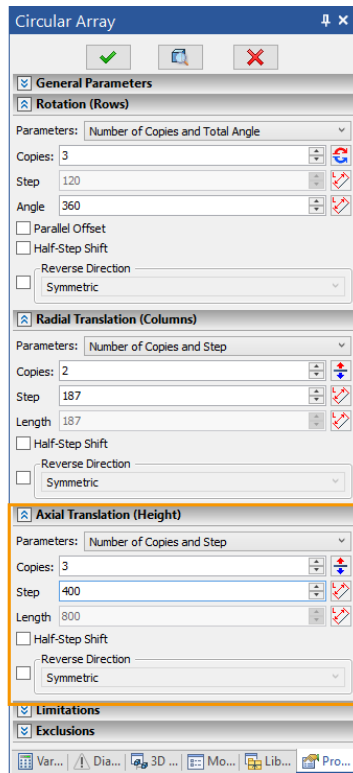


An important feature of the array of fragments is the ability to control the inheritance of parameters. For example, a fragment instance receives some of the parameters from the source fragment, while the other redefines itself.

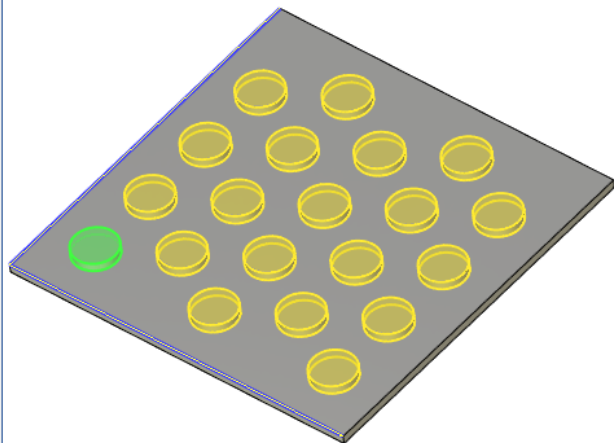
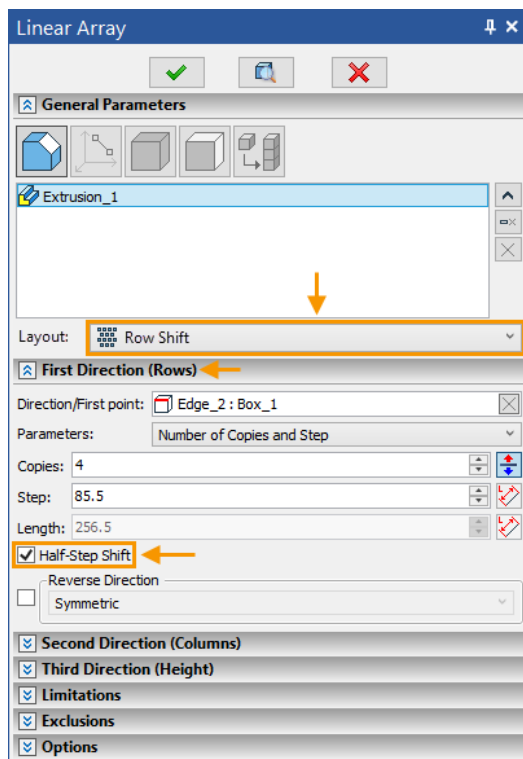


Linear and Circular Array

For linear and circular array operations, the dialogs of the general parameters and direction assignments were updated, and it is also possible to set a third direction vector.

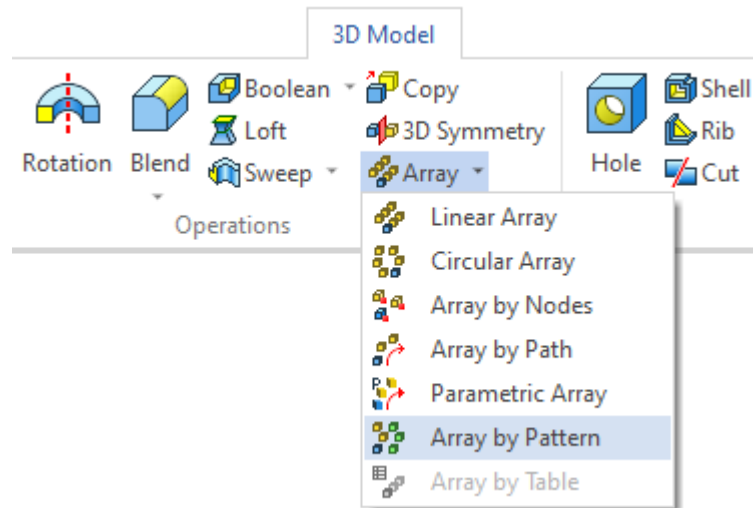


In the direction dialogs, the **Half-Step Shift** option has been added. When pressed, every second row/column/height is shifted by half a step. It also became possible to choose a layout. There are 4 options for schemes - **Standard**, **Row Shift**, **Column Shift**, **Custom**. Selecting a layout activates the **Half-Step Shift** option for the desired direction. Selecting more than one option for each direction activates a **Custom** layout.

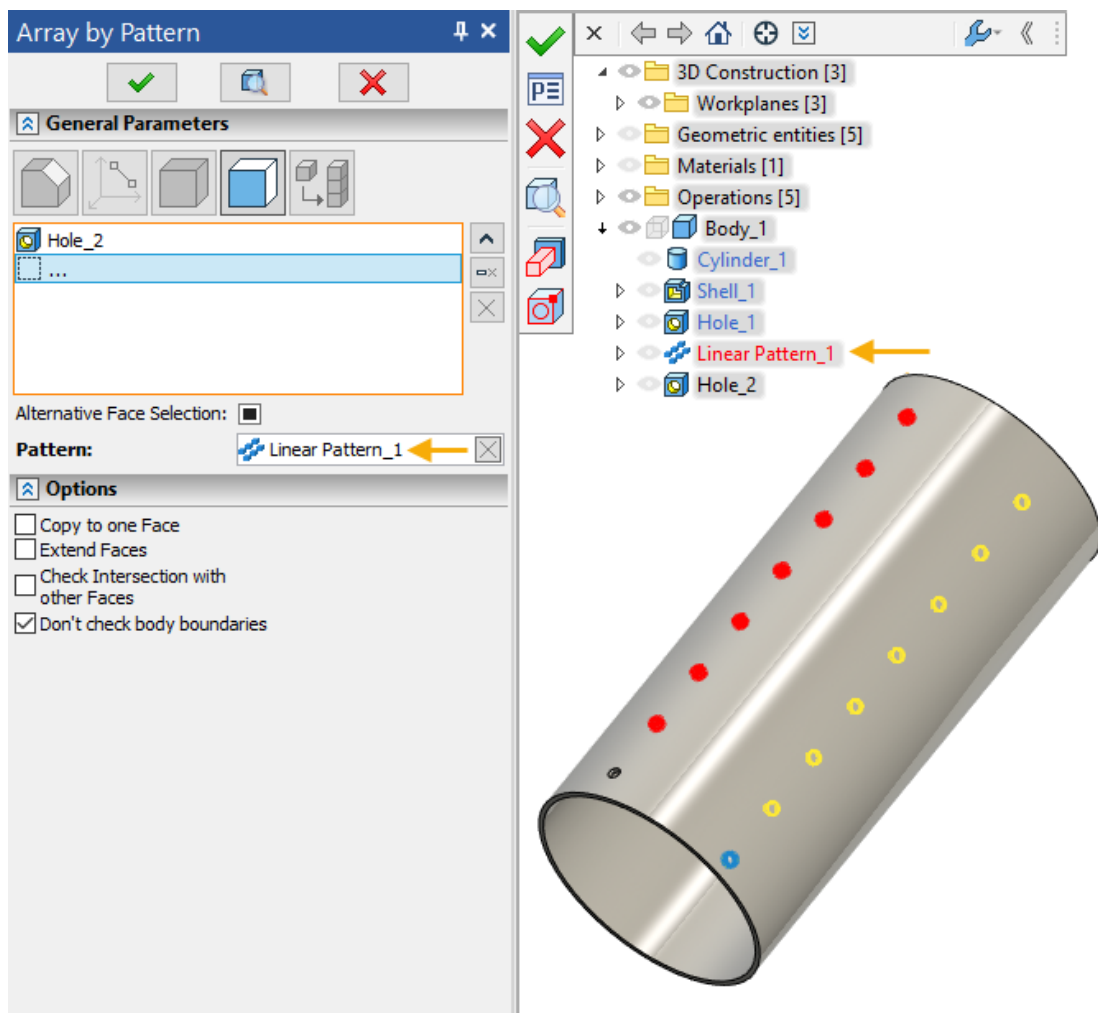


Array by Pattern

A new command **Array by Pattern** has been added, allowing you to create an array using an array already existing in the model as a prototype.



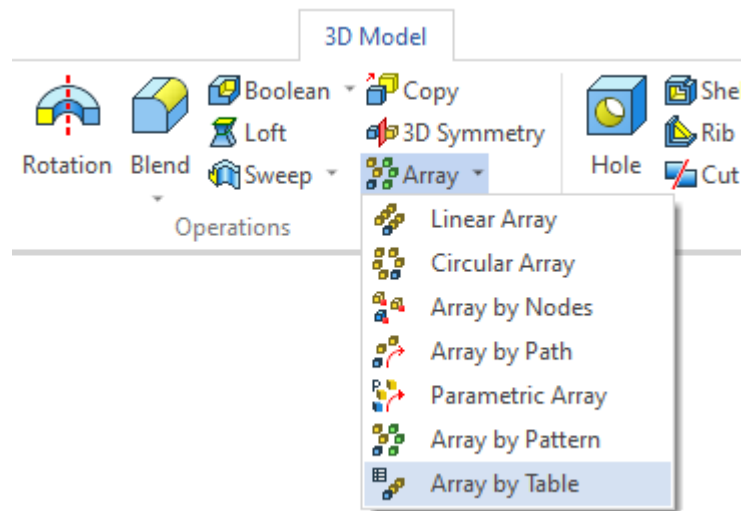
You can select any array as a pattern array.



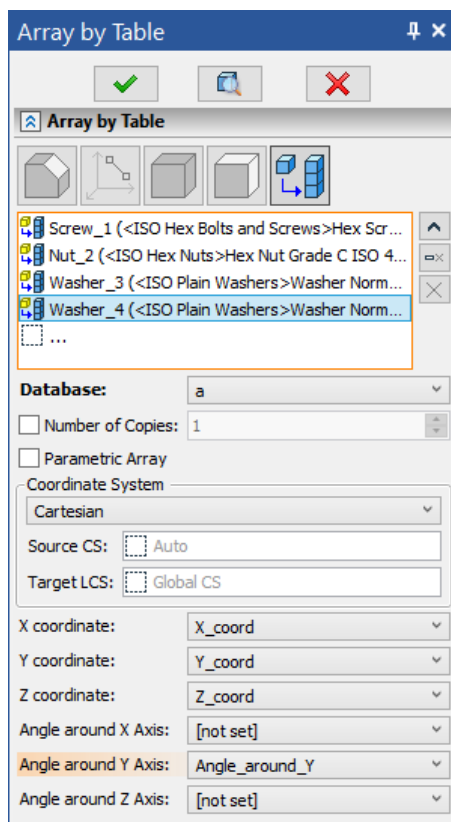
It is also possible to select faces as pattern.

3D Array by Table

Added a new command for creating 3D arrays - **Array by Table**.



With this command, you can use values from the database as array parameters.



a						
Nº	X_coord	Y_coord	Z_coord	Angle_around_X	Angle_around_Y	Angle_around_Z
1	60	60	30	90	90	0

There are 3 types of coordinate systems relative to which you can create an array from a database - Cartesian, Spherical and Cylindrical. When you select the type of coordinate system, a selection of the corresponding parameters appears.

Thus, 6 parameters can be set for a cartesian coordinate system:

- X coordinate

- Y coordinate
- Z coordinate
- Angle around X Axis
- Angle around Y Axis
- Angle around Z Axis

For a spherical coordinate system, 3 parameters can be set:

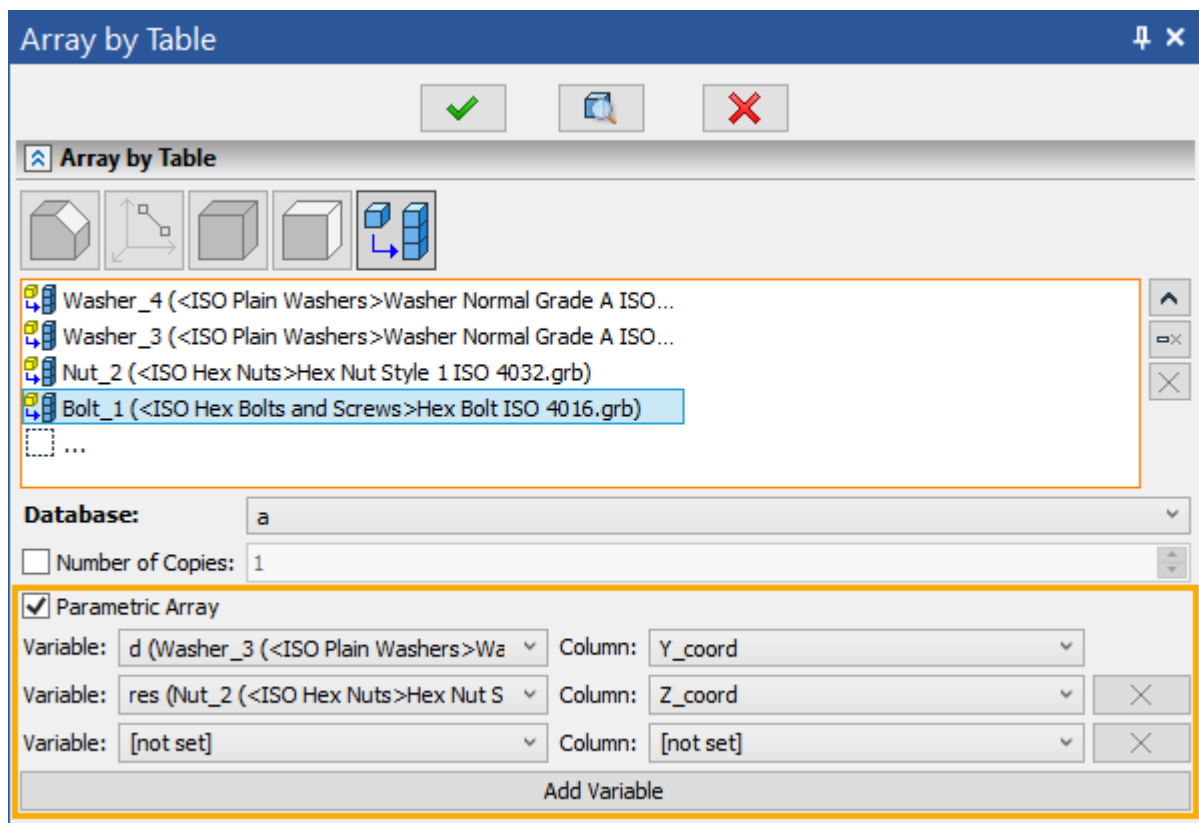
- Radial Distance
- Polar Angle
- Azimuthal Angle

For a cylindrical coordinate system, 3 parameters can be set:

- Radial Distance
- Azimuthal Angle
- Height

A prerequisite is to specify either the X coordinate or Y.

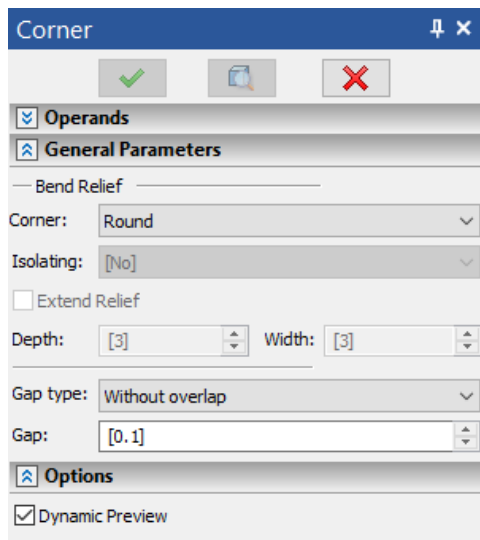
It is also possible to create a parametric array. For this, a special **Parametric Array** option has been added.



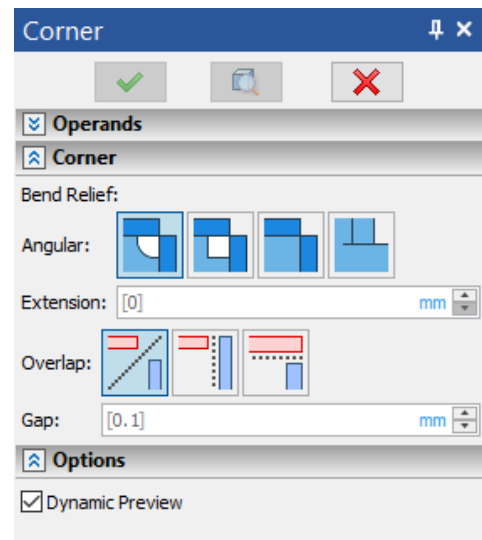
When the button is activated, the **Add Variable** button appears, which allows you to use the value of any model variable as the value of a database column.

Sheet Metal

The interface of various windows of parameters of sheet metal operations has been updated, in particular, the **Corner** operation has been redesigned.

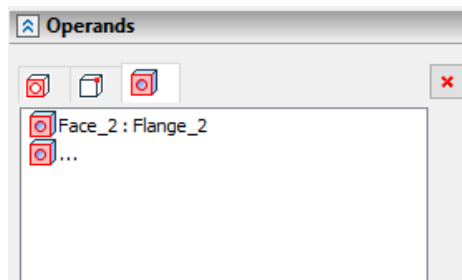


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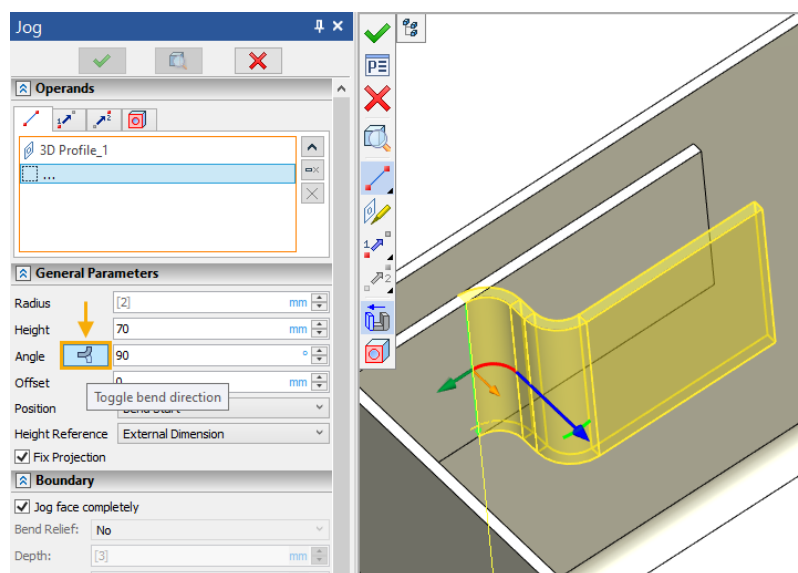
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You can now select boundary faces in an operation.




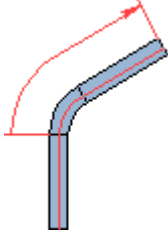
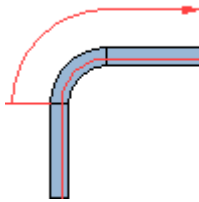
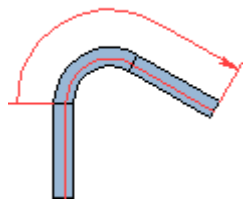

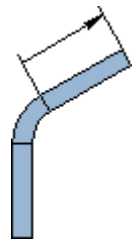
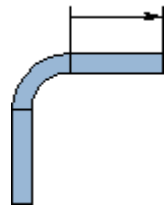
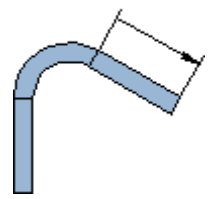

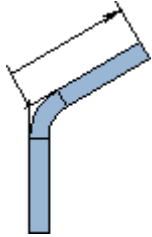
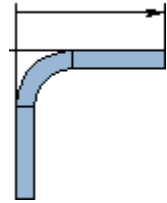
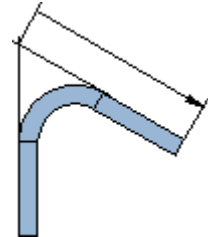

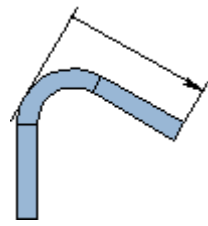

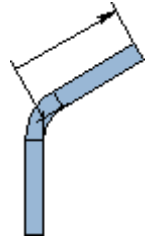
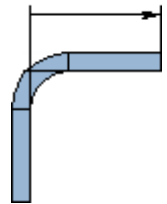
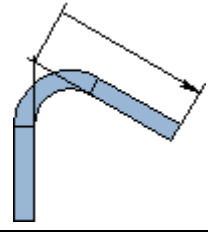

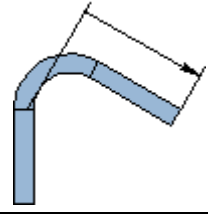
In operations where corner adjustment is available, such as **Flange**, **Hem**, **Convert Solid to Sheet Metal**, etc., the corner setting has also been updated.

In the **Jog** operation, the ability to toggle the bend direction has been added.




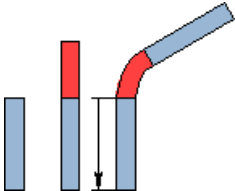
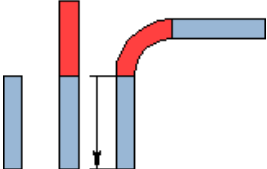
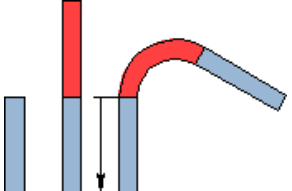

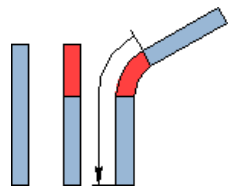
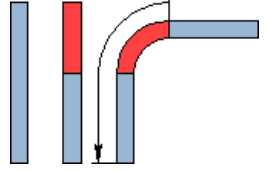
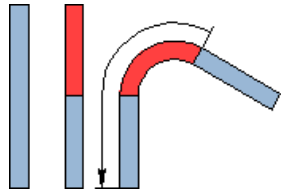

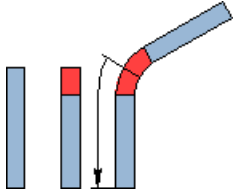
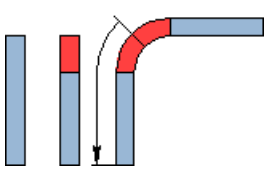
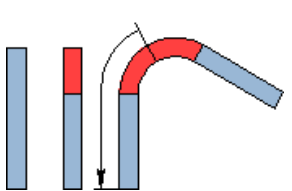

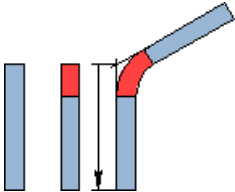
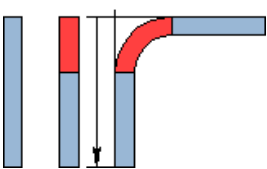
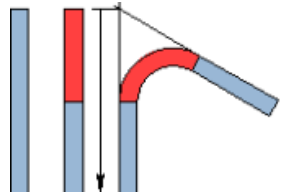

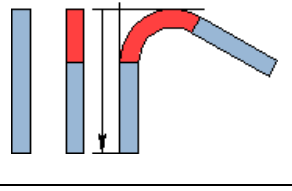
Flange Length Reference

The **Measured** parameter, which defines the method of applying a flange length, have been renamed to **Length Reference**. New options have been added for this parameter. The set of available methods is shown in the table below:


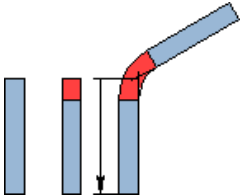
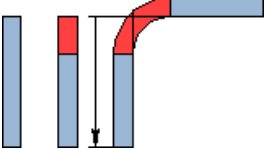
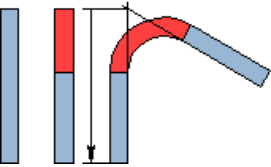

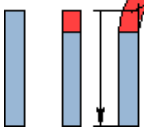
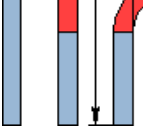
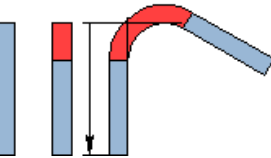

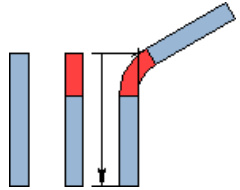
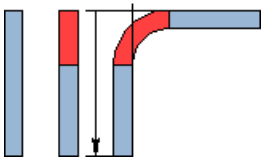
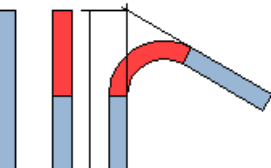
Icon	Name in T-FLEX CAD 17 (Name in T-FLEX CAD 16)	Method applied to different angles		
		0°-90°	90°	90°-180°
	Neutral Layer (Neutral Layer)			
	Flat Part (Flat Part)			
	Outer Side (External Side)			
	Outer Tangent			
	Inner Side (Internal Side)			
	Inner Tangent			

Position of Flange in relation to Baseline

New options have been added for the **Position** parameter, which defines the method of positioning a flange in relation to a baseline. The set of available methods is shown in the table below:

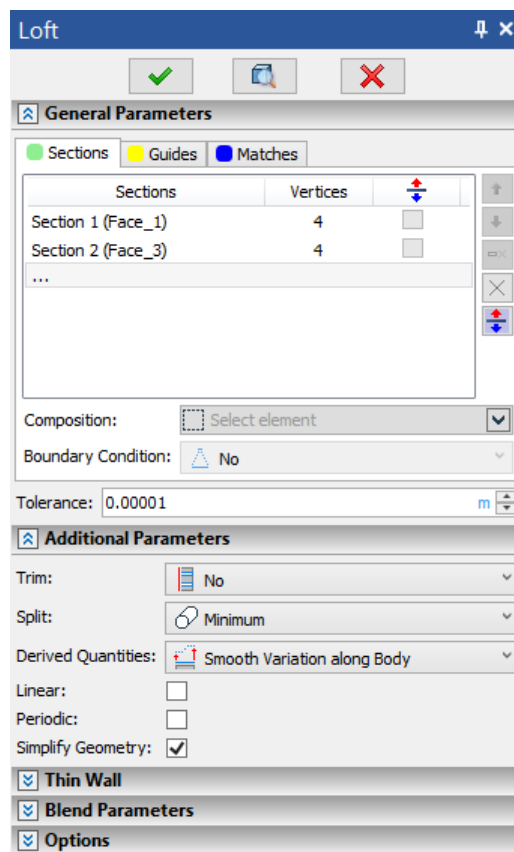
Icon	Name in T-FLEX CAD 17 (Name in T-FLEX CAD 16)	Method applied to different angles		
		0°-90°	90°	90°-180°
	Bend Start (Bend Start)			
	Bend End (Bend End)			
	Bend Center (Bend Center)			
	By Outer Side			
	By Outer Tangent			

Continued on the next page.

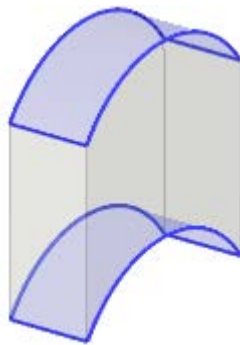
Icon	Name in T-FLEX CAD 17 (Name in T-FLEX CAD 16)	Method applied to different angles		
		0°-90°	90°	90°-180°
	By Inner Side (Internal Dimension)			
	By Inner Tangent			
	By External Dimension (External Dimension)			

Loft

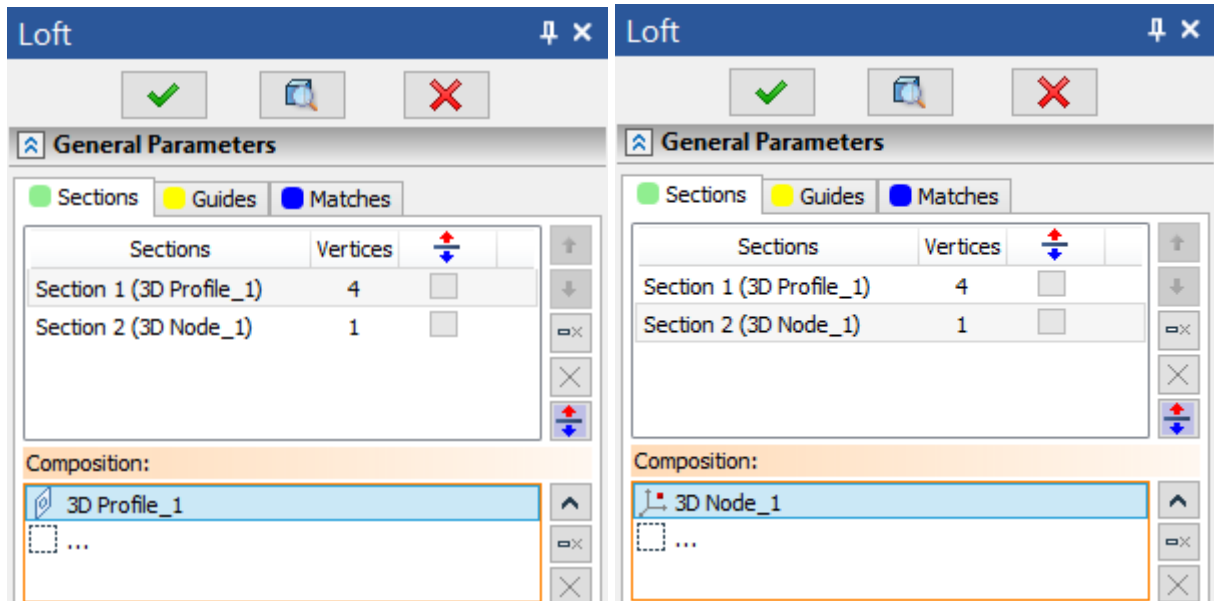
A lot of work has been done to update the **Loft** operation. Operation dialog has been completely updated.



Working with the operation has become more clear thanks to a global redesign of the interface. Added the ability to select non-planar sections.



Now, in order to select a point or planar/non-planar section, you do not need to use additional commands in the automenu, because selection of both types of sections is carried out in the **Composition** field.

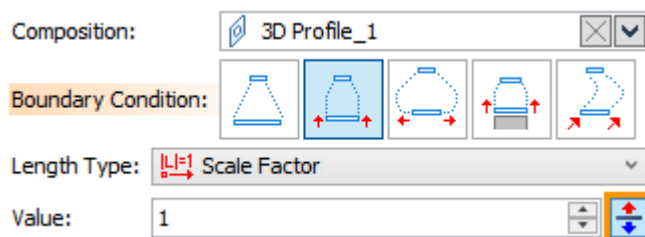


Now for each section, you can now set 4 boundary conditions – **By Normal**, **By Tangent**, **By Geometry**, **By Vectors**.

The **Normal (outside)** and **Normal (to profiles)** options have been moved to a separate boundary condition **By Normal**.



In all four types of boundary conditions, the option to reverse the direction of the vector or normal is available.

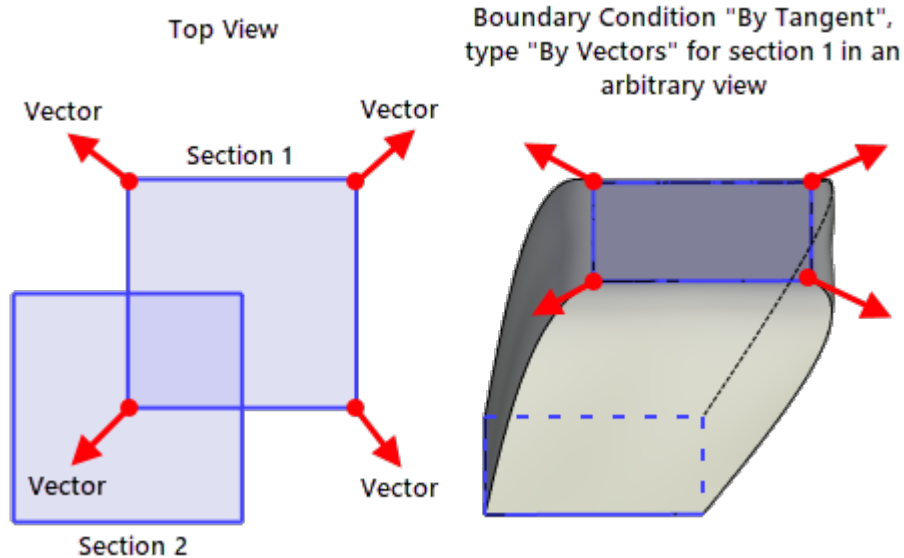


By Tangent option is now also included in the general list of boundary conditions.

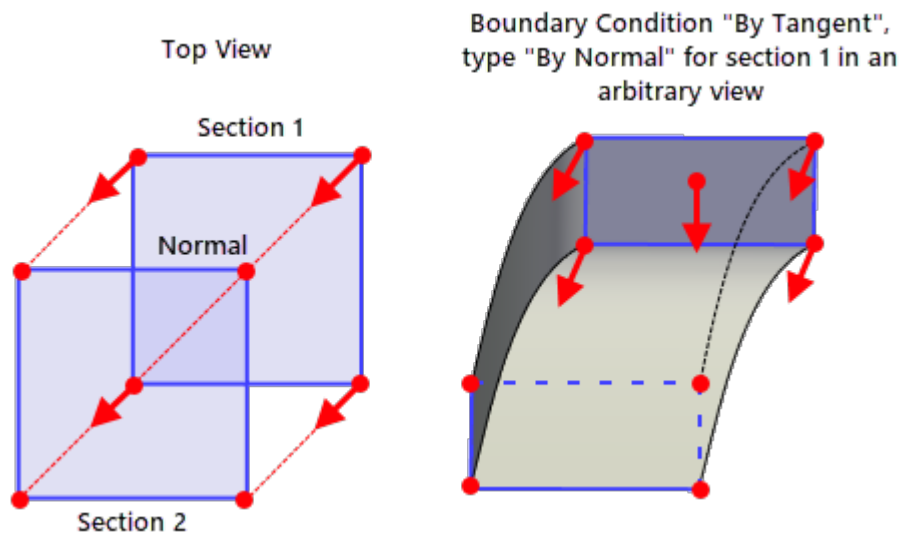


For this boundary condition, two types are available – **By Vectors** and **By Normal**.

The type **By Vectors** allows you to create a surface along planar and non-planar sections, taking into account vectors directed along the tangent. This is the "classic" type of the **By Tangent** boundary condition.



The type **By Normal** allows you to create a surface only by planar sections, taking into account the normal. A prerequisite for the creation of a surface is the presence of a normal at the section. The direction depends on the position of the next section.



To create a solid along the faces of other bodies, use the **By Geometry** boundary condition, which allows you to select a face or a set of faces.



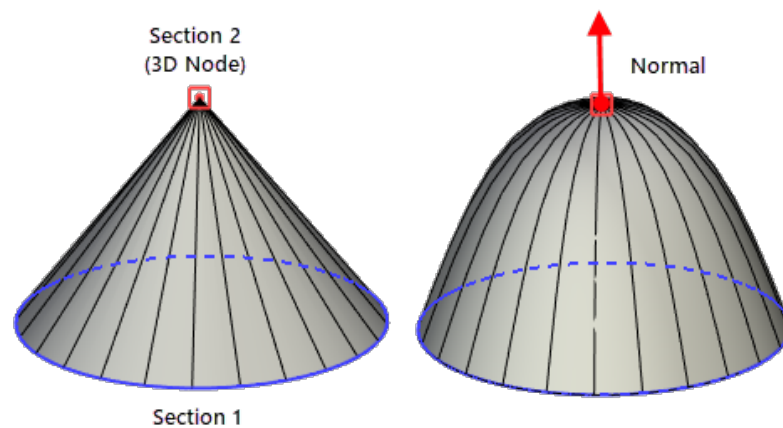
For this type of boundary condition, you can set the curvature **G1** or **G2**.

The window for specifying the vectors of boundary conditions has been moved to a separate boundary condition **By Vectors**.

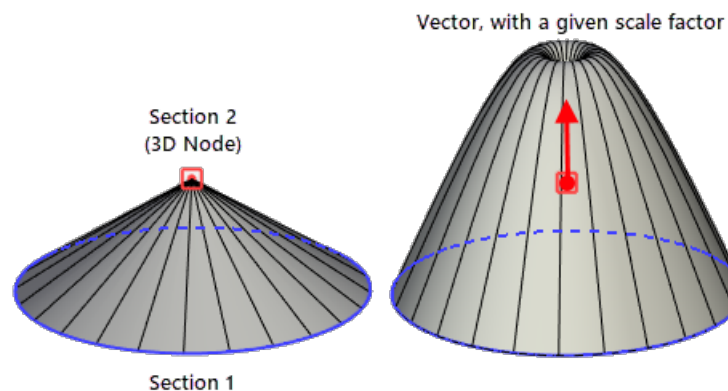


Depending on the type of section, the user will be offered a certain set of parameters. If a point is selected as a section, then the user can specify the type of condition **By Normal** or **By Tangent**.

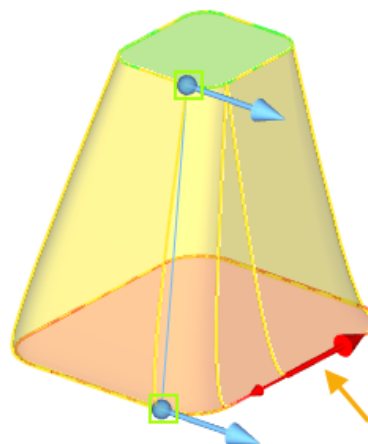
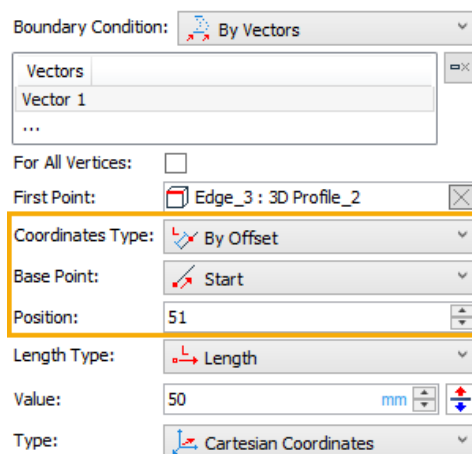
Condition type **By Normal** provides smooth ending for the created surface, limiting it to a tangent plane with a specified normal at the section-3D point.



Condition type **By Tangent** Produces a more general ending which may or may not be smooth, constraining the surface by limiting it with a vector of derivatives in a section-point.

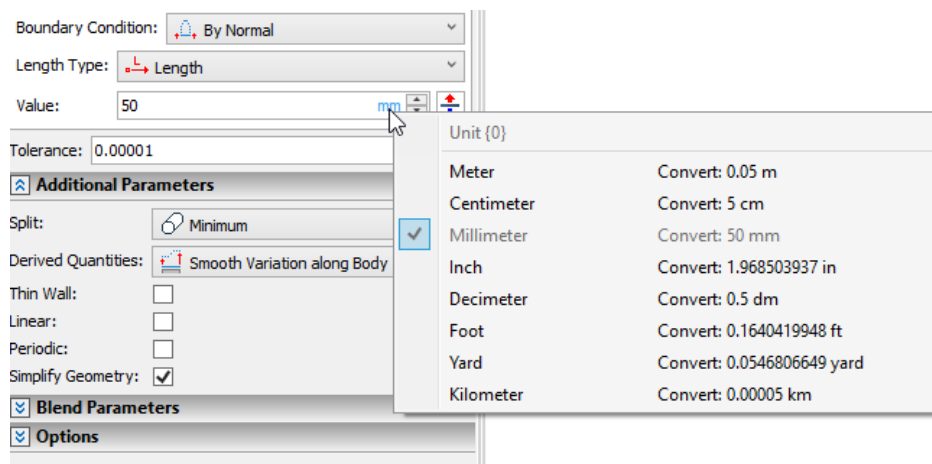


If a planar/non-planar section is selected as a section, then you can specify a set of vectors and select a first point for each. As the first point of a vector, you can select a point belonging to a section or an edge of this section and move along it using the manipulator or in the operation parameters window. In this case, you can select the type of coordinates **By Offset** or **Parametric**, and also specify the origin point **Start**, **Middle** or **End** similar to creating a 3D node on a curve.



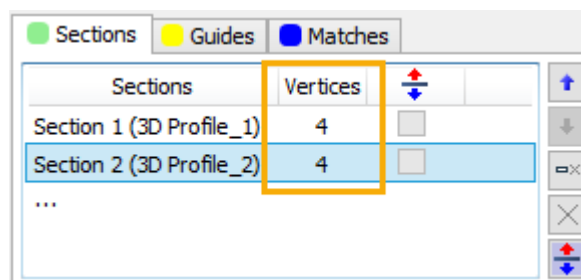
As before, for the boundary condition **By Vectors**, you can select the type of assignment - **Spherical** or **Cartesian** coordinates, **By Two Points** and **By Direction** of element.

For all boundary conditions, as before, you can select the type of length - **Length** or **Scale Factor** and set the **Value** for which a choice of systems of units is now available.



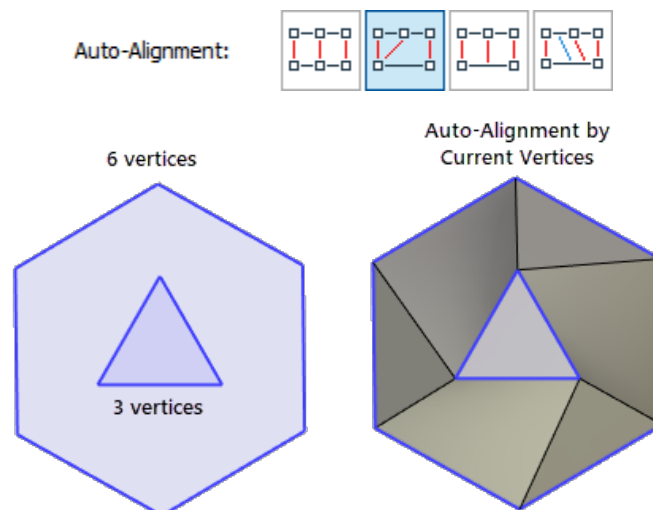
The **Guides** and **Matches** tabs have also been reworked. For a guide, you can specify the boundary conditions **By Geometry** and **By Vectors**, similar to the boundary conditions for sections, and for matching points, you can specify the boundary condition **By Vectors**.

When the **By Vectors** boundary condition is selected, the **For All Vertices** option is available for planar/non-planar sections and guides, which allows you to select one vector for all section vertices, except for the vertices of intersection with the guides and except for the matching points that have boundary conditions. In the new version of T-FLEX CAD, an emphasis is placed on maintaining the number of vertices for sections. For this, the number of section vertices is displayed in the section selection tab.

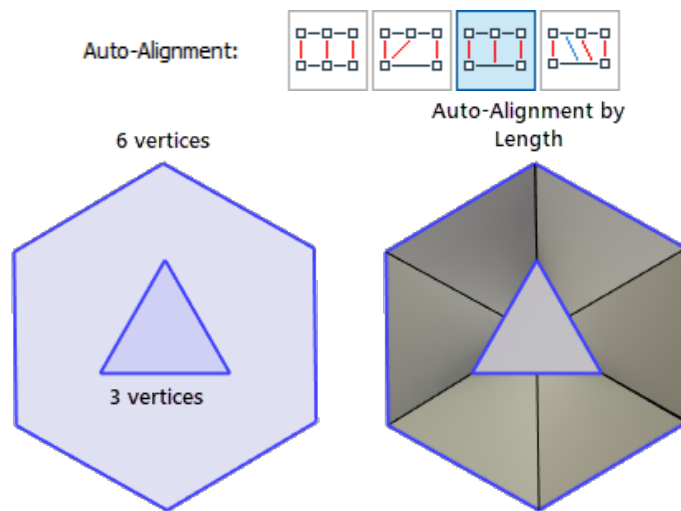


If the number of vertices is different, then for the correspondence points, you can set auto-alignment **By Current Vertices** or **By Length**.

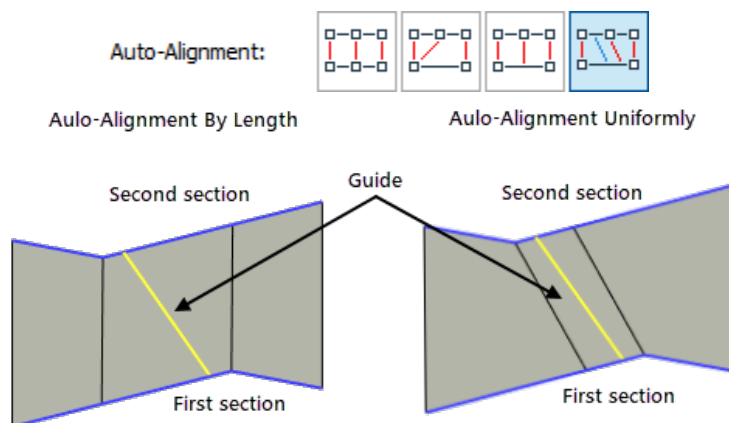
The **By Current Vertices** option allows you to automatically align the match points to the vertices of the sections.



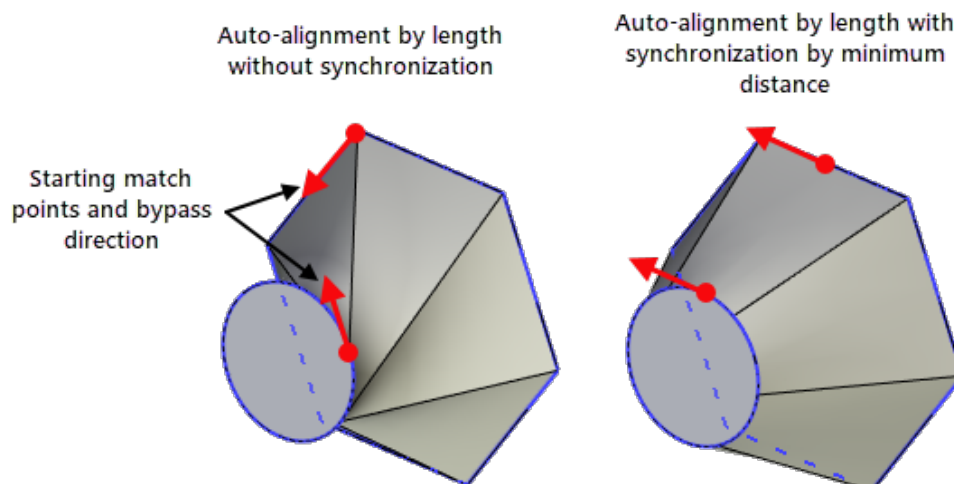
The **By Length** option allows you to automatically align the match points of the sections so that spacing between adjacent edges has the same ratio to total length.

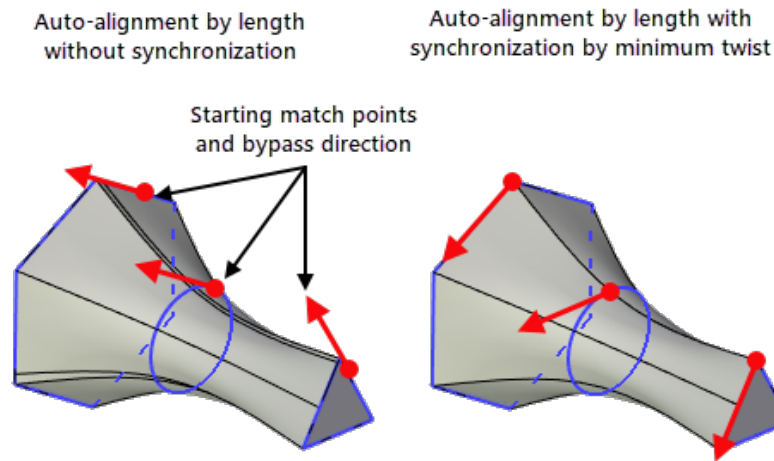


The **Uniformly** option allows to automatically align match points if at least one guide or one additional match is selected. If there is no guide and no matching, then the mode will work in the same way as the **By Length** mode.

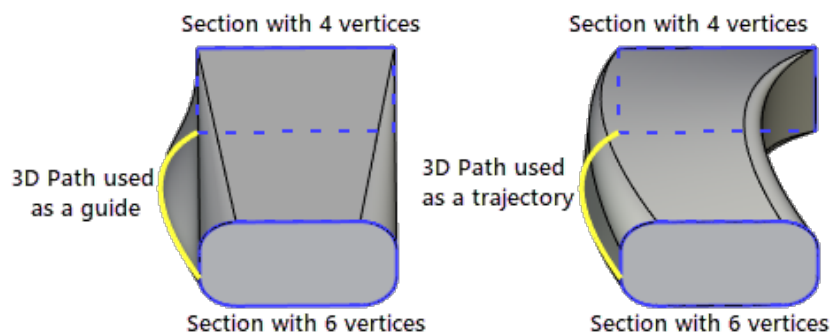


Added the **Synchronize** option, which allows you to automatically set the position of the starting points, ensuring the minimum distance between sections or minimum twist. This can be useful when circles or ellipses with only one starting point are used as sections.

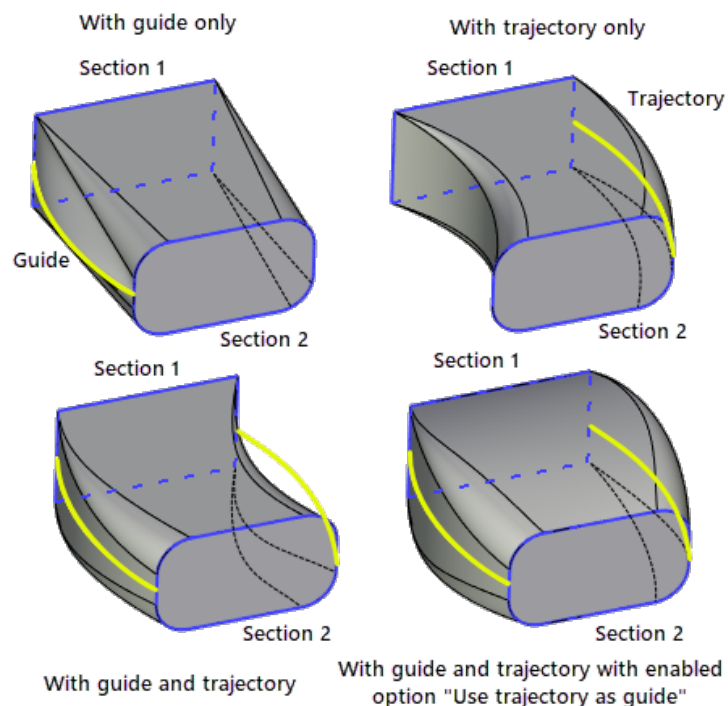




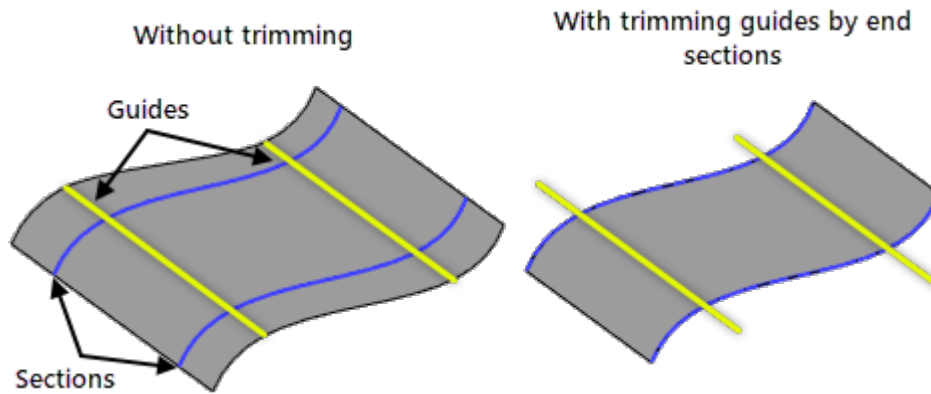
The option **3D Path** has been changed to the **Trajectory** option. The selection of the trajectory allows the use of wire geometry, which will determine the shape of the result along all vertices of the sections.



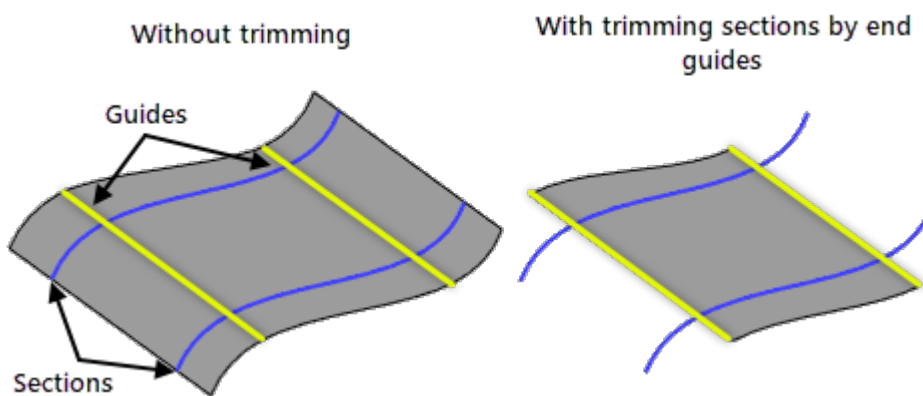
If necessary, you can add both guides and trajectories at the same time. If a trajectory is selected and at least one guide is selected, the **Use trajectory as guide** option will become active.



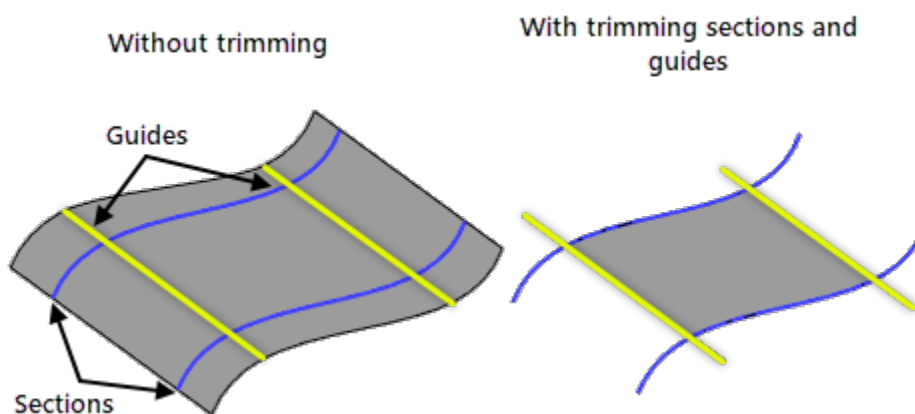
Added the ability to trim guides and trajectories if they are longer than the section boundaries. The new option **Trim - By End Sections** will only use the part of the guide or trajectory that is located between the first and last sections.



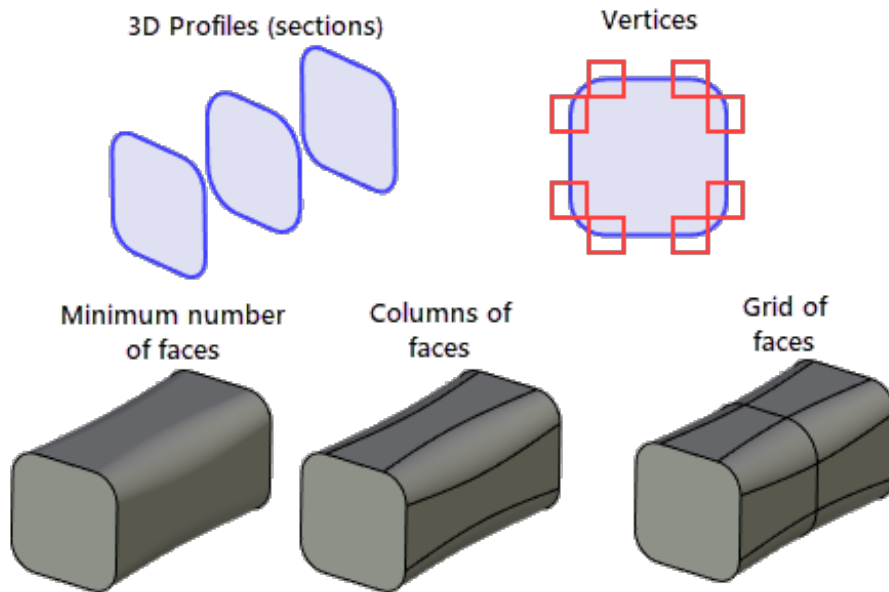
Similarly, you can trim sections using the **Trim - Sections by End Guides** guides option. To trim sections, you need at least 2 guides or 1 guide and a trajectory, with the **Use Trajectory as Guide** option enabled. Section trimming is possible only for open sections.



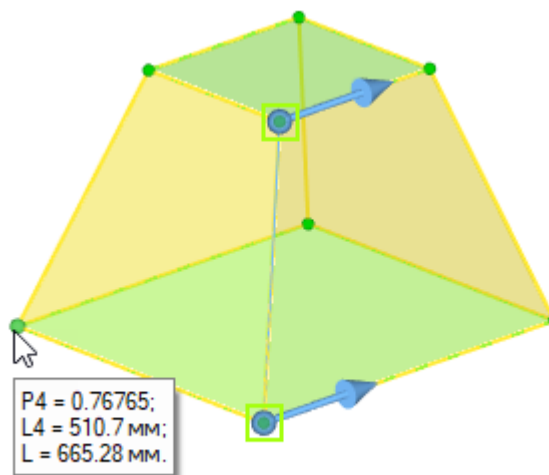
If necessary, you can trim both guides and sections at the same time as described above. To do this, use the **Trim - Sections and Guides** option.



Keep initial splitting option, which displays the number of faces, has been replaced with the **Split** option, which contains three types of face splitting - **Minimum**, **Columns** of faces and **Grid** of faces. The first type allows you to create a body with a minimum number of faces. Columns are produced by dividing the body along the sections direction at each profile vertex, and a grid of faces is produced by dividing the body into columns and then dividing along the body according to the position of the sections.



For sections, special manipulators are added in the original vertices. Hovering over them invokes a tip with additional information:



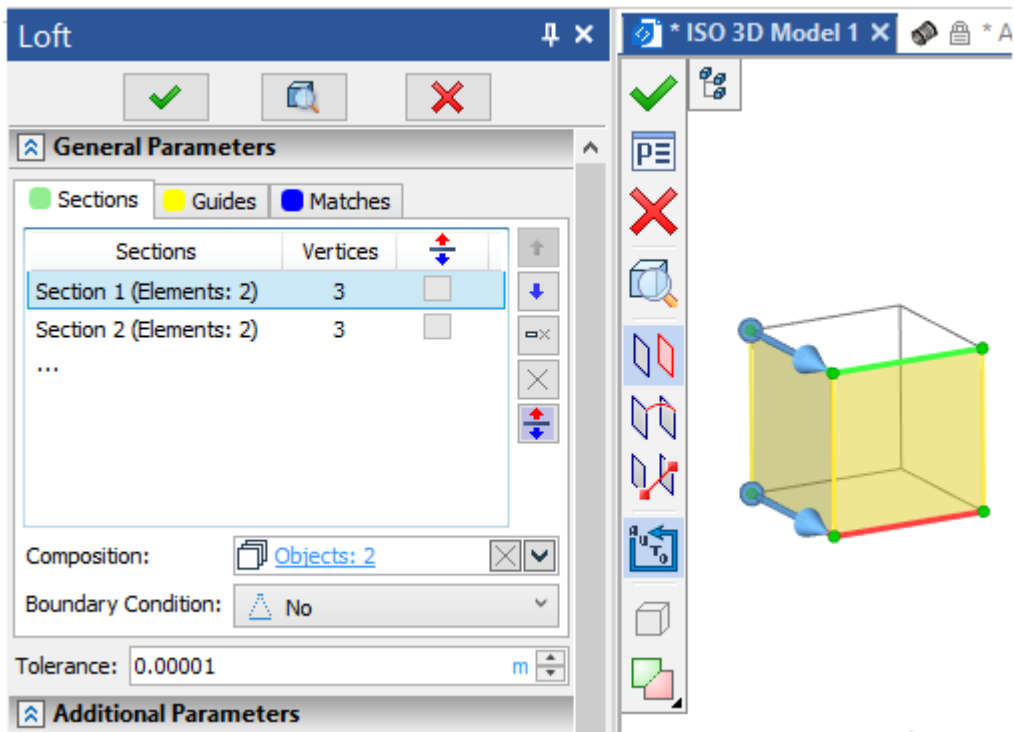
L - overall length of the section

L4 - length from the starting point to the current point, where 4 - the number of the current point

P4 - L4/L coefficient

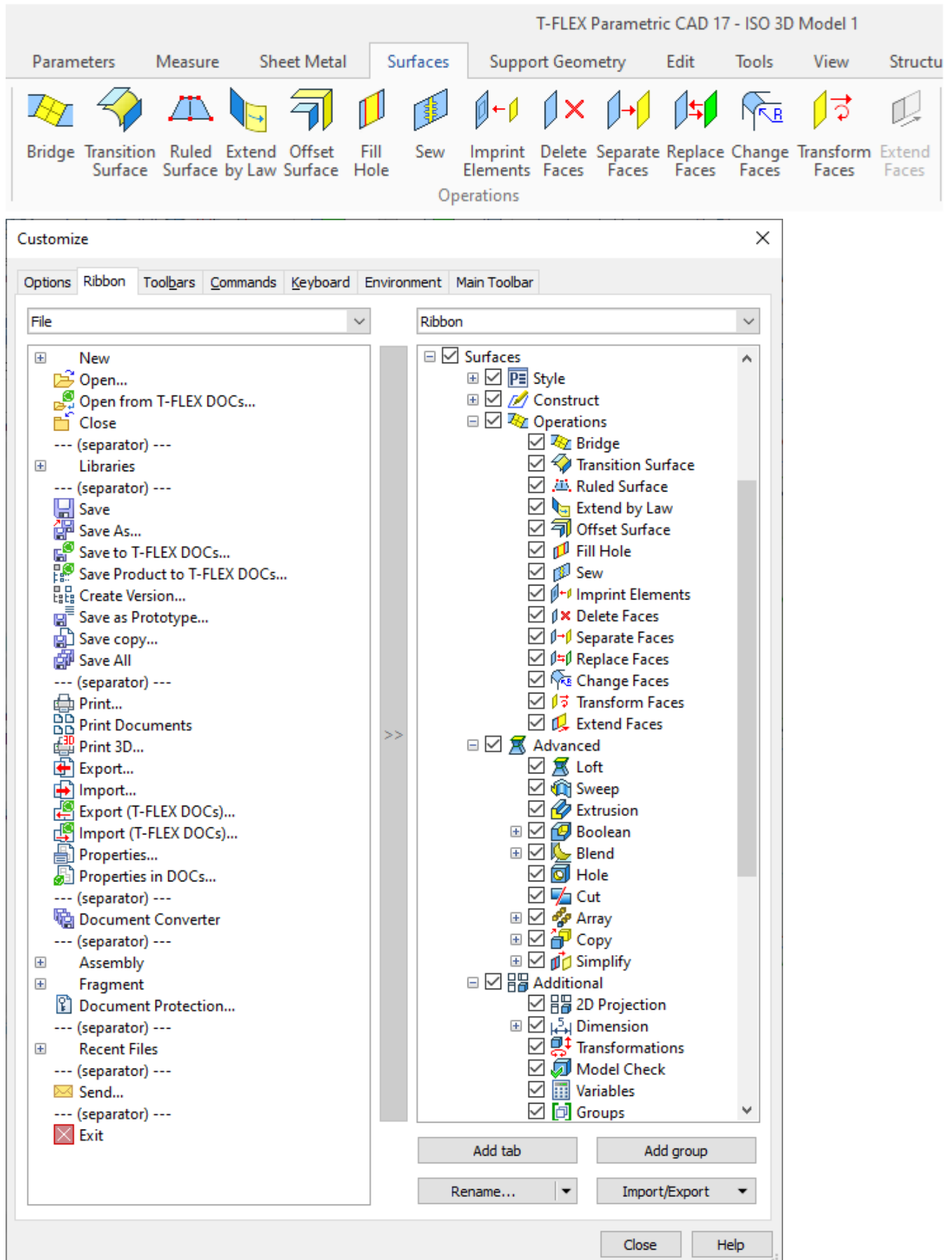


The new **Automatic Contour Search** option was added to the automenu. If it's enabled (by default), the system automatically decides whether selected element should be included into one of existing sections/guides or a new section/guide should be added.

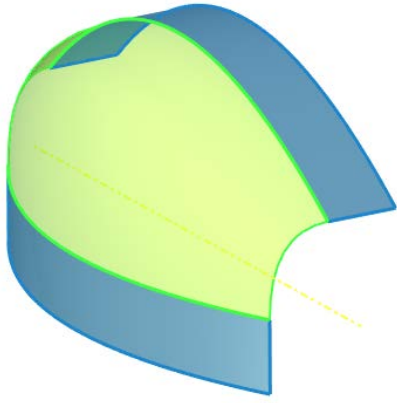


Surfaces

Now you can create surface models of any complexity in T-FLEX CAD. New commands for working with surfaces were created. Commands for working with faces and surfaces were improved. All surface commands are placed in a separate **Surfaces** ribbon tab.




Transition Surfaces

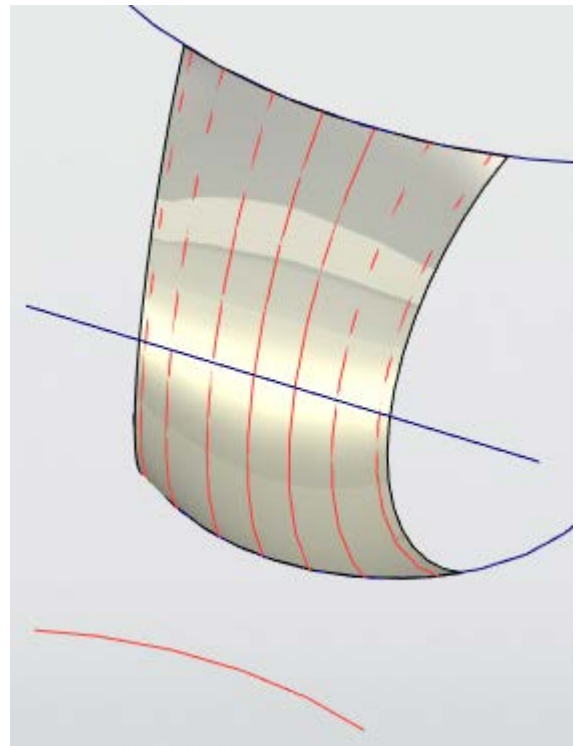
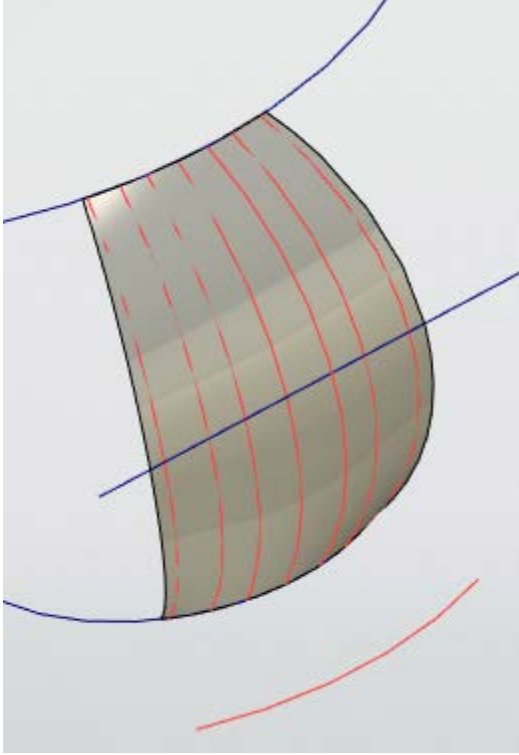


A new **Transition Surfaces** command was added. This command enables you to build surfaces based on conic sections: a parabola, hyperbola, ellipse, or with sections representing a circle, circle arc, or segment, with different geometric conditions and parameters.

The command contains ten options for constructing transition surfaces and will be developed. Depending on the selected option, the edges of the surface can be set as curves or be free. In this case, the shape of the surface edge can be set by a graph.

	Conic – Discriminant
	Conic – Path
	Conic – Tangent
	Conic – 4 Points
	Circle – Tangent to Surface
	Circle – Radius
	Circle – Radius and Angle
	Circle – Three Points
	Line – Angle
	Line – Tangent

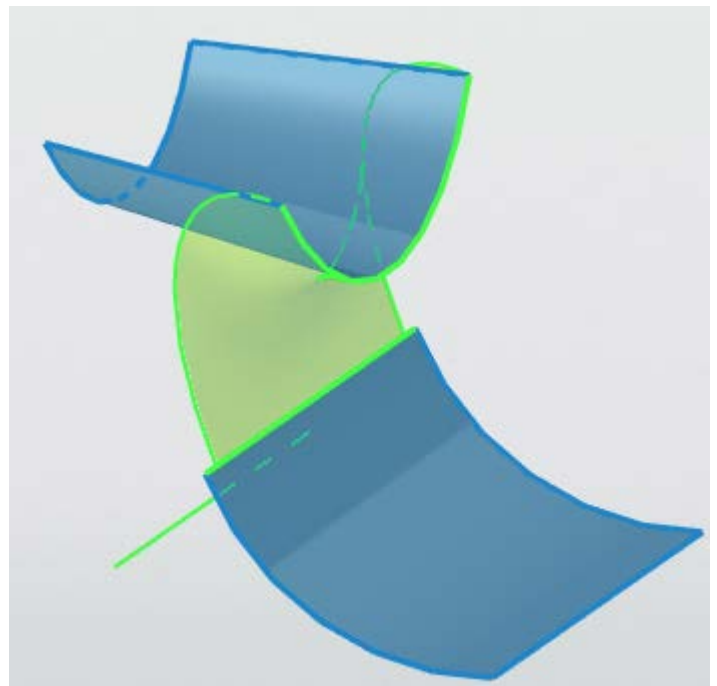
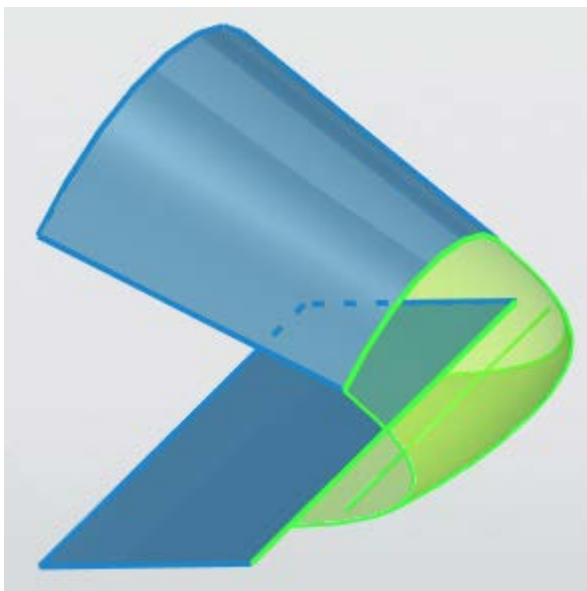
The basic element of any construction option is the reference curve. The reference curve can be defined by multiple smoothly conjugated edges, profiles, a trace, or a 3D path. The reference curve determines the direction of the tangents. The planes are located along the normals to these tangents. The forming profiles that determine the geometry of the created surface lie in these planes. The forming profiles and the planes in which they lie are not visible to the user. The figure shows the forming profiles conditionally: only the influence of the reference curve on the position of the profiles is reflected, their real frequency is much higher than in the figure.

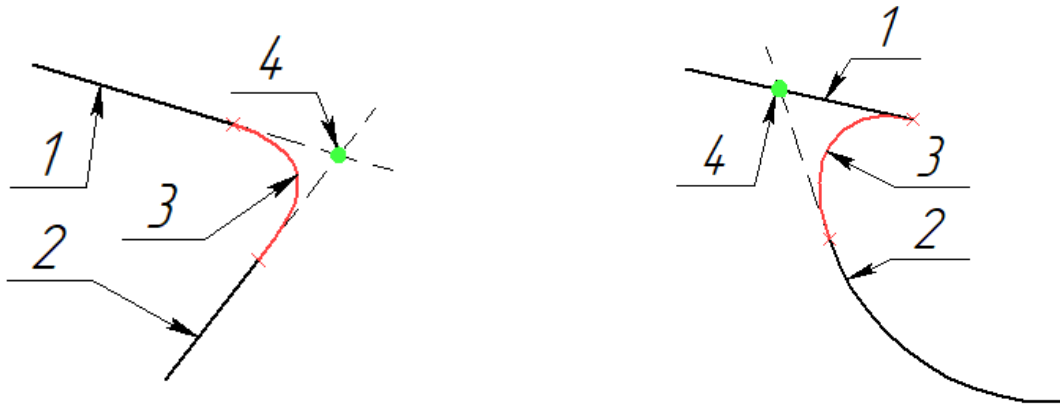


Influence of the reference curve on the location of invisible forming profiles

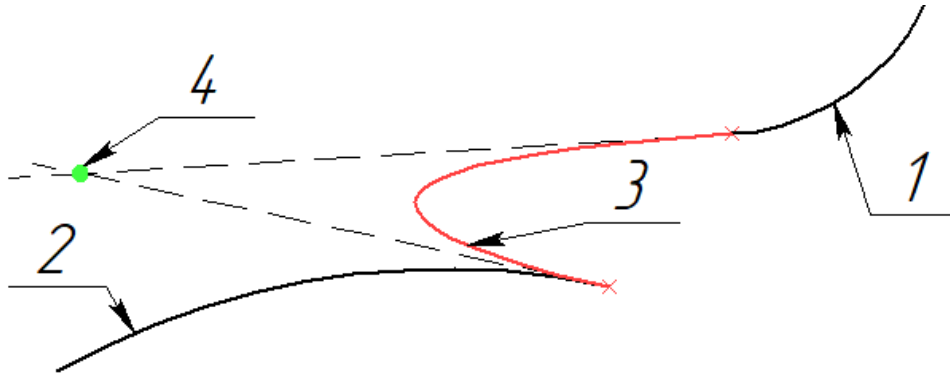
The law for constructing invisible profiles is determined by the selected method for constructing the transition surface, which is indicated above. Profiles can be parabolas, ellipse arcs, hyperbolas, circle arcs, circles, or segments.

The direction of contact of the conical surfaces of the first three types is determined by the location of the surfaces on which the guide curves lie. In this case, the bending direction of the created surface will be the only possible one. The crossing point of the tangents to the intersection of the surfaces in each plane of the invisible profiles determines the curvature direction.



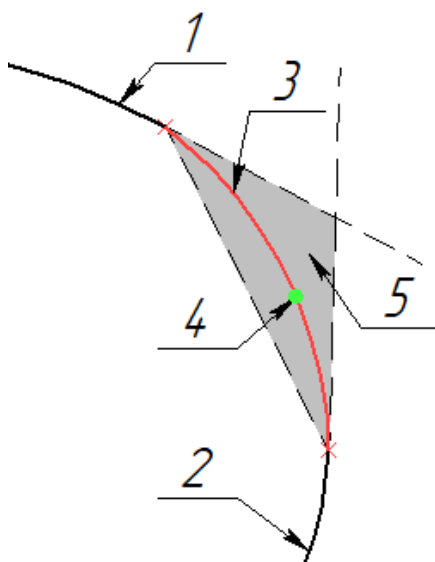


1 – first face; 2 – second face; 3 – the surface obtained (surface profile); 4 – tangent lines intersection point.



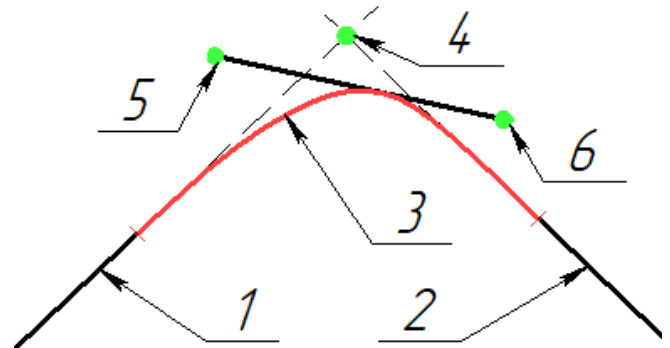
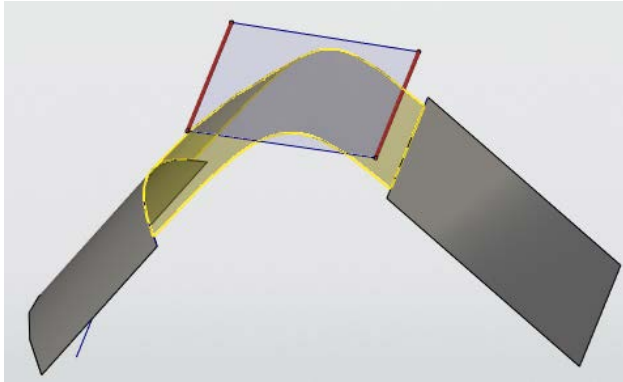
Conic – Discriminant. It creates a conical surface that has the form of a smooth rounding between two curves that lie in the specified surfaces. The degree of rounding of the resulting surface is determined by the **Discriminant** parameter. The parameter varies can be set from 0 to 1 (not including the extreme values of the specified range). If **Discriminant** is less than 0.5, the profiles that define the surface will be ellipse arcs. If the value is higher than 0.5, the profiles that define the surface will be hyperboles. If **Discriminant** is equal to 0.5, the profiles that define the surface will be parabola. The greater the **Discriminant** value is, the less flat surface is created.

Conic – Path. It creates a conical surface that has the form of a smooth rounding between two curves that lie in the specified surfaces. The shape of the surface rounding is determined by an intermediate path. The position of each point of the path is limited by the area of the triangle on the invisible plane of the forming profile.



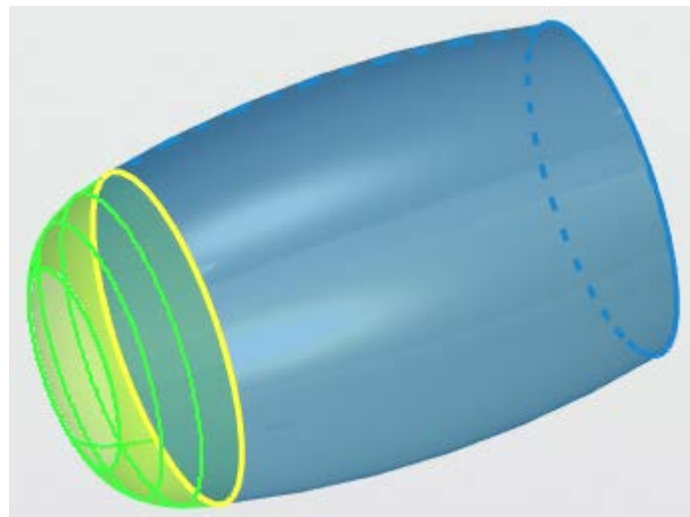
- 1 – first face;
- 2 – second face;
- 3 – the surface obtained (surface generating profile);
- 4 – point in the middle path;
- 5 – available region for the point in the middle path.

Conic – Tangent. It creates a conical surface that has the form of a smooth rounding between two curves that lie in the specified surfaces. The shape of the surface rounding is determined by another surface (forming surface) defined by two curves. The surface to be created must be tangent to this surface. The forming surface is invisible to the user if the selected curves do not belong to the already created surface. Each cross-section of the created surface must be tangent to the forming surface. Otherwise, the created surface will intersect the forming surface without a tangent condition.

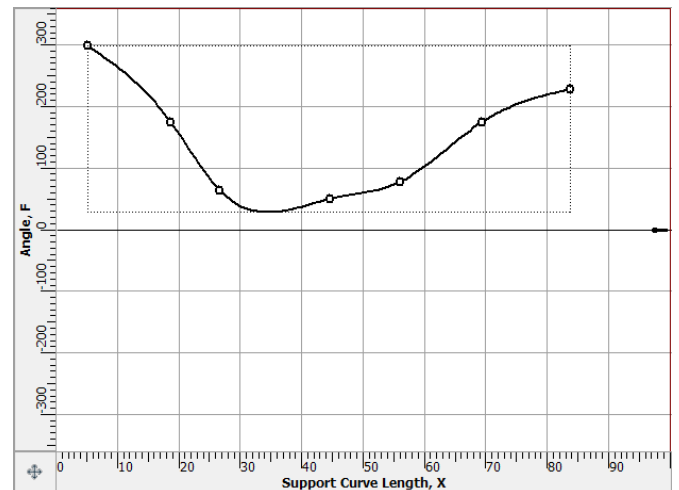
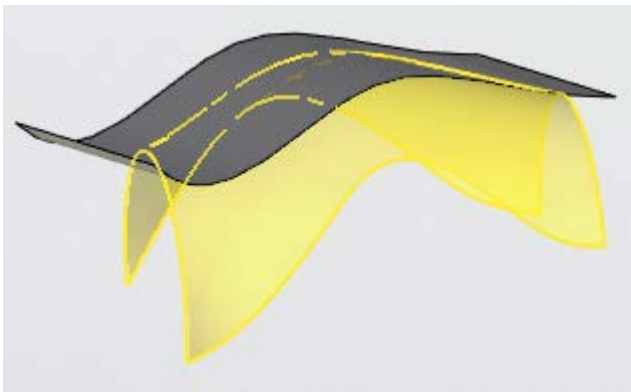
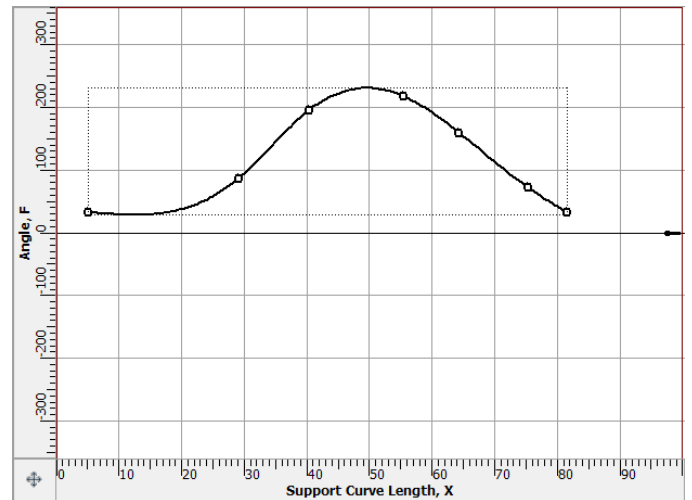
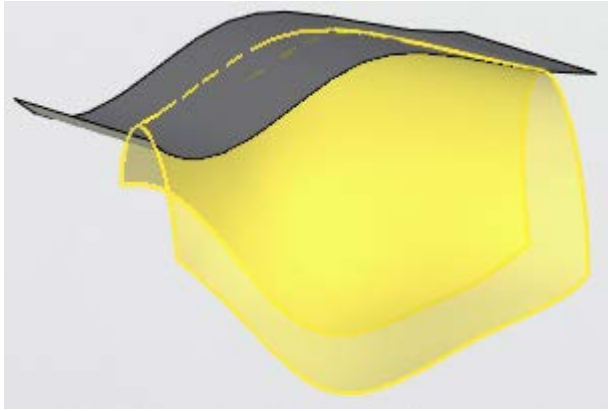


1 – first face; 2 – second face; 3 – the surface obtained; 4 – tangent lines intersection point; 5 – first reference curve point; 6 – second reference curve point.

Conic - 4 Points. It creates a conical section surface that smoothly touches a given surface along the specified guide curve (First Guide), while the shape of the surface is determined by three other guide curves.



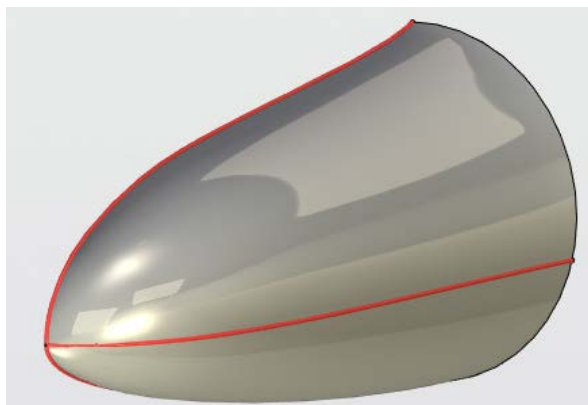
Circle – Tangent to Surface. Creates a surface that is tangent to the specified surface. The cross-section of the surface has the shape of a circular arc with a specified angle relative to the point of contact, the center of the arc lies on the specified curve. The arc angle can be constant along the entire length of the surface or variable. The variable angle is set using the graph.



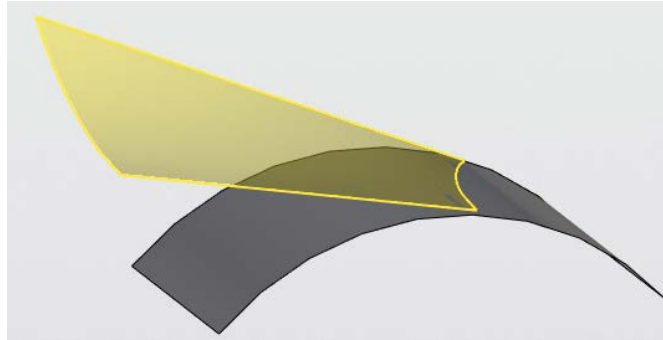
Circle – Radius. Creates a closed surface, which forming profiles are circles of a given radius. The radius can be constant or variable. The position of the forming profiles is determined by the reference curve. The center of the circles is determined by the specified curve.

Circle – Radius and Angle. Creates a surface that passes through the curve and is tangent to the surface that the curve belongs to. The cross-section of the surface has the shape of a circular arc with a specified radius and angle. The radius and angle can be constant along the entire length of the surface, or they can be set by a graph.

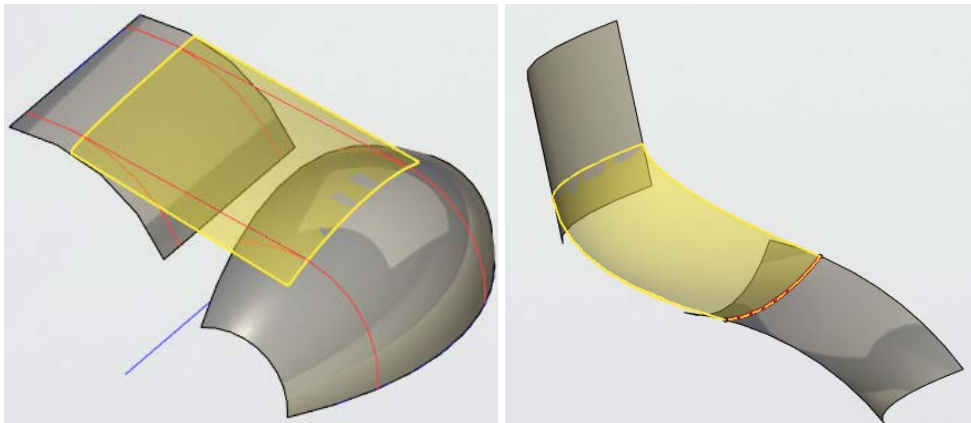
Circle – Three Points. Creates a surface that passes through three specified curves. The cross-section of the surface has the shape of a circular arc. The arc is created based on three points of intersection of the invisible profile plane with three specified curves. Curves can meet at a single point.



Line – Angle. Creates a surface whose forming profiles are segments. Each segment will be positioned at the specified angle to the selected surface. The angle can be constant or variable. At zero angle, the surface is created tangentially. The surface is bounded on one side by the initial guide, on the other by an intersection (or tangency) with the plane.

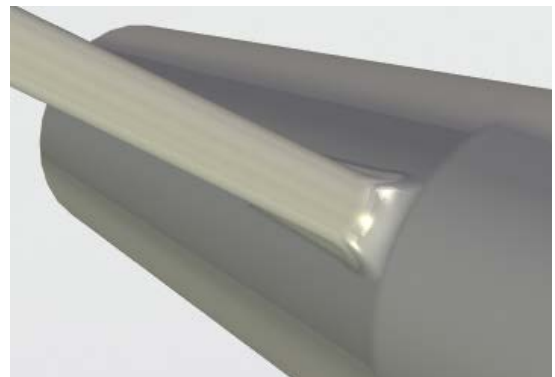
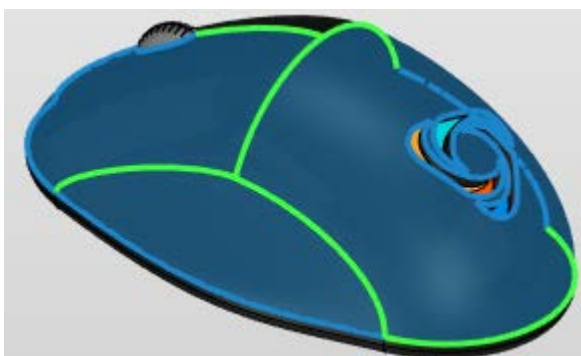


Line – Tangent. Creates a surface whose forming profiles are segments. Each segment will lie in a plane perpendicular to the reference curve. A segment will be tangent to two curves formed at the intersection of the plane, in which the forming segment lies, and the two surfaces specified by the user.



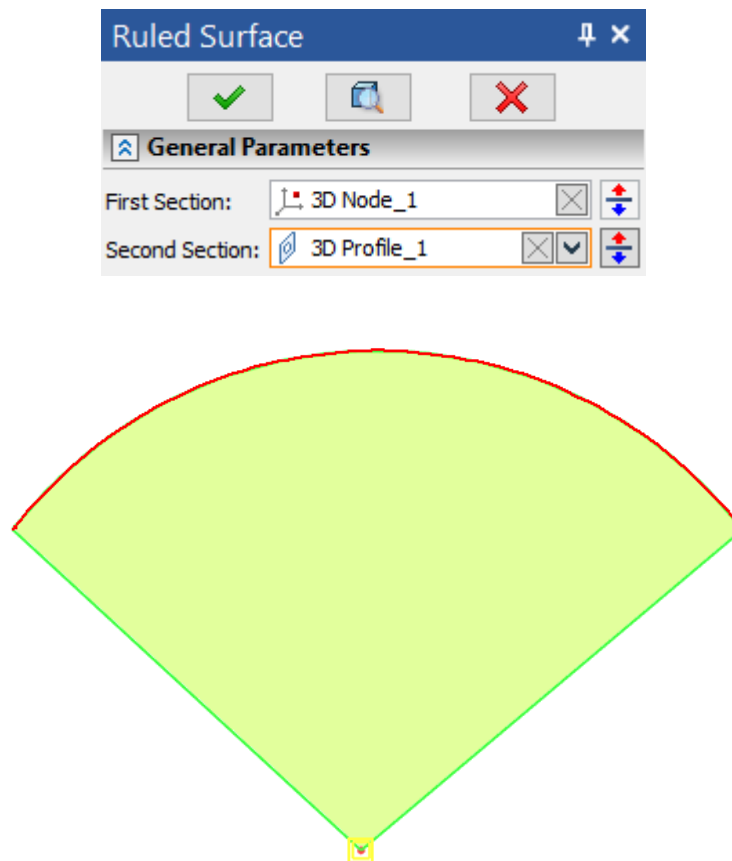
You can build paths on the created surfaces using the **Route** command. It is useful if you need to create a new guide to create the next transition surface.

The new **Transition Surface** command, along with the **Bridge** command, enables you to create complex surface models in different areas of the industry: from aviation to household appliances.



Ruled Surface

A new **Ruled Surface** command added. The command allows you to create surfaces using two selected sections. The forming line of the created surfaces is a straight line. Edges/3D Profiles/3D Paths, as well as any geometric points, can be selected as sections.

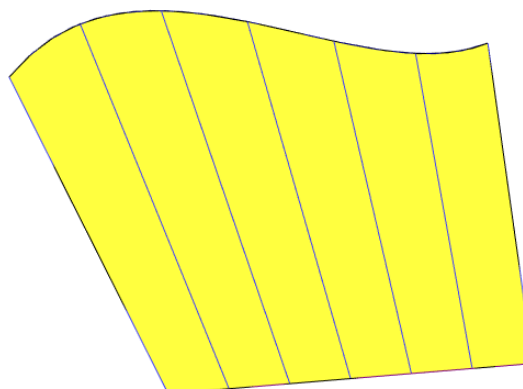


There is an opportunity to change the direction of the entire section. When you select the second section, the first one will automatically reverse.

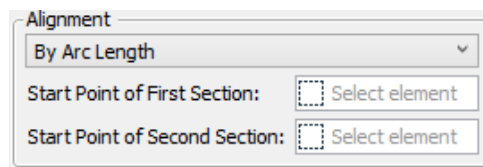
Sections can be smooth or with jogs.

There are two types of alignment - **By arc length** and **By support curve**.

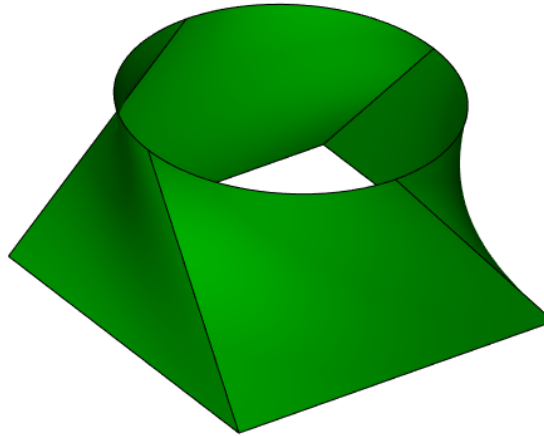
The essence of the **By Arc Length** type is the distribution of isoparametric curves, i.e. the intervals between adjacent isoparametric curves are equally related to the total length.



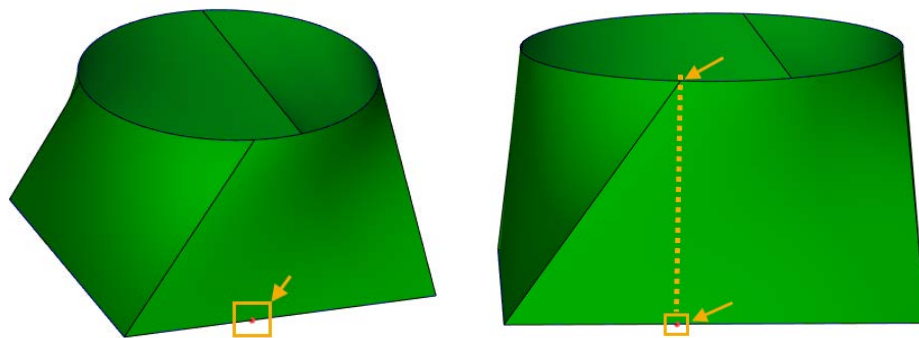
There is possibility to create surfaces between closed contours, provided that both contours are closed. In this case, fields will appear that allow you to set the start of the first and second sections.



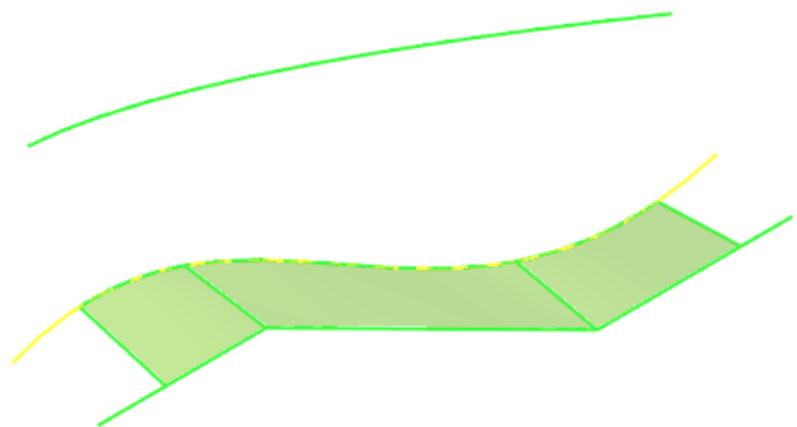
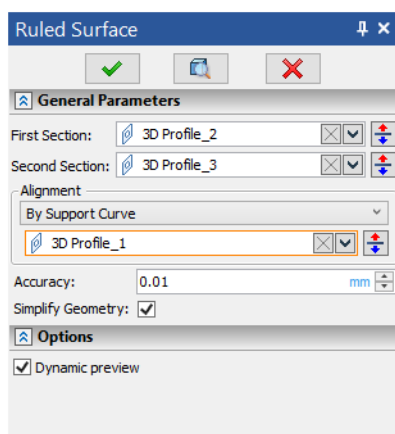
It is necessary to choose points as the start of sections. Below is a model without indicating the start of the section, i.e. the start of the second and first section is selected automatically.



If you select the 3D node located on the first section as the start of the second section, the result will look like this:



The essence of the type **By Support Curve** is that in each cross section to the support curve, a straight line will be displayed on the resulting surface. You can select a 3D Profile/3D Path/Set of edges as a support curve.



There is also possibility to specify closed sections with jogs.

Extend by Law

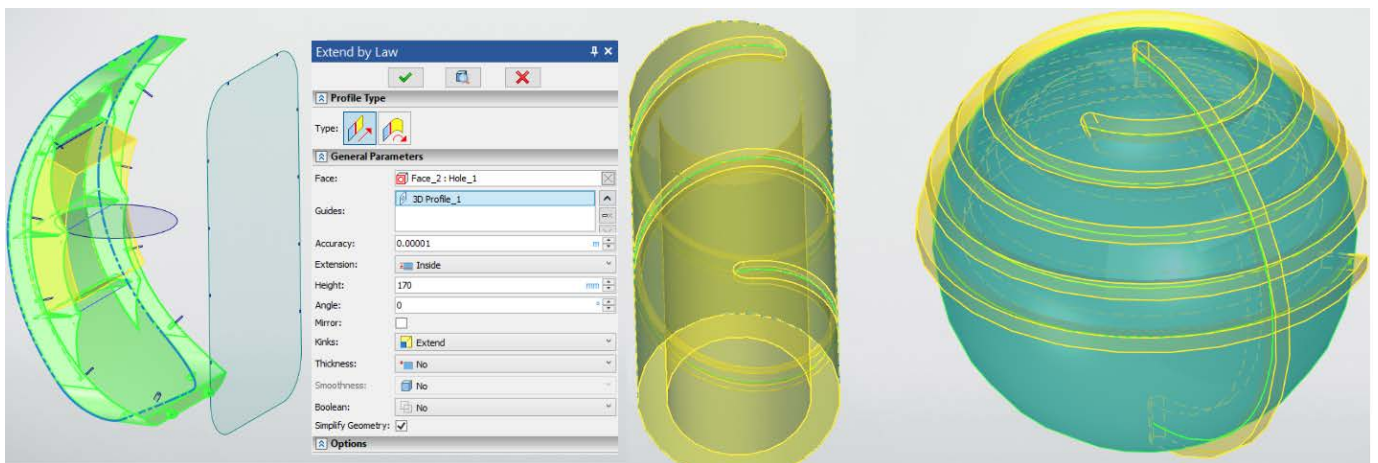
The command is designed primarily for surface modeling, but it also enables you to work with solid geometry. This command enables you to build a surface with one edge defined by a guide curve (of any complexity) that lies on a face.

This is useful when adding stiffeners to body parts that have complex spatial geometry. You do not need to perform a complex sequence of operations. It is only required to project the desired contour on the face or create a guide curve using the wire geometry commands and create the surface in one operation.

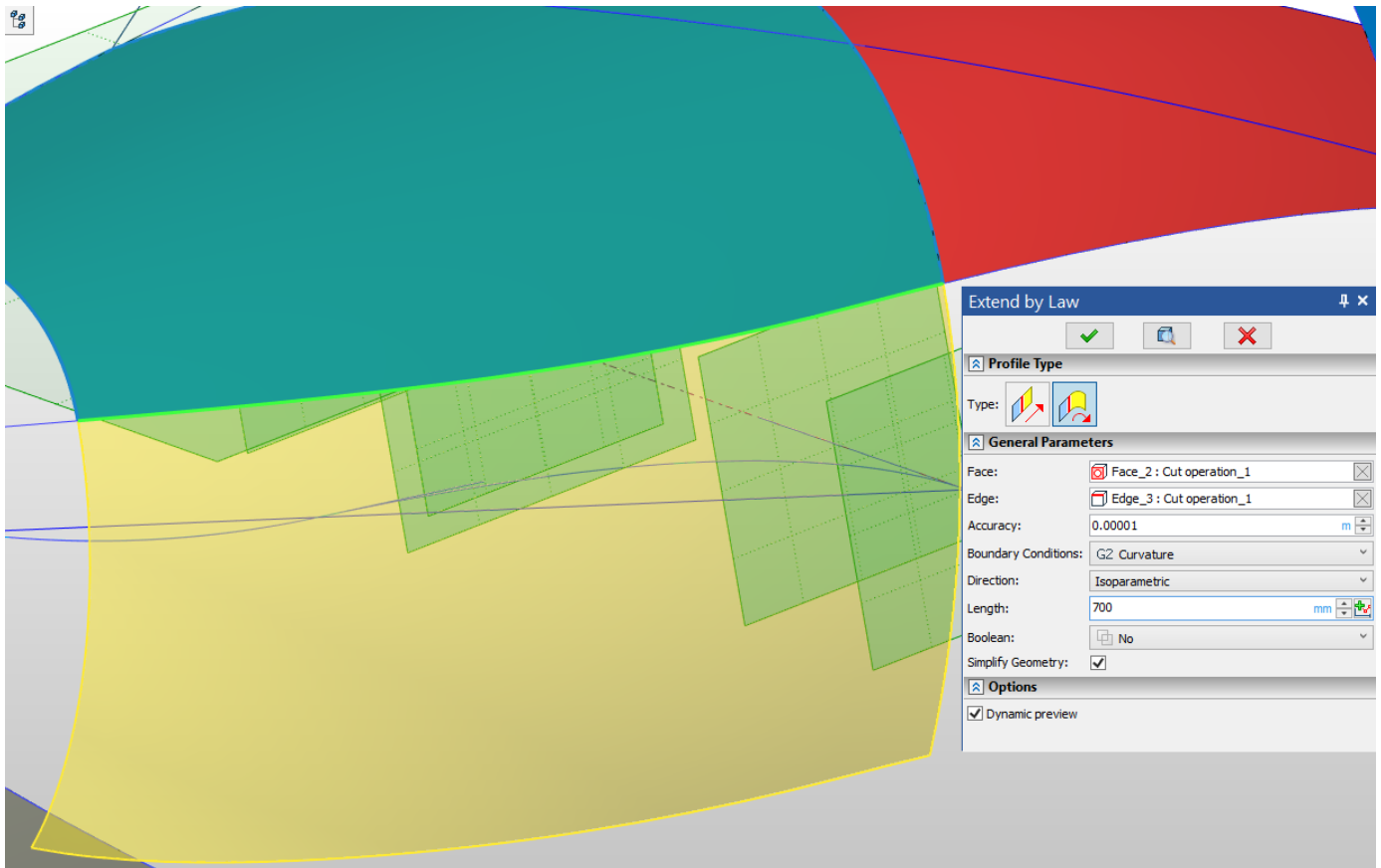
In solid modeling, the command is useful for creating grooves obtained by milling. The shape of the groove is set by the guide curve, and the width is set by the thickness parameter. If you specify the **Thickness** parameter, a solid body will be created instead of a surface. To create a cavity, specify a Boolean subtraction command in the command parameters. You can also round the edges of the groove to simulate the marking left by the milling cutter.

Two types of profile are available in the command: **Line** and **Circle Arc**.


Two types of creating an operation are shown below: when the surface is extended by a line and an arc of a circle. As a result, a surface is formed with optional possibility to change the angle of inclination to the surface normal. Also it is possible to add thickness and obtain a solid body with the options for rounding the edges, thereby reducing the number of clicks in some work scenarios and increasing the usability.

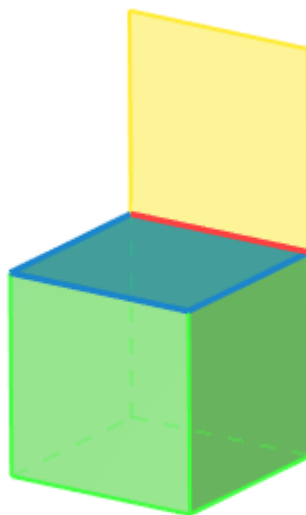


The second mode in addition includes several options. One of them creates the surface, inheriting the radius of curvature at each point of the specified edge of the face to which the continuation is created. Alternatively, user can manually specify what radius the surface should be created.

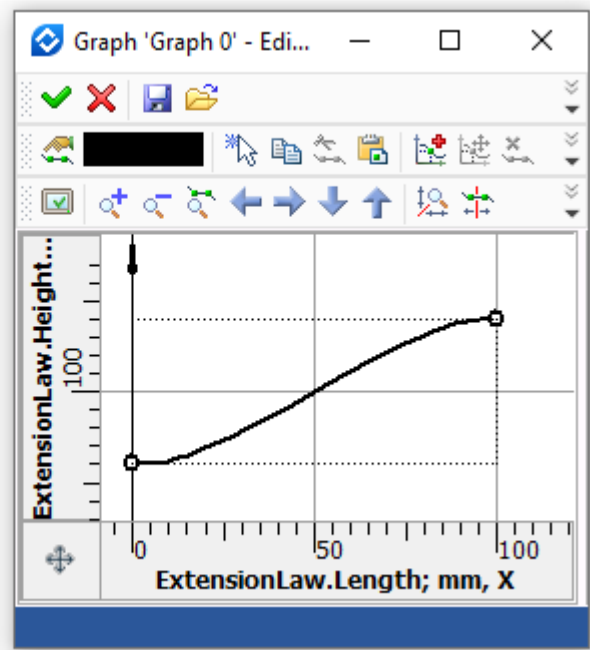
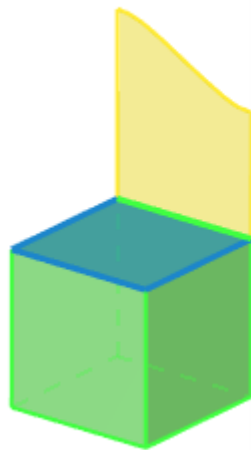


It is possible to define **Height** and **Angle** via a graph depending on a guide length.

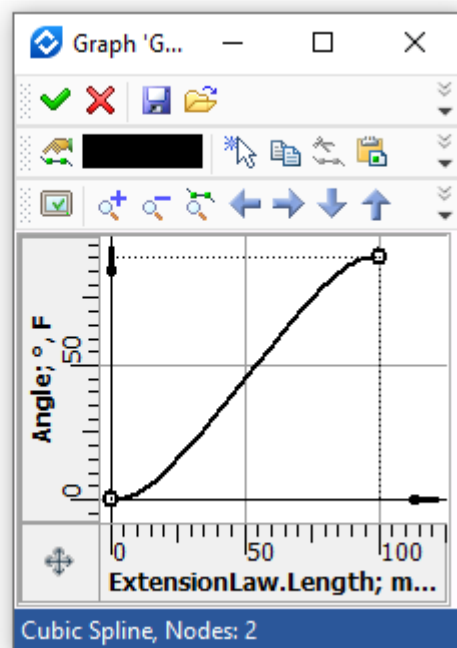
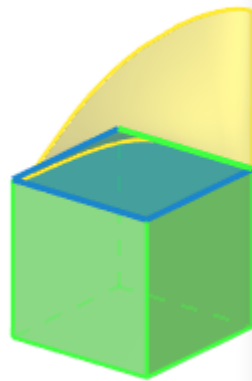
Clicking the  button invokes a graph type selection dialog. Then the graph can be edited using the T-FLEX CAD **Graph editor**.



Constant height and angle



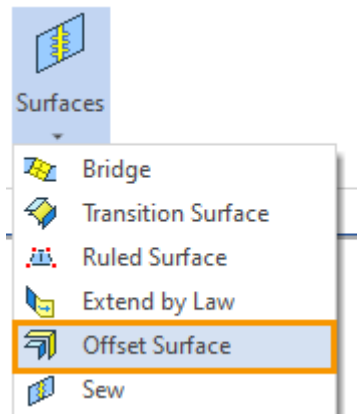
Height defined via graph



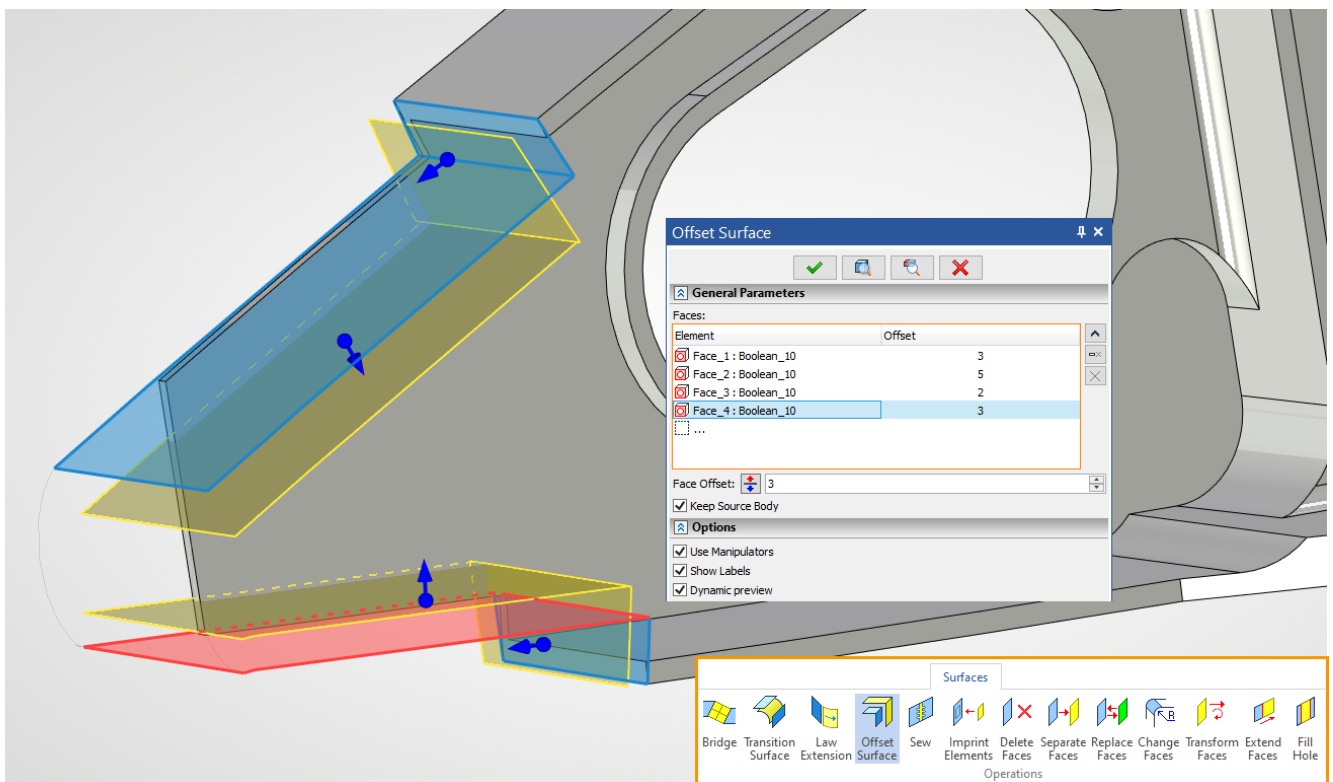
Angle defined via graph

Offset Surface

A new **Offset Surface** command added. Previously, such functionality was in the **Shell** command with some restrictions on the choice of faces of solids, now it is a separate command with more features without previous restrictions. You can call the command from the Ribbon. It is located in the **Special** group in the drop-down list of surface commands.



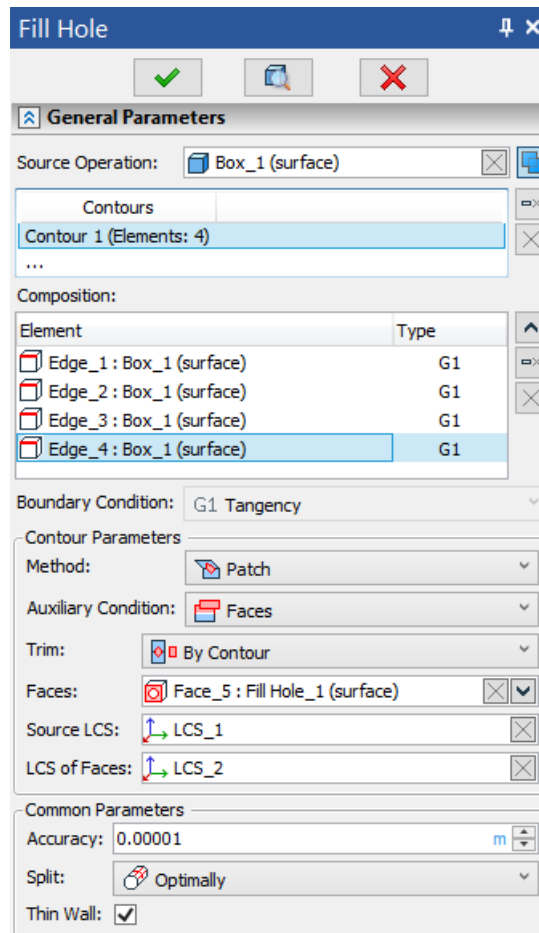
This command allows you to construct a surface consisting of a set of faces equidistant to the selected ones. The offset of each face relative to the original face may differ. You can set the offset value for each face or group of faces in the parameters or using the draggers in the 3D scene.



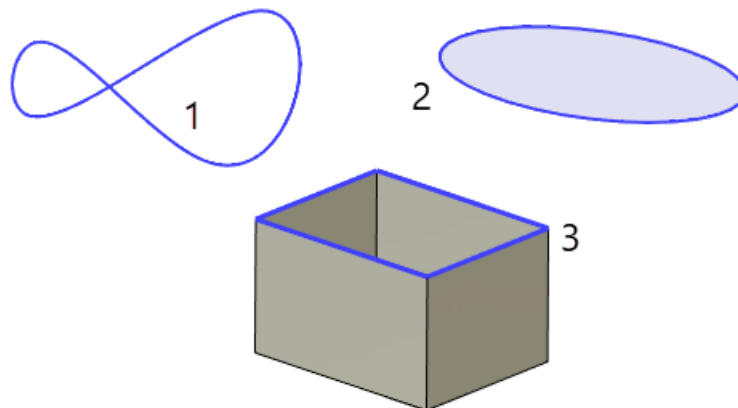
A new dialog allows you to manage the set of selected items: remove them from the list, add them, and separately edit parameters for each item or group of items.

Area Filling

A large-scale reworking of the **Fill Hole** operation has been done. Now it has become even more convenient to work with this operation. The interface has been updated and transferred to a new type of dialogs:

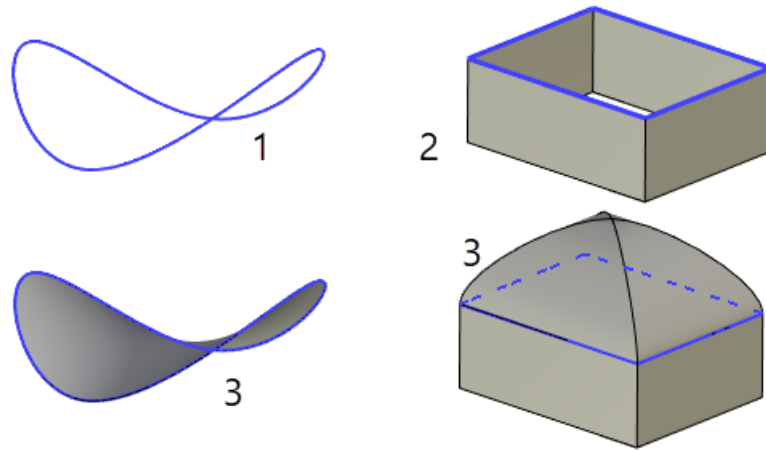


Now several 3D paths (closed), several 3D profiles (closed) or a set of edges (closed) can be specified as a contour. It is possible to simultaneously select 3D paths/3D profiles/edges as one contour.



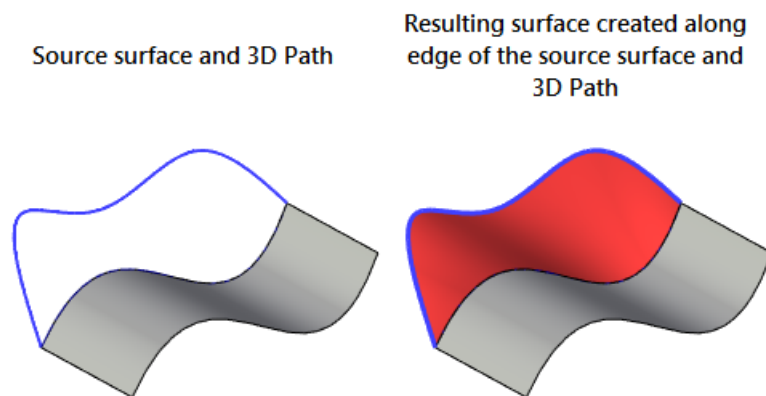
1. Closed 3D Path
2. Closed 3D Profile
3. Closed set of edges

So, for example, you can select a closed 3D path and fill it like in the picture below.

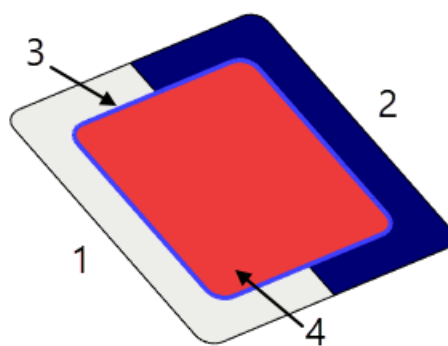


1. Closed 3D Path
2. Closed set of edges
3. Resulting surface

Added the ability to combine surface edges with 3D path or 3D profile edges.



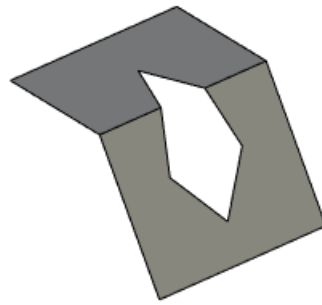
Added the ability to select edges from two different surfaces as one contour.



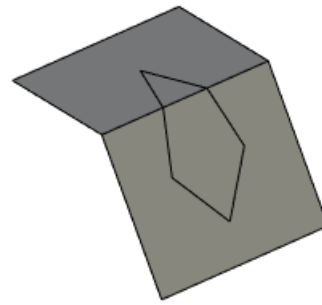
1. First surface
2. Second surface
3. Contour, which includes the edges of different surfaces
4. Resulting surface

There are now two methods for filling the area - **Patch** and **Extend**. The Patch allows you to fill an area in the classic way, leaving a "trace" of filling the area in the form of edges.

Source surface

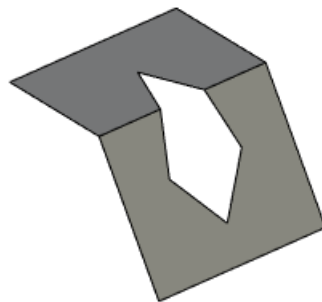


Filling hole with Patch method

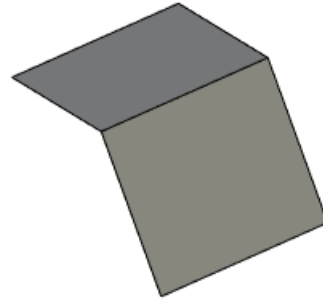


Extend is a special case and allows you to extend the surface.

Source surface



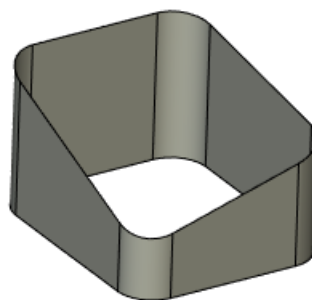
Filling hole with Extend method



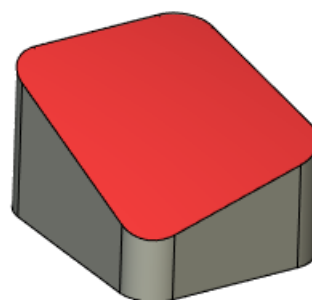
After choosing this method, the choice of additional options becomes unavailable.

Added a new variant of the priority fill surface shape - **Non-smooth**, with which the hole is filled, ignoring all smoothness requirements, using an analytical solution where possible.

Source surface



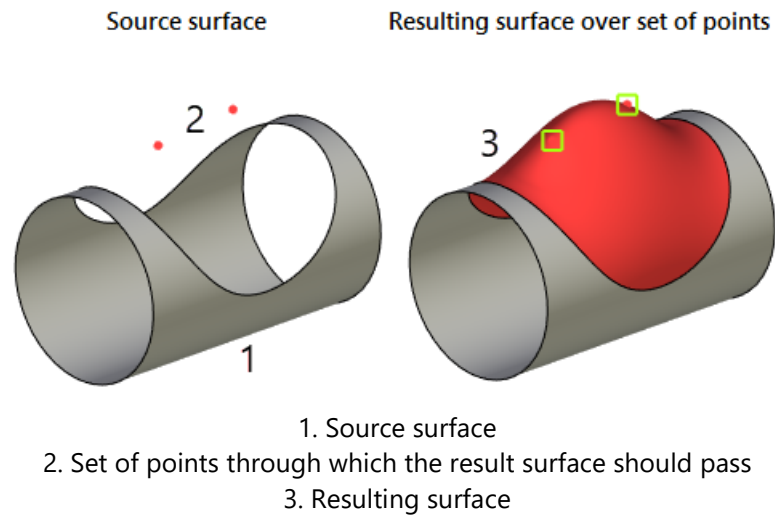
"Non-Smooth" shape of created surface



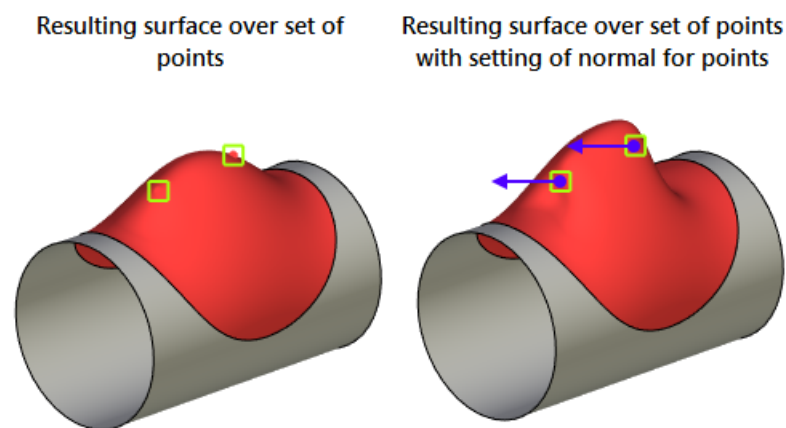
The generated surface shape **Preferable plane** have been renamed to **Plane**.

Added a new parameter **Auxiliary Condition**, which allows you to set the shape of the created surface. There are 6 additional conditions available - **Points**, **Vertex**, **Edges**, **Faces** and **Surface**.

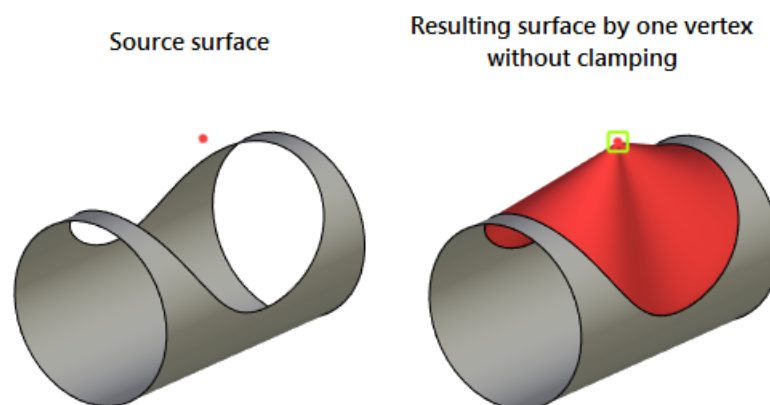
Auxiliary condition **Points** allows you to create a surface passing through a set of points, belonging or not belonging to the contour.



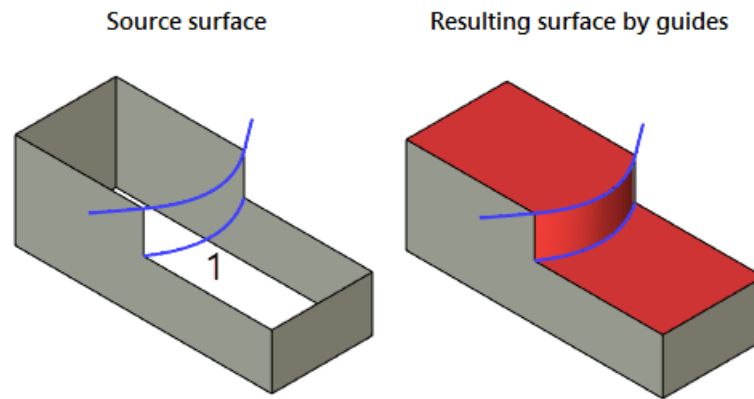
A normal can be specified for each point. After selecting the appropriate option, a menu for specifying vectors will appear, similar to the **Loft** operation. The only difference is that the length of the vector is taken from the unit length.



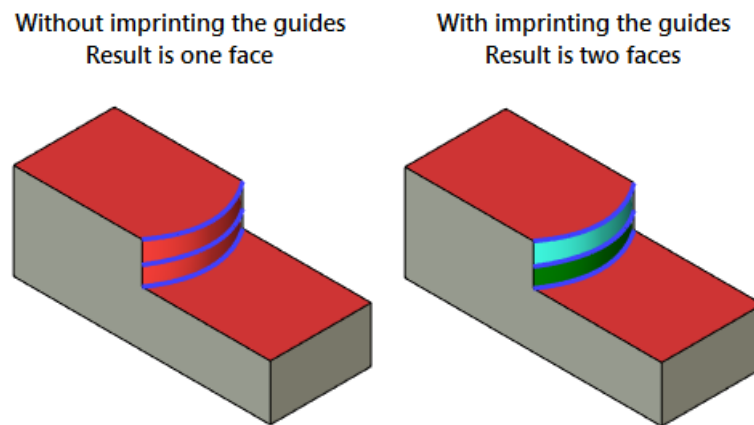
Auxiliary condition **Vertex** allows you to select one point, which belongs or does not belong to the contour, and create a surface relative to it. Under this condition, the type of **Clamp** can be selected. Clamp type accepts 3 modes - **No**, **Auto**, **Normal**. When the **No** mode is selected, the surface is created without additional smoothness conditions.



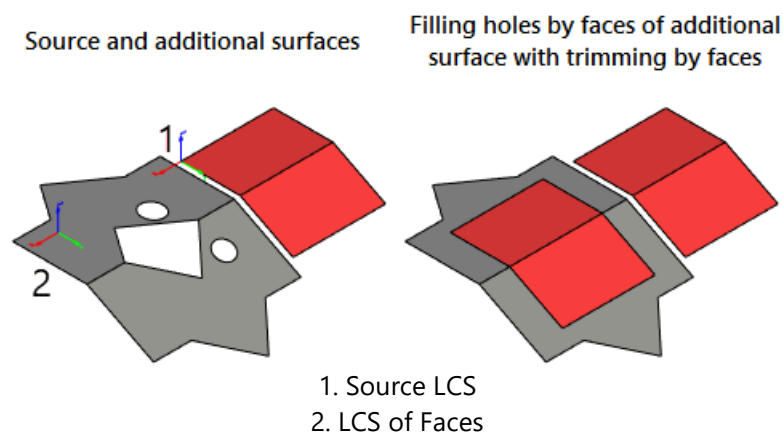
If the **Normal** mode is selected, the system works in the same way as the auxiliary condition **Points** using the normal. **Auto** mode allows the system to independently determine the required clamp. Auxiliary condition **Edges** allows you to set guides for the surface being created. The principle of operation is similar to the guides in the **Loft** operation. If the selected guides are longer than the boundaries of the contour, then the contour of the guide that is inside the contour will be used when creating the surface.



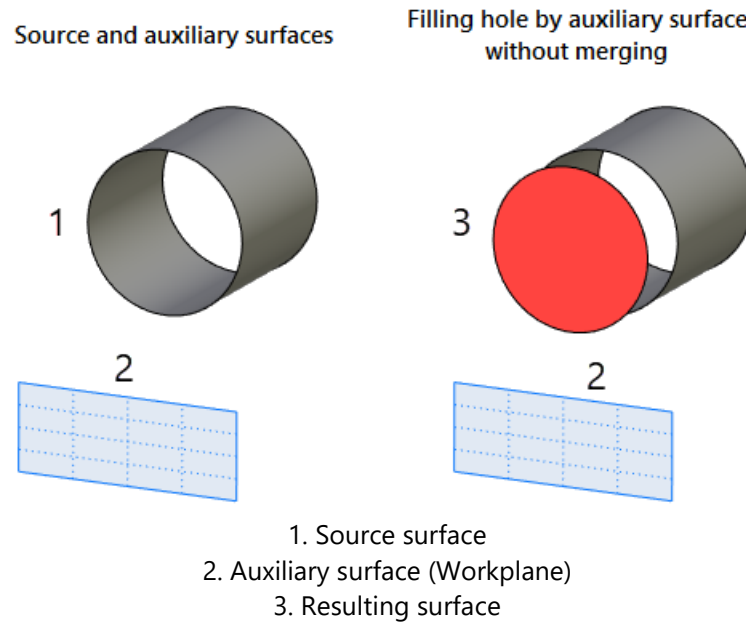
If necessary, guides can be imprinted on a face, that is, split a face along a guide.



Fill by sheet body has been renamed to an auxiliary condition **Faces** and updated. If the face is outside the area to be filled, then you can specify the initial **Source LCS** and the **LCS of Faces**, with the help of which the system will be able to correctly determine where the area is filled.



Fill by geometric surface has been renamed to an auxiliary condition **Surface** and updated.



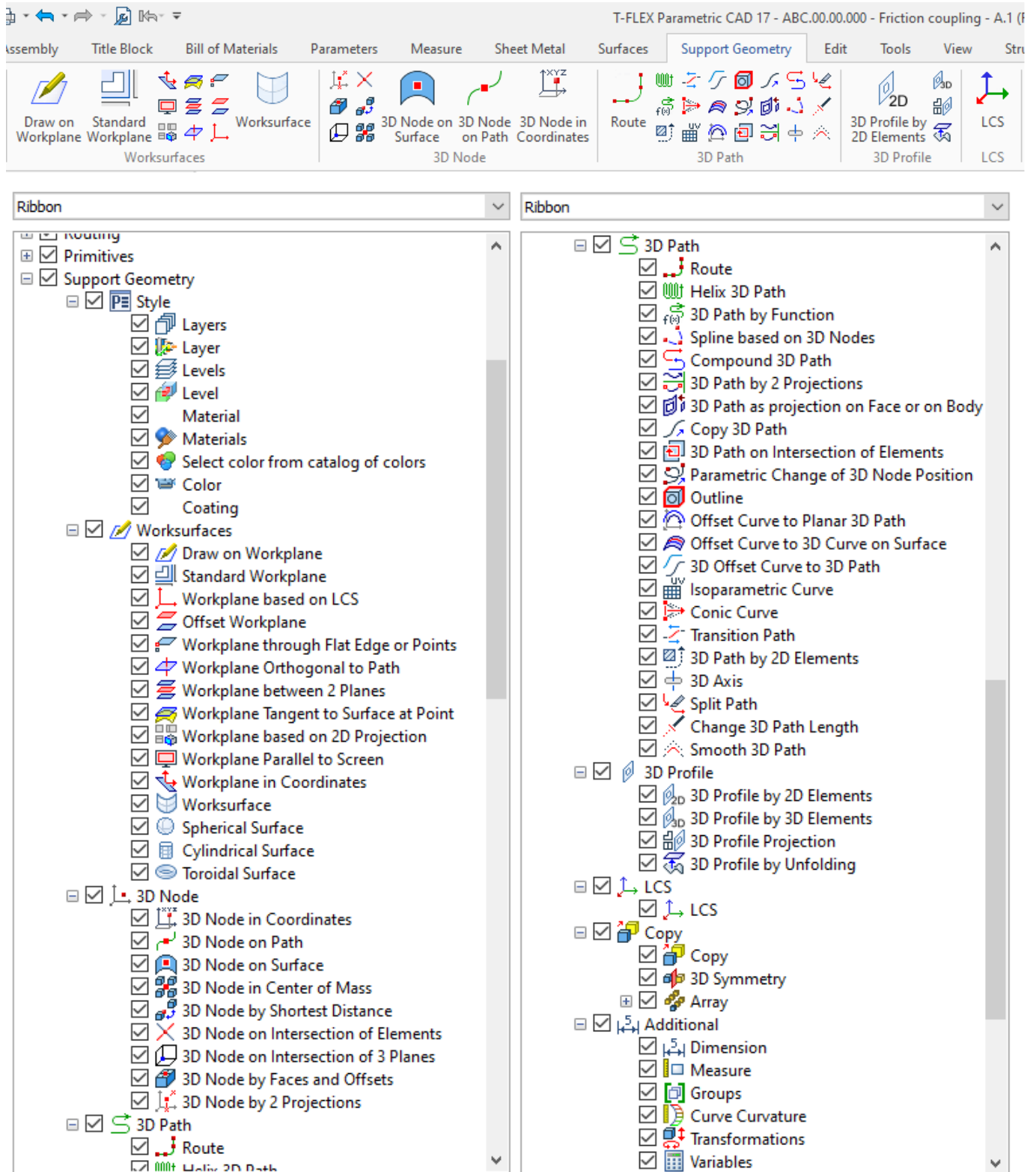
For this additional condition, you can specify an **auxiliary point** to refine the closest face to this point. This can be useful for toroidal and spherical surfaces.

The **Minimize Topology** option has been reworked and renamed to **Split**. Added new types of face splitting similar to the **Loft** operation.

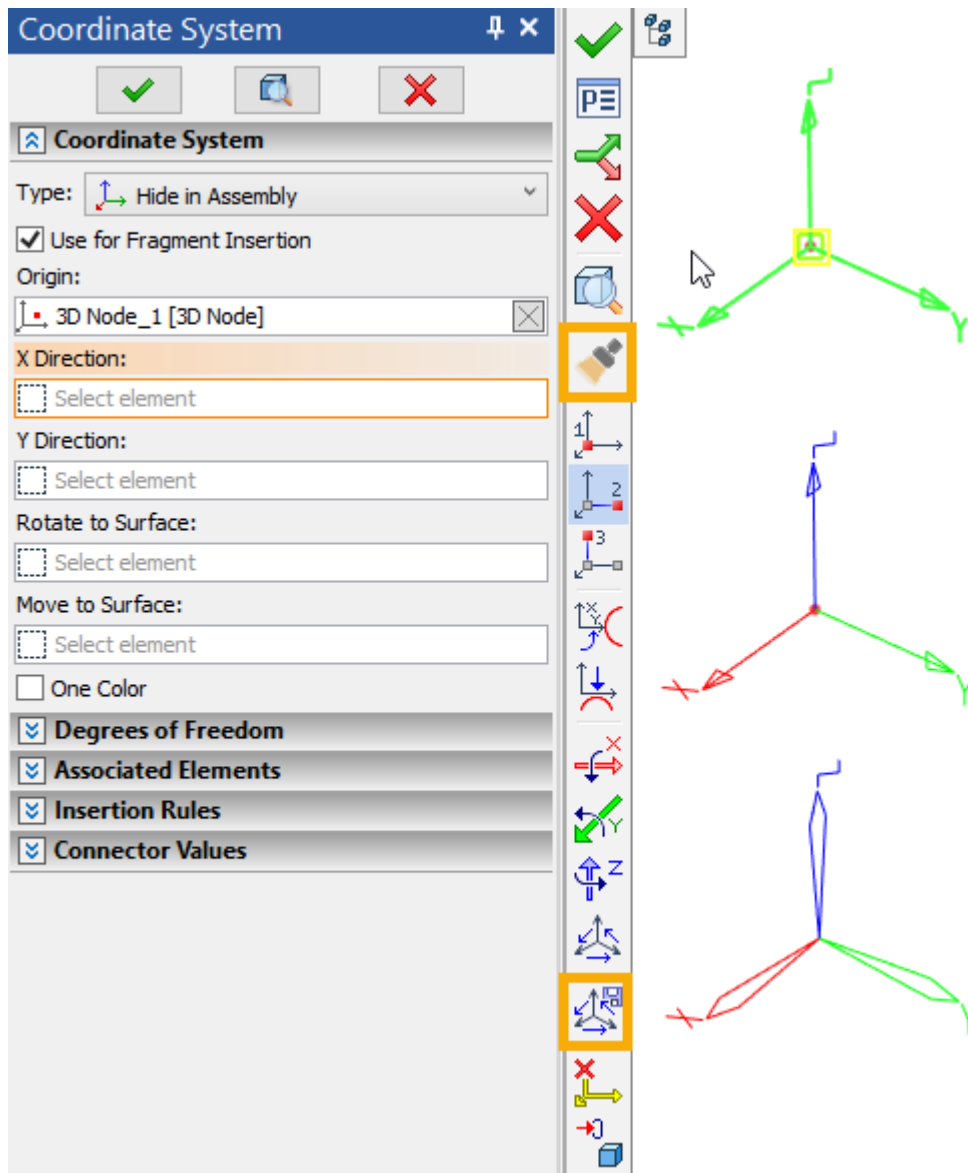
3D Constructions of Support Geometry

For surface modeling, it is important to have convenient tools for working with support geometry, so **3D Path**, **3D Profile**, **3D Node** and **Workplane** commands have been significantly improved. There are new ways to create support geometry elements. Command dialogs are more convenient.

Various **Worksurface**, **3D Node**, **3D Path** and **3D Profile** types can now be called directly from the new **Support Geometry** ribbon tab:



LCS



New options have been added to command's automenu:







<C> Format by Pattern

Select an existing LCS, upon activating this option. All parameters of the selected LCS, except connector values and origin position, will be applied to the current LCS.



<K> Save given orientation

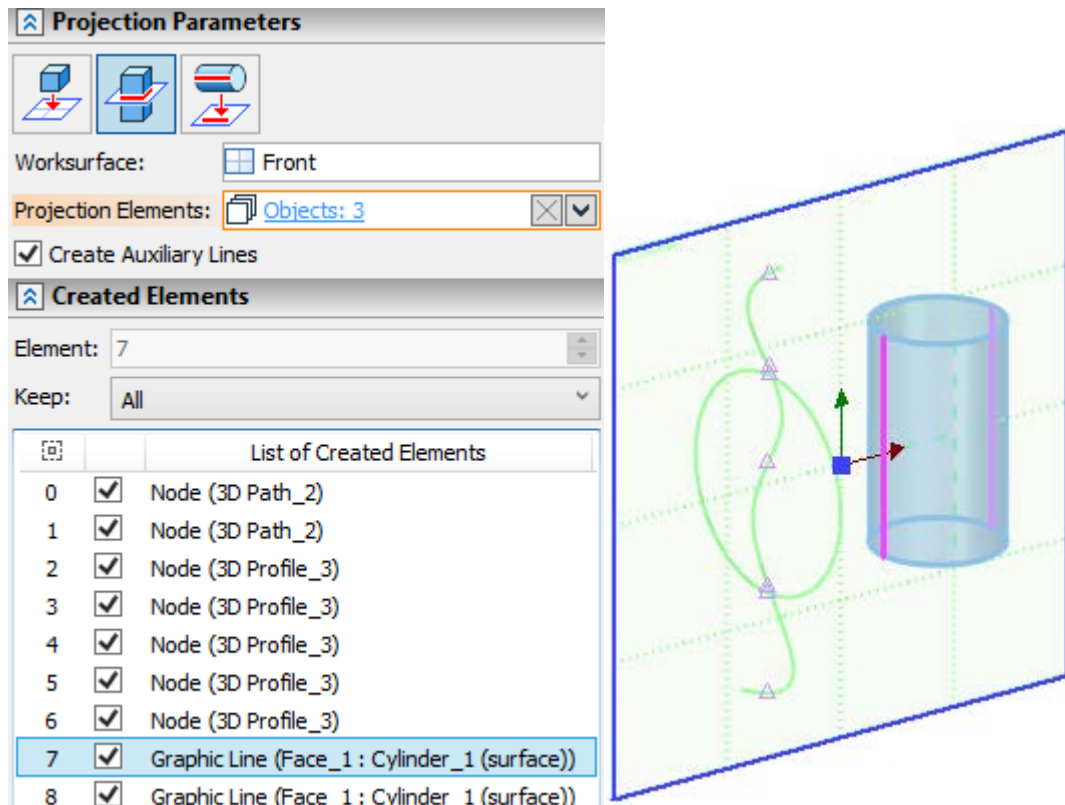
Upon activating this option, all rotations applied to the current LCS via , , , and  options will be automatically applied to the next LCS created after the current one.

Keep in mind, that this option doesn't affect rotations resulting from aligning axes to selected elements.

Intersections with Workplane

The **Intersections with Workplane** mode of the **Project Elements** command is improved.

Now the command detects all possible intersections in case of multiple intersections with the same element. You can select, which of the possible intersections should be constructed, using the new **Created Elements** tab in the bottom part of the command's parameters window.



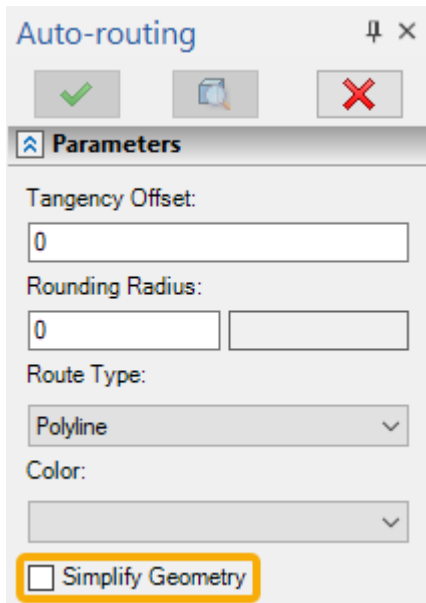
The tab contains the **List of Created Elements**, which shows the number, checkbox, type of created element (node, graphic line) and name of parent element for each possible intersection. Rows of the list can be selected using . The selected row is highlighted in the list and the corresponding element is highlighted in the 3D scene.


The **Keep** drop-down list contains following options:

- **Current**
Only the element corresponding to the selected row of the list will be created.
- **All** (default)
All elements will be created.
- **Selected**
Only elements, whose checkboxes are enabled in the list, will be created.
- **All Excluding Selected**
Only elements, whose checkboxes are disabled in the list, will be created.

The **Element** input box contains the number of the currently selected row of the list.

Route



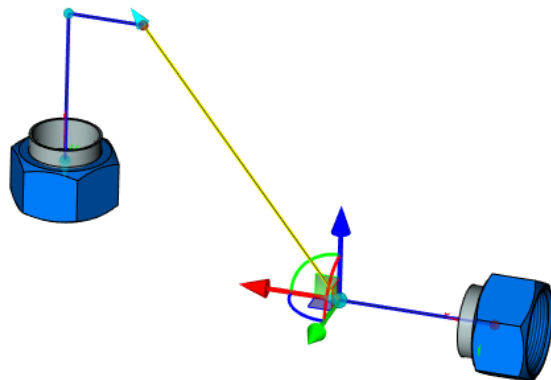
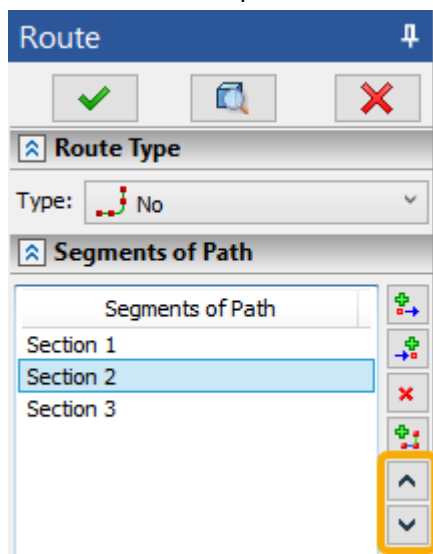
The **Simplify Geometry** option is added to the  **Auto-routing** command.

Simplifying merges segments of route if all of the following conditions are met:

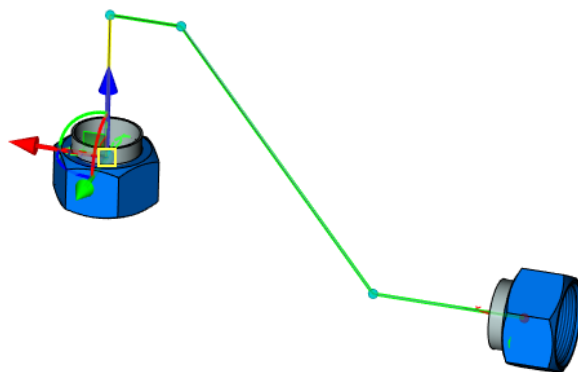
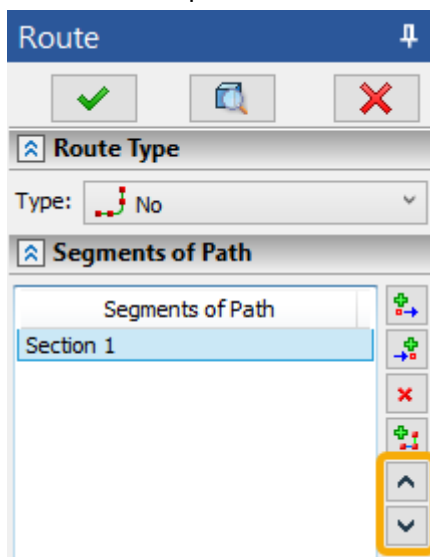
- **Tangency Offset** is equal to zero;
- **Rounding Radii** of segments are equal to each other;
- **Route Type** of segments is the same.


Let's create two similar routes - first without simplification, second with simplification.

The route created without simplification consists of three segments.



The route created with simplification has the same shape, but consists of the single segment only.

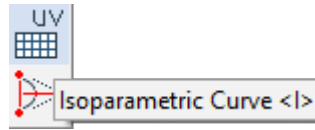


Figures above also show that the  **Route** command now has buttons for moving the selected segment up/down the list

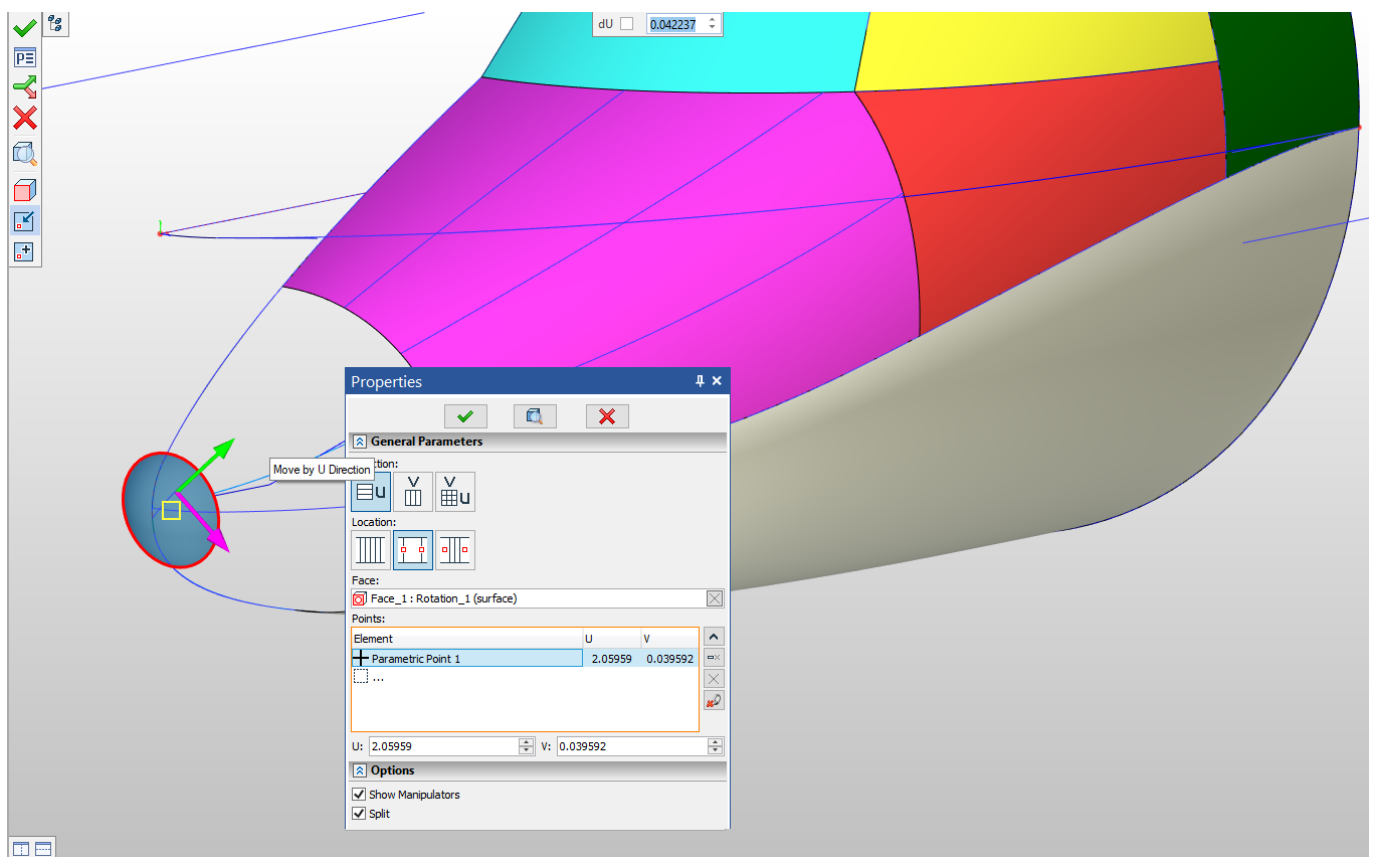
3D Path

Isoparametric Curve

A new option for creating 3D paths has been added to the command: **Isoparametric Curve**.



You can create a grid consisting of separate 3D paths on the selected surface. The grid can be created in one or two directions. You can specify a step for the grid. You can set the step for creating isoparametric paths based on values entered by the user or based on 3D points. 3D points can be specially created by the user or set based on an existing body. You can use isoparametric curves as the initial and final segments of transition paths, along which you can create a surface.








Conic Curve

Conic Curve is the new mode for building a 3D path. It is essentially a separate command that enables you to create the spatial curves of a conic section: hyperbola, parabola, arc of an ellipse. The mode is directly related to the new **Transition Surface** command, where the forming surface profile is also a conic section curve. New wire and surface modeling tools expand possibilities for working with structures where aerodynamic properties are important: curves and surfaces of a conic section are widely used in aviation as geometric objects with optimal curvature.

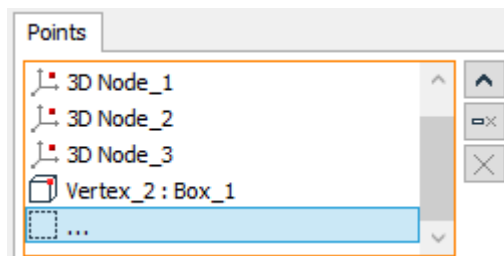
The new **Conic Curve** 3D path option was added. It has six options for creating a conic curve. You can select an option by clicking the corresponding dialog icon.



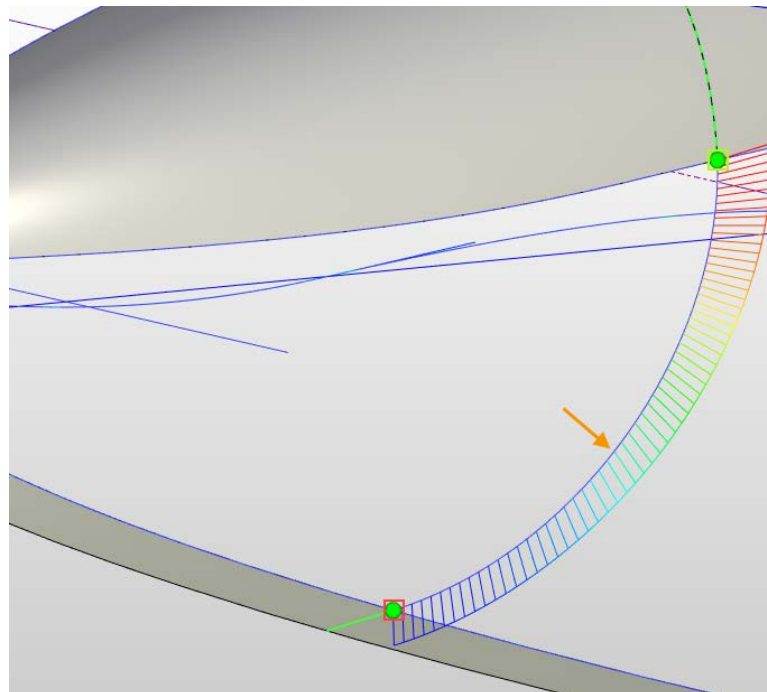
By 5 Points

	By 2 Points, 2 Guides and Discriminant
	By 3 Points and 2 Guides
	By 2 Points, Vertex and Discriminant
	By 3 Points and Vertex
	By 4 Points and Guide

The command dialog enables you to edit the selected construction elements.




The resulting curves will represent 3D paths for which all possible wire geometry operations will be available, and which can be used as construction elements in solid-state and surface modeling operations.

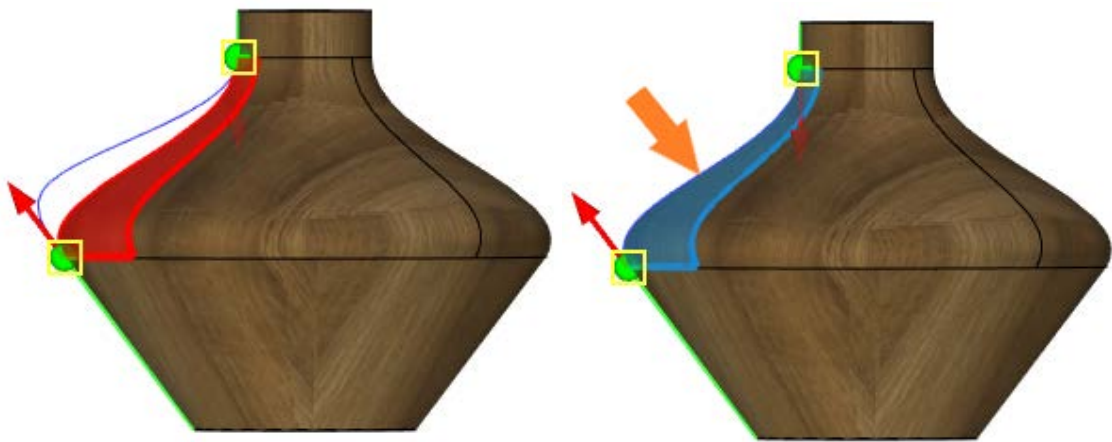


Transition Path

The functionality of the option for creating transition paths has been expanded. Tangent settings are available for the transition path in the **Spline** mode: for each endpoint, the smoothness types G0, G1, G2, and G3 are available. The length of the tangent vector is determined by the **Tangent magnitude** parameter.

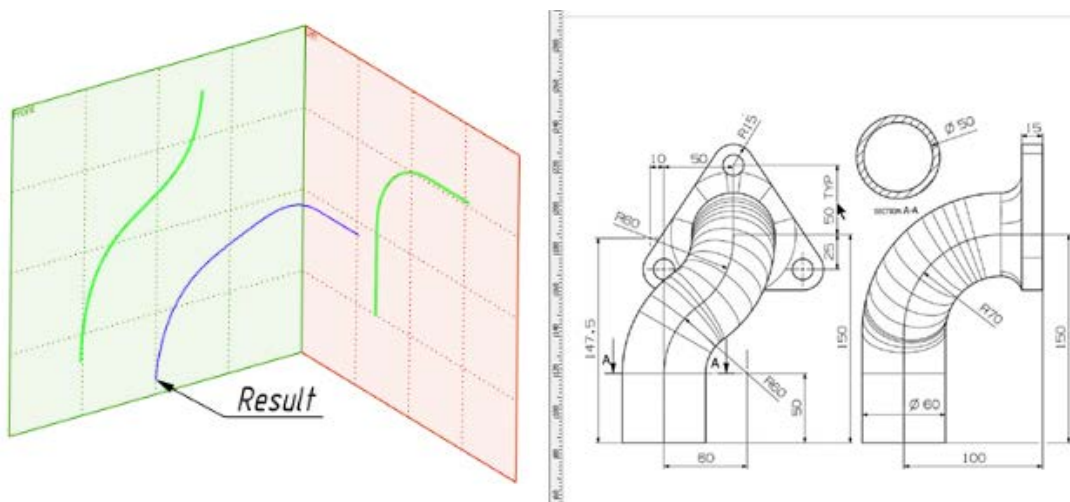


The transition path can be «put» on the selected surface using the **Select Supporting Geometry**  option.



3D Path by Two Projections

The **Create 3D Path using two projections** option features were extended. Now the condition for coincidence of start and end points of the projections is not mandatory. When working from 2D to 3D, you can select image lines.






Compound 3D Path

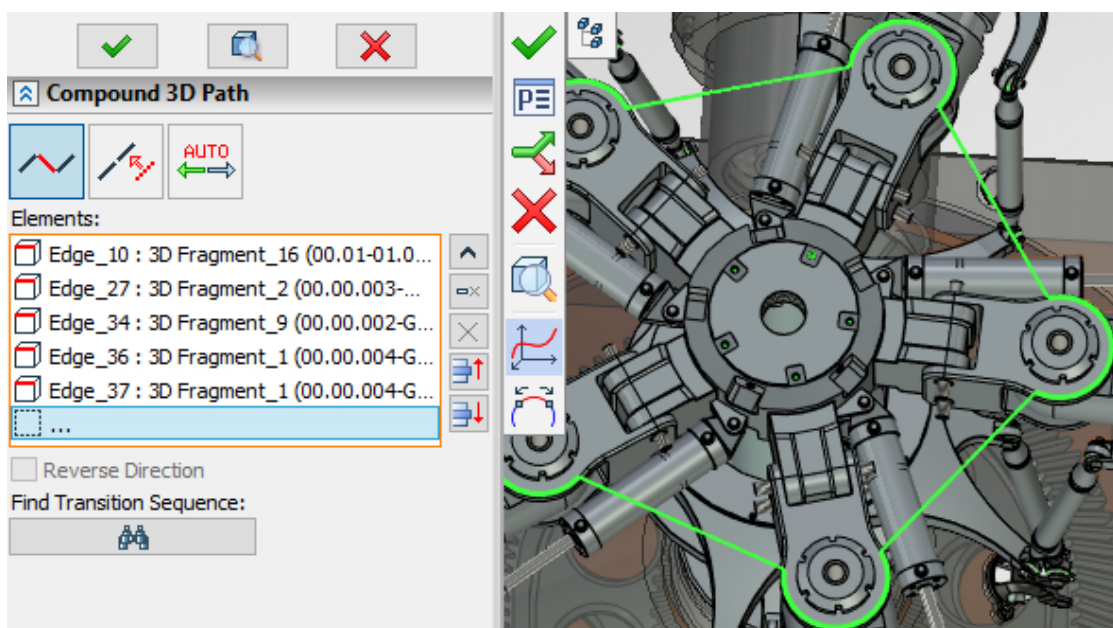
The mechanism for creating paths on multiple 3D paths and multiple edges was improved and optimized. Now you can create such paths in the same **Compound 3D Path** mode. A smooth sequence of paths and edges can be selected automatically when one of the sequence elements is selected. When you select

unrelated elements, there are several ways to join them: **Auto Joining Mode**, **Move**. When you select **Auto Joining Mode**, the elements are automatically joined by line segments. When you select the **Move** mode, the start point of the next section is moved to the endpoint of the previous one.

The **Auto** mode is also available. In this mode, only joined elements can be selected, and the orientation of elements that are included in the path is determined automatically, as it worked before.

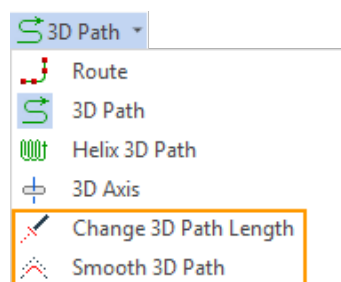
	Auto Joining Mode
	Move
	Auto

All the selected elements must be single linked, i.e. you cannot select different ways to link elements when creating a single path.



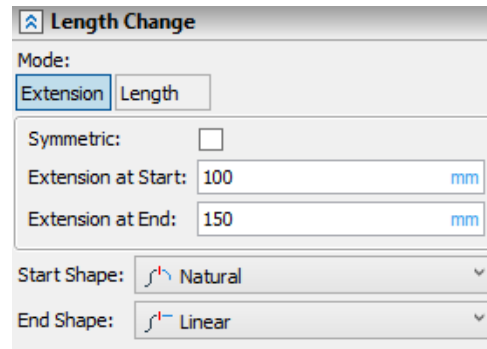
Modification


The path modification command can be called from the Ribbon in the **3D Path** list: **Change 3D Path Length** and **Smooth 3D Path**.

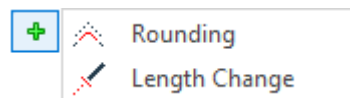



You can also call the **Modify** command from the path context menu.

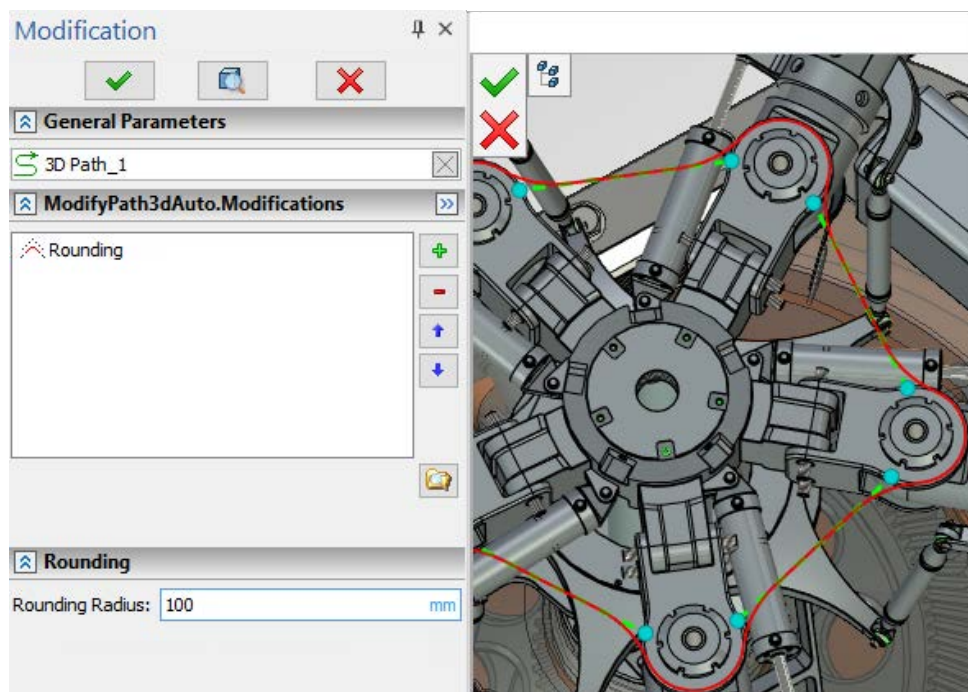
Smooth 3D Path allows you to round all sharp edges on the path, and **Change 3D Path Length** allows you to set the law of lengthening the path from its extreme points or change the total length of the path to a specified value. You can change the length symmetrically on both sides or specify an invariable side.



You can select only one smoothing modification and one length change modification for each path. Modification options can be added using the icon .



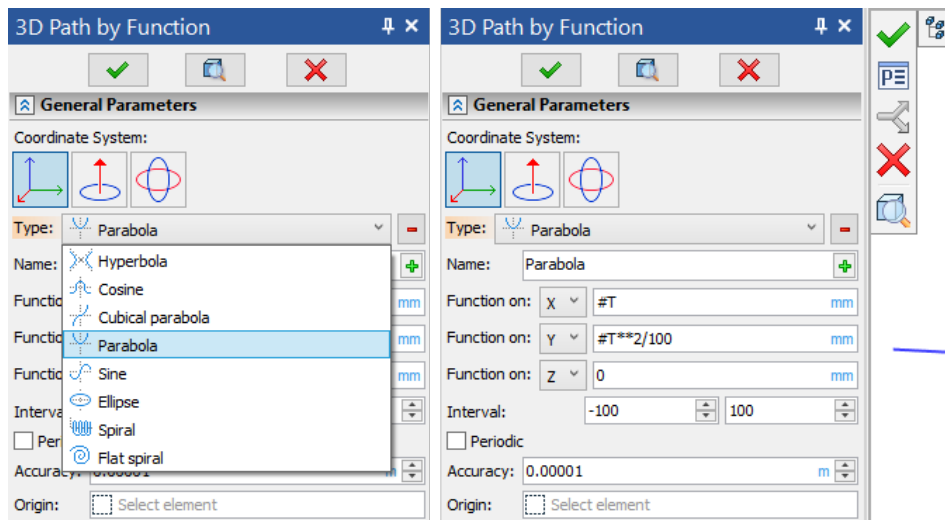
When you call the modification command again, the parameters of the previous modification will already be set for the path in the dialog. You can change the modification parameters, or you can delete one of the two modification options  or both modification options at once.



3D Path by Function

A new command **3D Path by Function** has been added, which allows you to create a 3D path by function in a given coordinate system.

There are 3 types of coordinate systems available - **Cartesian**, **Cylindrical**, **Spherical**. Each coordinate system has its own set of parameters. So, for example, for a Cartesian coordinate system it is proposed to specify formulas for **X**, **Y**, **Z**, for cylindrical with respect to **ρ** , **φ** , **Z**, and for spherical with respect to **ρ** , **θ** , **φ** . For all types of coordinate systems, it is possible to select, create and delete a type of function.

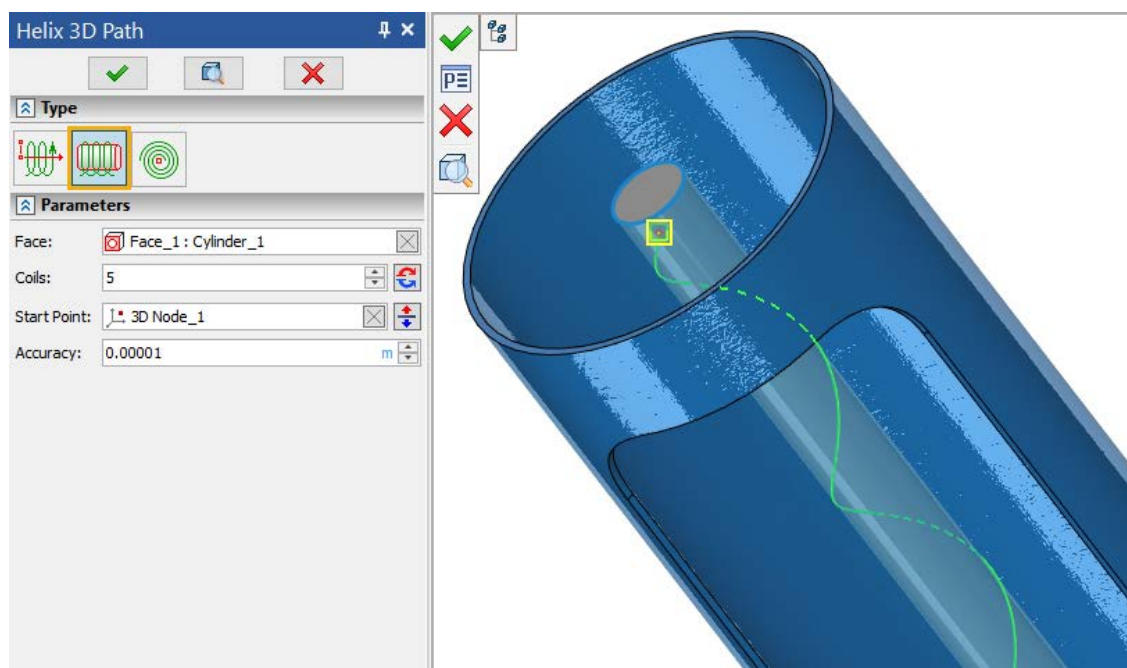


For each type of coordinate system, it is possible to set the interval, accuracy and select the origin of the 3D path. You can also close the 3D path using the **Periodic** option.

Helix 3D Path

Helix 3D Path by face

The ability to create helix 3D path by face was added.

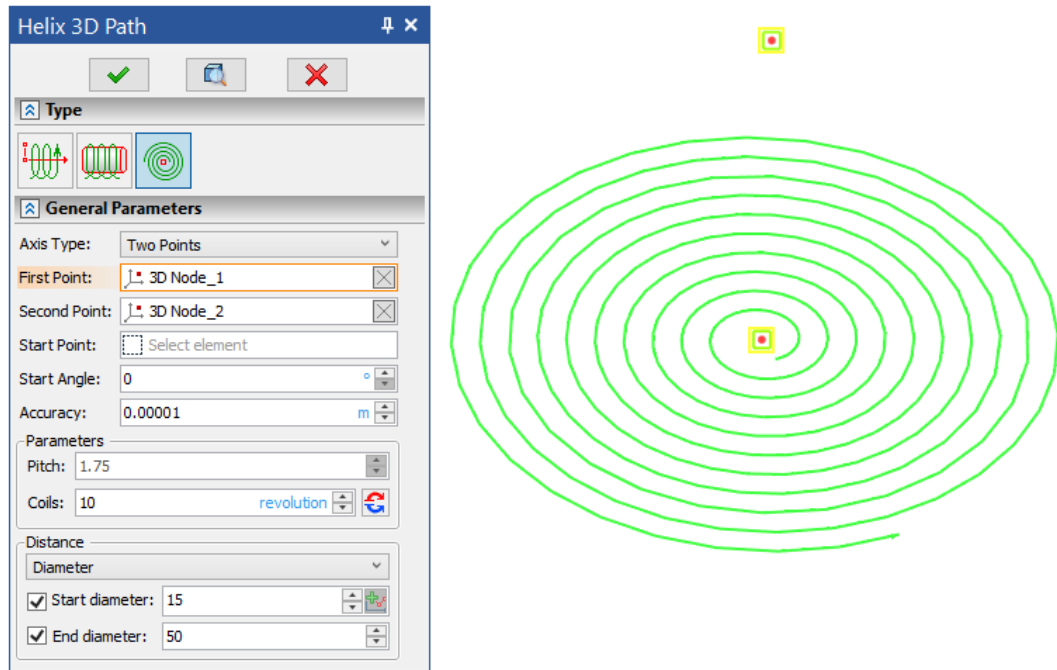


The surface on which the face lies should be closed in one of the directions, and the number of coils should be positive. As in the type of helix path **Along Axis**, there is the possibility of changing the direction of the coils.

You can also specify the start point of the helix path, which can be anywhere on the selected face. You can change the direction to the opposite.

Flat Helix 3D Path

Added the ability to create a flat spiral 3D path.



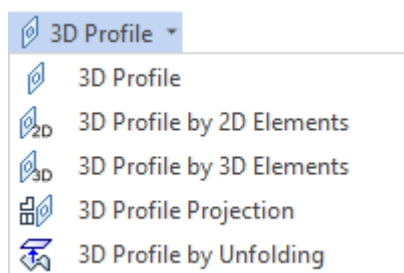
You can select 3 types as the axis - **Two points**, **Axis**, **Point and Direction**. As with other types of spiral path, you can specify a start point and a start angle.

3D Profile

Command features were expanded, and the command interface was optimized. All the command options are divided into four main principles for obtaining a profile:

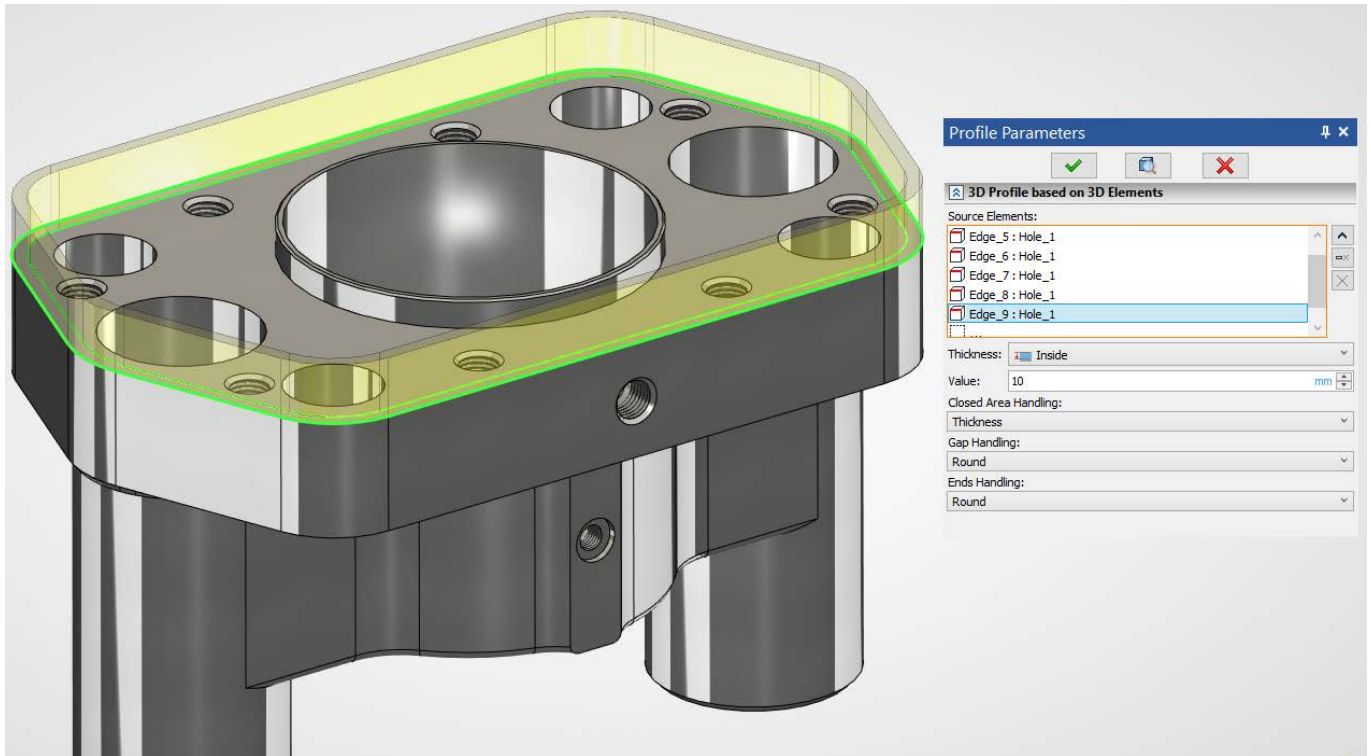
- 3D Profile by 2D Elements
- 3D Profile by 3D Elements
- 3D Profile Projection
- 3D Profile by Unfolding

The listed options for creating profiles can be specified in the command automenu, or selected in the Ribbon, or selected automatically based on the selected elements.



When creating a profile based on 2D elements, you can now explicitly specify the creation of a profile based on shading, text, or color.

When creating a profile based on 3D elements, there is a new **Offset** option for creating a profile as an offset curve from an existing profile. If the profile is multi-contour, the offset will be created from all the contours. The Offset direction is determined from the list as **Extend** or **Reduce**.



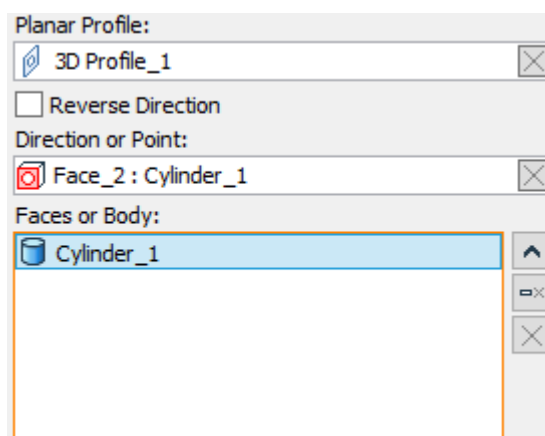
When projecting a profile to a face, the projection direction is automatically determined based on the selected face. If there is a situation where the projection direction can be interpreted in two ways, you can select the desired direction option using the change projection direction flag.

The 3D Profile command interface was updated. The icons for final geometry switching are now at the top of the dialog.



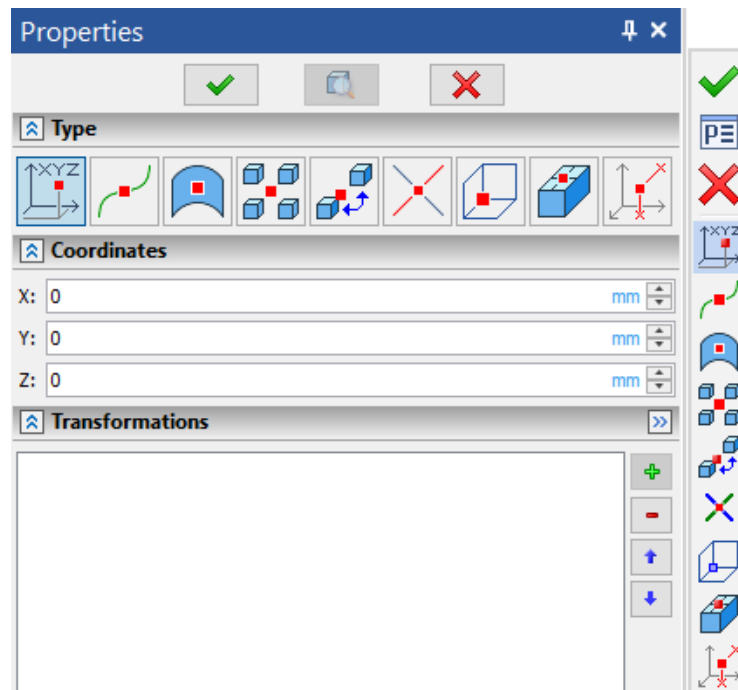
The **Thickness** list now contains options for thickness mode defining. By default, if there are no self-intersections, the option without thickness is selected.

All the selected items are specified in the dialog and can be deleted selectively or all together.



3D Node

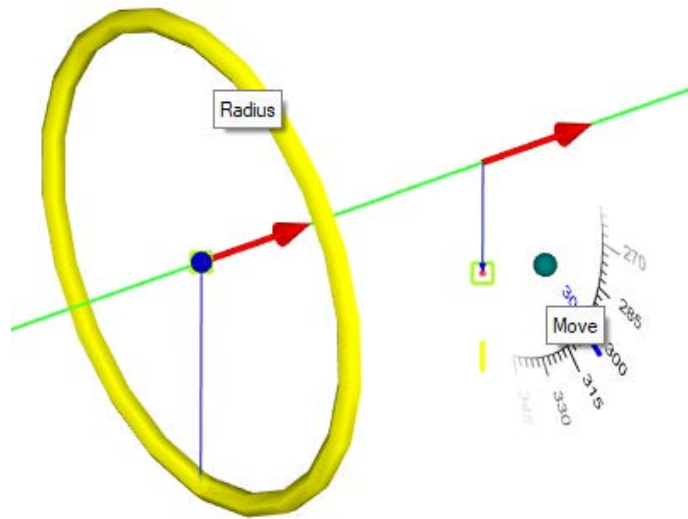
The command dialog was updated. Options for creating a 3D node are divided into 9 types. Each type has a corresponding icon in the dialog that is duplicated in the command automenu.



As in all renewed dialogs, creation modes can be selected by clicking the icon at the top of the dialog.

	In Coordinates
	On Path
	On Surface
	In Center of Mass
	By Shortest Distance
	On Intersection of Elements
	On Intersection of Three Planes
	By Faces and Offsets
	By Two Projections

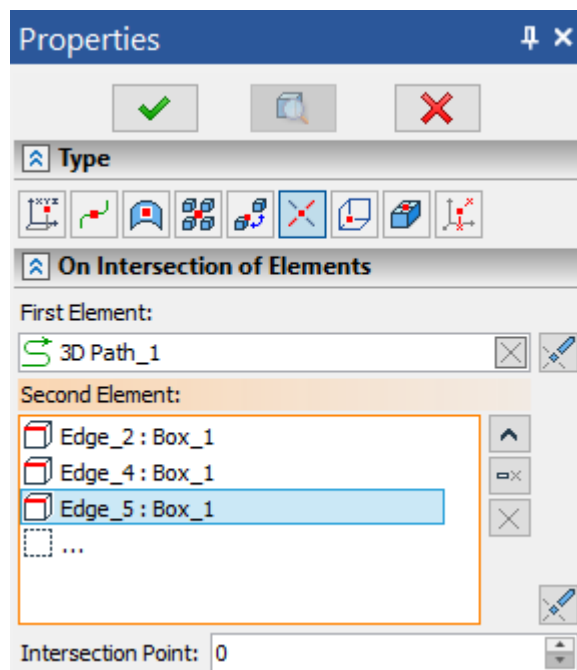
On Path. Draggers that enable you to set the position of a 3D node in a cylindrical coordinate system whose axis is tangent to the path were added. The position of the point of contact of the axis with the path is determined by the parameter "U", which changes in the selected system of units according to the total length of the path. The "R" parameter specifies the radius, and the "V" parameter specifies the angle. You can also change the units of measurement for these parameters.



On Surface. Draggers that enable you to move a node along the surface, as well as set the node offset along the normal to the surface, were added. The offset on the surface is set by the parameters "U" and "V" which lie in the interval from 0 to 1. The normal offset is set by the "W" parameter in the specified length units.

By Shortest Distance. The segment of the shortest distance is determined between the elements. A 3D node will be placed on it. This segment is not shown explicitly in the 3D scene. Using the Ratio parameter, you can determine the position of the 3D node on the specified segment in the range from 0 to 1.

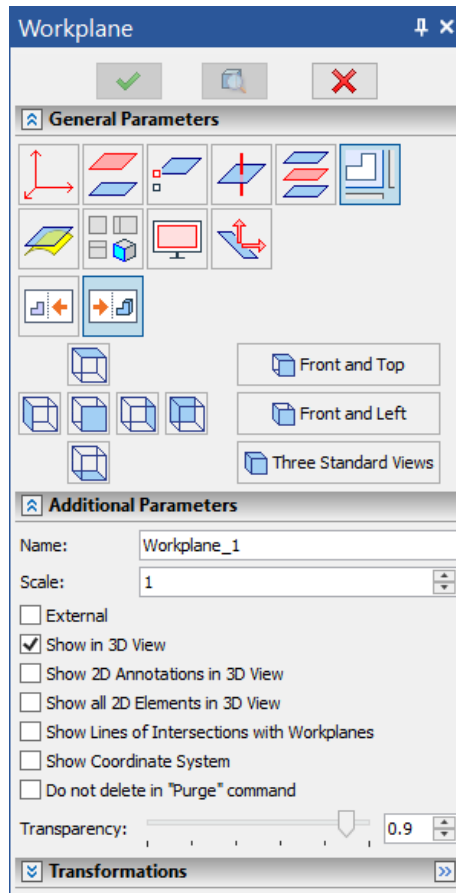
In the mode **On Intersection of Elements**, added the ability to select a set of operands as the second element.



Bindings are now available for 3D node manipulators.

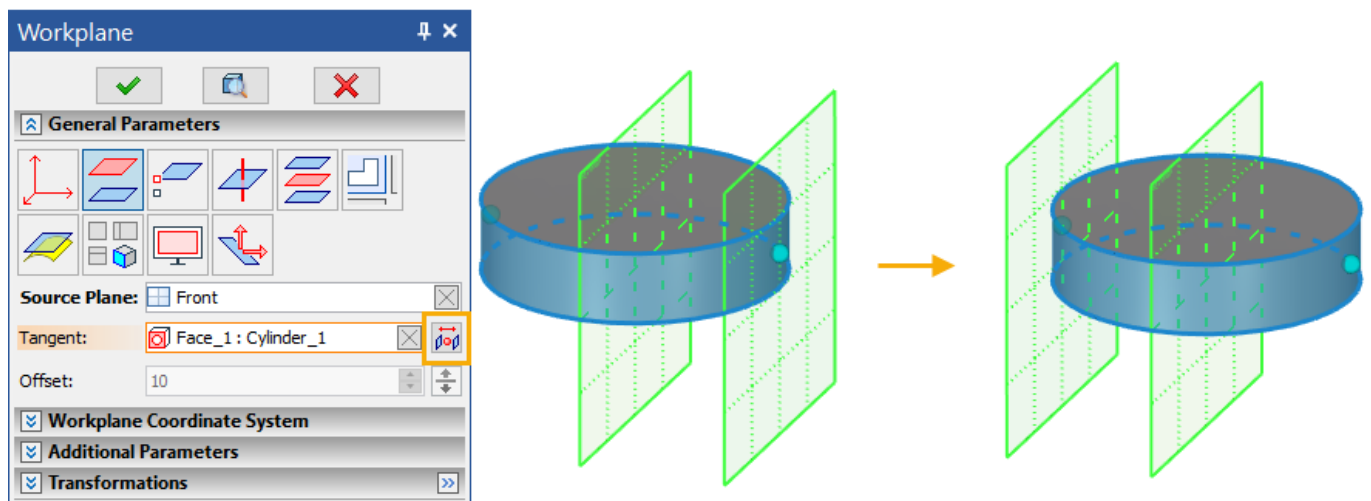
Workplane

The command for creating workplanes has a new interface.

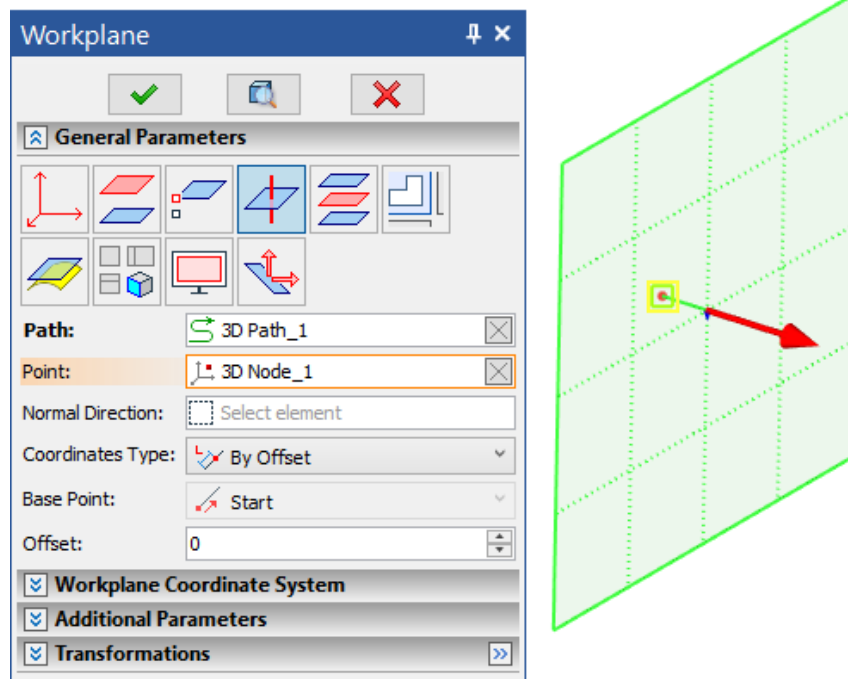


All automenu commands and their parameters have been moved to the command parameters window, now it's even easier to create workplanes.

Added the ability to rotate the position of a tangent surface 180 degrees.

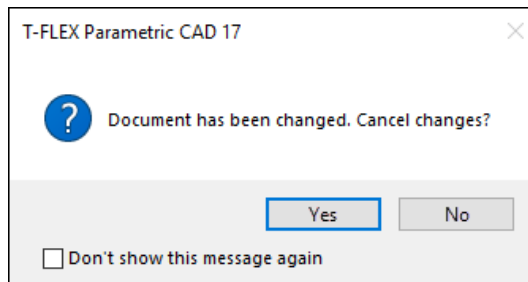


You can now create workplanes perpendicular to the path by specifying an offset.



Undoing Changes After Drawing on a Workplane

Now, when canceling drawing on the workplane, it is possible to cancel the added changes. It should be noted that all changes from the beginning of drawing will be canceled, and not only those that were on the workplane.

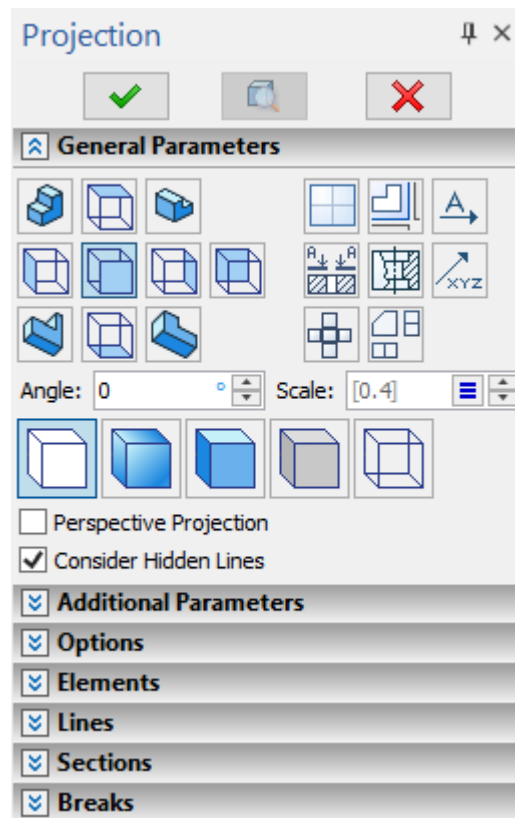


2D Design

Working in a 2D scene has become even more convenient. Added a command for creating a 2D array from a table. **Projection** command has been significantly reworked. The ability to place hyperlinks to various system elements and external resources has been added to the text editing context menu. The functionality of the drawing zones has been significantly expanded.

“Projection” Command Updating

A lot of work has been done on the **Projection** command. The command has been significantly reworked and transferred to a new interface.



The position of the buttons of the projection views can be changed depending on the **ISO (First Angle)** or **ANSI (Third Angle)** standard in the system settings.



ISO view buttons position

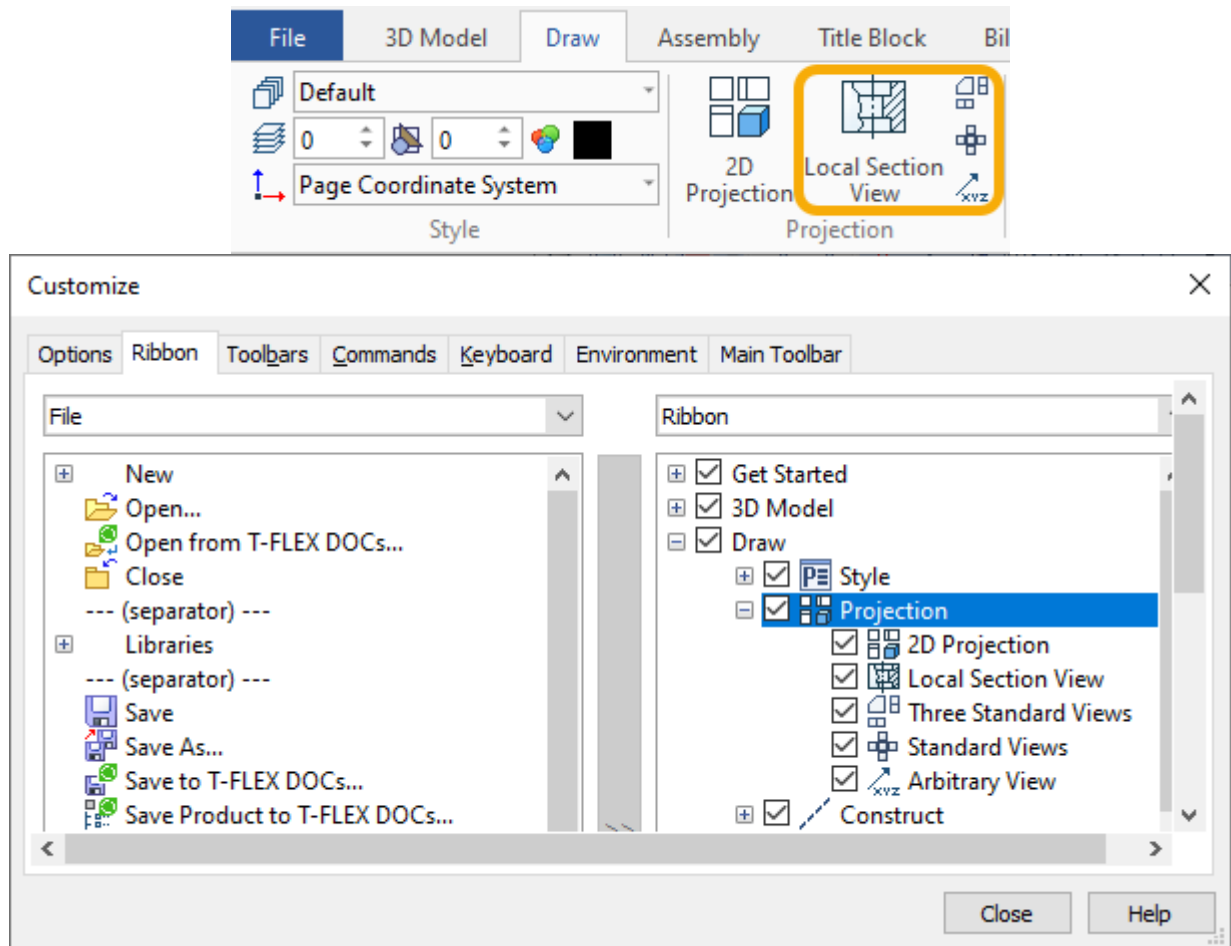


ANSI view buttons position

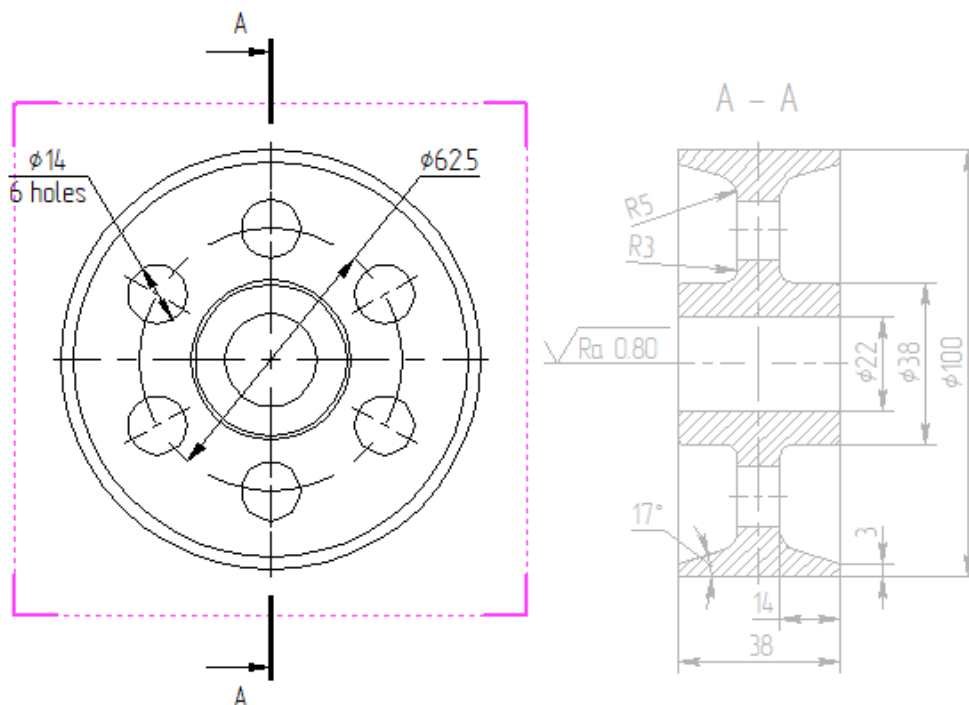
Added new types of isometric projections.

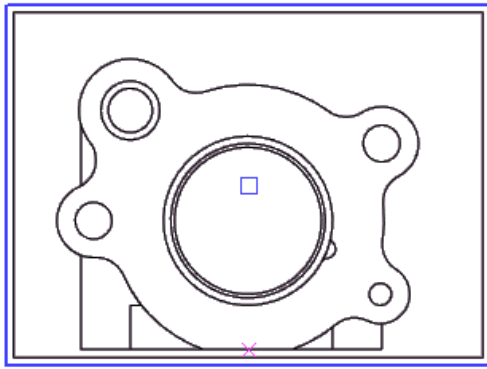


Local section, standard and arbitrary views creation modes are now available directly from the ribbon.

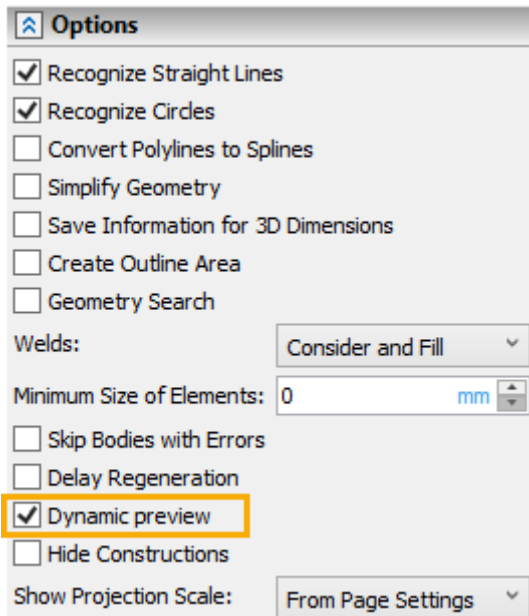


To modify the projection (drawing lines) or to add appearance elements, you now need to activate the projection page, which occurs either automatically when you try to place a line or appearance element on the projection, or with double click on the drawing view. Deactivation of the drawing view is performed with double click outside the drawing view or using the **Close drawing view** command in the **Draw** tab. When the projection page is active, it is automatically ensured that it is impossible to set dimensions and other symbols attached to different projections.





When creating a 3D projection, its preview is now displayed.

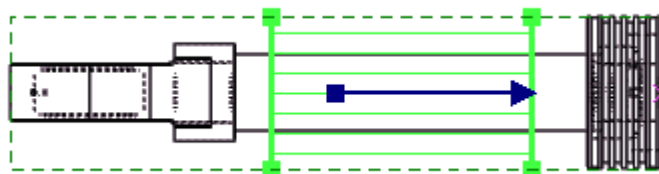


The **Dynamic Preview** checkbox is added to the command parameters for switching on/off a dynamic preview of a projection.

Disabling this checkbox is useful, when working with complex documents, if using the preview slows down the system.

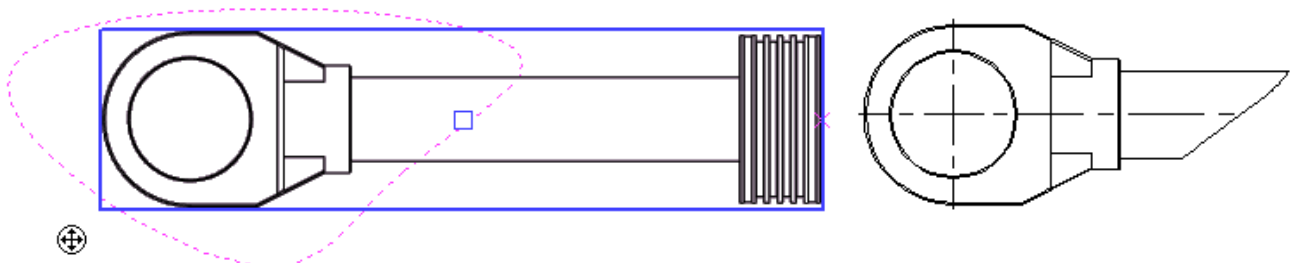
All dimensions specified on the projection page are attached to it, and when it is moved, they move with it without attaching manipulations. Moving the projection itself is now performed by moving the drawing view.

The mechanism for creating local section views and the mechanism for creating views with breaks has been significantly improved.



Added the ability to insert breaks on the local section view. Break line type can now be saved in projection defaults.

Added the ability to crop projection along the specified cut outline.



New automenu options are added to most projection types:



<T> Crop View



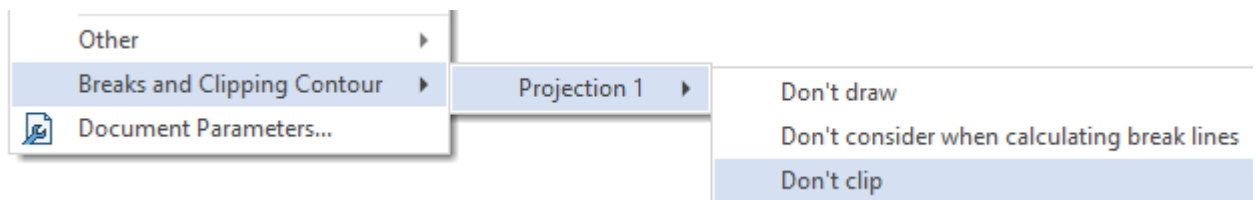
<Y> Reset view outline

New automenu option is added to **Local Section View**:

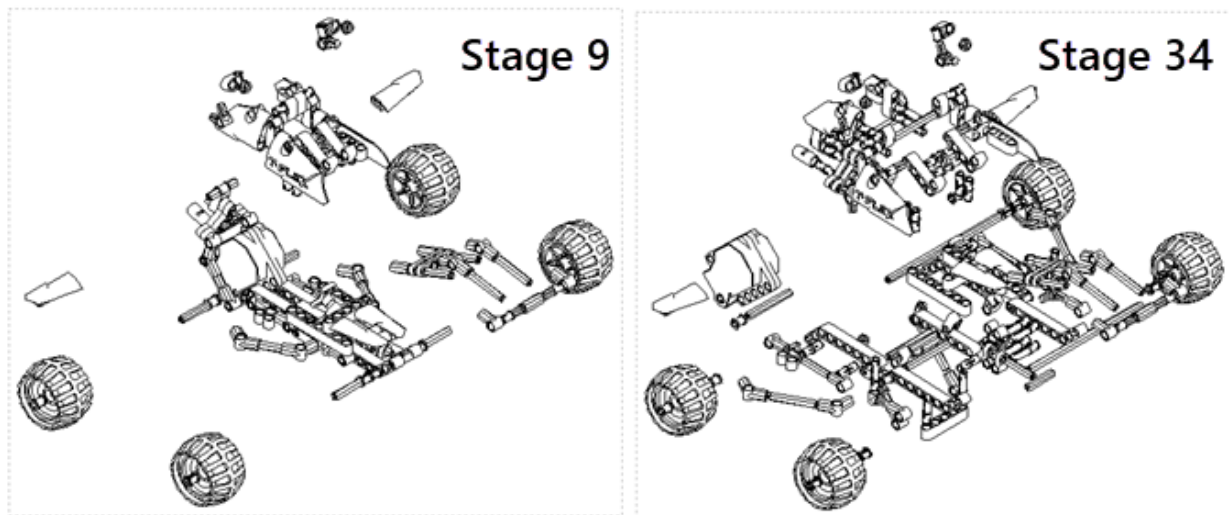


<P> Create Hatch by Spline

Breaks and Clipping Contour options are added to the contextual menu of active projection's elements. These options allow controlling element's clipping and visibility as well as excluding an element from break lines calculation.



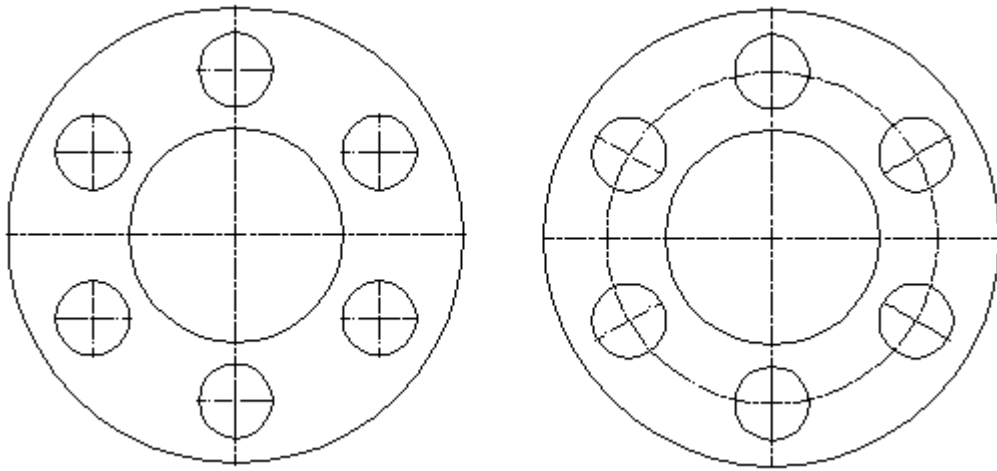
The ability to select the stage of the exploded view scenario has been added to the projection parameters, which can be used, for example, to create documentation with step-by-step illustrations of the assembly/disassembly process.



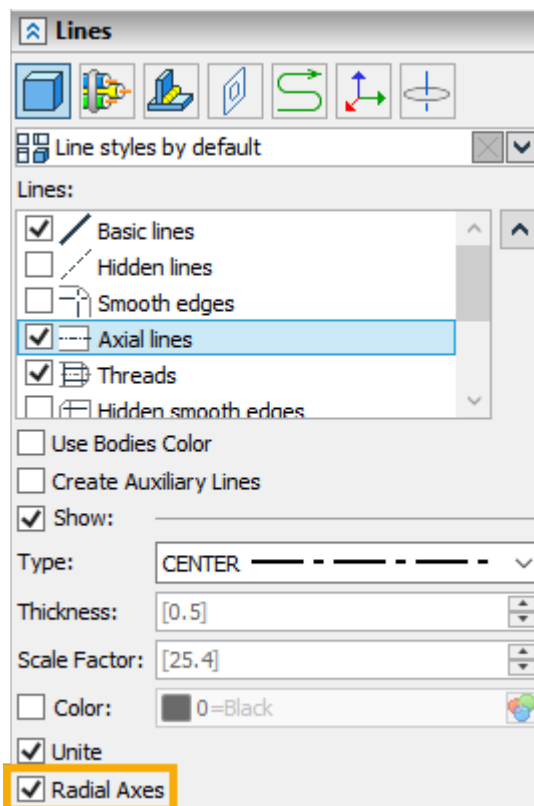
In addition, the ability to project the trajectory of movement and adjust the type of trajectory lines has been added.

The ability to exclude bodies with errors has been added to the projection parameters.

Added the **Radial Axes** checkbox (enabled by default). The checkbox appears in the **Lines** section of command's parameters window upon selecting the **Axial lines** in the list of categories. This option is useful, when projecting holes that are placed along a circle. The effect is shown in the picture below.

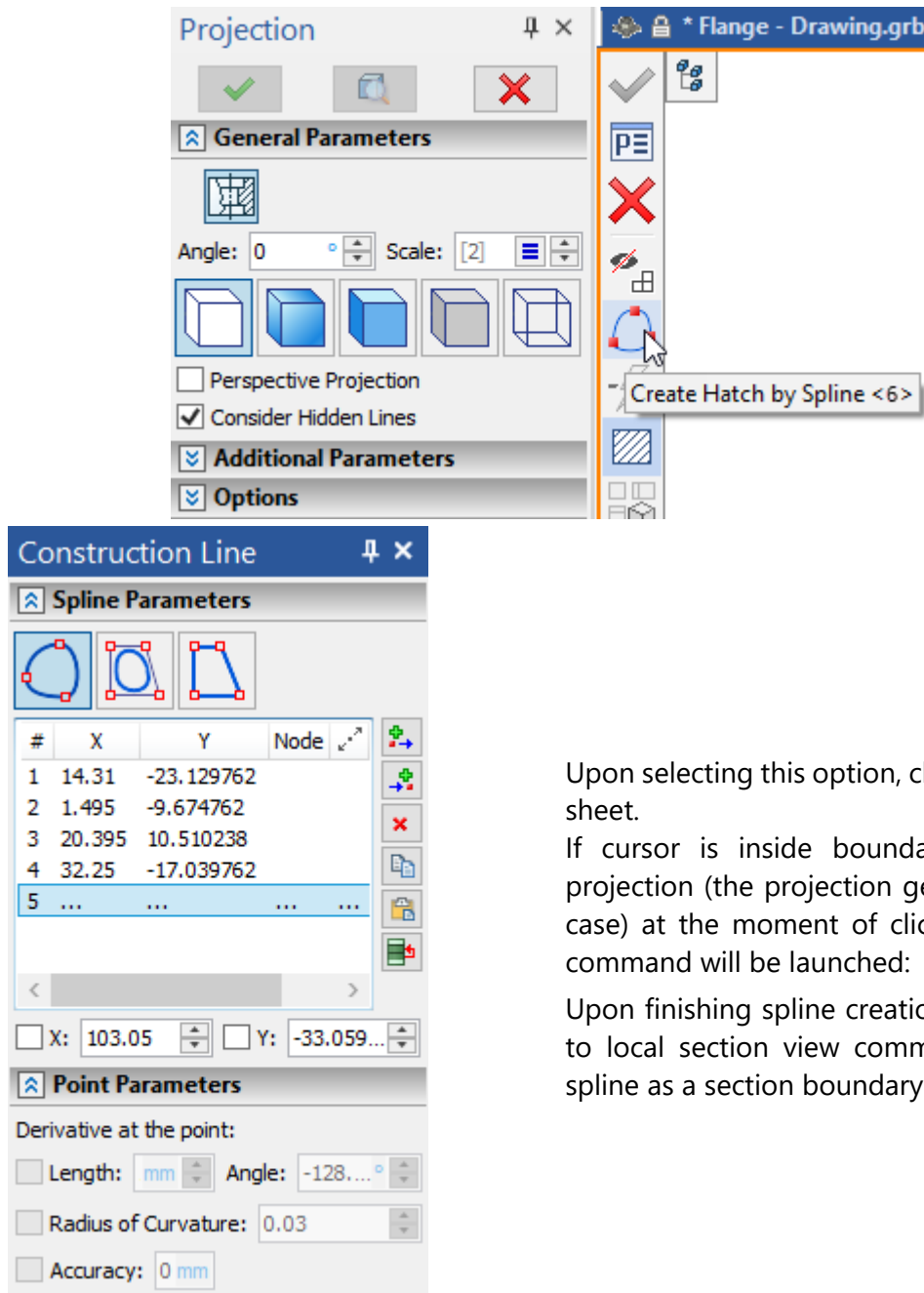



On the left Radial Axes disabled, on the right - enabled.



Create Hatch by Spline

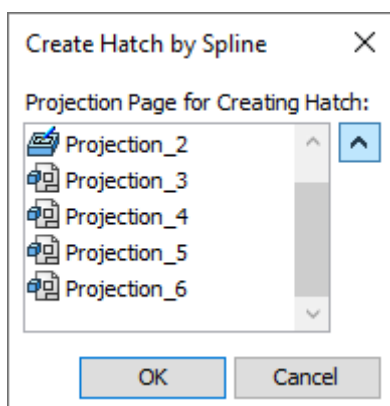
Create Hatch by Spline option is added to the Local Section View automenu..



Upon selecting this option, click  on a drawing sheet.

If cursor is inside boundaries of any existing projection (the projection gets highlighted in this case) at the moment of clicking, then the spline command will be launched:

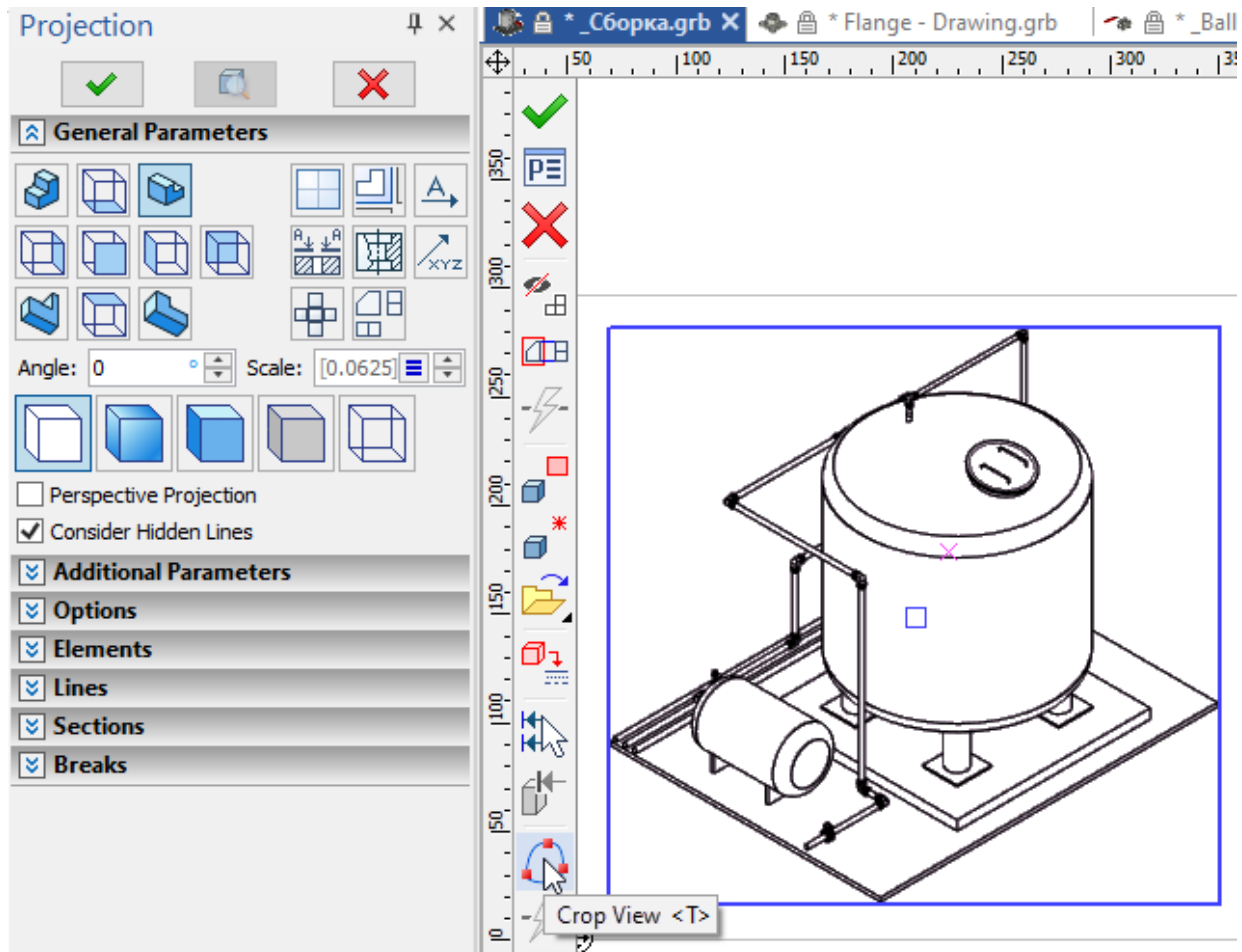
Upon finishing spline creation, the system returns to local section view command and selects this spline as a section boundary.



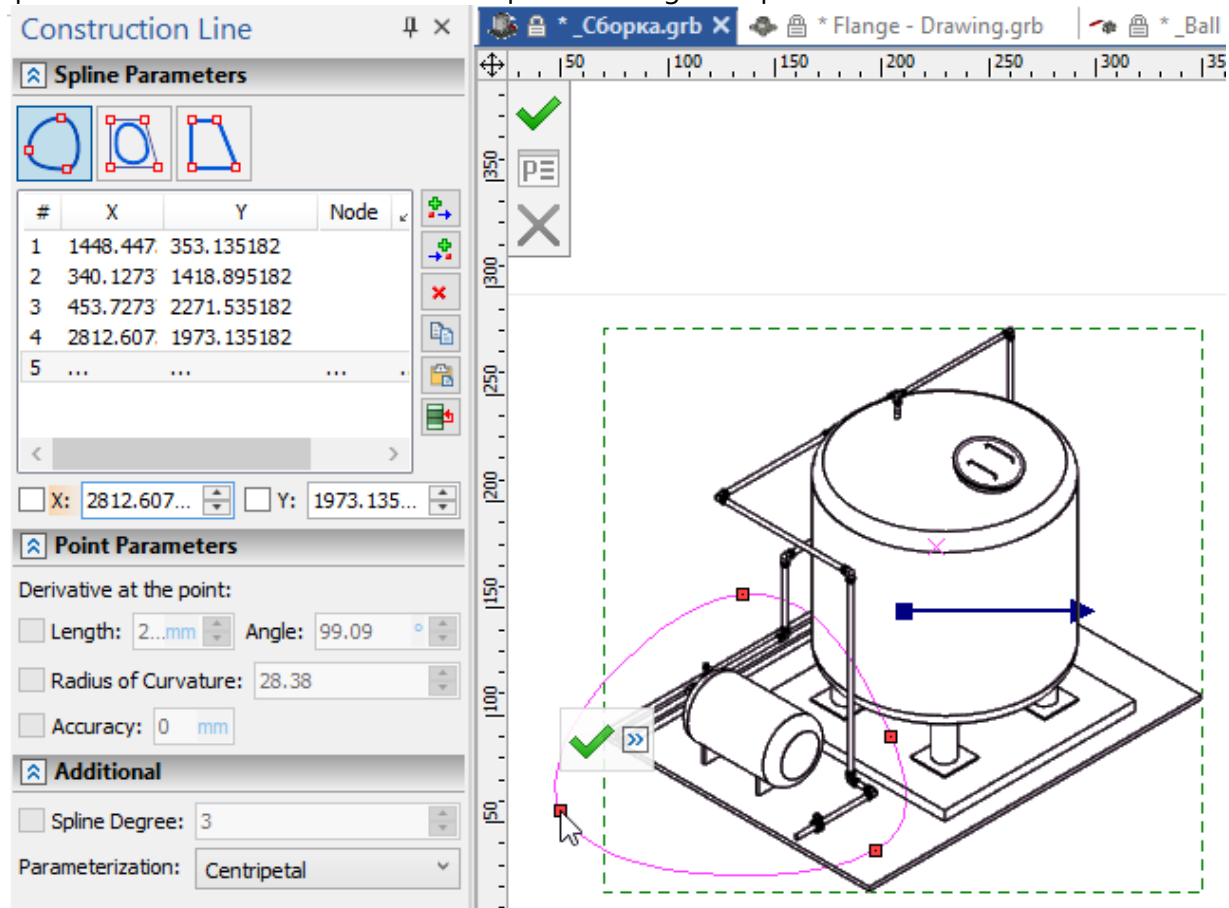
If cursor is outside boundaries of any existing projections at the moment of clicking, then the projection selection dialog appears. Upon selecting a projection, the system launches the same spline command as described above.

View Clipping Contour

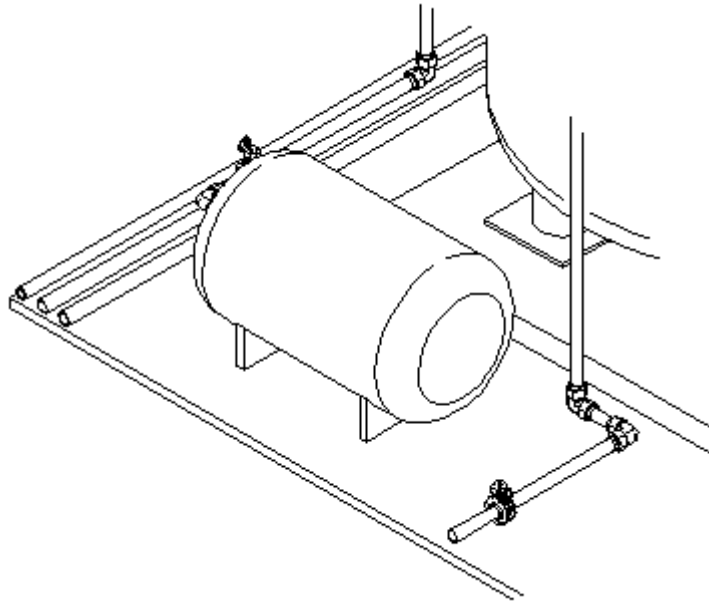
Crop View automenu option is added to most projection types:



Spline creation command is launched upon selecting this option:



Upon finishing spline creation, the system returns to projection command and selects this spline as a view outline.



Use the following automenu option to cancel view clipping:



<Y> Reset view outline

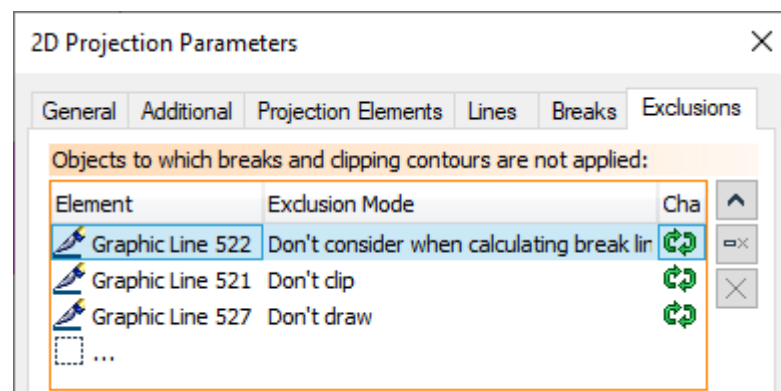
It can be used both, when creating a new projection, and when editing an existing one using the **Edit** option in projection's contextual menu.

Exclusions on 2D Projections and Drawing Views


Following **Breaks and Clipping Contour** options are added to the contextual menu of active projection's and drawing view's elements:

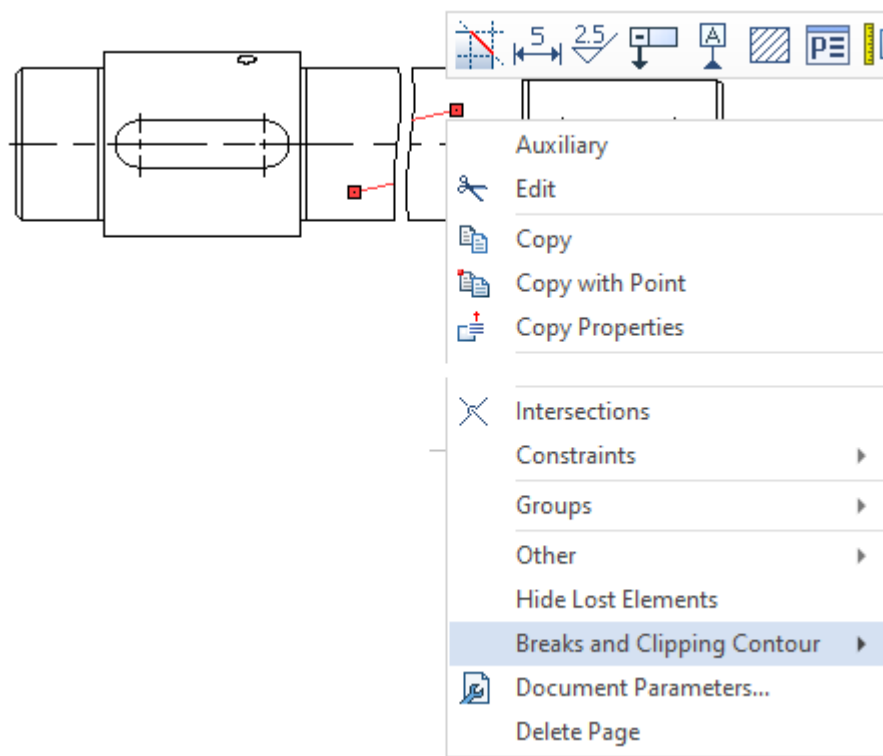
- Don't draw
- Don't consider when calculating break lines
- Don't clip

Elements affected by these options are added to the list in the **Exclusions** tab of projection's parameters:

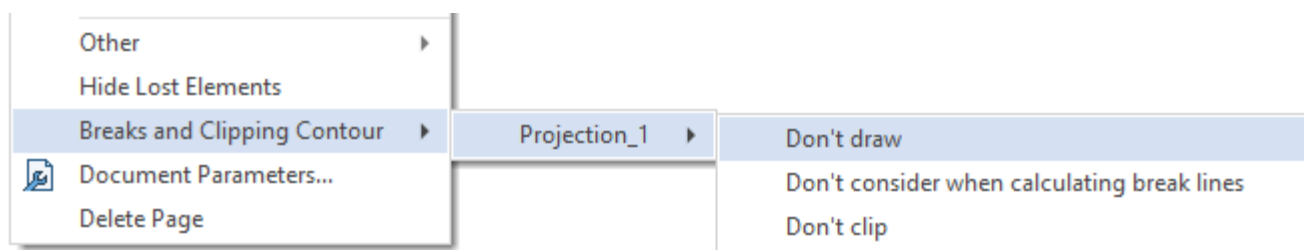


Don't draw

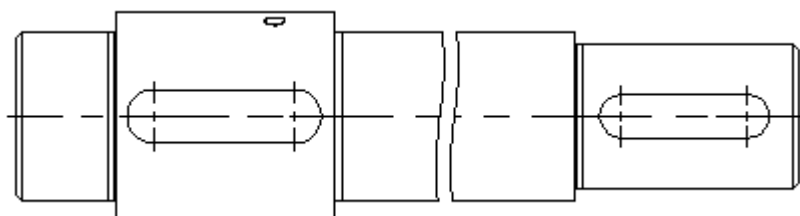
Click  on active projection's element:



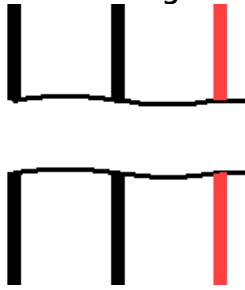
Select **Breaks and Clipping Contour** > **Projection_#** > **Don't draw** in the appeared contextual menu:



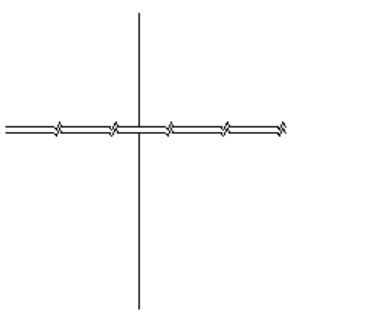
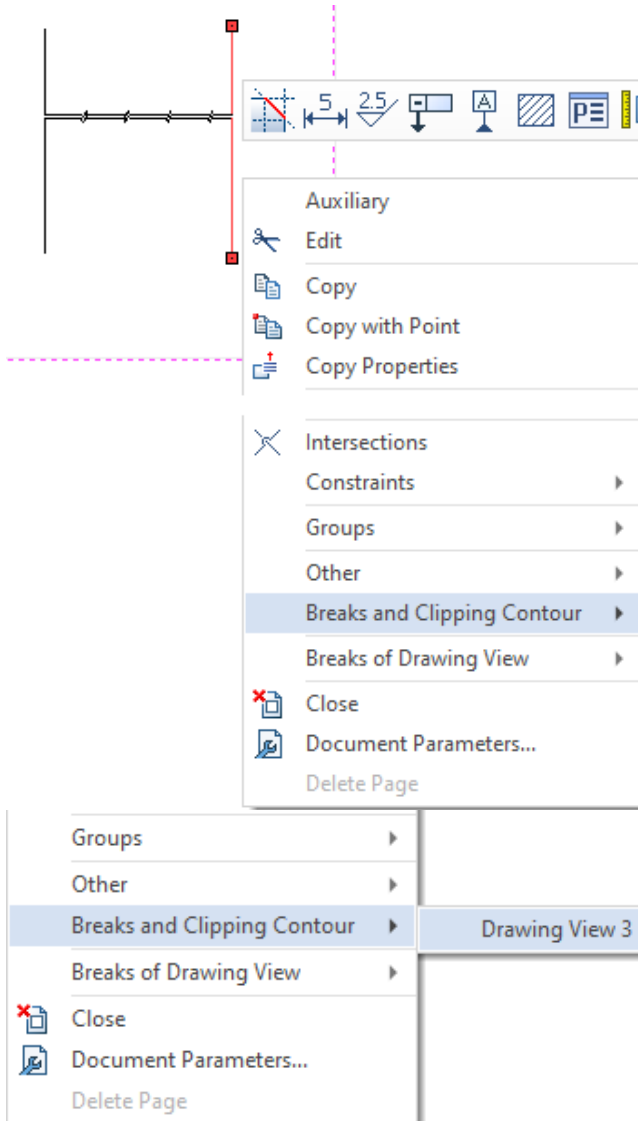
Selected element is not drawn anymore:




Don't consider when calculating break lines



The option is enabled for the red line




This option is available both on 2D projections and on basic drawing views. However, on 2D projections it only affects lines representing contour of 3D model body. User-created graphic lines on 2D projections are not considered when calculating break lines anyway, no matter if this option is enabled or not.

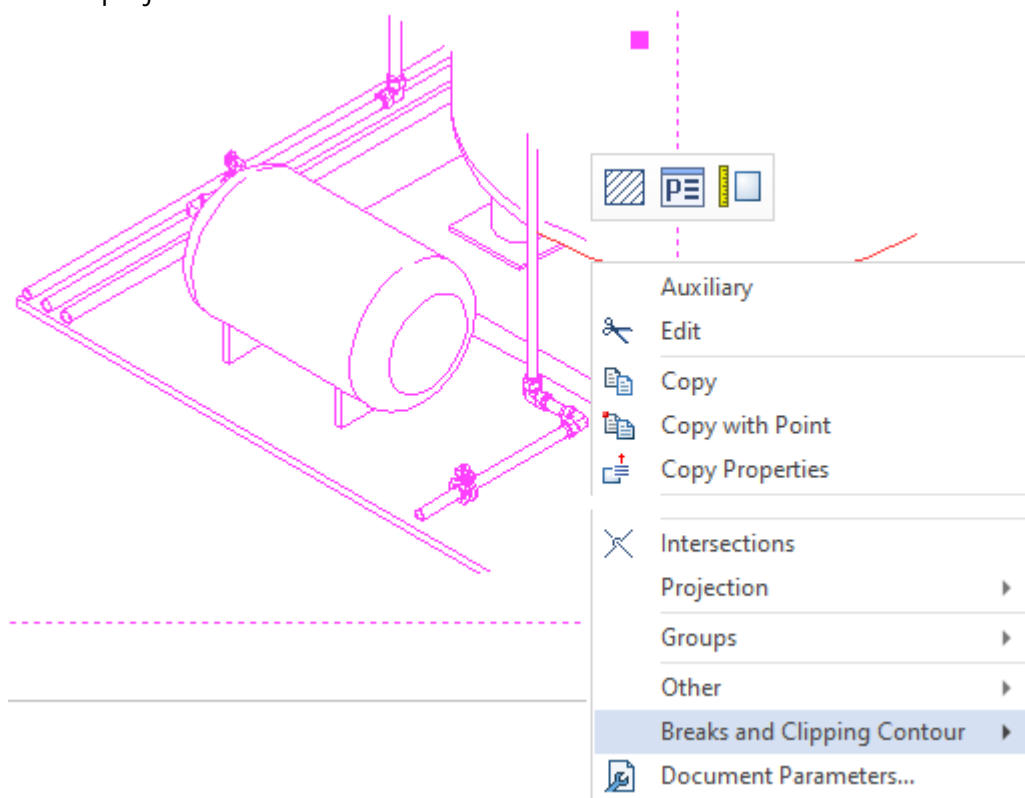
Click  on active drawing view's element.

Select **Breaks** and **Clipping Contour** > **Projection_#** > **Don't consider when calculating break lines** in the appeared contextual menu.

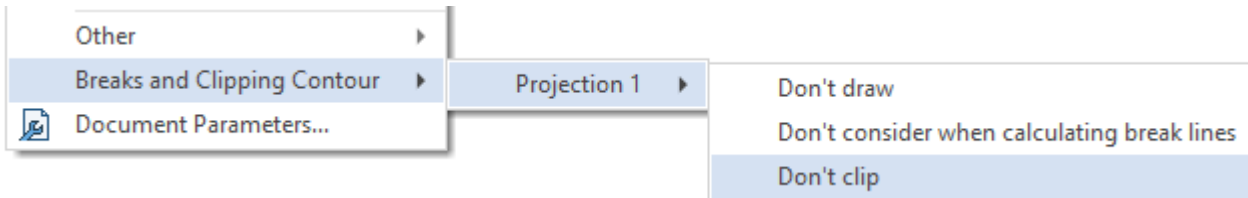
Break line doesn't reach the selected element anymore.

Don't clip

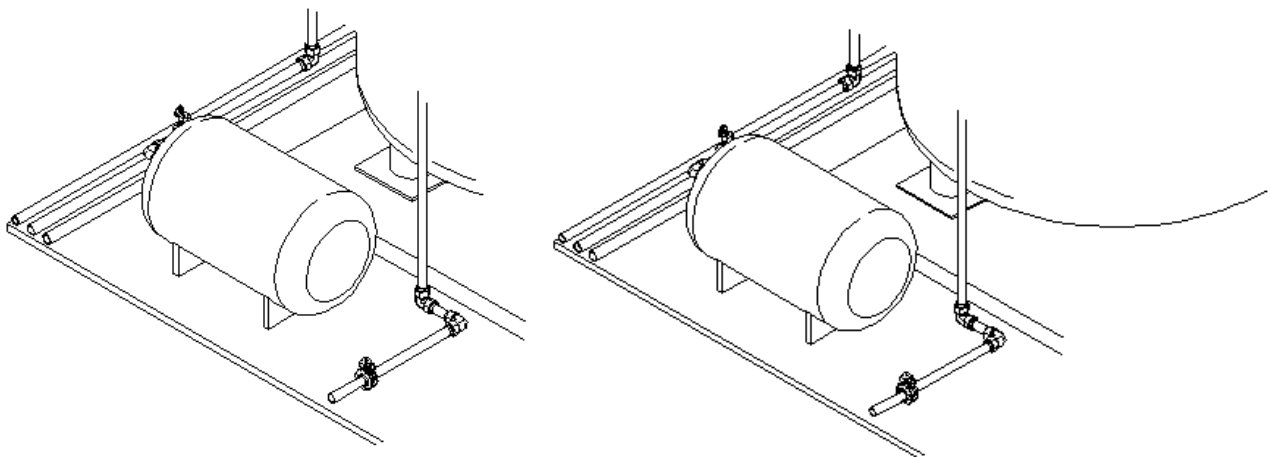
Click  on active projection's element:



Select **Breaks and Clipping Contour** > **Projection_#** > **Don't clip** in the appeared contextual menu:

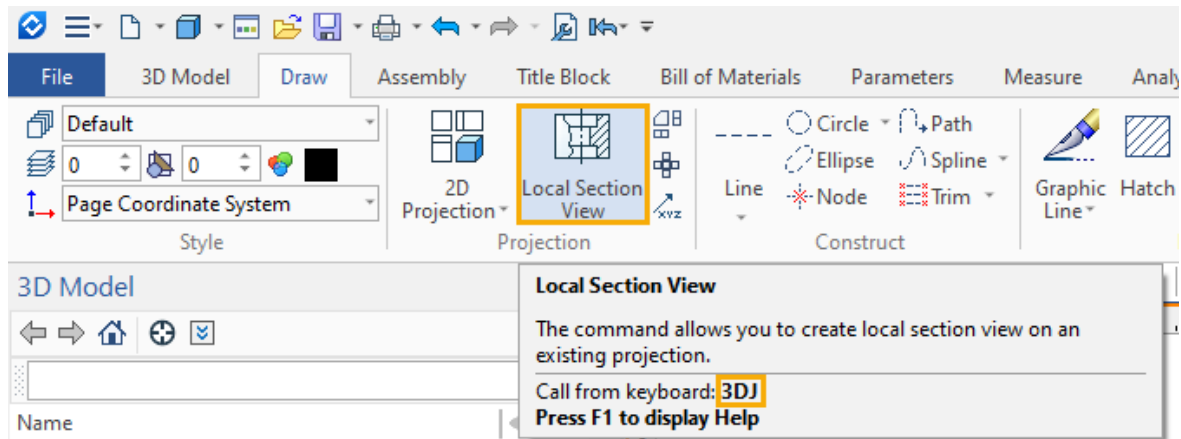


Selected element is not clipped anymore:



New Local Section View Command

You can now use the new **Local Section View** command instead of the cumbersome old local section creation mode of the **2D Projection** command. The new command provides only the section-related tools, without the rest of the parameters and controls common for 2D projections of all types. The command can be called from the ribbon or via keyboard separately from the main **2D Projection** command.

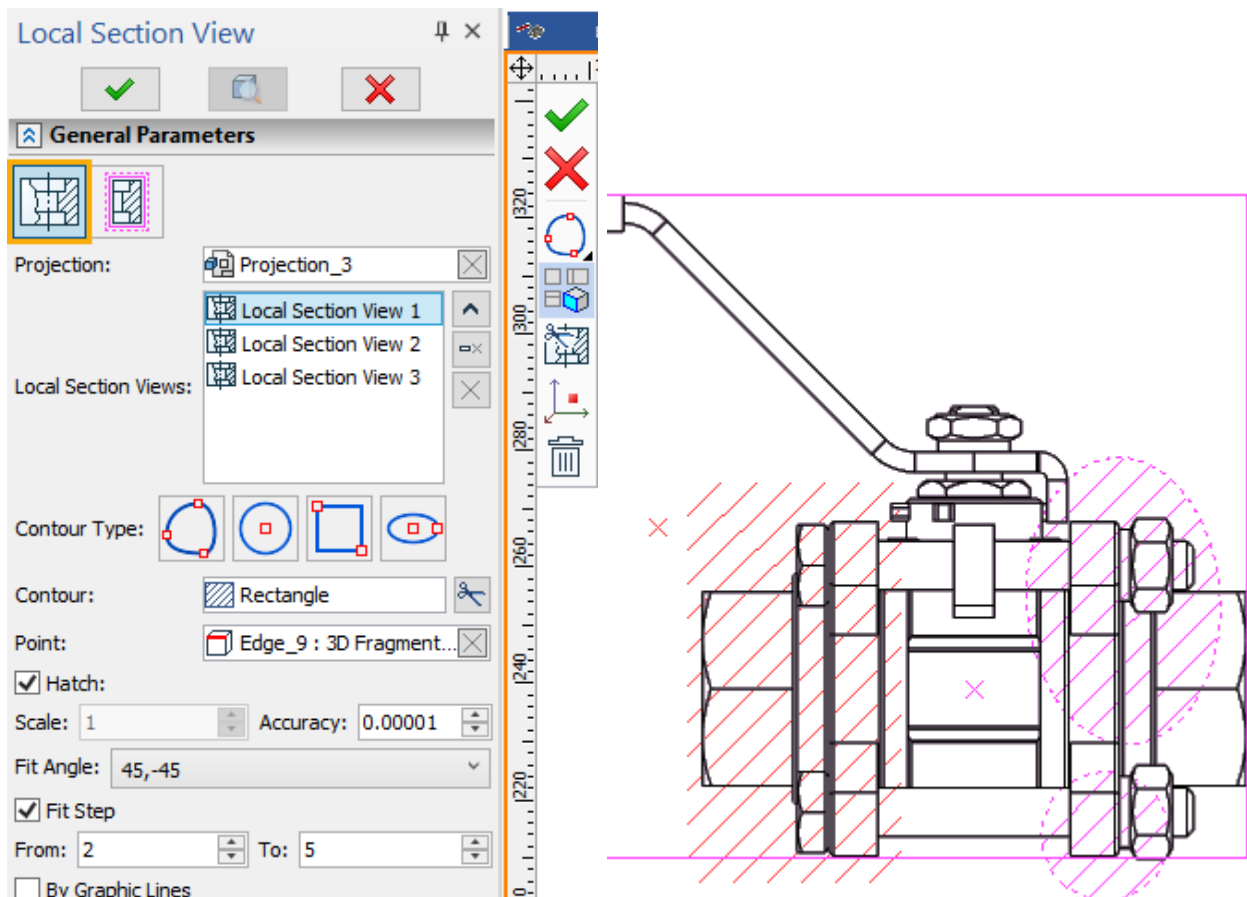


The command has two modes: **Within Projection** and **Application**.



Within Projection

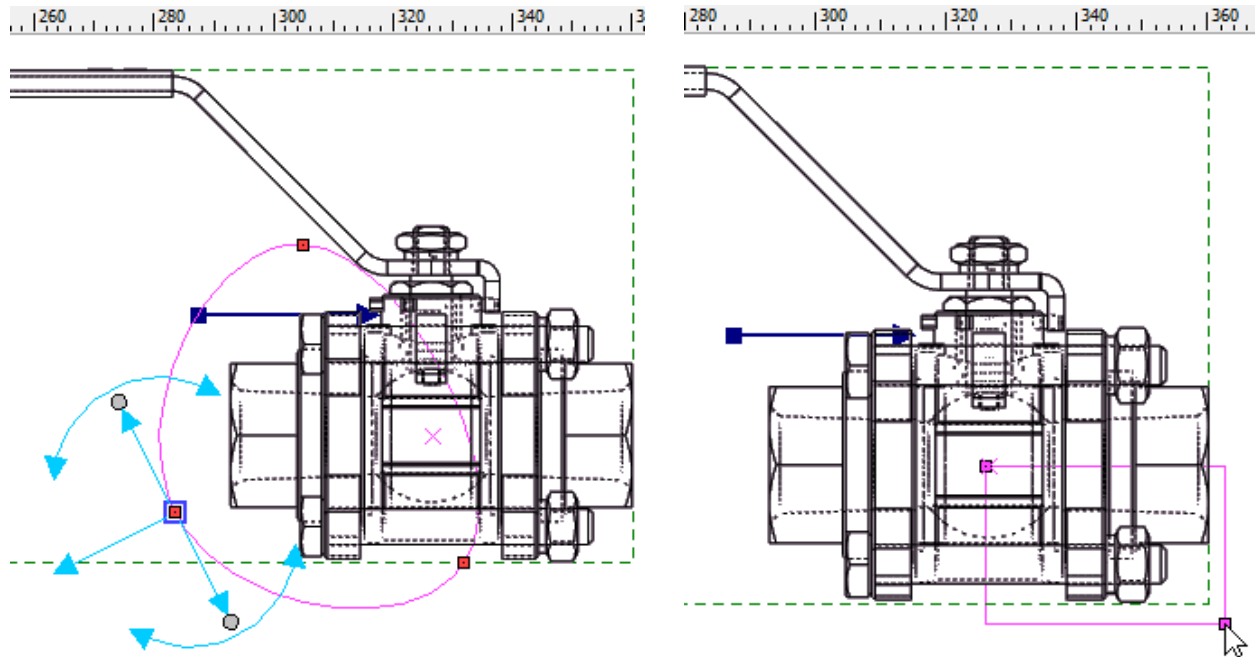
This mode is active by default, upon calling the command. It replaced the mechanics of creating local section views within the main projection, which were previously introduced in the 17.0.70.0 release of the T-FLEX CAD. A section created in this mode is a part of a main projection. The mode allows to cut multiple areas of a single projection by different cutting planes. The mode can be used both for creating new sections and editing sections created earlier within projections.



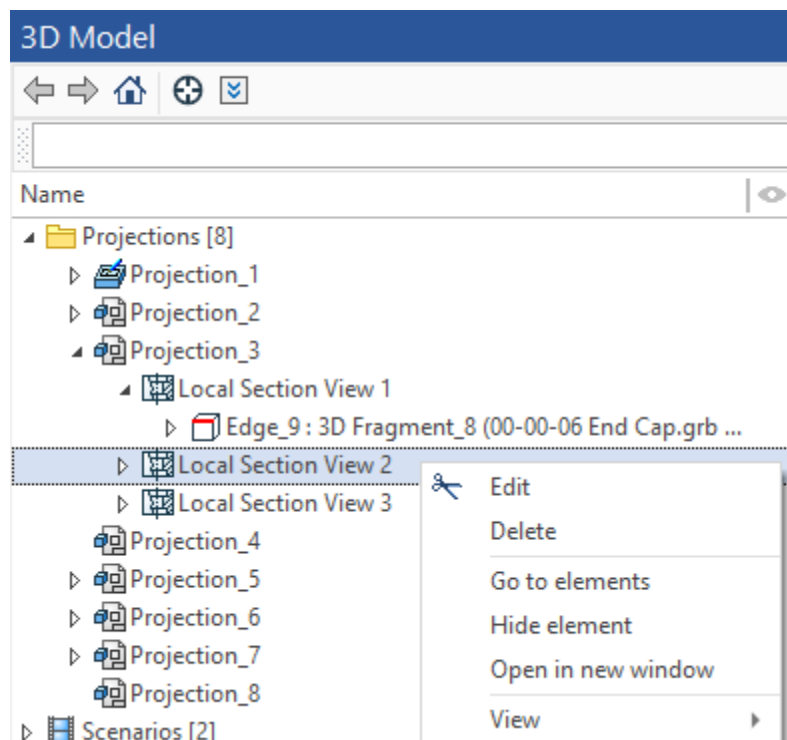
Upon creating a new local section, it is assigned hatch parameters set in the parameters of its parent projection. You can edit hatch parameters in the command's parameters window, if necessary.

Hatch parameters displayed in the **Local Section View** command are not associated with a particular section. Instead, they're applied to the whole projection. So, changing hatch parameters in this command changes them in projection's parameters dialog and vice versa.

The contour of sectioned area is created in the process of creating a section. It can be a spline, circle, ellipse or a rectangle.



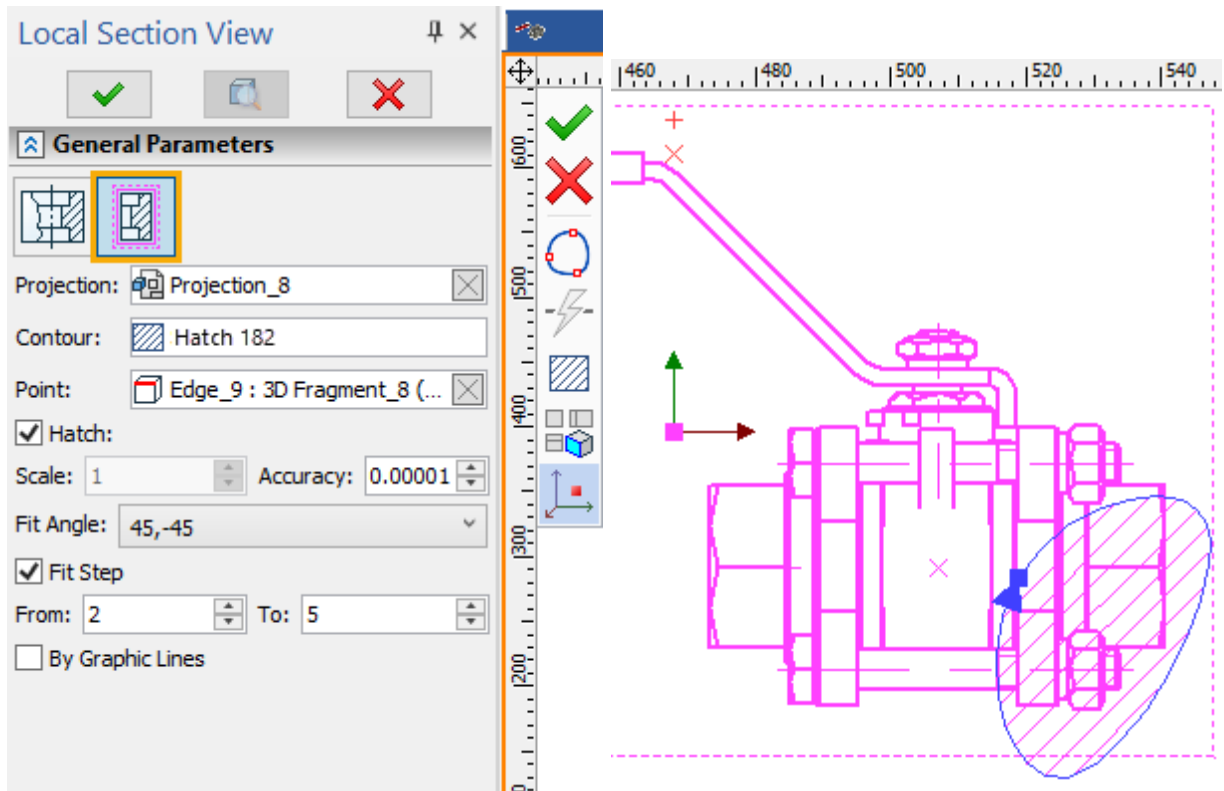
Local sections created within a projection are displayed in the 3D Model Tree as child objects of such projection. The **Delete** command available in contextual menu of a section in the tree allows to delete section without launching the **Local Section View** command. The **Edit** command available in the same menu invokes the Local Section View command for the parent projection. Local sections within projection do not have own parameters dialogs. They inherit parameters set in the dialog of the parent projection.



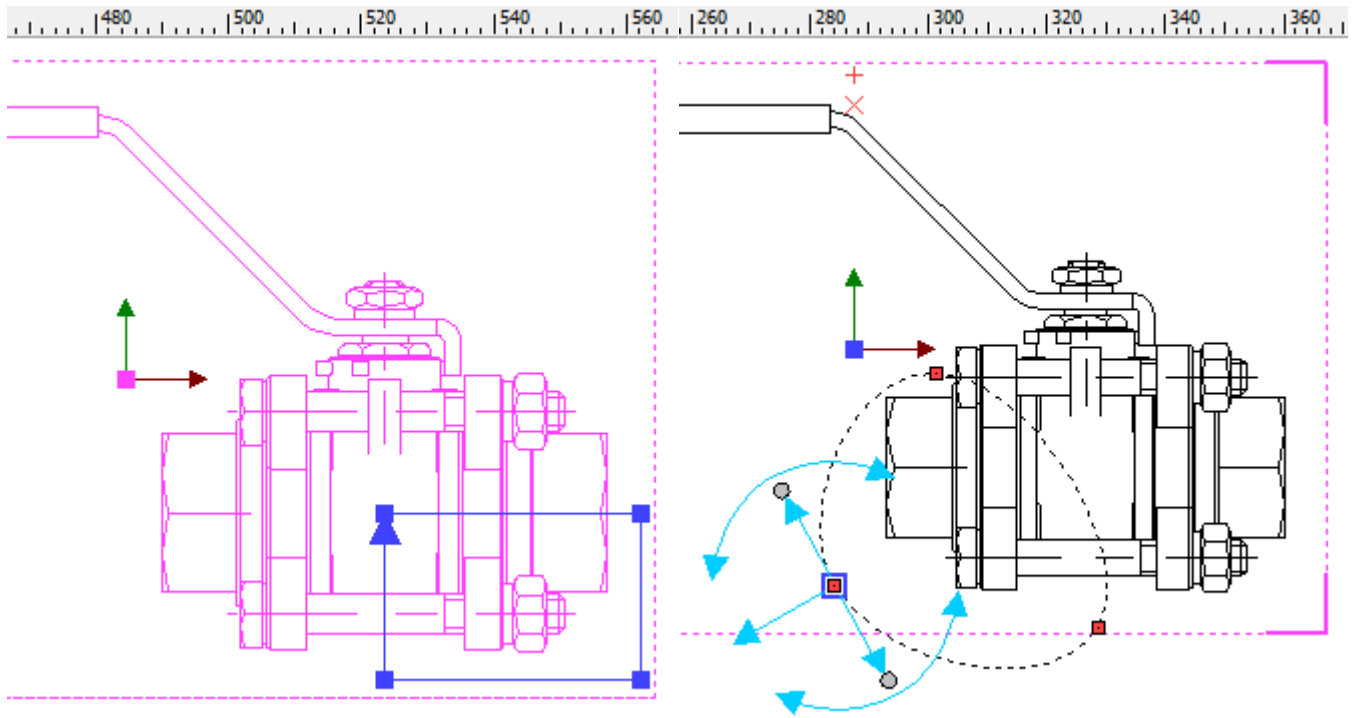



Application

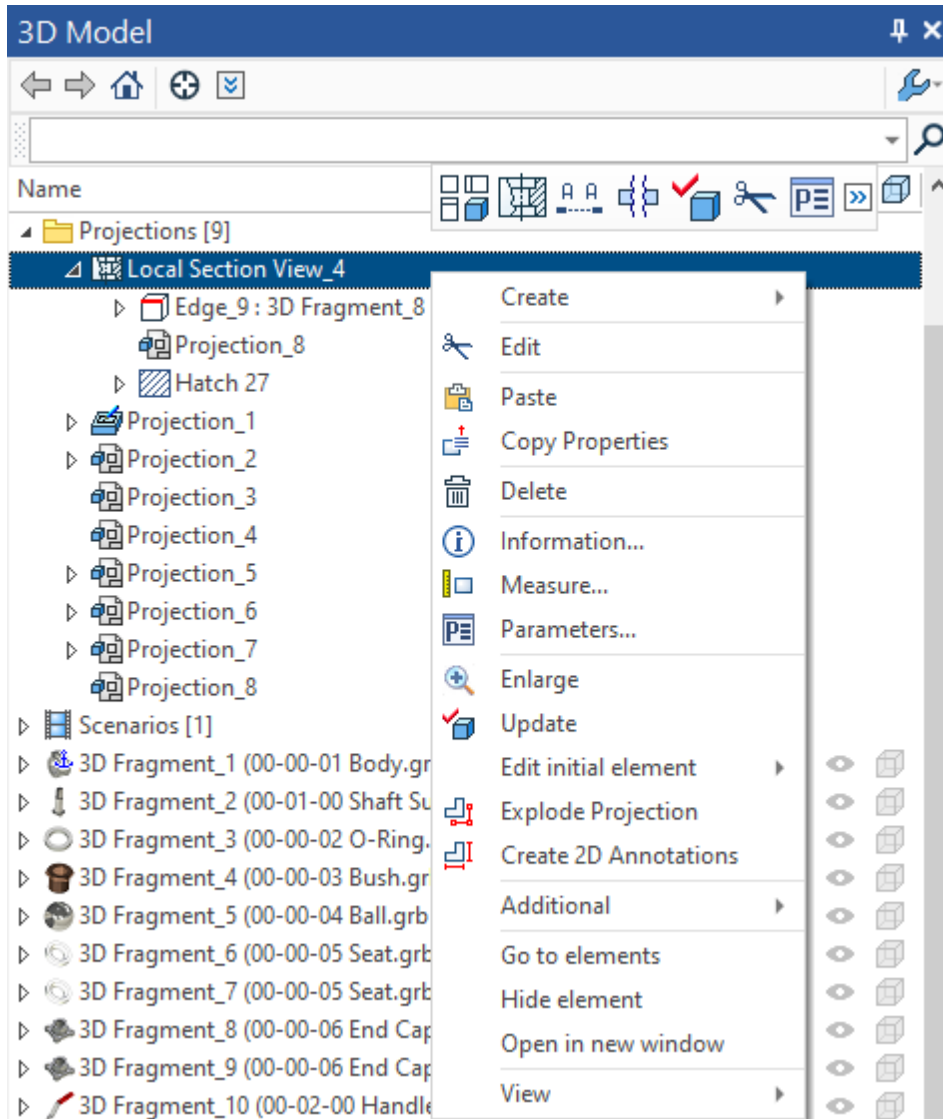
A local section created in this mode is a separate projection, which overlaps the main projection. This mode cuts a single area only. In order to cut multiple areas of a single projection, you would have to create individual local section projections for each area. This mode can only be used for creating new sections.



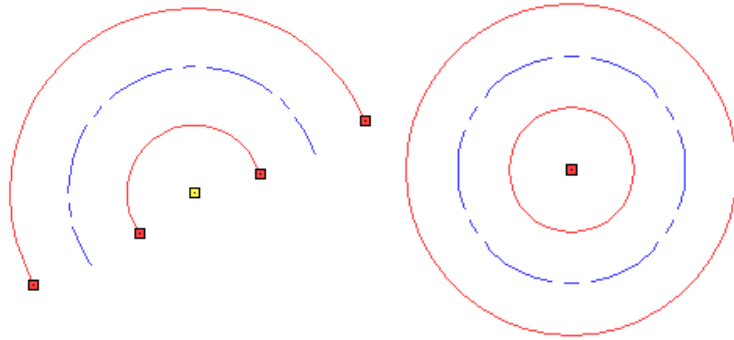
You can either create a spline defining contour of sectioned area in the process of creating a section, or use a pre-created hatch as a contour.





Local sections created in the application mode are displayed in the 3D Model Tree as separate projections of a special type. Contextual menu of such projections in the tree is the same as of other projections. The  **Edit** command of the contextual menu invokes the local section editing mode of the **2D Projection** command. Parameters dialog of a local section created in the application mode is also the same as of other projections.

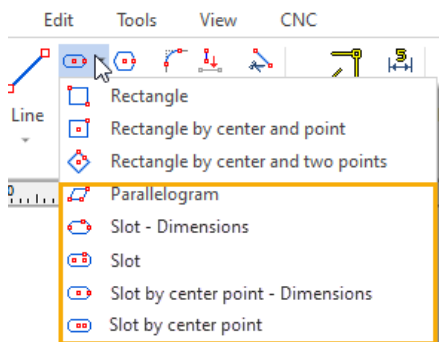


Axis



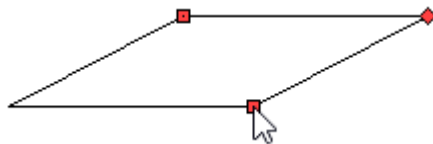
The  **Create Axis of Two Graphic Lines** mode of the  **Axis** command now allows to create axis between two concentric arcs:

New Sketch Commands



New sketch commands are added:

- **Parallelogram,**
- **Slot - Dimensions,**
- **Slot,**
- **Slot by center point – Dimensions,**
- **Slot by center point.**



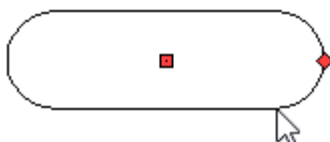
To create a **Parallelogram** element, you need to specify three points. All three points define the X and Y coordinates. For the second and third points, you can specify either dx and dy offset, or angle and length



To create the **Slot - Dimensions** element, you need to specify two points that define the dimensions of the slot. Then you need to determine the width of the slot or the radius of the circles. The first and second points specify the X and Y coordinates. For the second point, you can specify either dx and dy offset, or angle and length.



To create the **Slot** element, you need to specify two points from which the circles that define the slot dimensions will emerge. Then you need to determine the width of the slot or the radius of the circles outgoing from these points. The first and second points specify the X and Y coordinates. For the second point, you can specify either dx and dy offset, or angle and length.



To create the **Slot in center point - Dimensions** element, you need to specify two points, one of which will define the center of the slot, and the second will define its extreme circles. Then you need to determine the width of the slot or the radius of the circles. The first and second points specify the X and Y coordinates. For the second point, you can specify either dx and dy offset, or angle and length.



To create the **Slot in center point** element, you need to specify two points, one of which will define the center of the slot, and a circle will emerge from the second (symmetrically on the other side). Then you need to determine the width of the slot or the radius of the circles. The first and second points specify the X and Y coordinates. For the second point, you can specify either dx and dy offset, or angle and length.

Spline Painting

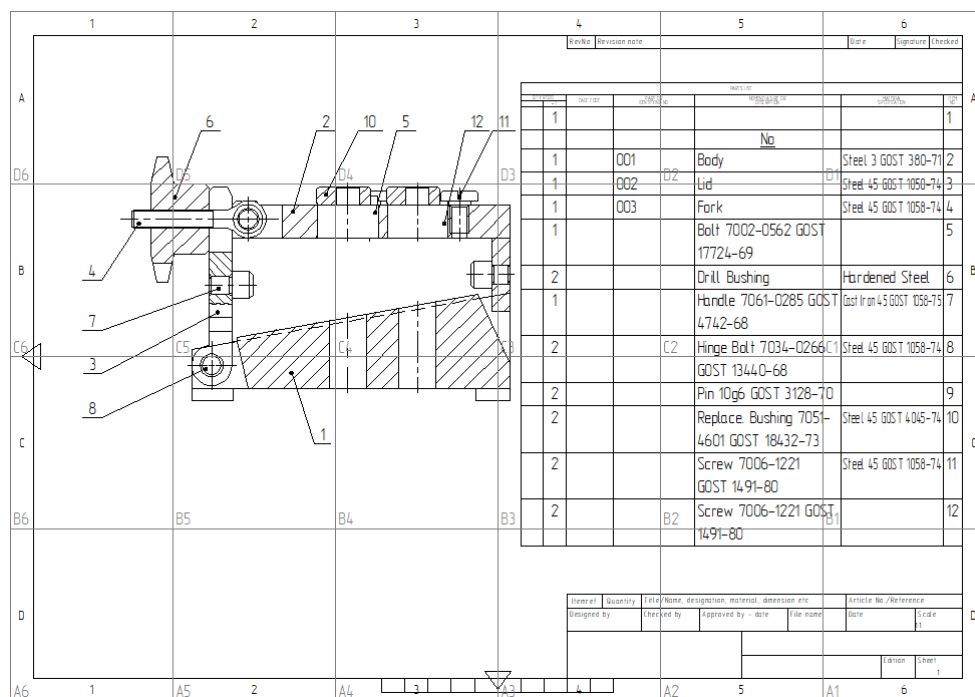
Spline command now supports a continuous curve input holding the left mouse button pressed, i.e. implemented is the feature of spline painting.



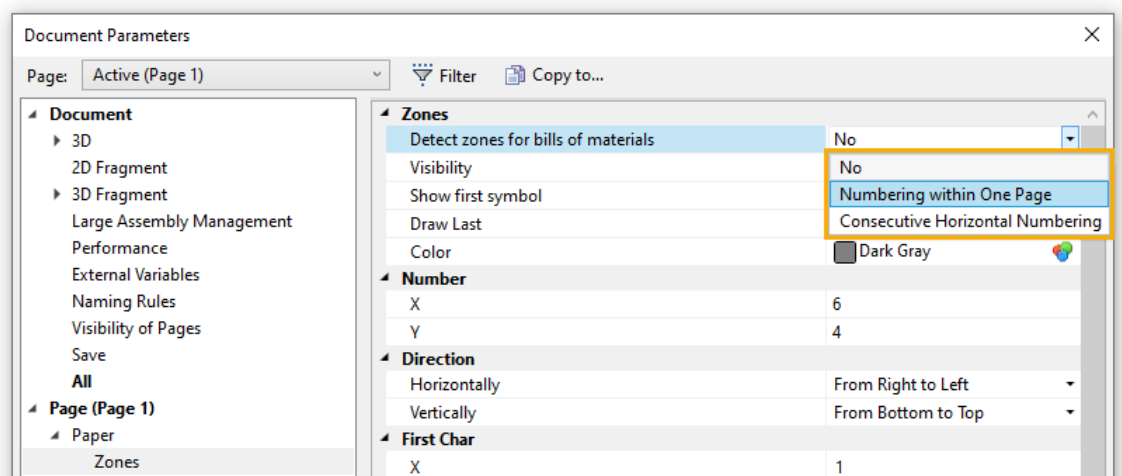
The spline points are added automatically according to the current image scale on the screen. Spline generation accuracy by the cursor movement is set in the **Options** dialog in the **Snap** tab. Accuracy is set in pixels, i.e. it depends on the current image scale in the working window.

Updating the Functionality of Drawing Zones

A lot of work has been done on the arrangement mechanism and zones detecting in the drawings.

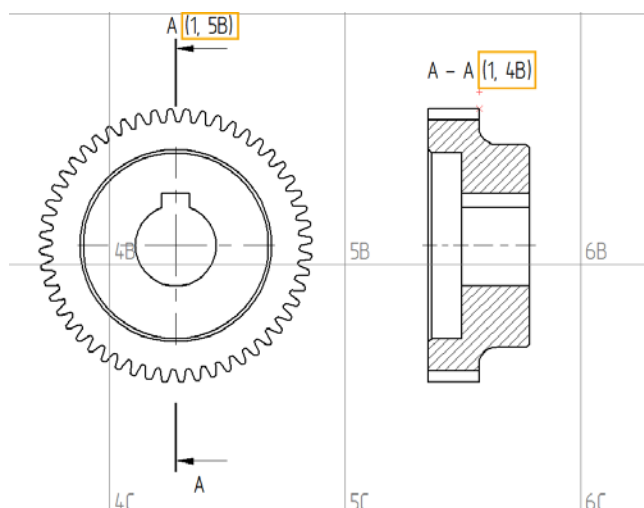
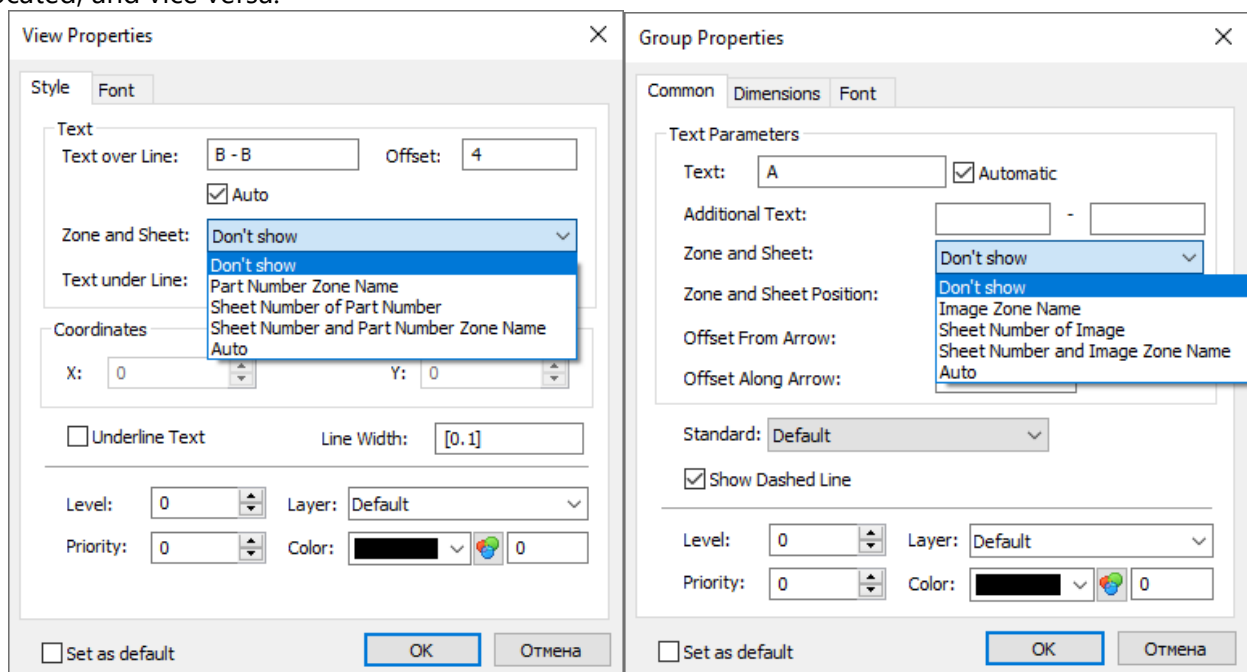


In the document parameters, you can now select numbering within one page or consecutive horizontal numbering as a detection of zones for bill of materials.

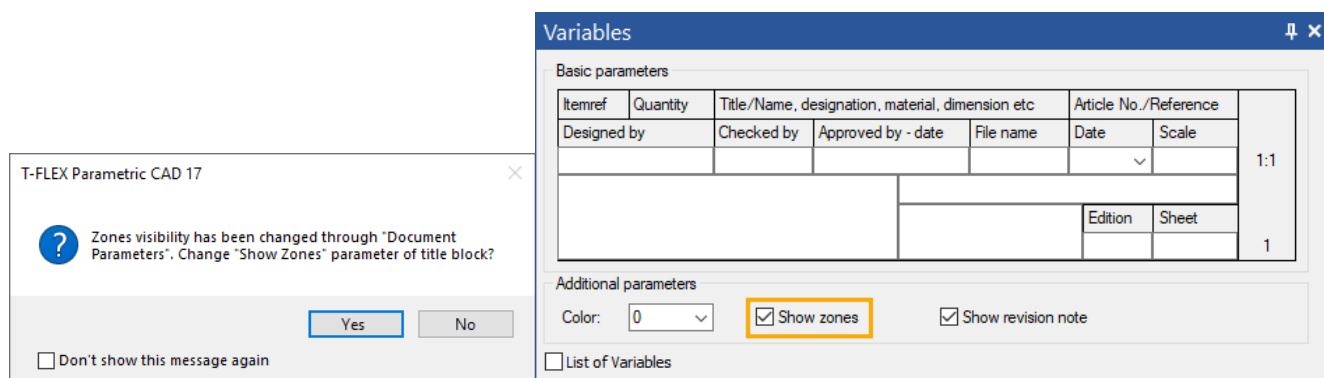


Also added the ability to change the page orientation without losing the division of the page into zones.

The designation of the view/cut/section can now display the name of the zone where the additional image is located, and vice versa:



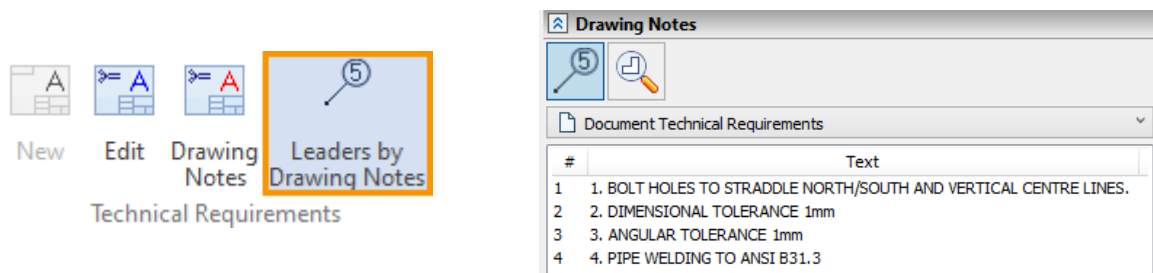
Also, synchronization between the division of the page into zones and zones of title blocks is implemented.



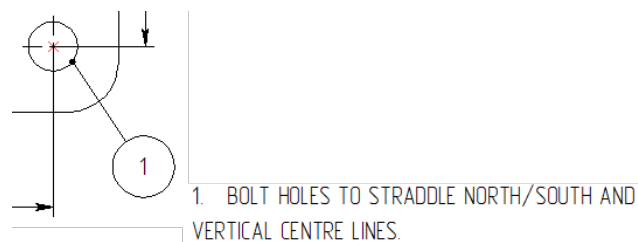
Also added the ability to change the page orientation without losing the division of the page into zones.

Notes on Technical Requirements

A feature of posting the notes on technical requirements in the document was added. It is similar to posting positions in the specification.



Technical requirement points selection is available from the command parameters:

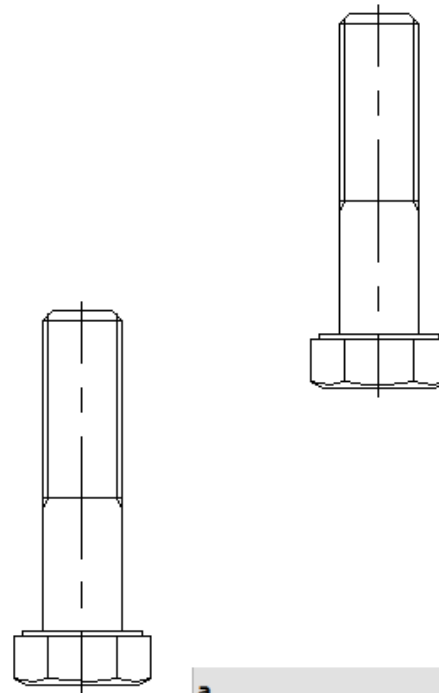
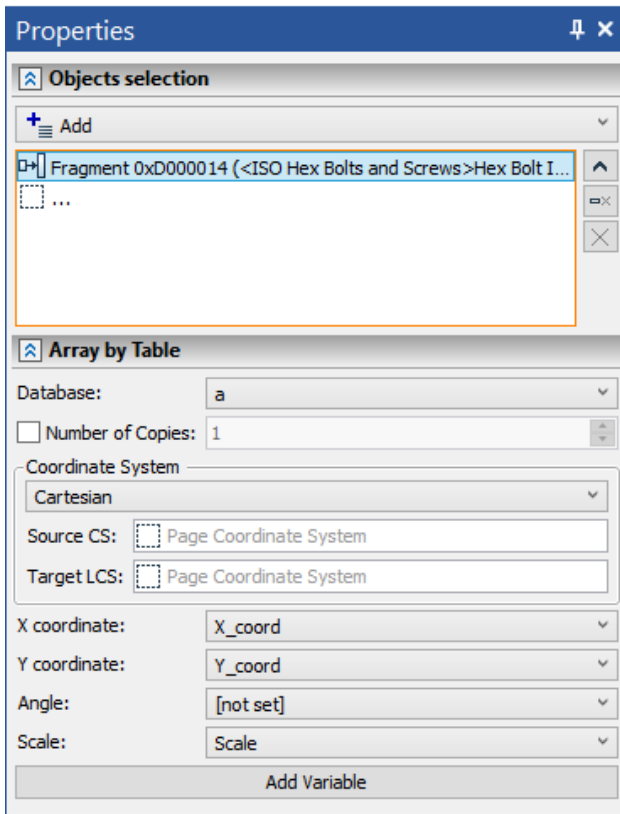


Technical requirements now have an indicator visible in the mode of showing non-printing characters.

- 1 → BOLT HOLES TO STRADDLE NORTH/SOUTH AND VERTICAL CENTRE LINES.¶
- 2 → ELEVATIONS ARE TO CENTRE LINE UNLESS OTHERWISE NOTED.¶
- 3 → DIMENSIONAL TOLERANCE 1mm¶
- 4 → ANGULAR TOLERANCE 1mm¶
- 5 → PIPE WELDING TO ANSI B31.3¶

2D Array by Table

Added a new command for creating 2D arrays **Array by Table**.



a				
Nº	X_coord	Y_coord	Angle	Scale
1	25	35	45	1

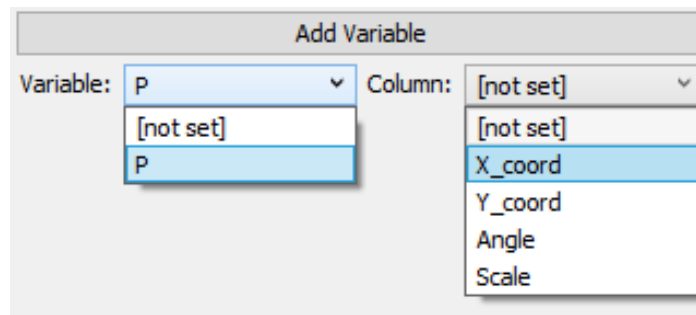
Using this command, you can use values from the database as array parameters.

There are 4 parameters for specifying an array according to the table:

- X coordinate
- Y coordinate
- Angle
- Scale

A prerequisite is to specify either X coordinates or Y.

The **Add Variable** button allows you to use the value of any model variable as the value of the database column.



Hyperlinks in Texts

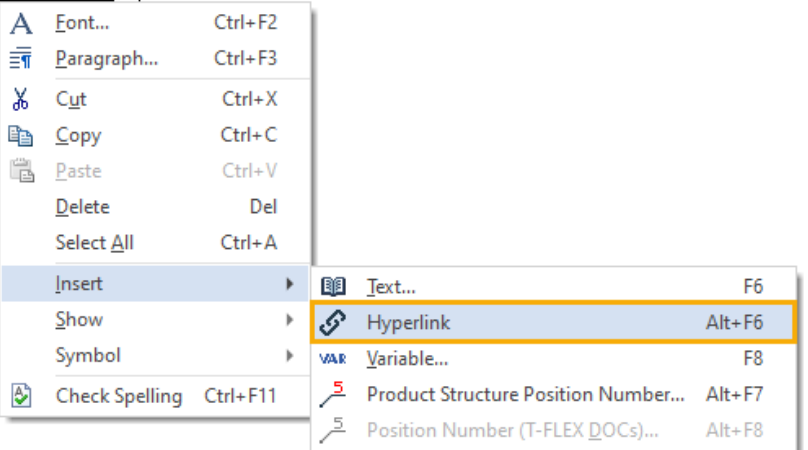
A command for creating hyperlinks has been added to the text editing context menu, allowing you to jump to CAD objects and pages, call macros, and also open external files and network resources (URL).

1. Applicable standards/specifications [ANSI Y14.5M-1982](#)







Dimension and Tolerances

2. Cast fillets to be [R0.9](#) and Corner Radii [R1](#) unless otherwise noted

3. * Dimensions according to standard









Hyperlink Parameters

Type:      

Address:

Text:







Hyperlink Parameters



Type:      

File:

Text:







Hyperlink Parameters



Type:      

Objects:  

Text:

Hyperlink Parameters

Type:      







Dimension:  

☒ Nominal

☒ Tolerance

☒ Deviations

Hyperlink Parameters

Type:      







Page:


First Corner: X: Y:

Second Corner: X: Y:

Text:

Hyperlink Parameters

Type:      

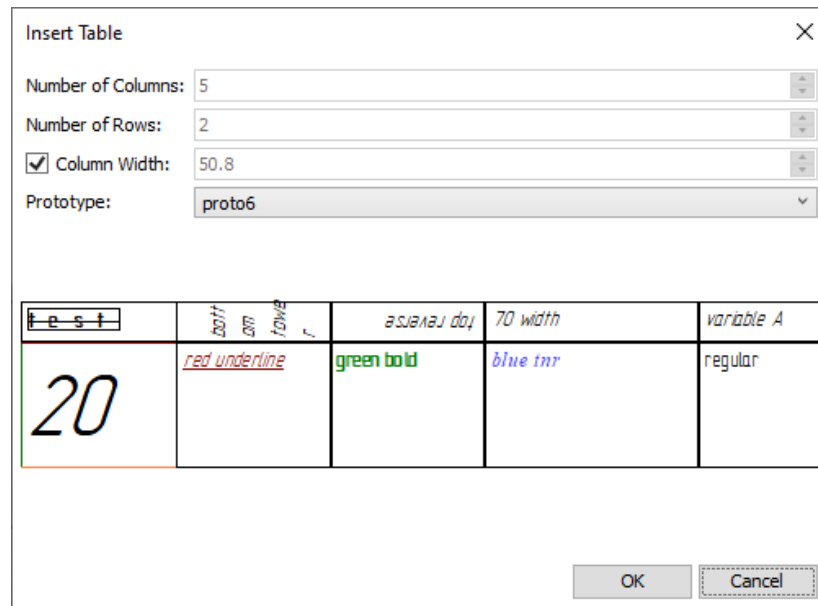
Macro: 

Text:

The link is clicked with the right mouse button while holding down the Ctrl button.

Insert Tables from Prototype

Added the ability to insert tables from a pre-created prototype.

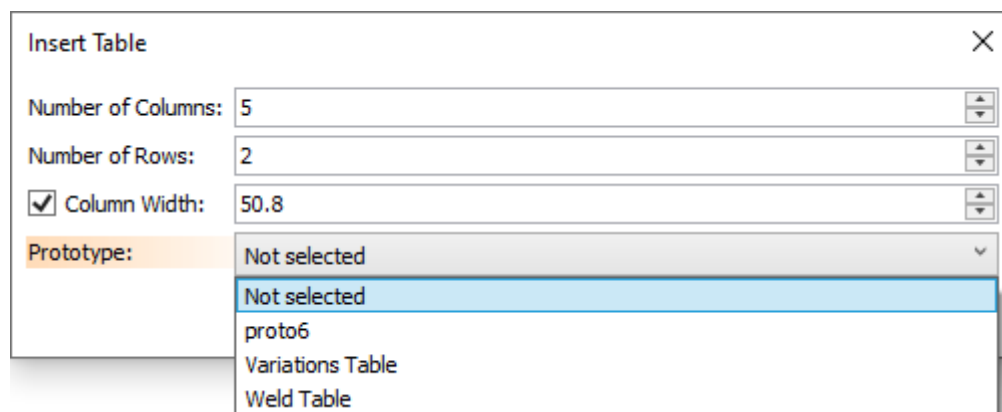


To create a prototype, you need to perform the following sequence of actions:

- Create a document containing a drawing view;
- Create table;
- Save the document in the table prototypes folder (C: \Program Files\T-FLEX Parametric CAD 17\Program\Template\Tables).

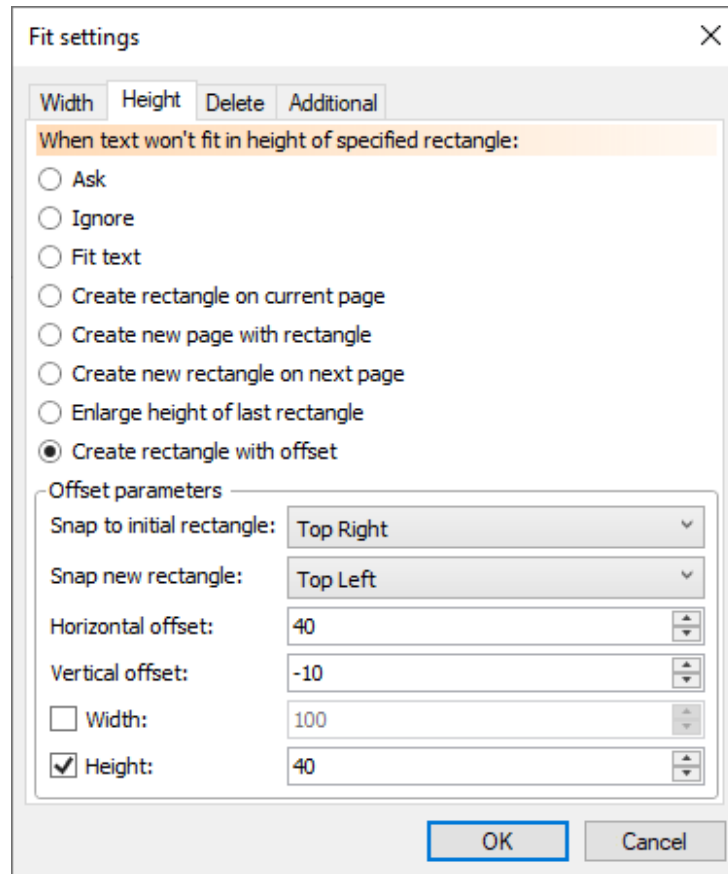
Only one table should be present in the generated document.

After the performed actions, when inserting a table in any document, the choice of a prototype will be available.

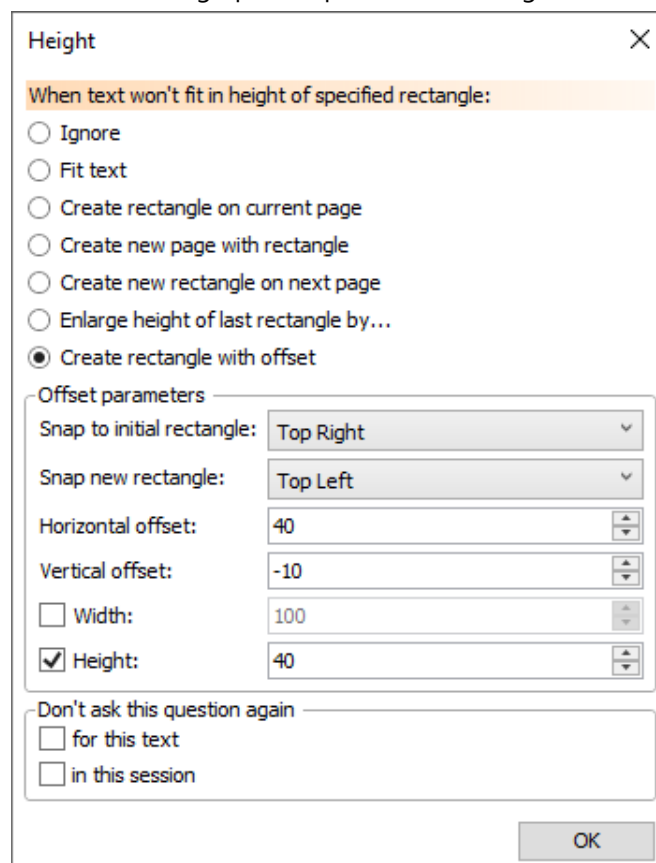


Paragraph-text

The **Create rectangle with offset** option is added to the list of options applicable, when paragraph-text doesn't fit in height of the rectangle:



Paragraph size parameters dialog

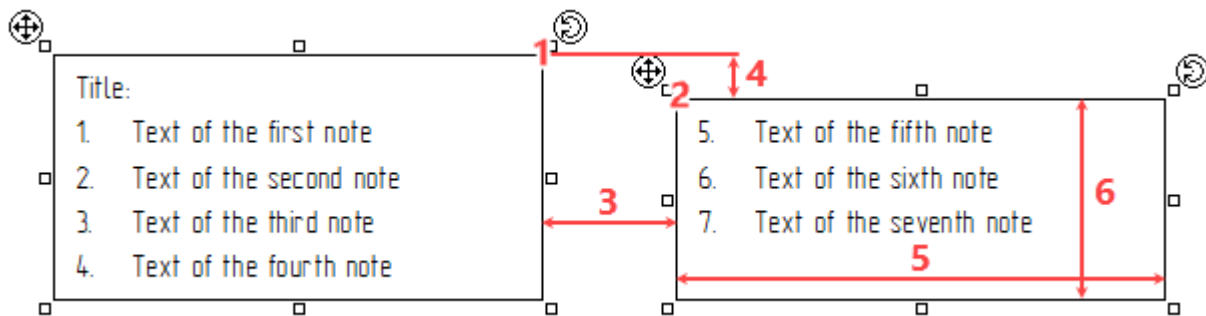


The dialog invoked, when paragraph-text doesn't fit in height of the rectangle, if the **Ask** option is applied in the paragraph size parameters dialog

This option makes the system automatically create a new rectangle, when the text doesn't fit in the initial one. A new rectangle is positioned by defining offsets of one of the angles of a new rectangle in relation to one of the angles of the initial rectangle (**Bottom Left, Top Left, Top Right, Bottom Right**).

Following parameters are available:

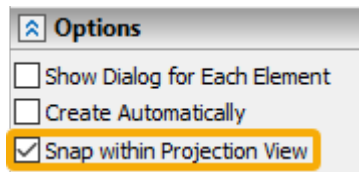
- **Snap to initial rectangle**
An angle of the initial rectangle, which will define a position of a new one.
- **Snap new rectangle**
An angle of a new rectangle, which is positioned in relation to an angle of the initial rectangle defined above.
- **Horizontal offset** and **Vertical offset**
Offset of a new rectangle from the initial one.
- **Width** and **Height** of a new rectangle
By default, dimensions of a new rectangle are equal to dimensions of the initial rectangle. So, dimension values are displayed in these input boxes but can not be edited. In order to type in custom value, enable the checkbox located to the left side of a corresponding parameter's name.



The result of creating a new rectangle with parameters shown on the previous image

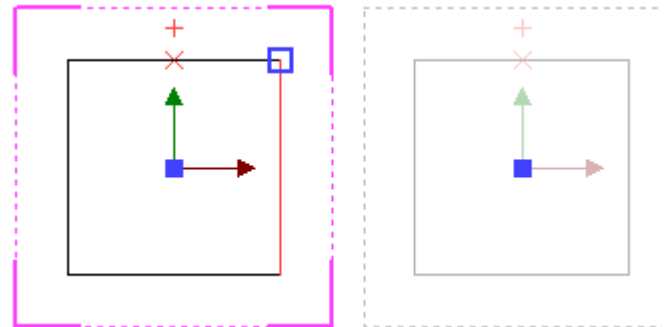
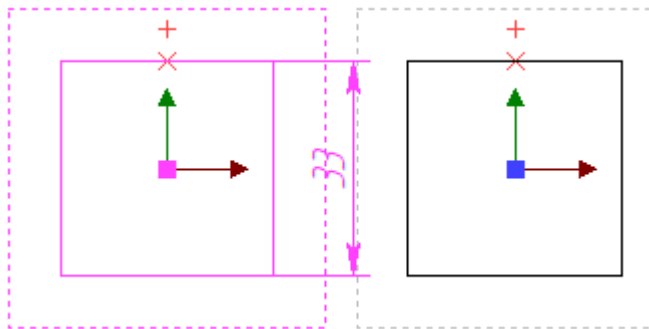
- 1 - **Top Right** angle of the initial rectangle
- 2 - **Top Left** angle of the new rectangle
- 3 - **Horizontal offset** of the new rectangle from the initial one (40 mm)
- 4 - **Vertical offset** of the new rectangle from the initial one (-10 mm)
- 5 - **Width** of the new rectangle is defined by the width of the initial one (100 mm)
- 6 - **Height** of the new rectangle is defined by the custom value (40 mm)

Snapping of Dimensions and Leader Notes



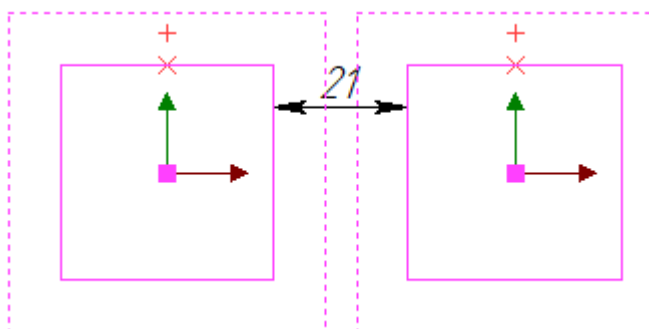
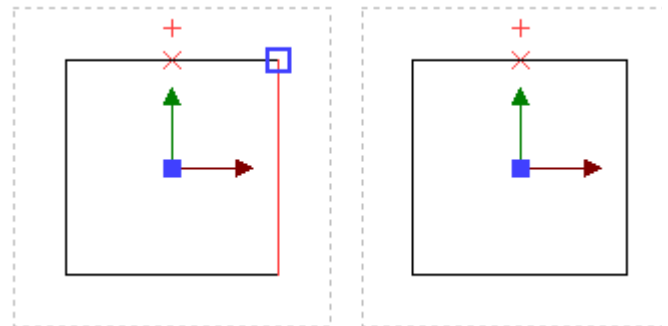
The **Snap within Projection View** checkbox is added to the **Options** tab of **Dimension** and **Leader Note** commands' parameters windows.

The checkbox is enabled by default, so a drawing view gets activated upon selecting a first reference element.



In result, **Dimensions** and **Leader Notes** are created within a single projection and selecting reference elements on other projections is impossible.

If the checkbox is disabled, then a drawing view doesn't get activated upon selecting a first reference element, so you can select a second reference element on another projection.



In result, created **Dimensions** and **Leader Notes** belong to the whole drawing page, not to a particular projection.

This option can be also disabled by holding **<Shift>** while selecting a first reference element.

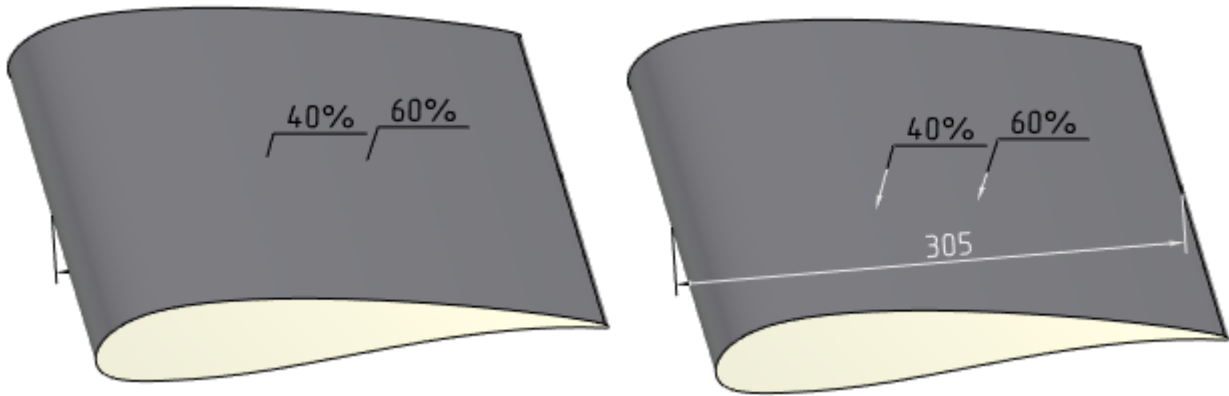
Keep in mind, that values of dimensions created without snapping on projections with breaks do not take "broken" part of the model into account.

When dimension values do not represent actual dimensions of a model due to disabled snapping, info messages are displayed in the **Diagnostics** window:

Diagnostics			
#	Message	Element	
1	Nominal value must be set manually. Dimension is snapped to different projections or falls into break area.	Linear Dimension...	
2	Nominal value must be set manually. Dimension is snapped to different projections or falls into break area.	Linear Dimension...	

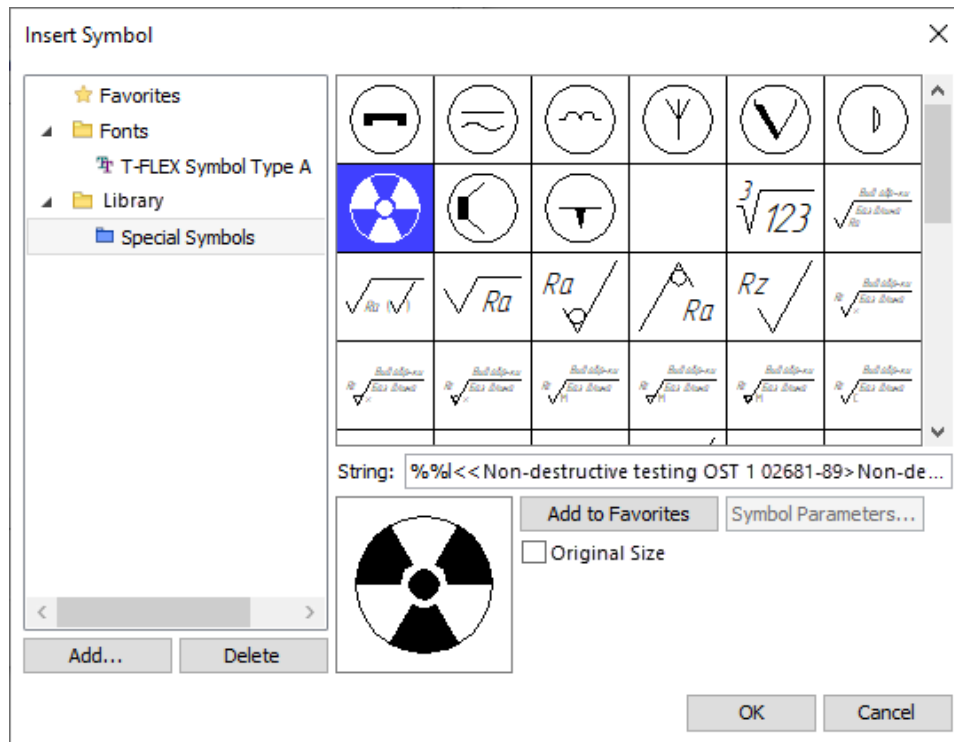
3D Annotations Display

The display of 3D annotations (dimensions, leader notes, roughness and GD&T symbols) hidden behind non-transparent bodies and surfaces, can now be enabled (**Options > 3D > 3D > Display annotations over 3D model**).



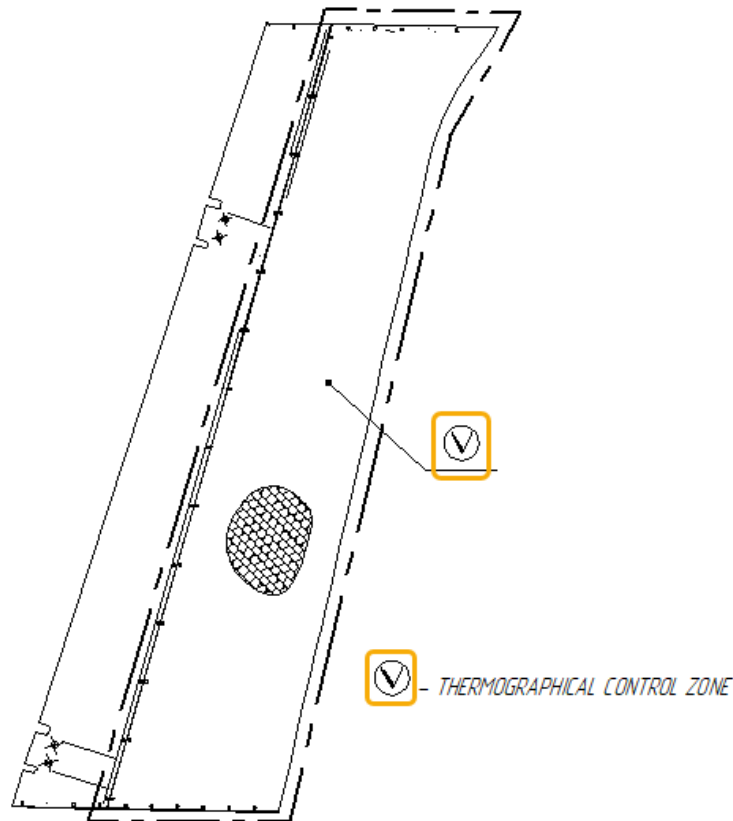
Symbols Insertion



The old symbols insertion dialog has been replaced by the new one:



New dialog supports custom symbol libraries.

Custom symbols can be inserted into string text, as well as into textual input boxes of various detailing elements (e.g.: dimensions and leader notes).



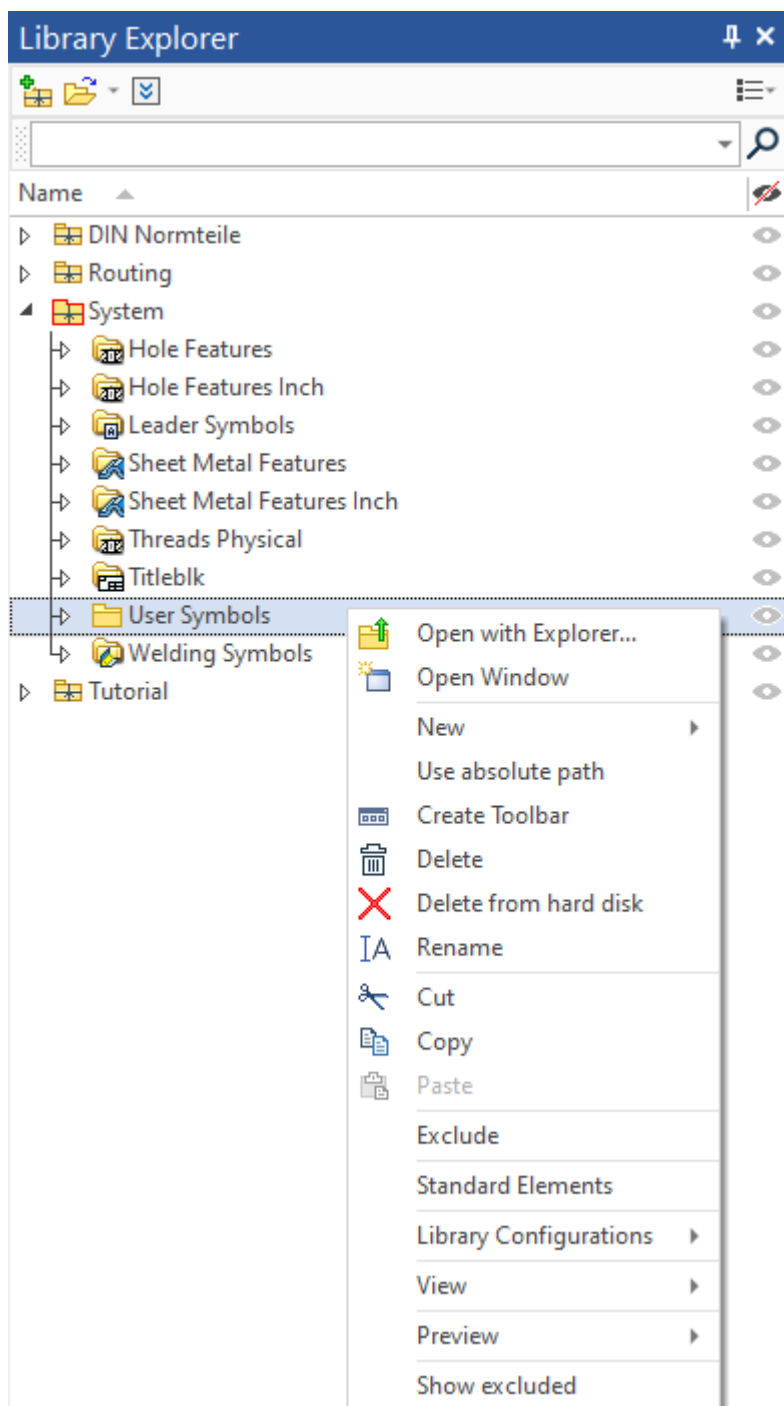
In order to invoke the dialog, click  the desired textual input box in parameters window or parameters dialog, then either press <Alt> + <F9>, or click  within the input box to invoke its contextual menu and then select the **Insert Symbol...** item.


In order to use custom symbols, first you have to create a custom library, using the standard T-FLEX CAD procedures. A library can be organized in one of the following ways:

- Each symbol is created in a separate document;
- All symbols are created in the same single document, each symbol is created on a separate page.

When creating each symbol in a separate document, you can add custom dialog of symbol's external variables. Such dialog can later be invoked from the symbols insertion dialog using the **Symbol Parameters...** button, in order to customize values of inserted instance's variables.

Next you have to add the created library into the **System** configuration and assign the **Symbols** attribute to the library in its contextual menu.




Then the folder of the library can be added into the tree shown in the left side of the symbols insertion dialog. In order to add a folder, click the **Add...** button and select path to the desired folder using the standard Windows Explorer interface. Elements of the tree can be selected by clicking . In order to delete a folder, select it in the tree and click the **Delete** button.


The tree contains following sections:

- **Favorites**
You can add particular symbols here, both standard and custom.
- **Fonts**
Contains standard set of symbols of the font selected for symbols in the document parameters (**Document Parameters > Page > Symbols > Characters > Font**). The default font is **T-FLEX Symbol Type A**. If you change the font in document parameters, the font of previously inserted symbols will also change.
- **Library**
Contains symbol libraries supplied together with the T-FLEX CAD and user-added folders.

Upon selecting a folder in the tree, the list of its symbols is displayed in the upper right side of the dialog.

Symbols can be selected in the list by clicking .

Following elements are displayed in the lower right side of the dialog:

- **String**
Code of the selected symbol. Instead of the symbol itself, the textual input boxes of the T-FLEX CAD interface will contain this code. The symbol itself is displayed only in 2D and 3D windows..
- **Preview of the selected symbol**
- **Add to Favorites**
Adds the selected symbol into the **Favorites** section of the tree. In order to delete a symbol from favorites, double click it with .
- **Symbol Parameters...**
Invokes the dialog of selected symbol's external variables.
- **Original size**
This checkbox is disabled by default, so the size of symbols is defined by target text's font parameters. If you enable this checkbox, symbols will be inserted either using the size defined in the external variables dialog, or using the size applied in symbol's document.
- **Ignore Font Angle**
This checkbox replaces the Initial size checkbox for symbols from the standard set. The checkbox is enabled by default, so the symbol is inserted without applying the italic slope regardless of target text's parameters.

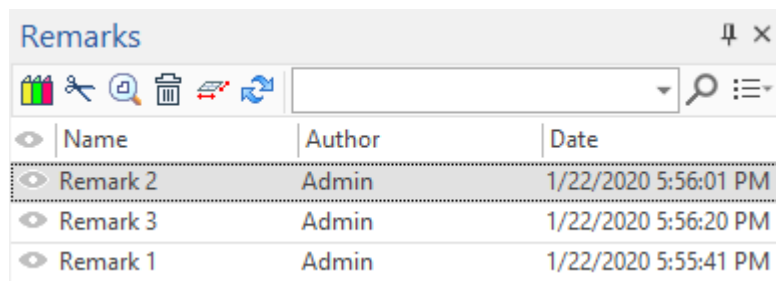
Select the desired symbol, customize its parameters (if necessary) and click **OK**. The symbol will be inserted.

New Mechanism for Remarks

The development of 3D model remark mechanisms is a transition to a new way of working with design documentation without drawing design and extra papers. The designer can indicate all technical requirements and comments directly on the 3D model, and then pass it to the executive or for revision. For the convenience of managing extended remarks, a special window for working with comments has been developed. Comments can be created using the remarks tool: dimensions, labels, texts, etc. You can use a spline drawing to simulate a handwritten label.

"Remarks" Window

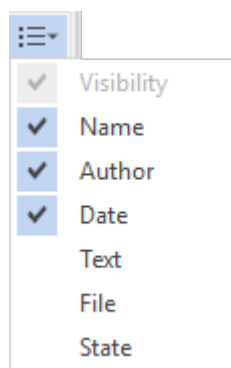
To work with remarks: create, delete, and edit notes, you need to use a special **Remarks** window.



A set of icons at the top of the window enables you to manage comments.

	New Remark
	Edit
	Show
	Delete
	Create Remark Plane
	Update

The window enables you to search for and configure the displayed columns.



Visibility. The first column of the window is always available. Click the icon to determine the visibility of the remark in a 3D scene or drawing.

Name. Name of comment.


Author. Author of comment.

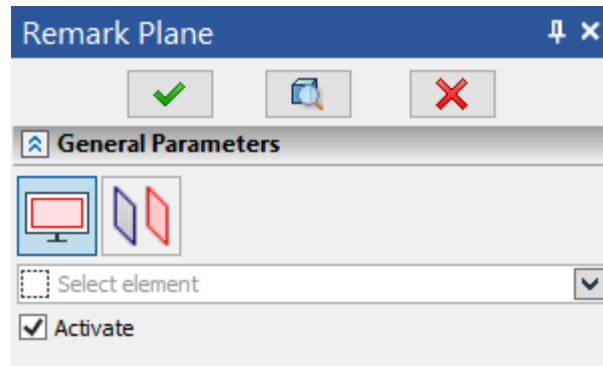
Date. Date of comment creation.

Text. A remark object is a graphical object, so it is sometimes convenient to explain it with text.

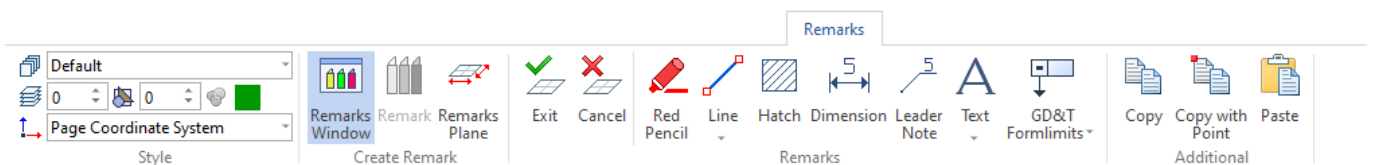
File. Specifies the location and name of the file containing the remark. From the point of view of CAD system objects, a remark is an object created in a context. By default, the remark file is saved to the folder where the file to which the comments were created is located.

Creating Comments

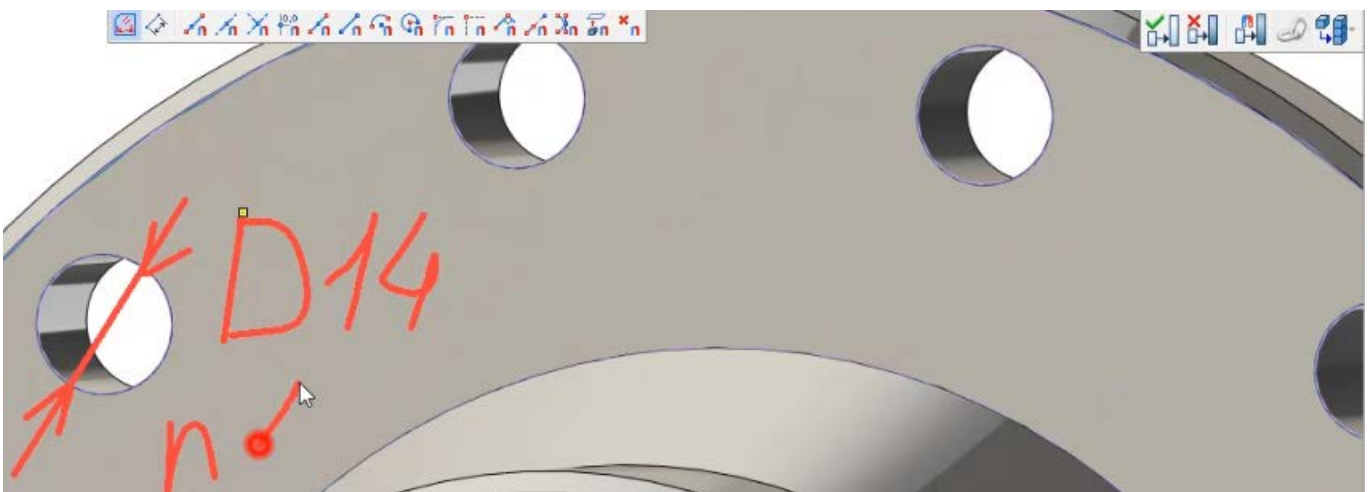
To create a new comment, use the icon  in the **Remarks** window. The system automatically activates the dialog for selecting the plane on which the remark will be created.



The **Remarks** tab automatically becomes active in Ribbon.



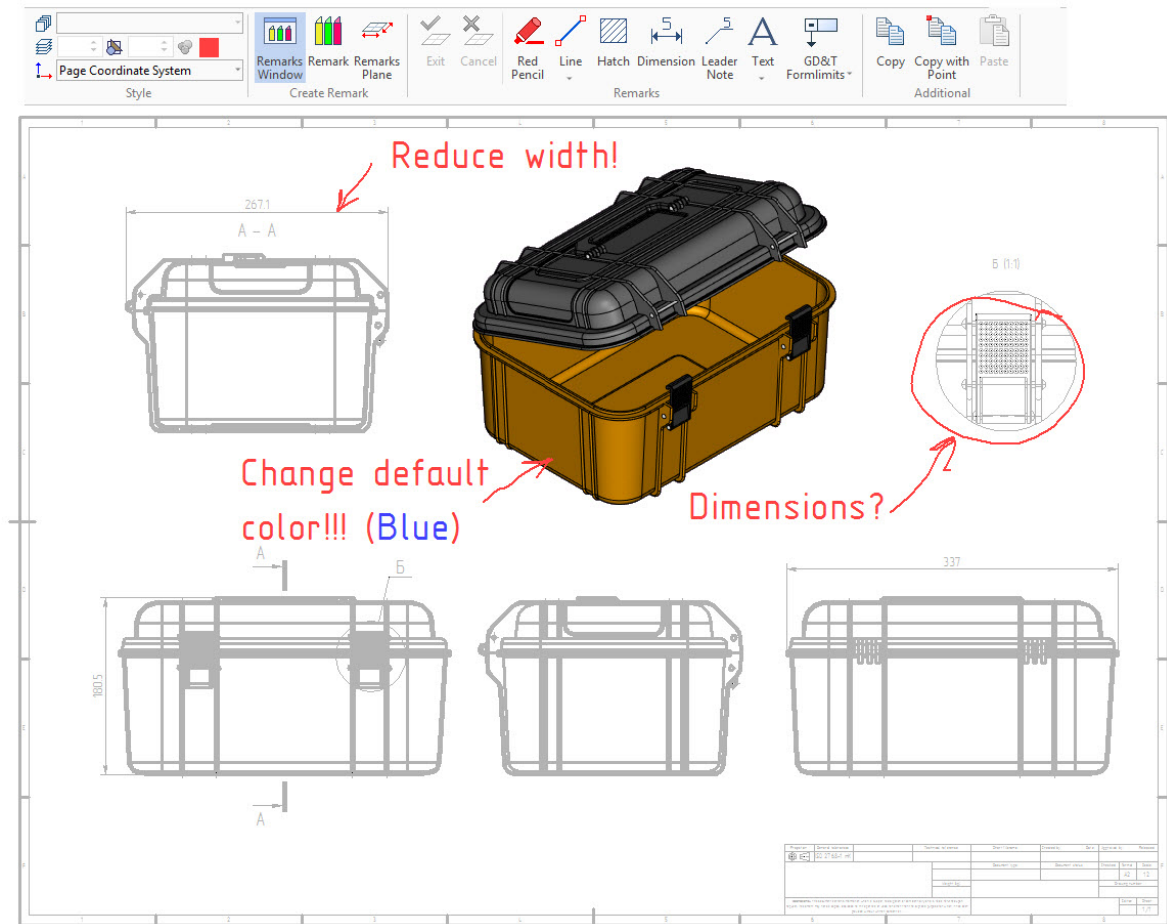
Remark and sketch tools are available on the tab. The combination of remark elements and sketch elements enables you to create a comment on a model or drawing in a free, hand-drawn style. For example, in the spline drawing mode (**Red Pencil** command), you can create a remark that indicates a defect in the model, on the selected remark plane.



After creating a comment, you need to exit the context. As mentioned above, a comment is an object created in the context and written to a special file. To exit the context, you need to use the panel for working in context.



You can also create remarks in drawings.



If a remark is created in the integration mode with T-FLEX DOCs, the remarks are saved as T-FLEX DOCs objects without using files representing a remark fragment.

Remarks in the T-FLEX DOCs Integration Mode

When working with T-FLEX DOCs data, it may be necessary to peer review system objects. For example, in the process of document correction and approval, employees may need to initiate the process of issuing remarks on the document, and the department head may give tasks to his employees to create remarks. Thus, the T-FLEX DOCs remarks management mechanism helps organize teamwork with system data in terms of their approval and peer review.

The remark management mechanism of T FLEX DOCs allows you to create remarks for system objects, as well as generate requests for creating remarks. Remarks can be created for any system objects, including T-FLEX CAD files.

Creating remarks for T-FLEX CAD files in integration mode is done through the viewing toolbar tools. You must select the object associated with the T-FLEX CAD file, or the file itself, and open the **View** tab on the property panel.

Product Composition

In the new version of T-FLEX CAD, **Product Structure** is renamed to **Product Composition**.
The option **Concatenate Unique Values with Quantity** in the column of summing values when merging has been added.

Column Properties

Properties

Name:Format

Synonym Name:

Category:

Type:String

Unit:None

Column Width:Auto

☐ Show column in Product Composition window

☐ Group by column

☒ Sum values when merging

Concatenate Unique ValuSummationConcatenate stringsConcatenate unique stringsConcatenate Unique Values with Quantity

Value

Default Value:

List of values:

Number format

☐ Enabled

☐ Trim trailing zeros

Precision:6

Data Assignment

Format

OK

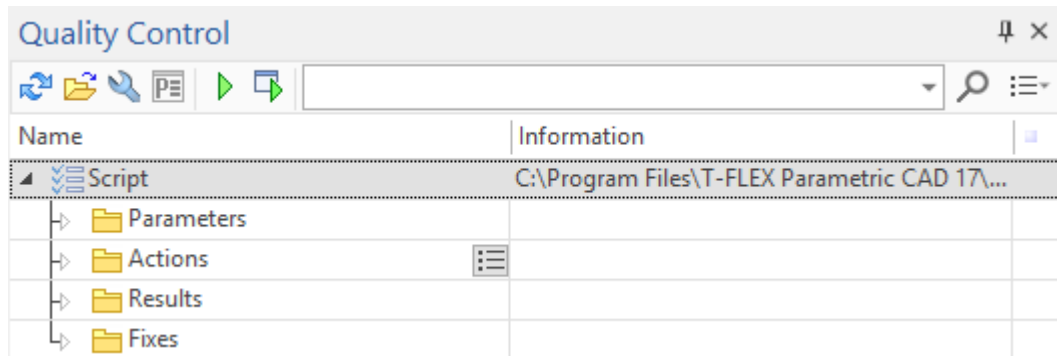
Cancel

This option sums the unique values based on the count when string concatenated.

Product Composition

Quality Control - Checking Models and Drawings

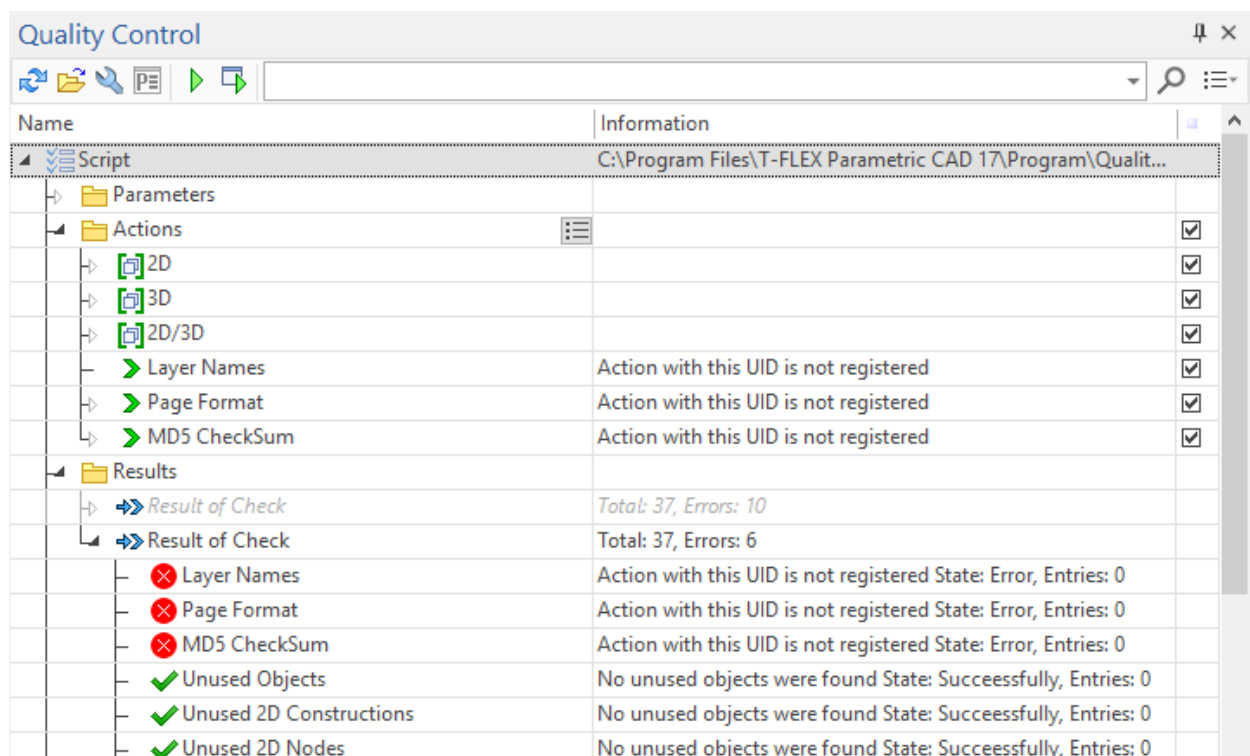
A new tool for automatic design tool check and file security checks in secret enterprises was added: **Quality Control**. In a new window of the system, you can call a special extension file "*.tfqms" - a verification script that contains control methods and controlled parameters. Each company can develop its own verification algorithm or use a standard one.



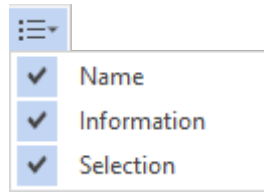
A special **Quality Control** window is used to display test results, set test parameters, load and execute test scripts. All actions for the checking of models depend on the scenario. The script can automatically check fonts, the thickness of lines, dimensions, technical requirements, absence of "adjusted dimensions" when the nominal value is corrected manually, absence of intersection of bodies, absence of model and variable recalculation errors, etc. In addition, the script can automatically check files for hiding confidential information, which is necessary for the security service. You can check for hidden texts, layers, and working planes, check for encrypted texts in element names, and check for "extra", unused structures.

New "Quality Control" Window

The **Quality Control** window is used for working with quality control scripts. The tree-view window displays the quality control script, enables you to run the script, and displays the result of the check.



The following columns are available in the window.







Name. Displays the name of the script, action, result, and any other script object.

Information. Provides an explanatory comment on the action, displays the result of the action and corrections.

Selection. The column contains a special field next to each action. Setting or removing the flag in the field determines whether the action will be performed or not.

The following commands are available in the window.

	Update
	Open Script
	Settings
	Script Parameters
	Run Script
	Run Script for Folder

A search is available. The search is similar to the search in other windows.

Quality Control Script

The quality control script is a file in the ".tfqms" format where the test set is registered. This file is an XML program code for calling model validation methods through a special library based on the T-FLEX CAD Open API.

```
<Action Name="Unused Objects" Uid="e61cc80f-4f0b-4e02-bd6c-f6fcb97b3581" LogicUid="e64f240d-dd59-4175-af2e-ed6d0e7f1e84">
  <Parameters>
    <Parameter Key="Point 3D" Name="3D Points" Type="Bool" Value="true"/>
    <Parameter Key="Path 3D" Name="3D Paths" Type="Bool" Value="true"/>
    <Parameter Key="Profile 3D" Name="3D Profiles" Type="Bool" Value="true"/>
    <Parameter Key="LCS 3D" Name="LCSs" Type="Bool" Value="true"/>
    <Parameter Key="Workplane 3D" Name="Workplanes" Type="Bool" Value="true"/>
    <Parameter Key="Knot 2D" Name="2D Nodes" Type="Bool" Value="true"/>
    <Parameter Key="Draw 2D" Name="Construction Lines" Type="Bool" Value="true"/>
    <Parameter Key="Page 2D" Name="Pages" Type="Bool" Value="true"/>
    <Parameter Key="Invisible Variable" Name="Invisible Variables" Type="Bool" Value="true"/>
    <Parameter Key="Visible Variable" Name="Visible Variables" Type="Bool" Value="true"/>
    <Parameter Key="Named 2D" Name="Named 2D Elements" Type="Bool" Value="true"/>
    <Parameter Key="Default 2D" Name="2D Constructions" Type="Bool" Value="true"/>
    <Parameter Key="Unknown" Name="Others" Type="Bool" Value="true"/>
    <Parameter Key="Delete Named 2D" Name="Delete Named 2D Elements" Type="Bool" Value="true"/>
    <Parameter Key="Delete On Active Page Only" Name="Only on Active Page" Type="Bool" Value="true"/>
  </Parameters>
</Action>
```

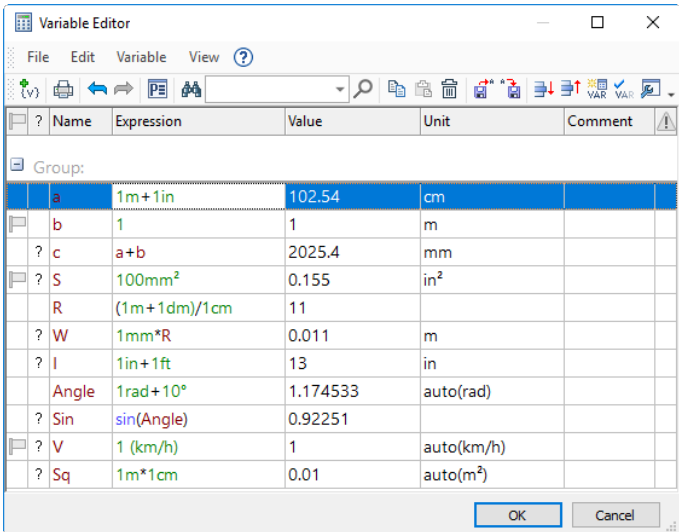
The standard quality control script can be independently expanded or adjusted by the user, or written again, taking into account the rules for calling the loaded library and displaying data in the **Quality Control** window.

Variable Editor

The variable editor has become even more convenient. Now, when the wrong separator of the integer and fractional part is entered, it is corrected automatically. For each variable, you can specify a measurement unit, and the system automatically checks the correctness of measurement units in the calculated variables. Finding variables has become easier thanks to the quick search field. You can now borrow variables from other fragments directly from the editor.

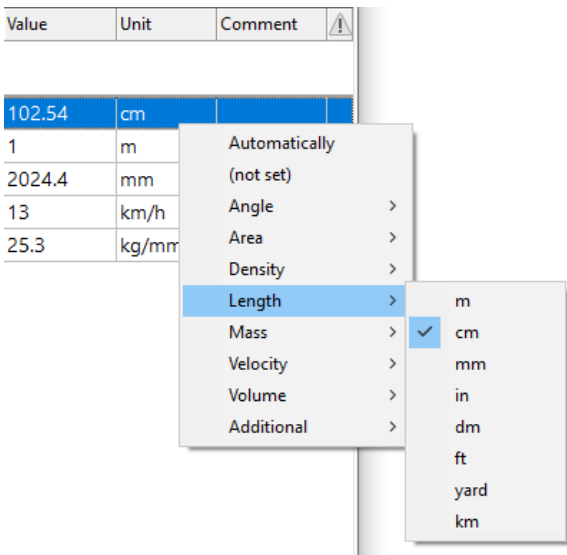
New "Unit" Parameter

The **Unit** column was added to the variable editor.



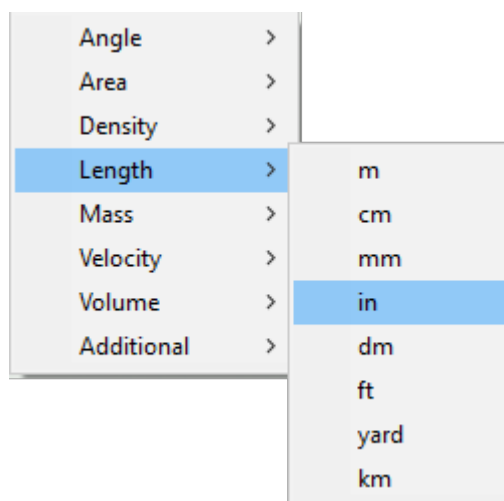
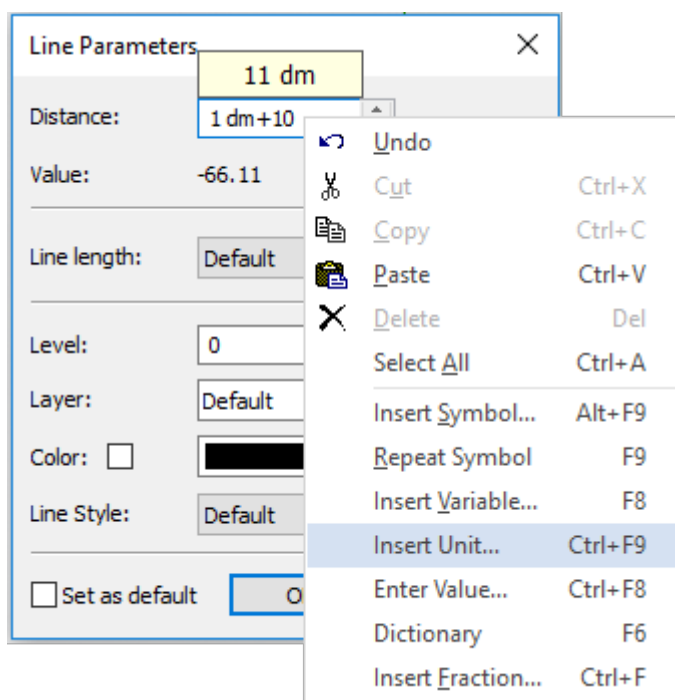
	Name	Expression	Value	Unit	Comment
	a	1m+1in	102.54	cm	
	b	1	1	m	
?	c	a+b	2025.4	mm	
?	S	100mm ²	0.155	in ²	
	R	(1m+1dm)/1cm	11		
?	W	1mm*R	0.011	m	
?	I	1in+1ft	13	in	
	Angle	1rad+10°	1.174533	auto(rad)	
?	Sin	sin(Angle)	0.92251		
?	V	1 (km/h)	1	auto(km/h)	
?	Sq	1m*1cm	0.01	auto(m ²)	

The column indicates the measurement unit for each variable. The following values are available: **not set**, **Automatically**, or a measurement unit. In the case of an **Automatically** value, the variable measurement unit is set according to the expression. For all the variables of previous versions, the parameter is set to **not set**. Clicking in the **MU** column field enables you to change the measurement unit of a variable using the context menu.

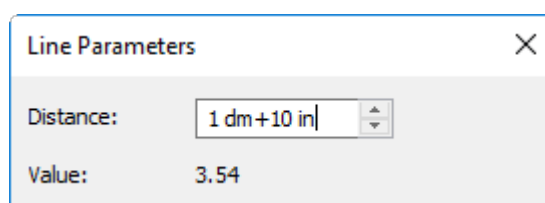


Support for working with measurement units was added to the syntax of variable expressions. The measurement unit in an expression is specified by an abbreviated name, using Latin or Cyrillic letters. If the name contains "/" or parentheses, it must be enclosed in parentheses. When calculating the values of expressions, the unit conversion is used, where possible.

The **Paste the measurements unit** item was added to all the value input fields, supporting variables. When you use it, a menu with the list of measurement units appear.



The selected measurement unit is added to the expression.

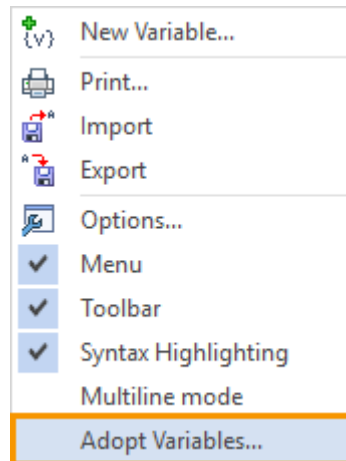


When calculating expression values, the compatibility of measurement units is checked. If there is a discrepancy, the system displays a warning message in the diagnostics column.

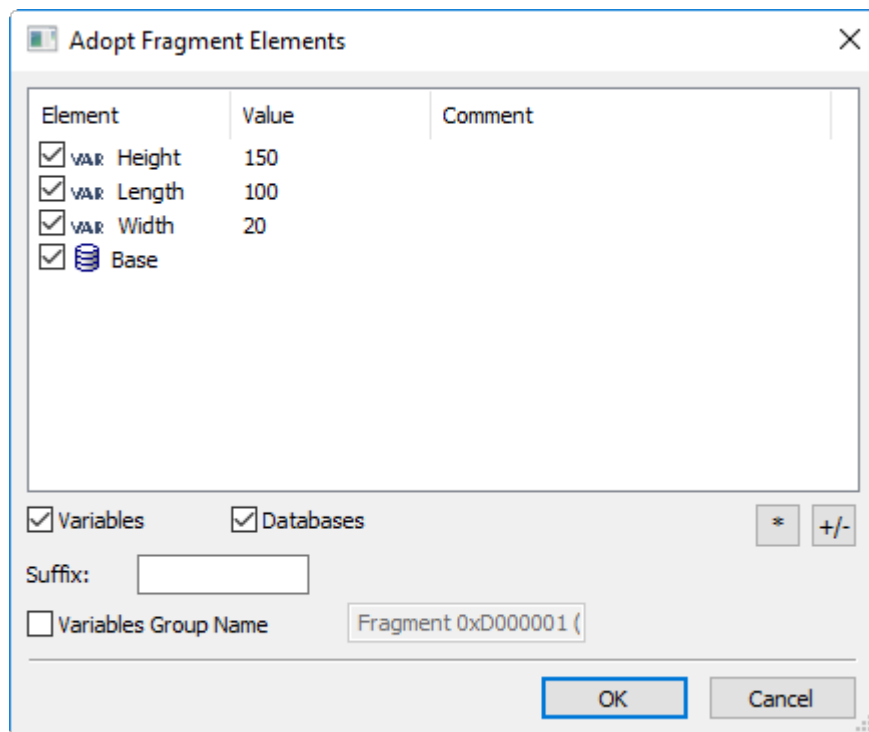
When using variables that have a measurement unit set as a parameter for model elements, the value is automatically converted to the model units of measurement.

Variables Borrowing

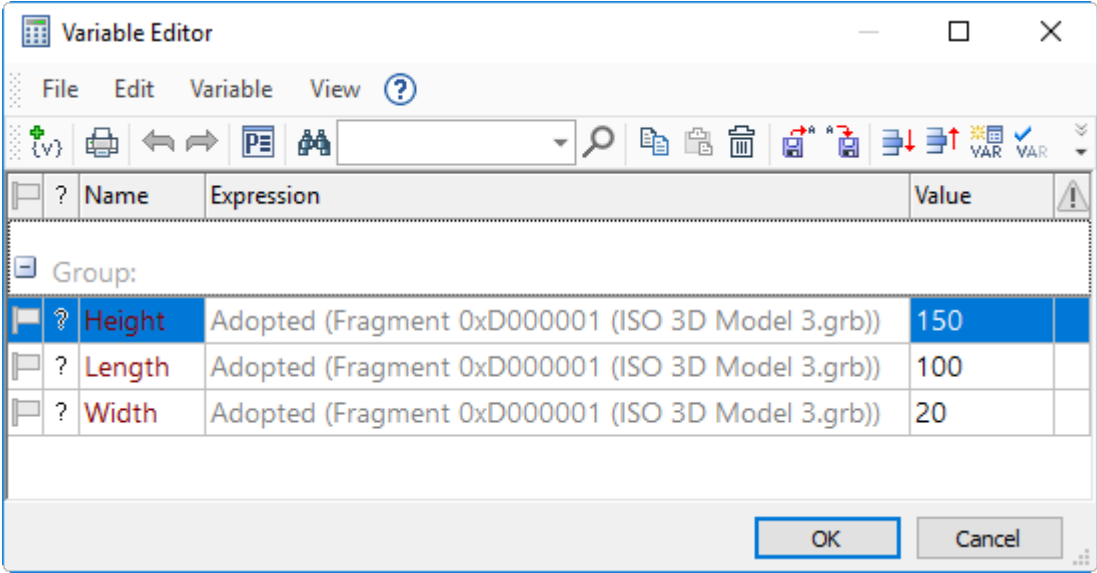
The **Adopt Variables** command was added to the variable editor. The command is called from the editor context menu.



This command enables you to select a file from which you can select variables and databases to borrow. The command can be called from the context menu of the variable editor. When you call the command, a dialog appears where you should specify the variables and databases that you want to borrow.



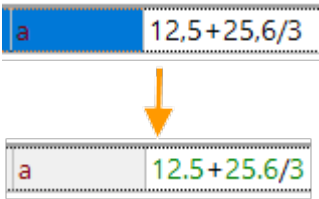
The values of the borrowed variables will depend on the values of the variables in the fragment from which they were borrowed. The file from which the variables were borrowed does not have to be inserted as a fragment in the assembly. Borrowed variables and databases will be marked as borrowed with the name of the source fragment.



Base Adopted (Fragment 0xD000001 (ISO 3D Model 3.grb))				
Nº	Column1	Column2	Column3	
1	0	0	0	

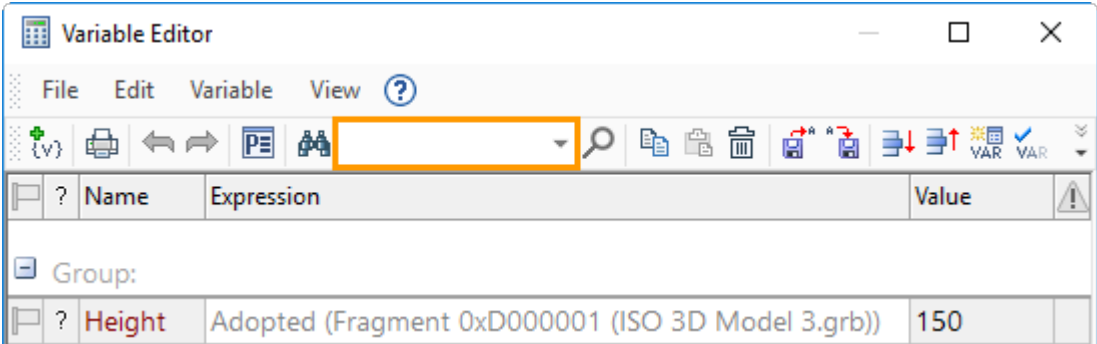
Separator Automatic Replacement

The automatic replacement of comma to a point as the decimal separator for the integer and fractional parts of a number was added. The separator in expressions must be a point. An algorithm that automatically corrects the user's error when writing expressions was added: if a comma was placed instead of the point separator, an automatic replacement will be performed.



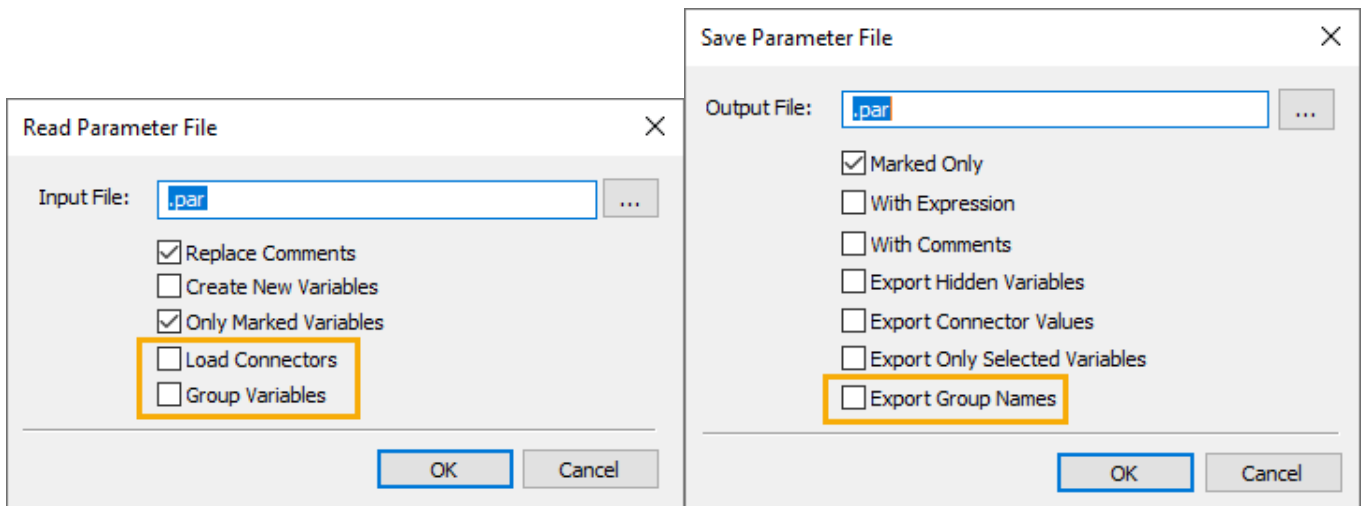
Quick Search For Variables

A quick search field was added to the variable editor. A search can be performed by any column of the variable editor: variable name, expression, value, etc.



Extension of the Variables Import/Export Functional

Now, when exporting variable values to a file, the ability to load connectors and group variables has been added. When importing variable values to a file, the ability to display group names is also added.



Conversion of Variable's Units of Measurement

Added the **tounit(,)** function, which allows to convert variable's units of measurement.

The function has two arguments: a variable, whose value (with a set or not set unit) will be converted, and a target unit. The target unit can be defined either as a text string (eg "mm"), or as another variable, whose unit will be used.

If units of the first argument are compatible with the target unit (i.e. they have the same type), then the conversion to the target unit is performed. For example "kg" -> "t".

If units are not compatible, then the the target unit is assigned without conversion. Unit incompatibility error is not generated.

Flag	Name	Expression	Value	Unit
Group:				
Flag	value_1	123 kg	123	auto(kg)
Flag	value_2	456 g	456	auto(g)
Flag	value_3	1 t	1	auto(t)
Flag	\$unit	"r"	r	
	example_1	tounit(value_1,value_3)	0.123	auto(t)
	example_3	tounit(value_2, "t")	0.000456	auto(t)
	example_4	tounit(value_1,\$unit)	123	auto(kg)

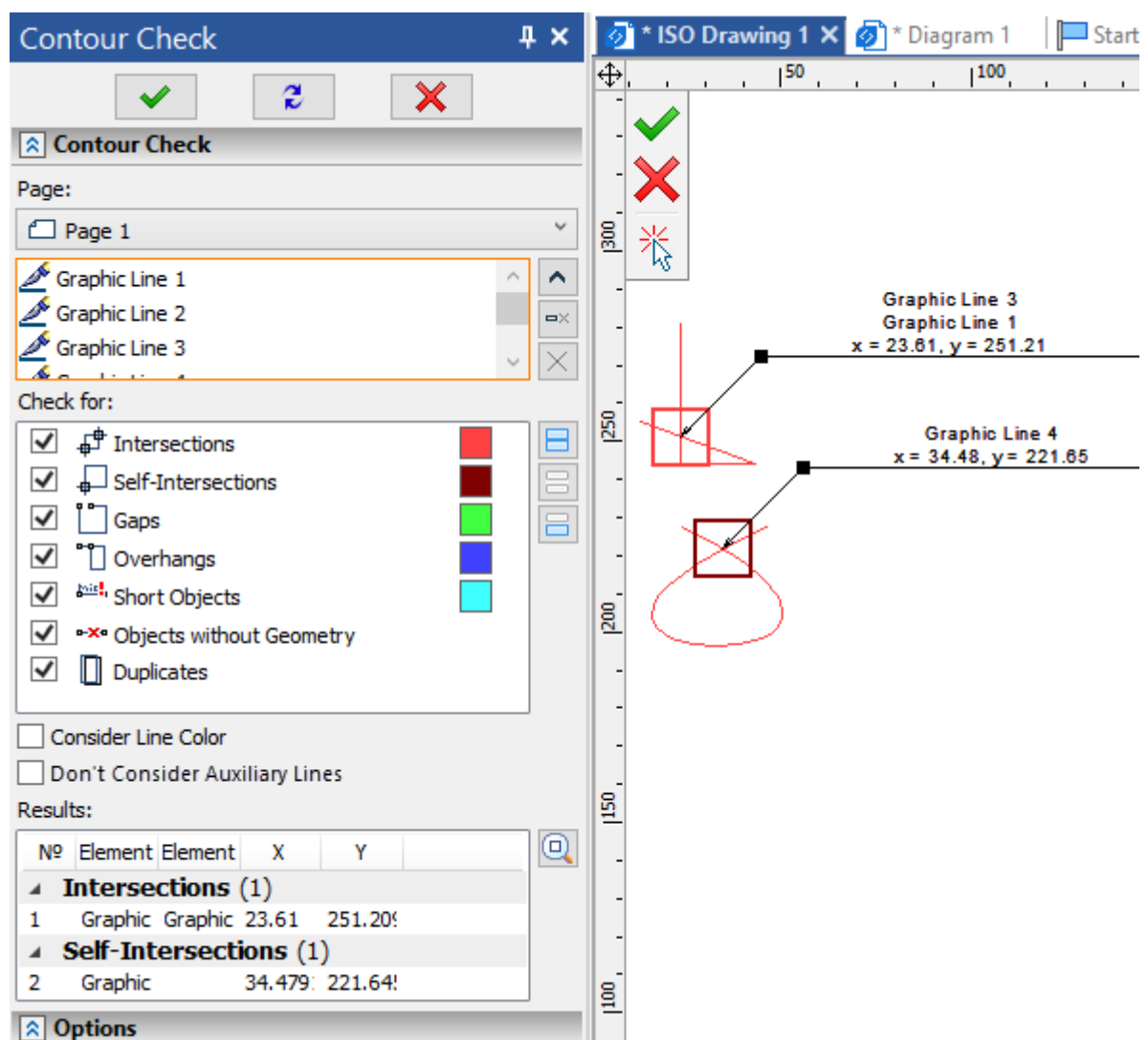
Measurement

The model measurement tools were significantly expanded: existing commands were improved and new ones added. New ways of measuring were added to the **Measurement** command. Now you can create 3D construction elements based on the results of the measurement. In addition, measurements have become more clear and understandable due to the **Decorations**. Analysis of the curvature of surfaces and curves can now be realized in various ways, providing both accurate values and a visual picture convenient for engineering. You can analyze the accuracy of matching elements using a special command.






Contour Check

A new command has been added:








Icon	Ribbon
	Measure > Contour Check > Contour Check Workplane > Analysis > Contour Check
Keyboard	Textual Menu
<QP>	Tools > Geometry Analysis > Contour Check







The command checks contours formed by graphic lines for errors on an active workplane, on a drawing page or within active drawing view. It is useful on active WP, when checking contours, that will be used for creating 3D profiles. When working with a drawing, you can select a **Page** to be checked using the drop-down list in the top of the command's parameters window. If there are no active drawing views on the selected page, then the command checks elements not belonging to views. If there are an active drawing view, then the command checks content of such view. When working on an active workplane, the current workplane is selected automatically and there are no other options in the drop-down list.

Below the **Page** drop-down list there is the list of lines to be checked. Upon calling the command, lines can be added to the list using . Calling the command with pre-selected lines automatically adds such lines into the list. Launching the check with empty list leads to checking all available graphic lines. The list of lines to be checked can be managed using the **Expand** , **Collapse** , **Delete element from the List**  and **Clear**  buttons.




In the panel below you can select categories to **Check for**:

-  **Intersections**
Intersections of two or more lines.
-  **Self-Intersections**
Lines intersecting themselves.
-  **Gaps**
Distance between lines within the specified range.
-  **Overhangs**
Partially coinciding lines.
-  **Short Objects**
Lines, whose lengths fall into the specified range.
-  **Objects without Geometry**
Objects, whose geometry cannot be constructed. For instance, if you construct an arc by three nodes first, and later make such nodes lay on the same straight line, then the arc will become the object without geometry.
-  **Duplicates**
Fully coinciding lines.

There are color samples near some of the categories in the **Check for** panel. The results of the corresponding type get outlined in the drawing or WP using such colors, upon performing the check. Colors can be customized using the system options dialog (**Options > Colors > Contour Check**). The system checks for categories, whose checkboxes are enabled. Checkboxes could be enabled or disabled by clicking  them directly or using buttons in the right side of the panel ( **Select All**,  **Unselect All**,  **Invert Selection**).

Following checkboxes are available below the Check for panel:

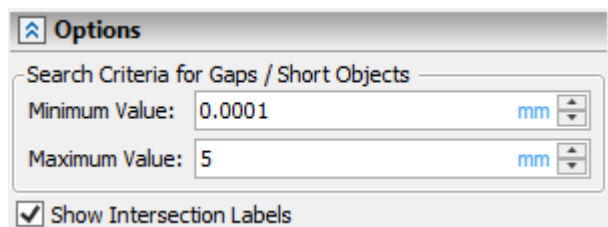
- **Consider Line Color**

The checkbox is disabled by default. Enabling it excludes from the results  **Intersections**,  **Gaps** and  **Overhangs** between lines of different colors.



- **Don't Consider Auxiliary Lines**

This checkbox is enabled by default, so auxiliary lines are ignored by check. If you disable this checkbox, all selected lines will be checked.

Following **Options** are available in the eponymous tab of the command's parameters window:








- **Search Criteria for Gaps / Short Objects**

 **Gaps** and  **Short Objects** are detected if gap value or object length fall within range between **Minimum** and **Maximum Value**. The range includes the minimum value, but doesn't include the maximum.

- **Show Intersection Labels**

If this checkbox is enabled, then labels will appear on a drawing or WP upon performing the check. Each label marks an intersection point and contain its coordinates as well as names of intersecting lines. Precision of coordinates is defined by document parameters (**Document Parameters > Page > View > Linear > Precision**).

Set the desired check parameters and press , **<Ctrl>+<Enter>** or **<Enter>**. The list of results will appear in the **Results** panel. Results in the list are grouped by categories. You can fold a group by clicking the  button, which is located to the left side of a group name. When folded, the button changes to . Click it again to unfold a group. The number of results in a group is shown in brackets to the right side of a group's name. Index number (**Nº**), names of affected lines (**Element 1, Element 2**), as well as **X** and **Y** coordinates (if applicable), are shown for each result in the list. Endpoint's coordinates of one of the affected lines are shown for gaps, coordinates of an intersection point are shown for intersections and self-intersections. Results can be selected in the list using . As you select a result in the list, affected lines get selected in the drawing/WP and an outline (if applicable) changes color to light magenta. You can  **Zoom In** camera to the selected result using button located at right side of the results list.

Upon receiving check results you may change command's parameters and launch check again (, **<Ctrl>+<Enter>** or **<Enter>**), or exit the command (, , **<Esc>**, **<Shift>+<Esc>**, **<X>**).

Measure Command

The **Measure** command has new measurement properties and new visualization tools.

Decorations

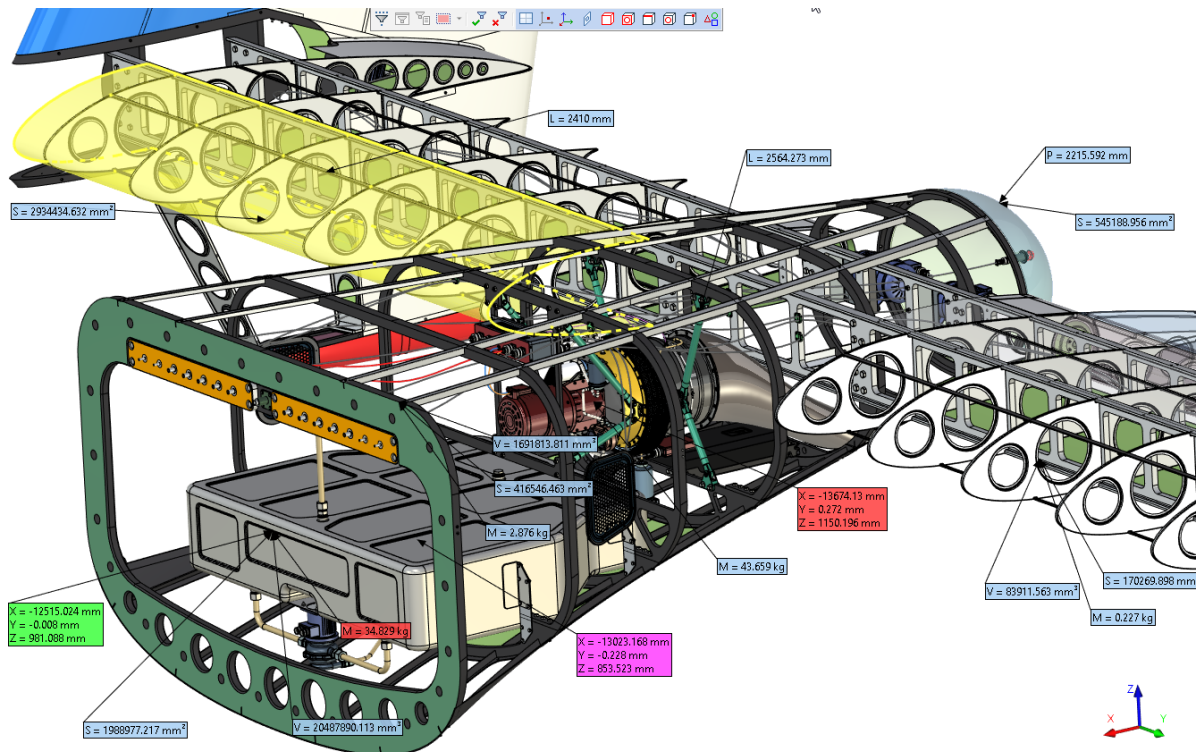
Now decoration of measurements can be displayed.

Value: 50 mm

Expression: `get("Edge_2", "EndY")`

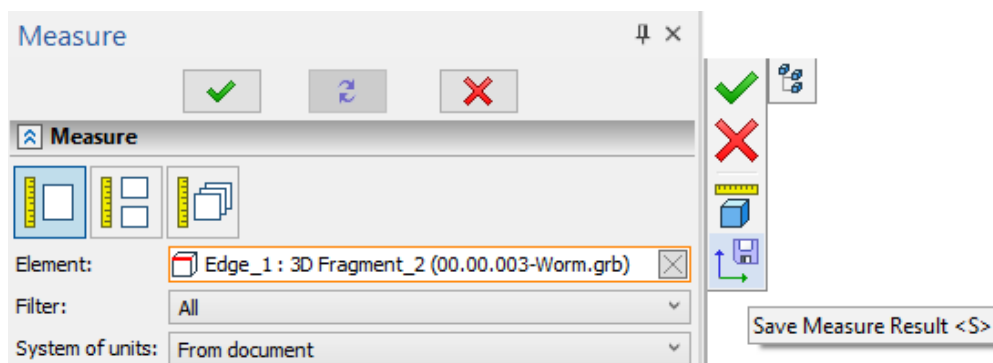
☒ Show Decorations

The decorations are a small window with brief information about the measured object.



Save Measure Result

There is a new possibility of saving the measurement results with the decorations.

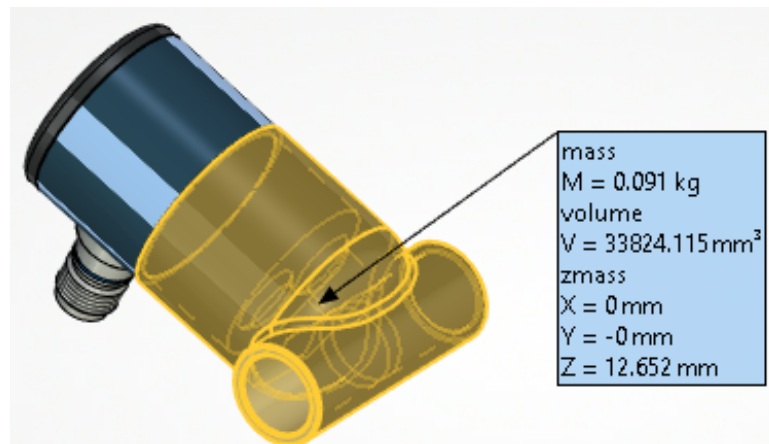


To save the measurement result, click the save button in the automenu. When the command is finished, the measurement result is created in the model elements.



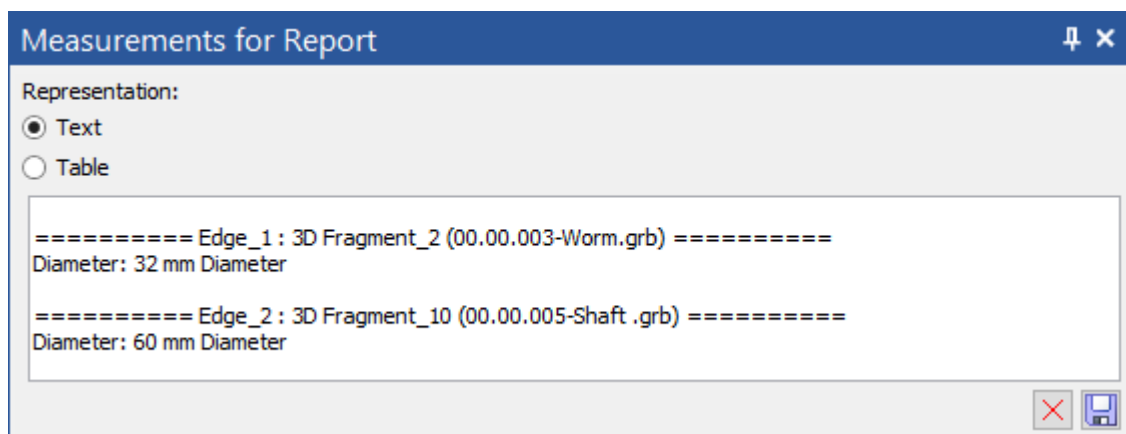
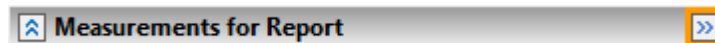
Multiple Selection

Measurement results can be merged into one decoration. If you select several measured properties using "Ctrl" or "Shift" and set the **Merge Labels** option flag, then the properties will be displayed in one decoration window.



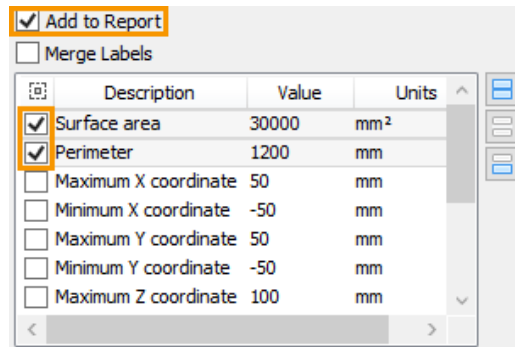
Report on Measurements

Several measurements can now be executed sequentially with the capability to compare the results of their analysis. The **Measurements for Report** window was added for simultaneous display of measurement results. The window is initially minimized in the command dialog, but you can drag the window to a convenient location on the screen or to a second monitor.




The window has two options for representing **Text** or **Table**. Both options of the results output enable us to understand for which items all the measurements were performed and what were the results. The **Table** option displays results with horizontal scrolling of measurements and automatic grouping by measurement method, while the **Text** option displays measurements with vertical scrolling.

To add measurement results to the report window, set the **Add to Report** flag. The list of results added to the measurement report is specified by flags in the special column **Select Property for Report**.




The set flags are applied to all the subsequent measurements performed in a single session of the **Measure** command. In addition, the command stores the list of results specified for different types of measured elements. When you re-measure elements of the same type, the desired results automatically appear in the report window. It is convenient when you need to measure several groups of the same type of elements and analyze them by the same parameters.

You can save the measurement results from the **Measurements for Report** window by clicking the icon . The results are saved in the «*.txt» file.

3D Node Based on Measurement Results

Now you can create a 3D node using the measurement results.

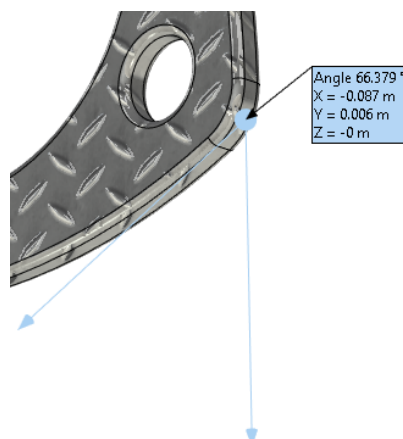
- for properties: StartX, StartY, StartZ, EndX, EndY, EndZ-a 3D node is created on the path (at the beginning or end, respectively);
- for properties: LocationX, LocationY, LocationZ-a 3D node is created on the surface at the starting point of the surface;
- for the Xmass, Ymass, and Zmass properties, a 3D node is created in the center of mass;
- for Distance, a 3D node is created on the first point of the segment that represents the minimum distance between the selected elements (the first point of the segment belongs to the first selected element).

If one of the properties listed above is selected, an additional button  appears in the automenu. A node is created when you click it.

Measuring Angle Between Curves

The angle between curves is measured as the angle between their tangents.

- The angle is measured for two curves between tangents at the first intersection point of the curves.
- If there is no intersection, it is measured at the intersection point of the tangent of the first curve and the second curve.
- If the tangent does not intersect the second curve, the angle is measured at the point where the tangents intersect.



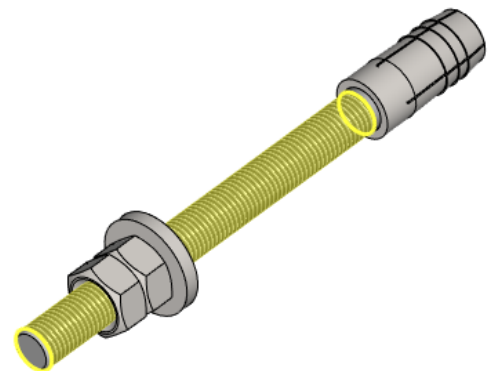
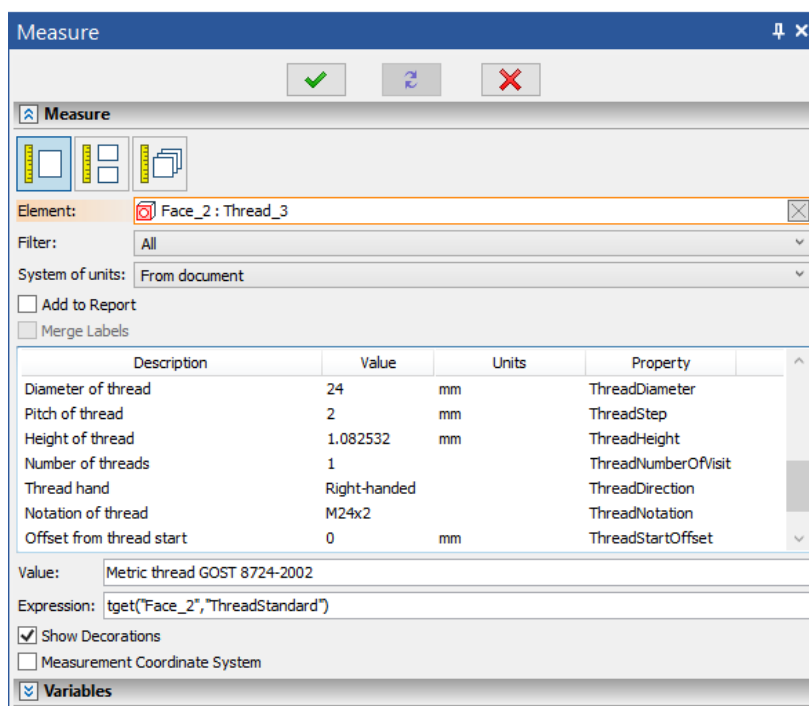
Measurement of Cone Radii

It is now possible to measure conical surfaces. Two radii and the diameters of the conical surface are measured: bigger (ConeRadiusMax and ConeDiameterMax properties) and smaller (Radius and Diameter properties). If the cone is not truncated, the values are the same. The height of the truncated cone (ConeHeight and ConeHeightFull properties) and the height of the full cone are also measured.

Description	Value	Units	Property
Half-angle	45	°	SemiAngle
Diameter	100	mm	Diameter
Major radius of cone	150	mm	ConeRadiusMax
Major diameter of cone	300	mm	ConeDiameterMax
Height of cone	100	mm	ConeHeight
Height of full cone	150	mm	ConeHeightFull

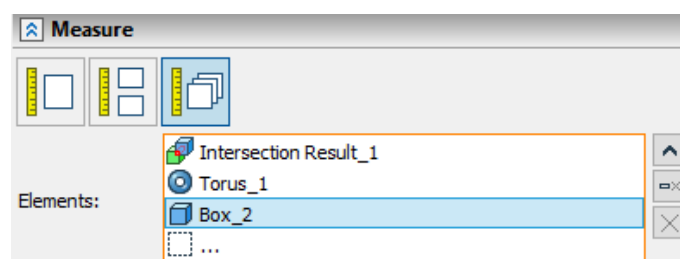
Thread Measurement

Added the ability to measure threads. You can select both threaded faces and the **Thread** operation itself.



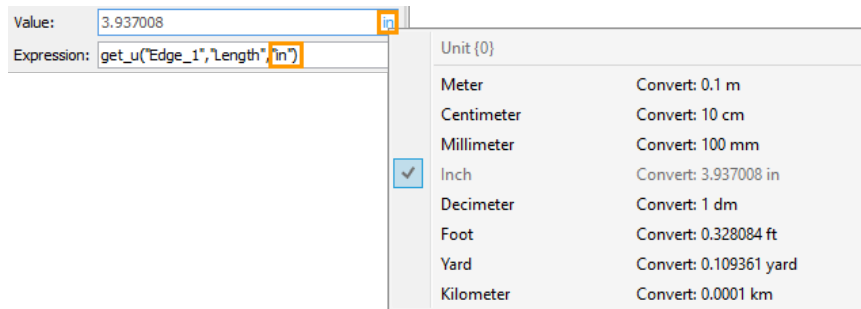
Measurement of Intersection Value

You can measure the volume of the intersection of two bodies explicitly. If you have created an intersection result for multiple bodies, you can measure the amount of intersection between the two selected bodies. To do this, select the intersection result in the **Measure** command, and then select the two desired bodies.



Units of Measurement Selecting

The Measurement command now enables you to select units of measurement. You can separately specify units for the **Value** field. The selected units of measurement are specified in the **Expression** field.

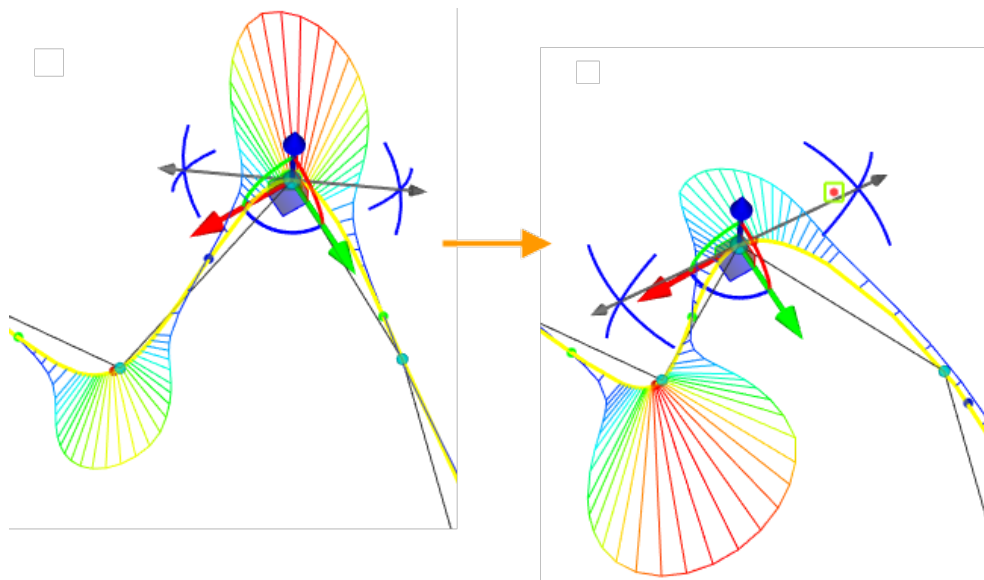


Set of Objects Measurement

A ratio measurement was added to measure multiple objects: dimensions, coordinates of the center of mass, and search for the minimum and maximum distances.

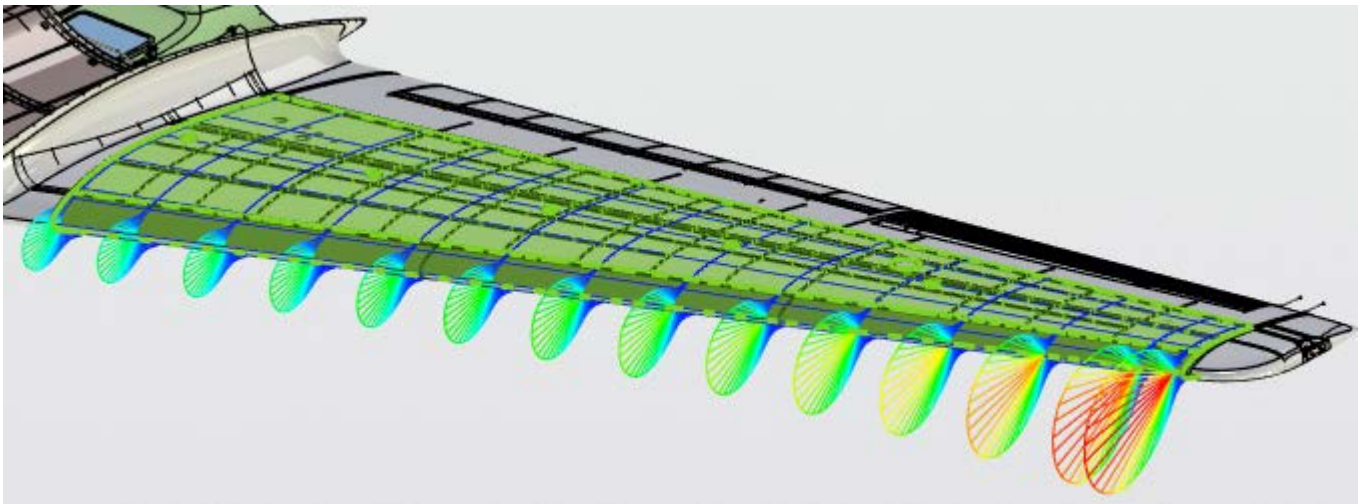
Curvature Analysis

You can view the curvature measurement for **3D Path** and **Route** elements directly at the time of editing. Now you can edit three-dimensional curves based on the resulting curvature, according to the measurement data, without leaving the editing mode.



Surface Curvature Analysis

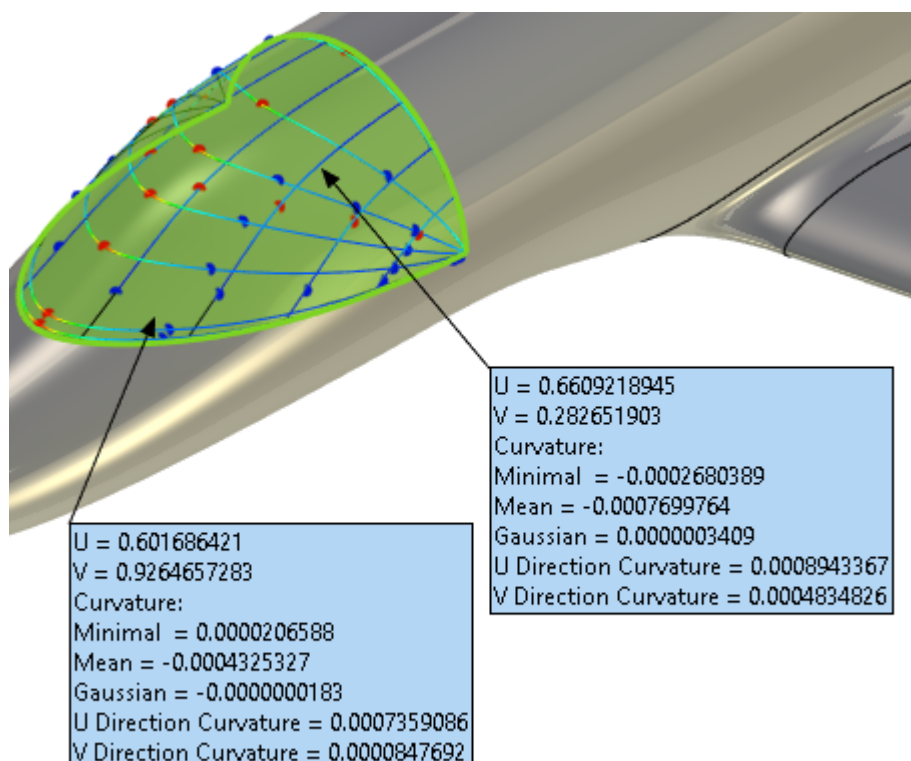
A new tool for analyzing surface curvature is available: **Surface Curvature by Sections** command.



The command can display the curvature and curvature radius of the surfaces in sections in mutually perpendicular U and V directions. The number of sections in each direction is customizable.

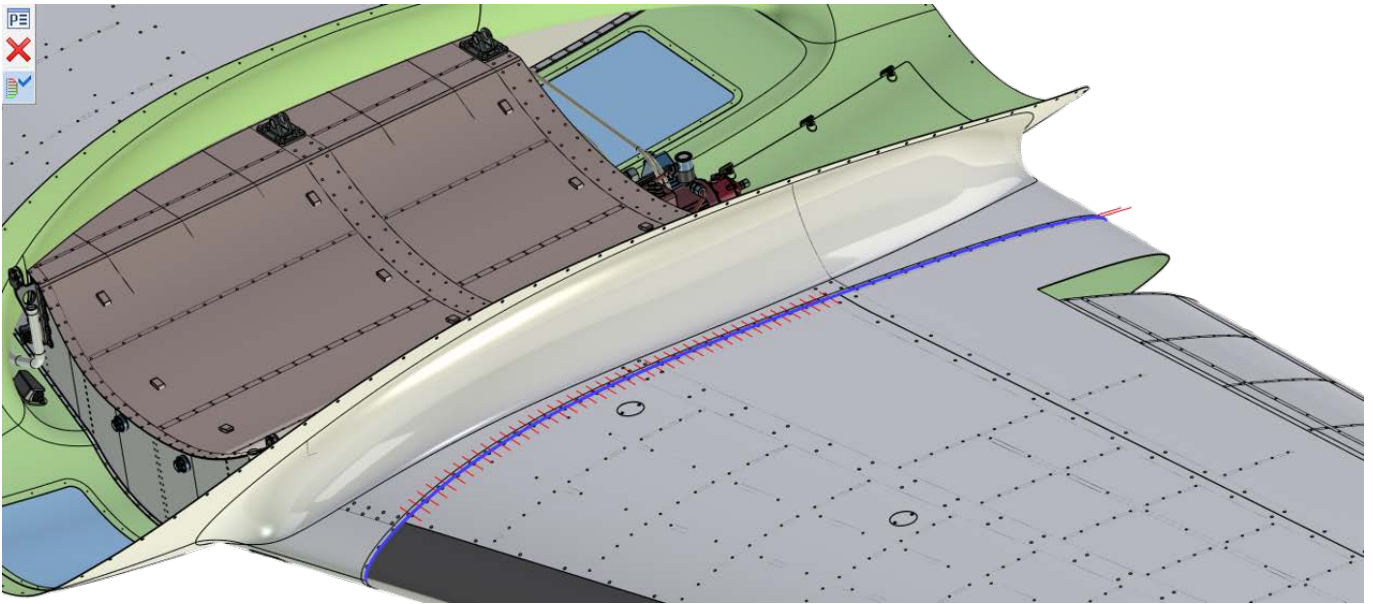


Visual display settings are available: color scale, image scale, image line density, output range. In addition, the display of measurements in points is available.



Deviation Control - Analyzing the Coincidence of Geometric Elements

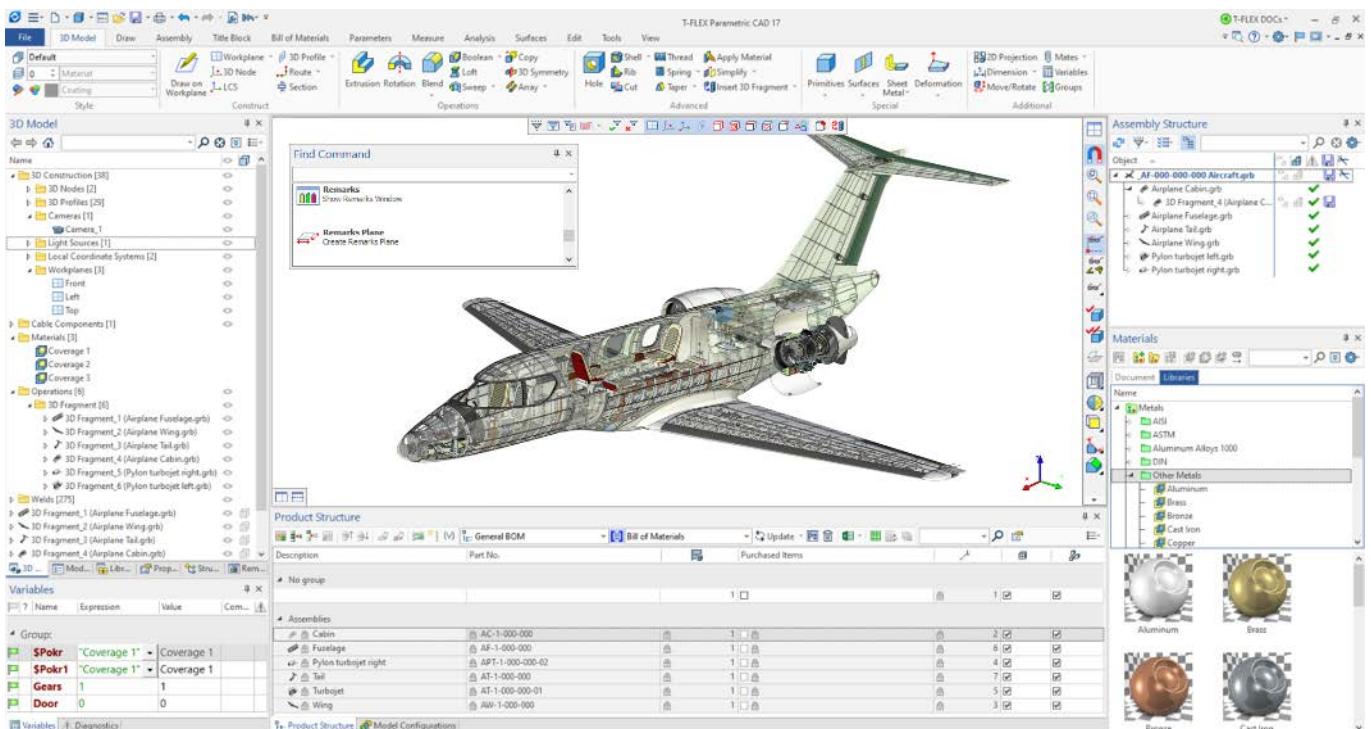
A new command for analyzing the coincidence of geometric elements has added – **Deviation Control**. The command allows you to analyze the accuracy of the coincidence of the edges of the selected faces. The number of deviation analysis points can be set by the user and determined automatically based on a preliminary analysis of the curvature of the contacting elements. The settings for the value of permissible deviations are also available.



Nº	Result	X	Y	Z	Distance	Angle
10	✓	-5692.555517	906.934368	-792.82477	1.002273	0.16057
11	✓	-5708.337283	906.730307	-788.95385	1.0022	0.160215
12	✓	-5724.169018	906.537259	-785.2922	1.002151	0.156238
13	✓	-5740.047254	906.35508	-781.83713	1.002122	0.152495
14	✓	-5755.968503	906.183597	-778.58534	1.002111	0.148966
15	✓	-5771.929414	906.02264	-775.53365	1.002114	0.14313
16	✓	-5787.926389	905.871912	-772.67641	1.002128	0.137872
17	✓	-5803.95596	905.731093	-770.00751	1.002148	0.132629
18	✓	-5820.014971	905.599884	-767.52133	1.002169	0.127419
19	✓	-5836.10047	905.477988	-765.21216	1.002188	0.122268
20	✓	-5852.209705	905.365104	-763.07433	1.002207	0.117194
21	✓	-5868.240123	905.260024	-761.10216	1.002228	0.112311

Interface and interaction with the system

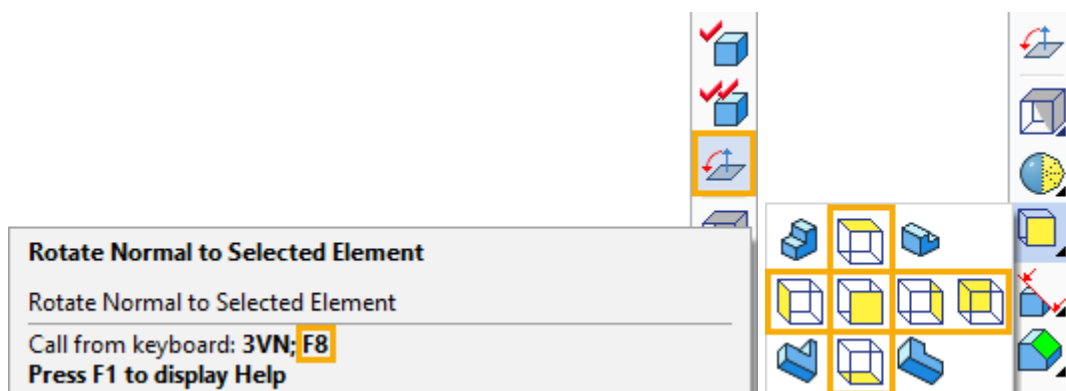
There are new customizable instruments for selecting the objects. Now you can select objects with either right mouse button, or left mouse button with the help of a rectangular, a lasso or a cutting line with or without selecting invisible objects. Auxiliary window visibility is now set by means of hot keys. The element list under the cursor is now customizable: the user can specify the list appearance time and its transparency in the system settings. **Materials** and **3D Model** windows are improved considering the users' feedback.



Rotating Camera in 3D Window

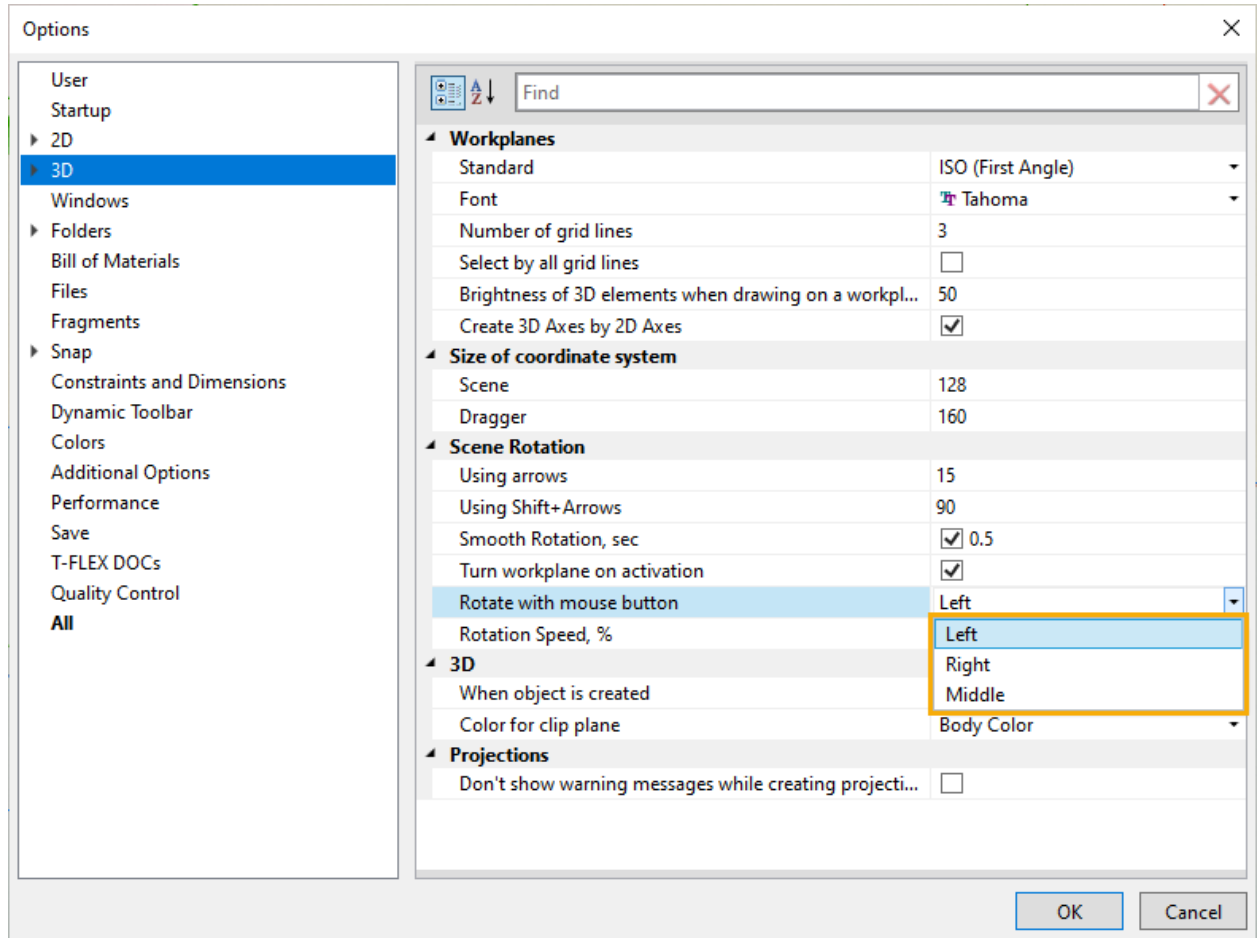
Rotate Normal to Selected Element

Now the command can be used, even if there are no elements selected. In such case, the command rotates the view up to the nearest standard flat view point (front, back, top, bottom, right, left view). Also, the command can now be called by pressing the <F8> button



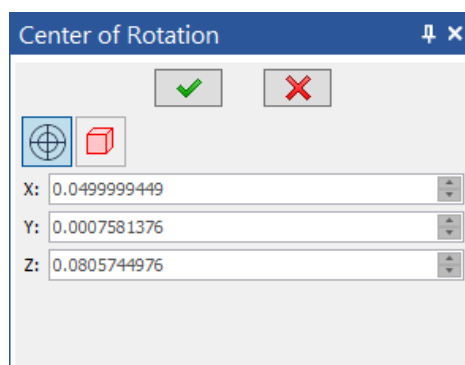
Rotate with Mouse Button Setting

Outline is input with the right mouse button pressed and the left button is used for the scene rotation. You can change the settings for scene rotation button: there is a **Rotate with mouse button** list in the **Options**. The list is in **3D** tab in the **Scene Rotation** group. If the right mouse button is used for rotation, the left button is used for selecting the objects. Also added the ability to rotate the scene using the middle mouse button.



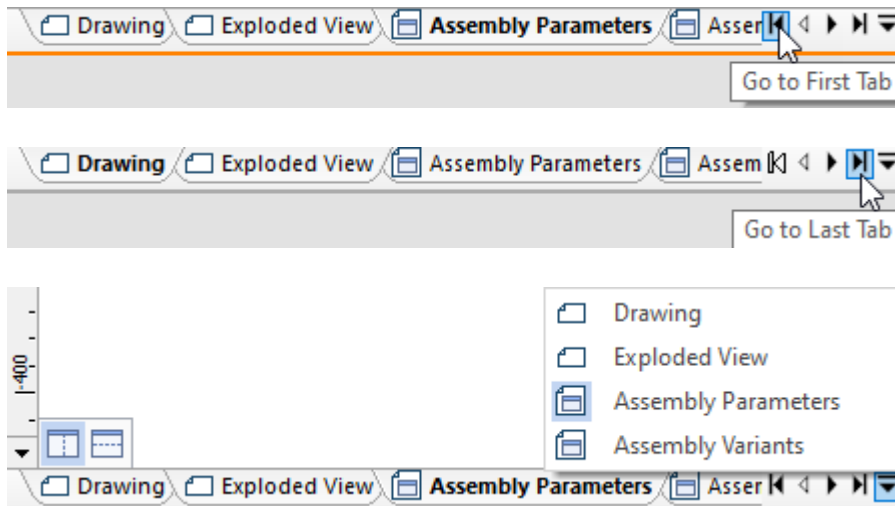
Set Center of Rotation

Set Center of Rotation command has been updated. Using the command has become more intuitive thanks to the updated interface.





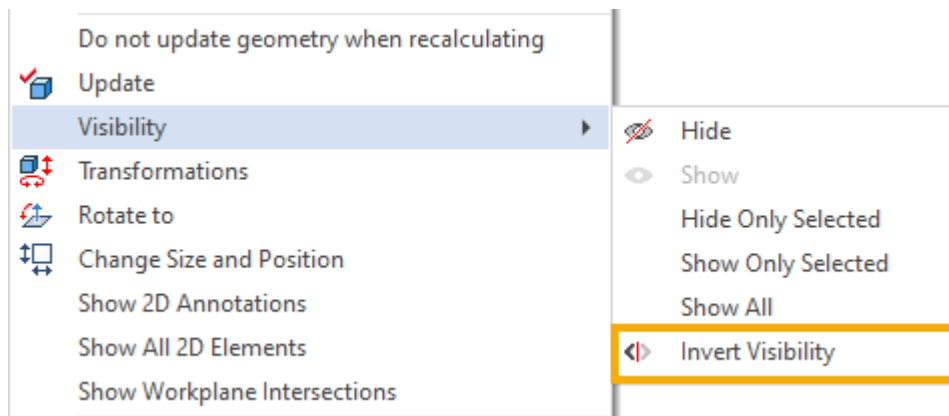
Quick Navigation between Pages

New buttons for quick navigation between pages are added to page tabs bar:



Invert Visibility

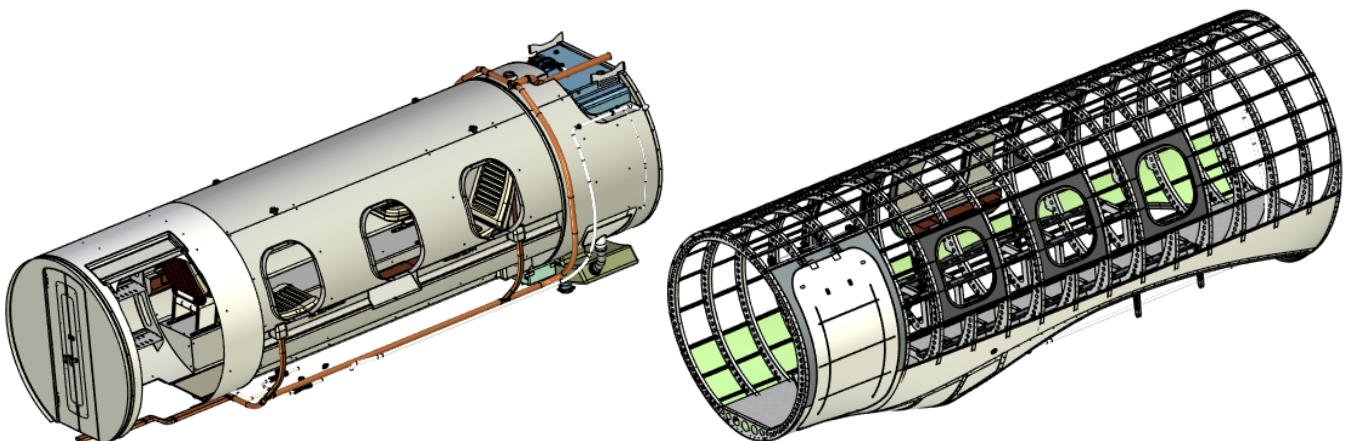
The new  **Invert Visibility** command is now available in the contextual menu, which appears upon right-clicking  3D elements or an empty space of the 3D scene.



Alternatively, the command can be called using the <3CB> or <Ctrl> + <Shift> + keys.

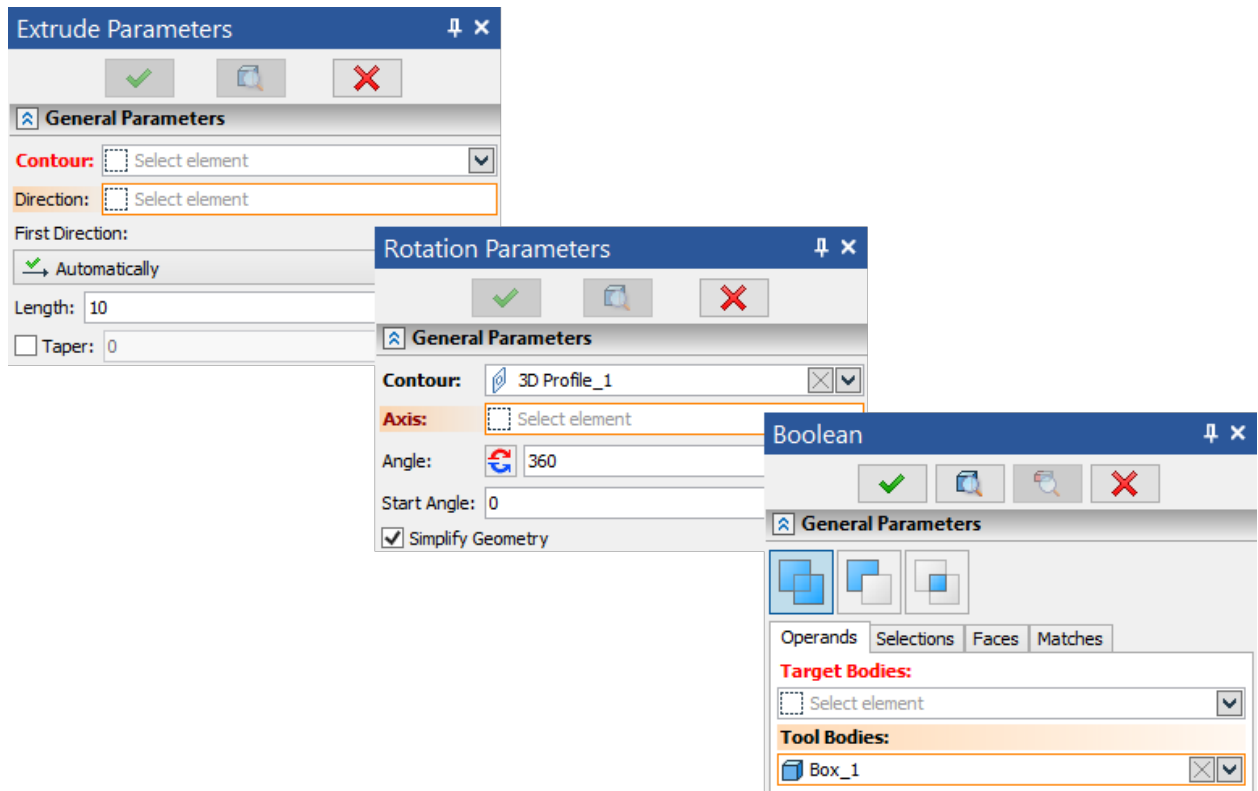
The command switches visibility of each element in 3D scene to an opposite one, i.e. hides visible elements and shows hidden elements.

The command does not affect elements, whose visibility is controlled by variables.



Highlighting the Parameters of Operations/Commands

Highlights for various operation parameters have been implemented.

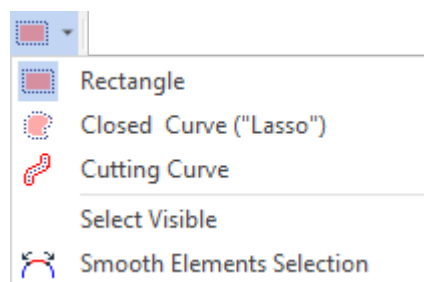


The active input field is now highlighted with an orange gradient. If a required entry field is not filled in, then the field name is highlighted in bold red. If this field is filled in, then the name is highlighted in black bold.

The updated highlighting makes it much easier to navigate in operations and commands.

Selecting 2D and 3D Objects – Updated Selector

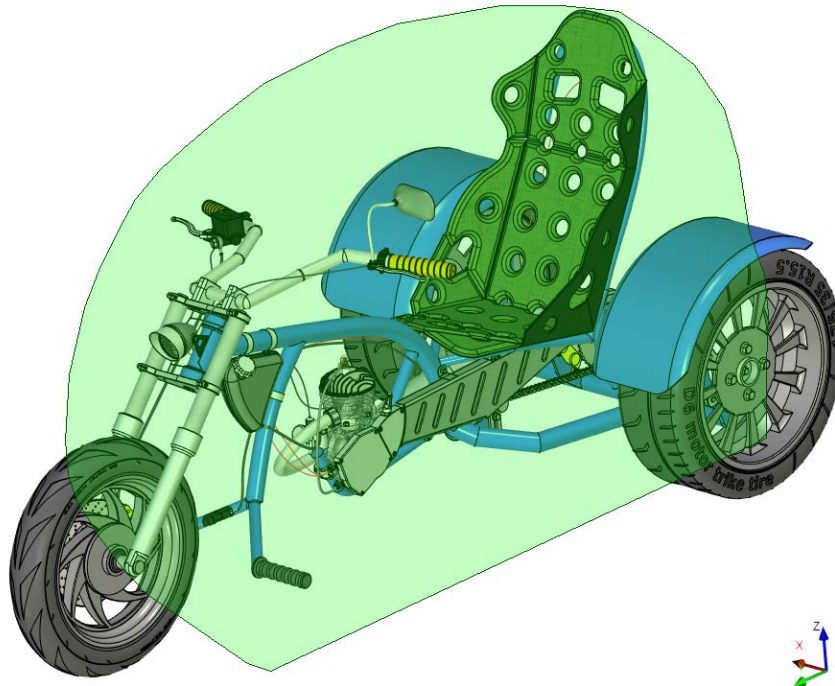
The feature of selecting elements by a random outline was added. An element selection mode control button was added to the filter panel. Selection modes are available for all the operations and in the command stand-by mode.



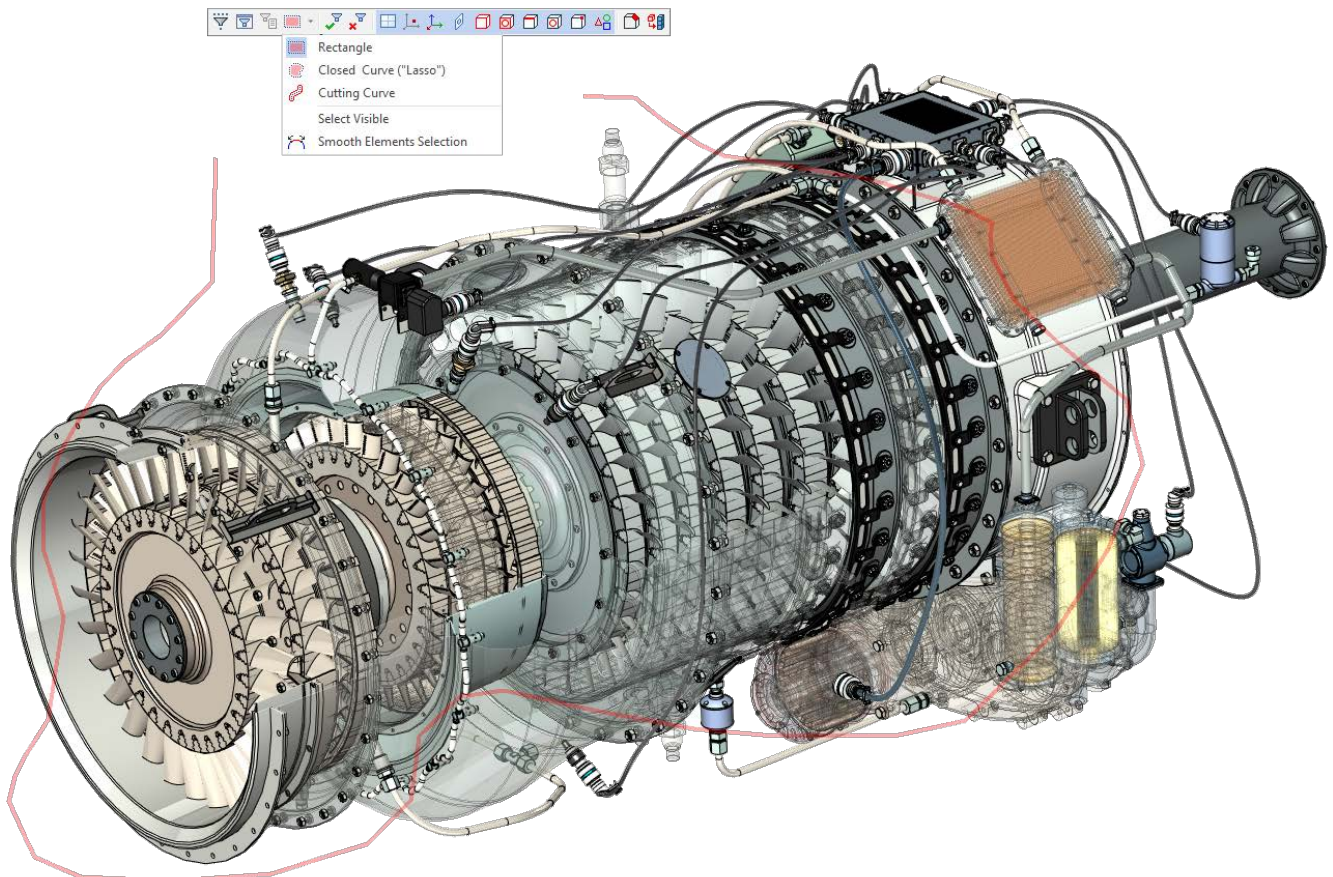
The following variants are available.

Rectangle. A common selection mode actual for the previous versions.

Closed Curve («Lasso»). A random closed outline input (the first and the current outline points are connected automatically): all the elements within one outline are selected.



Cutting Curve. A random open outline input: all the elements “touched” by the curve are selected.



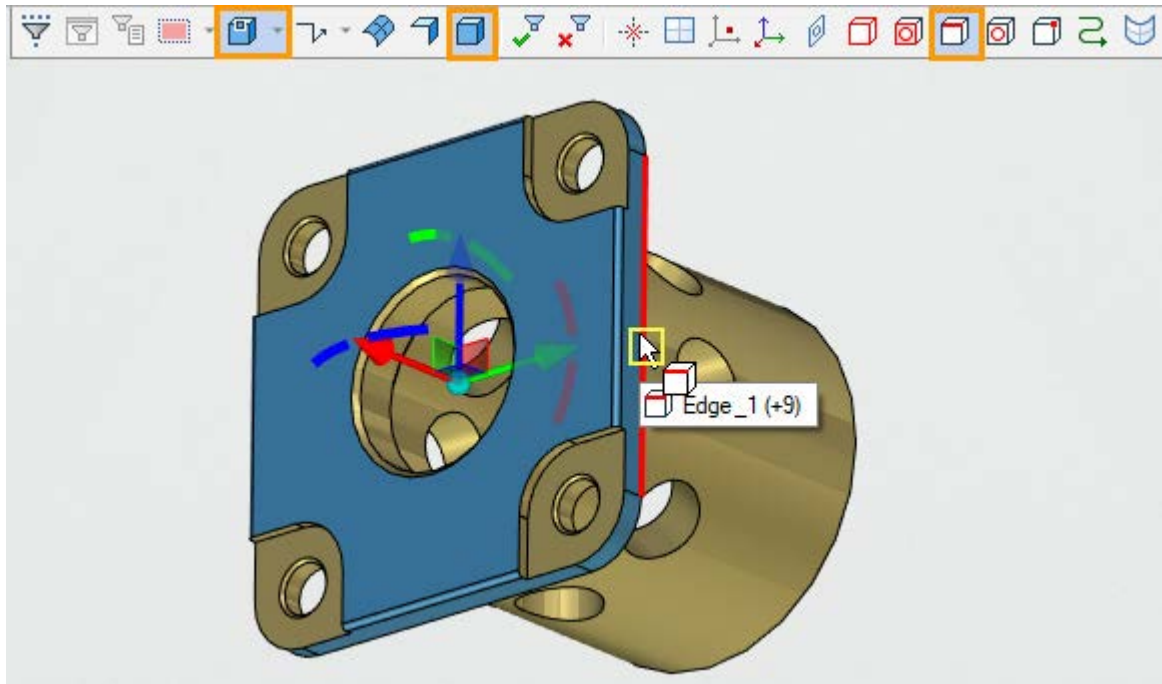
The following options are also available in the selection way list of the filter toolbar.

Select Visible. Enables the selection mode for visible elements only: not hidden by other elements or having reversed orientation.

Smooth Elements Selection. Enables automatic selection mode for smoothly joint elements succession.

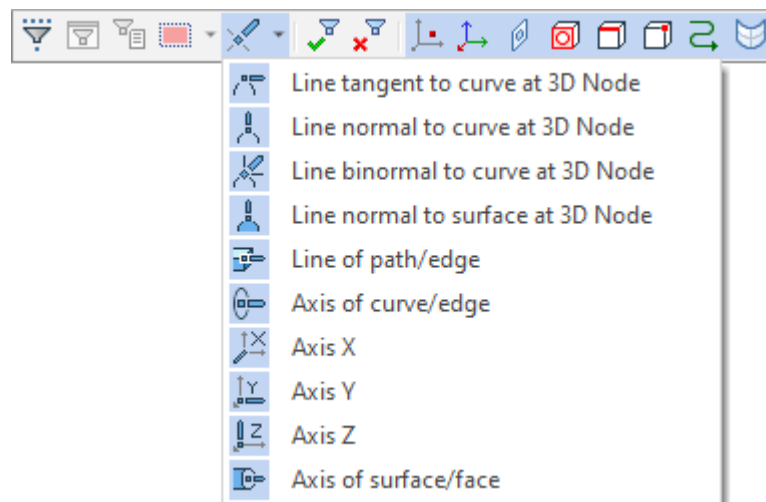
Filter of Selection by Geometric Type

An expanded selection filter within geometric types is now available in 3D modelling operations at the stage of selecting geometric objects. This means that it is now much more comfortable to find the necessary object with the cursor. E.g. when it is necessary to specify a point in the middle of the edge, you can set selecting only edges in the binding filter and specify **Edge middle** in the geometric objects filter.



Then all the other variants of selecting points and edges become unavailable and do not disturb.

Every geometric type has its own selection setting list. Geometric types can be seen in the left part of the filter panel. Geometric type selection setting is available automatically with the relevant geometric types set. E.g. a geometric type **Axis** with its own list of the objects available for selection appears automatically in the **Rotation** operation at the stage of rotation axis selection.



Another set of available geometric types is in the **3D Node** operation: **Point**, **Wire**, **Surface**, **Sheet**, **Solid**. There is a list of objects for **Point** and **Wire** geometric types.



To open the geometric type object list, press the triangle near the icon of the geometric type.

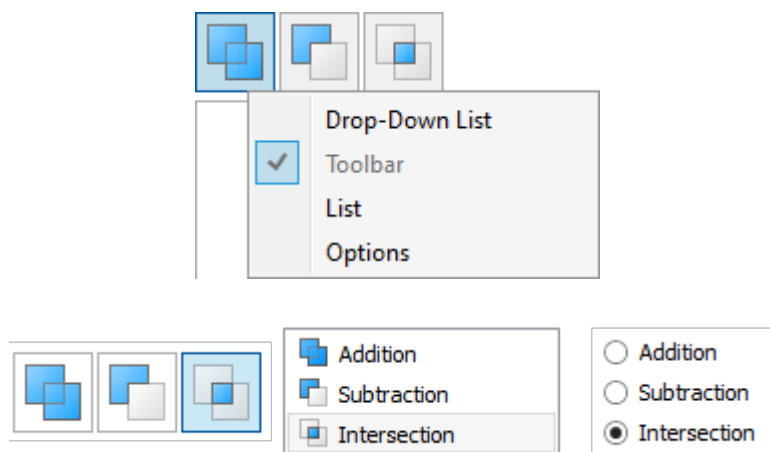
When necessary, you can call the element list immediately by using <Tab> key or by one click on the center mouse button.

After the element list appears, you can change element selection in the list with the help of keyboard or mouse. If it is necessary to switch forward between the elements in the list with the help of keyboard, use <Tab>, in case of backward switching use <Shift> + <Tab>. Confirm element selection from the list by mouse click or by pressing <Enter>.

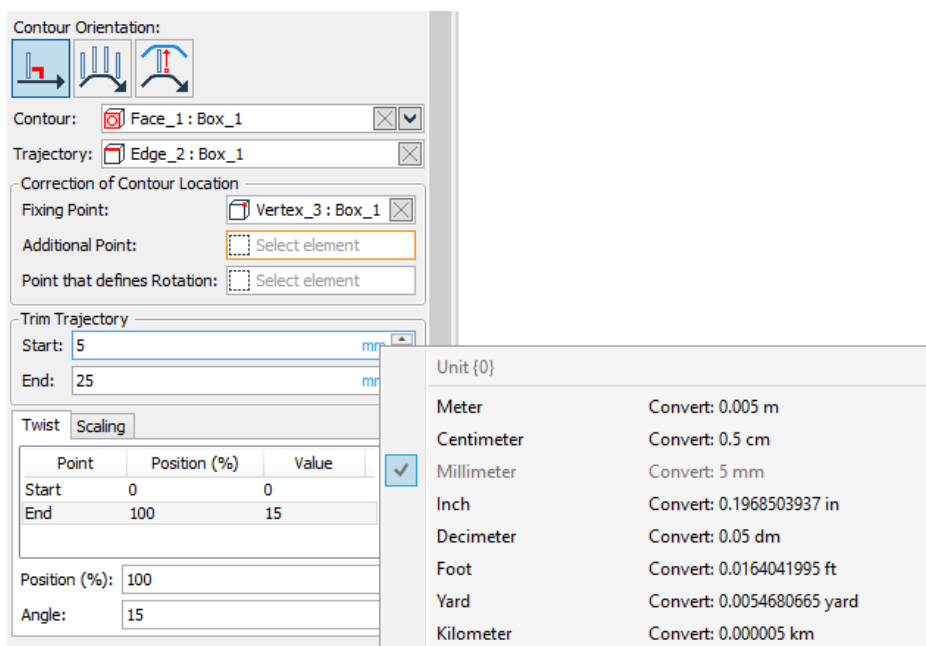
Command Dialogs

Work is underway to renew interface of dialogs of various commands: **3D Node**, **LCS**, **3D Profile**, **3D Path**, **Rotation**, **Boolean**, **Sweep**, **Cut**, **Fill Hole**.

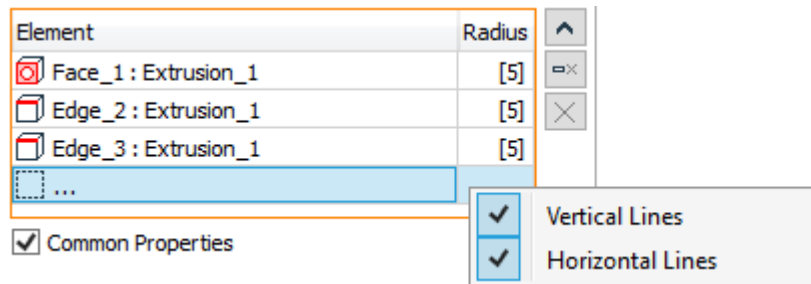
In new dialogs, the switching of modes and options of commands is more visual and convenient.



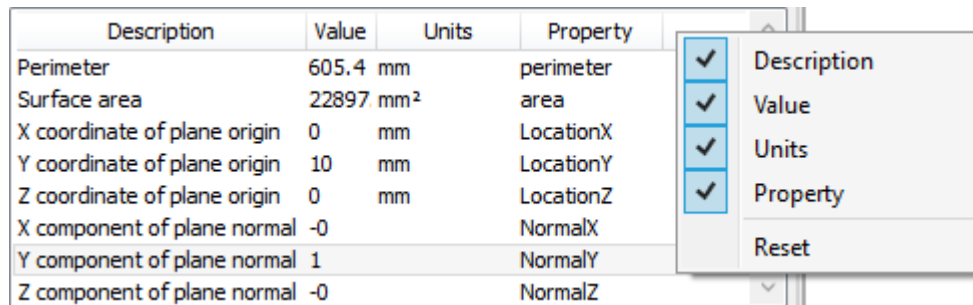
Selected input elements are placed in editable lists, which are located on tabs if this implies the logic of working with the command. In all input fields, a choice of units is available.



In the element selection fields, a grid display setting is available.

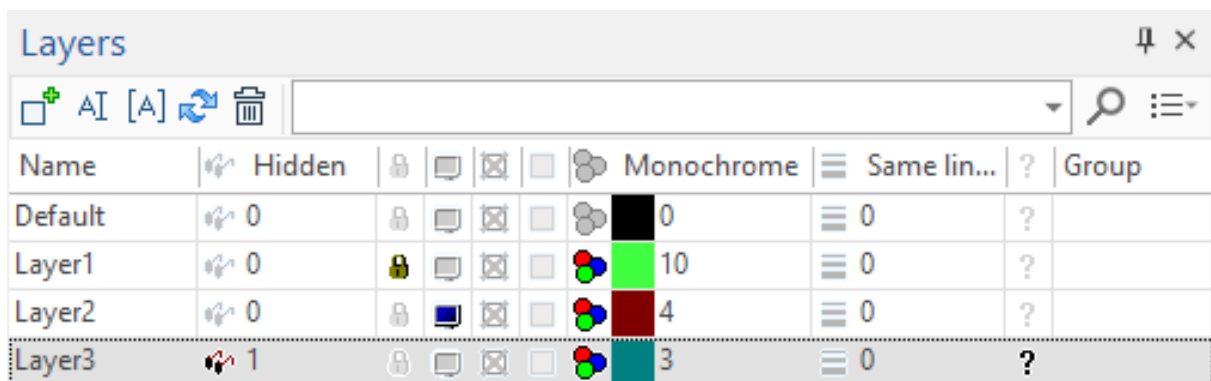


In the parameter settings fields, visibility and column order control are available.



New Window «Layers»

An auxiliary window **Layers** was added.



Layers command can be found in the window control submenu.

The window is modeless, it enables users to control layers and their parameters in the transparent mode. The layer parameters are shown in the columns.

The window supports:

- managing the columns: composition, order and width of the columns is saved and restored between working sessions;
- search by the name or part of the name;
- grouping by group name;
- multiple choice to set parameter values for several selected elements simultaneously.


A new **Group** parameter was added to the layers. Grouping works in the same way as the groups in **Variable Editor** window which makes it easy to navigate through the window in case of many layers.

«Model Configurations» window

The window has replaced **Configurations** and **Variations** command dialog.



The new window simplifies the search for the required configuration by the known dimension: the **Variables** option flag activates a special table view in the form of a comparison of variable values and configuration names. The execution table can be edited in the drawing, and all changes will be transferred to the **Model Configurations** window. By double-clicking on the drawing execution table, the **Model Configurations** window will be called up.

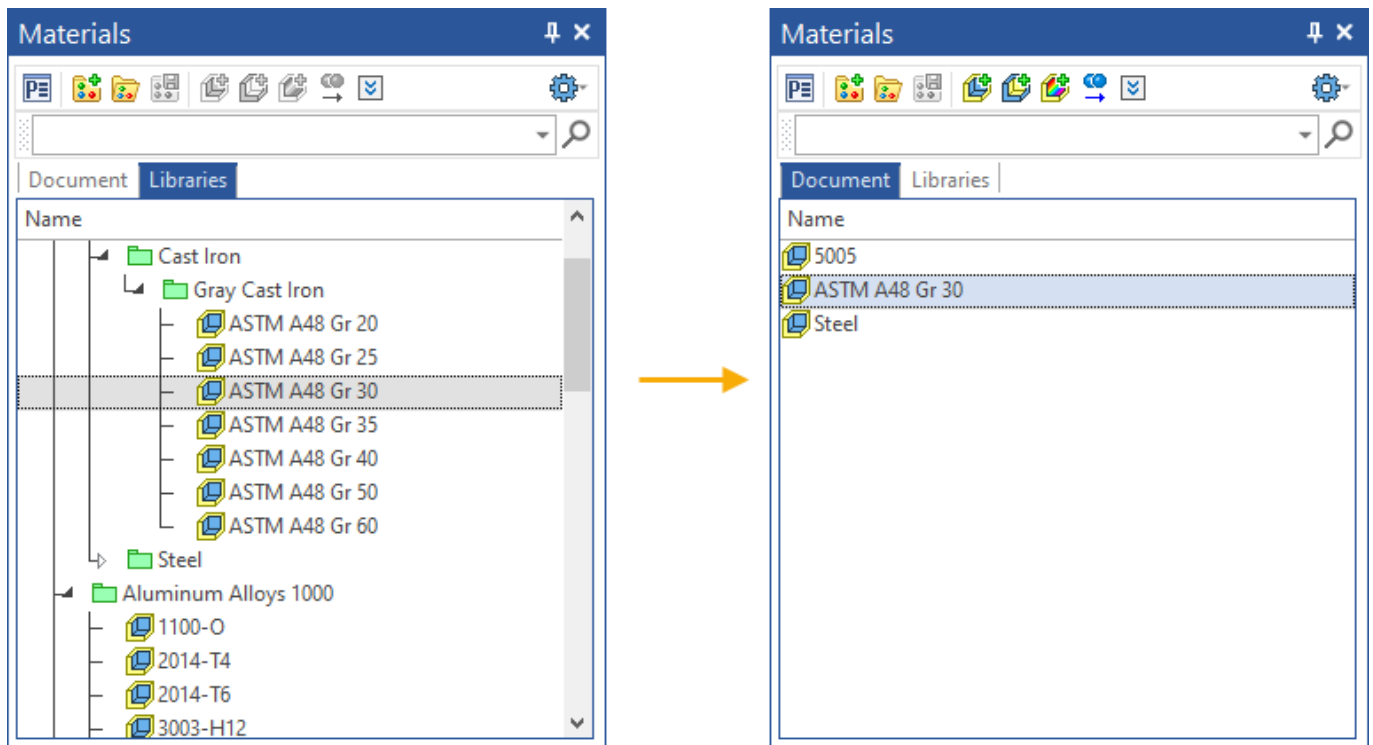
Work in the **Model Configurations** window is available without saving the file. In addition, import of performances using the **Import**  command is available.

The image is a screenshot of the T-FLEX CAD software interface. On the left is a 3D model of a mechanical assembly, a cylindrical component with multiple flanges and bolts. In the center is a 2D technical drawing of the same assembly, showing various views (front, top, side, and cross-sections). On the right is the 'Model Configurations' window, which contains a table of configurations. The table has columns for 'Base...', 'Name', 'Variation name', 'Subtitle', and 'Product Structure'. The first row is selected, showing 'ABC.00.00' as the base and name, with variation names ranging from -01 to -07. The 'Product Structure' column shows a hierarchical list of components for each variation.

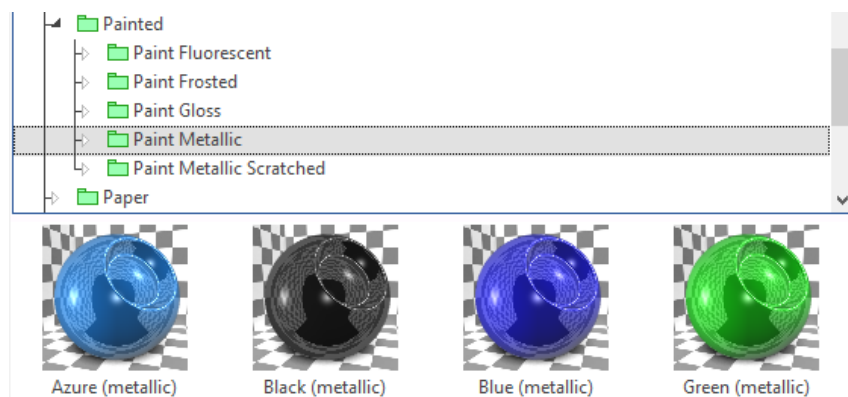
Base...	Name	Variation name	Subtitle	Product Structure
<input checked="" type="checkbox"/>	ABC.00.00			ABC.00.00
<input type="checkbox"/>	ABC.00.00	-01		ABC.00.00-01
<input type="checkbox"/>	ABC.00.00	-02		ABC.00.00-02
<input type="checkbox"/>	ABC.00.00	-03		ABC.00.00-03
<input type="checkbox"/>	ABC.00.00	-04		ABC.00.00-04
<input type="checkbox"/>	ABC.00.00	-05		ABC.00.00-05
<input type="checkbox"/>	ABC.00.00	-06		ABC.00.00-06
<input type="checkbox"/>	ABC.00.00	-07		ABC.00.00-07

«Materials» Window Improvement

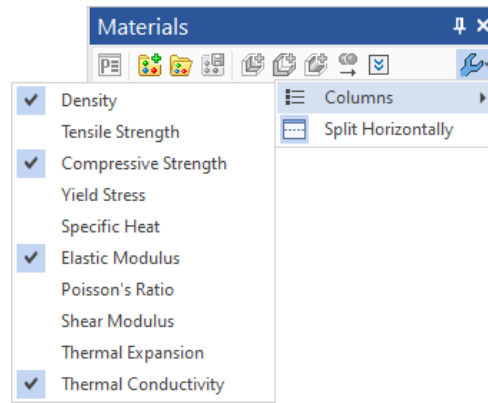
Materials window was considerably improved. **Document material list** and **Material list of open libraries** panels were joined into one panel with **Document** and **Libraries** tabs. The **Document** tab contains the materials used in the current document; the **Libraries** tab contains all library materials available in T-FLEX CAD. Any material from the **Libraries** tab can be dragged to the **Document** tab using drag-and-drop.



A preview icon was added to the left from the name of the material. Besides, material preview is available in the bottom panel of the **Materials** window. In case a particular material is chosen, it is the only one to be displayed at the preview panel. If a material directory is chosen, preview of all the materials from this directory is displayed as big icons.



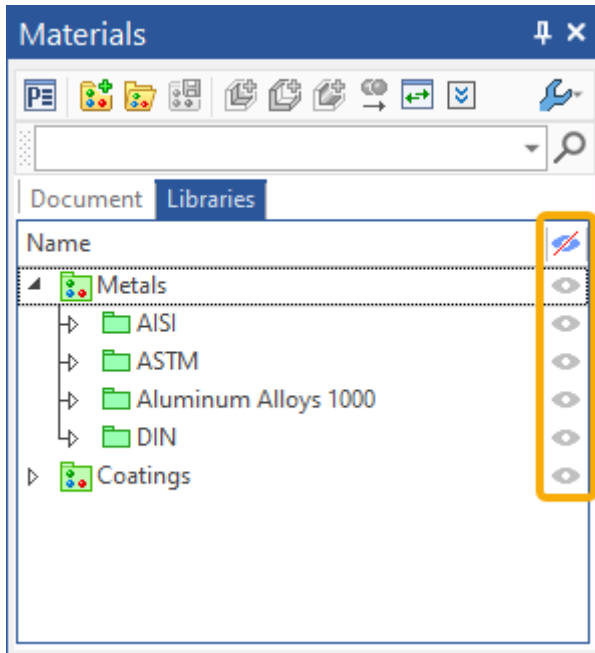
At the top of the window the columns-to-be-displayed setting list was added.






This makes it possible to display material properties relevant for the user. Pressing the column name enables sorting the materials in every directory in the ascending order by this particular material property or by pressing it twice – in the descending order. This mechanism simplifies finding the required material by the required property.

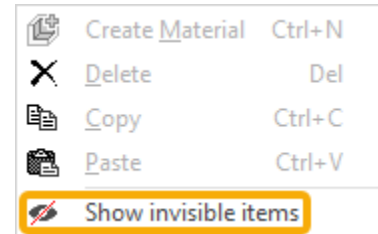
Name	Density	Compressive Strength	Elastic Modulus
Metals			
AISI			
Steel			
Stainless Steel			
AISI 201	8000 kg/m ³	640 N/mm ²	200000 N/mm ²
AISI 202	8000 kg/m ³	700 N/mm ²	200000 N/mm ²
AISI 303	8000 kg/m ³	585 N/mm ²	200000 N/mm ²
AISI 304	8000 kg/m ³	215 N/mm ²	200000 N/mm ²
AISI 309S	8000 kg/m ³	515 N/mm ²	200000 N/mm ²
AISI 316Ti	8000 kg/m ³	515 N/mm ²	200000 N/mm ²
AISI 321	8000 kg/m ³	515 N/mm ²	193000 N/mm ²
AISI 409	7760 kg/m ³	520 N/mm ²	217000 N/mm ²
AISI 420	7650 kg/m ³	1165 N/mm ²	218000 N/mm ²
AISI 430	7800 kg/m ³	480 N/mm ²	200000 N/mm ²
AISI 439	7700 kg/m ³	480 N/mm ²	206000 N/mm ²
AISI 441	7700 kg/m ³	530 N/mm ²	220000 N/mm ²
AISI 904L	7900 kg/m ³	490 N/mm ²	200000 N/mm ²


Material Libraries Visibility Control




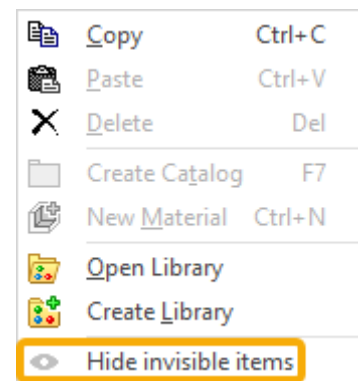
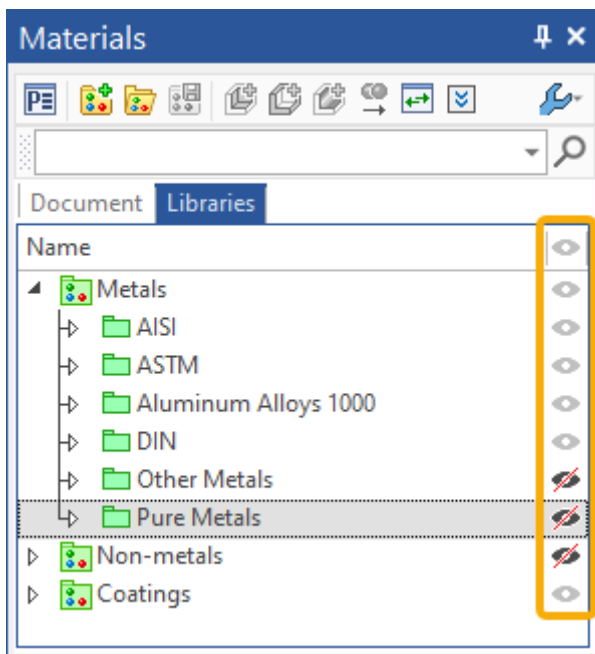
The  **Visibility** column is added to the **Libraries** tab of the **Materials** tool window. Click an  icon in this column in order to hide a corresponding library or a folder of a library.

You can  **Show invisible items** using the contextual menu of libraries list.



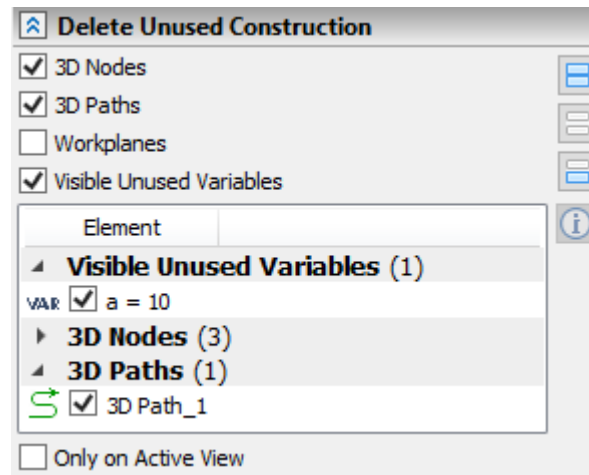
In result, all elements will be displayed regardless of their visibility. Click an  icon in the **Visibility** column in order to make a corresponding invisible element visible again.





You can switch back to the default representation, where invisible elements are hidden, using the  **Hide invisible items** command in the contextual menu of libraries list.



«Purge» Window

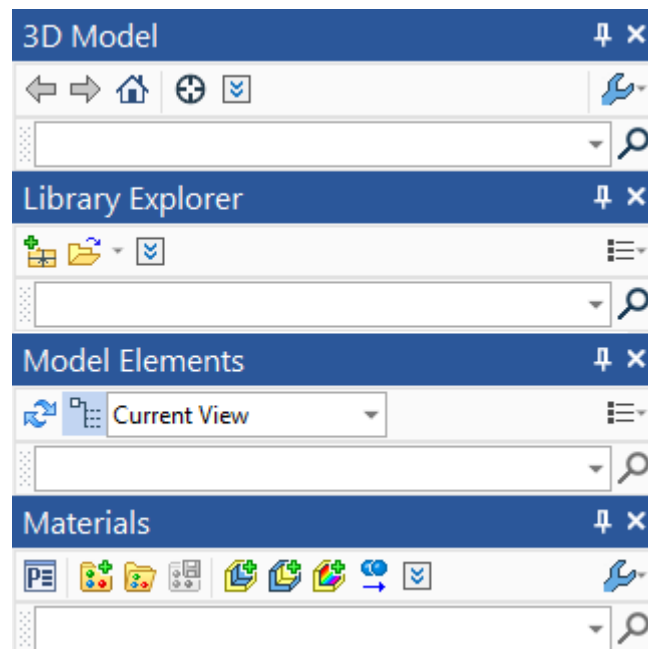
The entire list of unused items is now available for deletion in the **Purge** command. The items to be left can be chosen from the list. Visible user variables that are not used are now included in the list of items that can be deleted.



In the commands dialog, you can cancel the deletion of an item, or a group of items. Inverting the list , clearing the list , selecting all items , and information about the selected item  are available.

Tool Windows Updated View

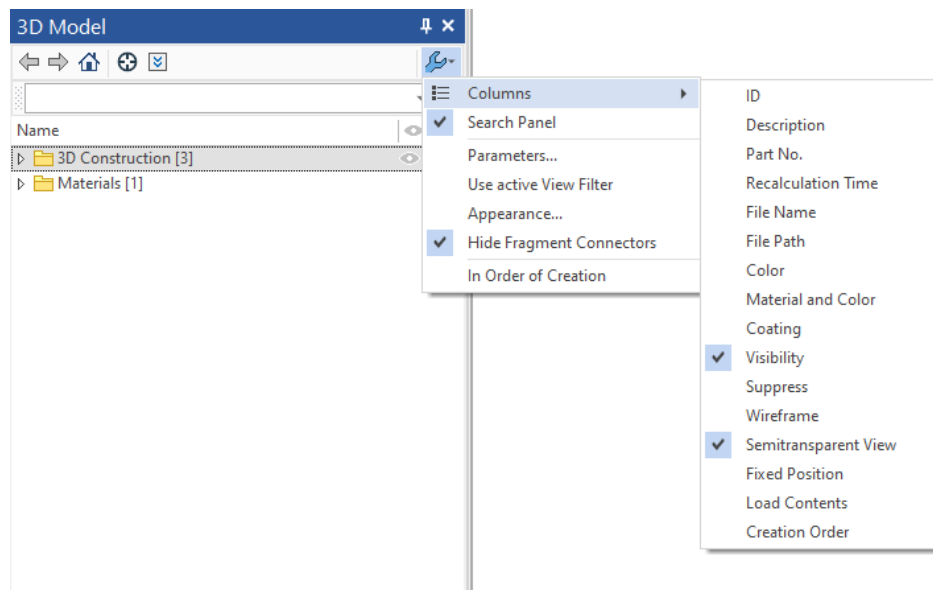
All tool windows have been moved to the new interface.



The search panel is moved to a separate toolbar.

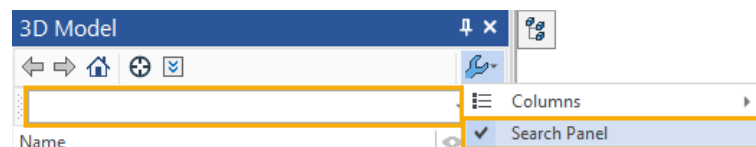
«3D Model» Window and Model Tree

The control panel of the **3D Model** window and the model tree has been reworked.

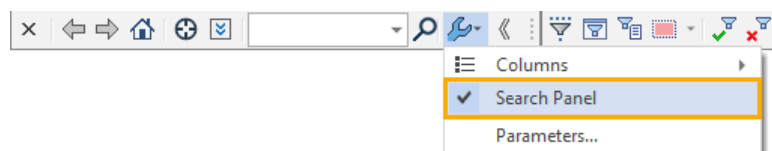


The parameters called by the right mouse button in the 3D model window moved to the parameter window, where the column settings were previously located.

As with all tool windows, in the **3D Model** window, the search panel is moved to a separate toolbar. Now it can be hidden by clicking on the corresponding item in the context menu.

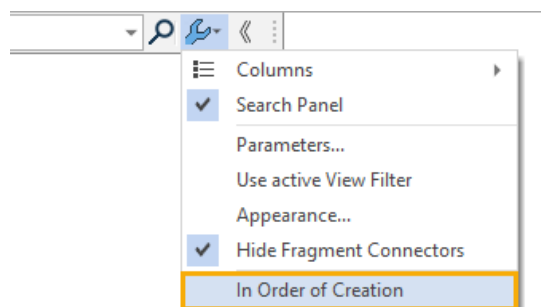


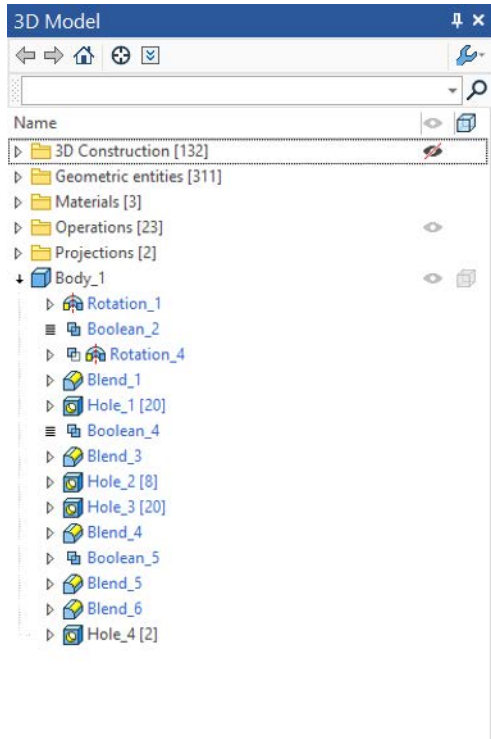
Similar changes were made to the search bar in the model tree.



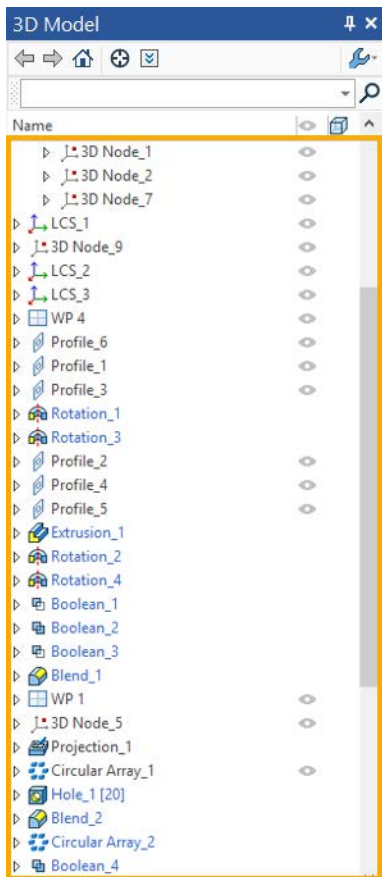
In the case of the model tree, the search bar is now disabled by default.

Also added the ability to sort created items in the order they were created.



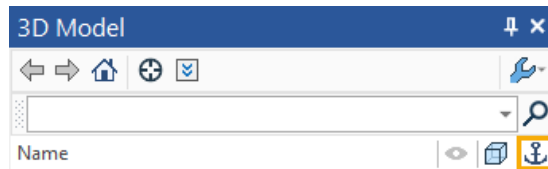


Standard View



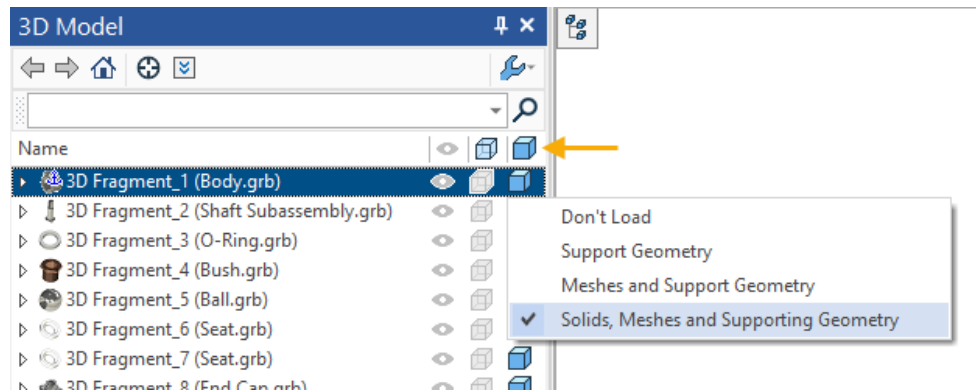
In Order of Creation View

Added **Fixed Position** column.



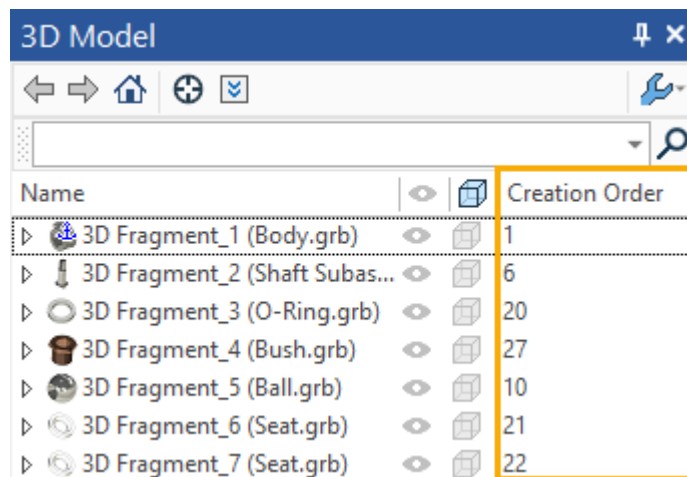
The column ensures quick assigning and control of fixed bodies.

Added **Load Contents** column.



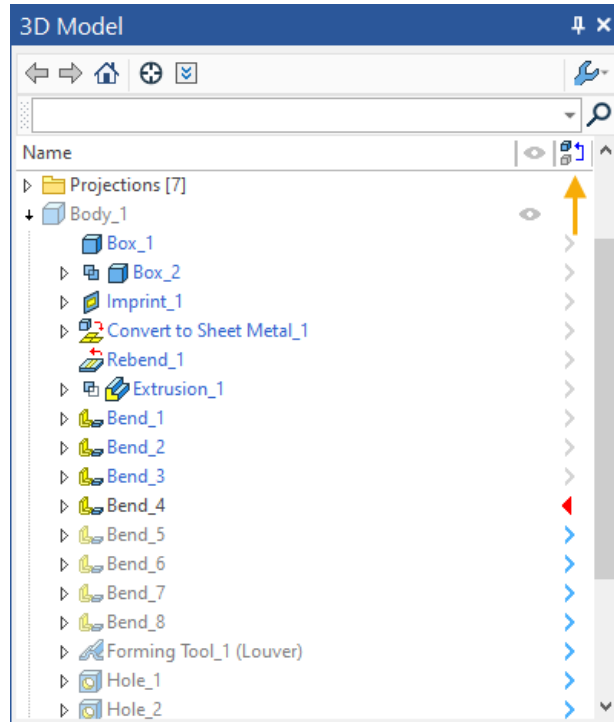
The column allows you to quickly select content download options for each fragment for performance and memory management.

Added **Creation Order** column.



The column allows you to display the order in which fragments are created in the model.

Added **Rollback** column.

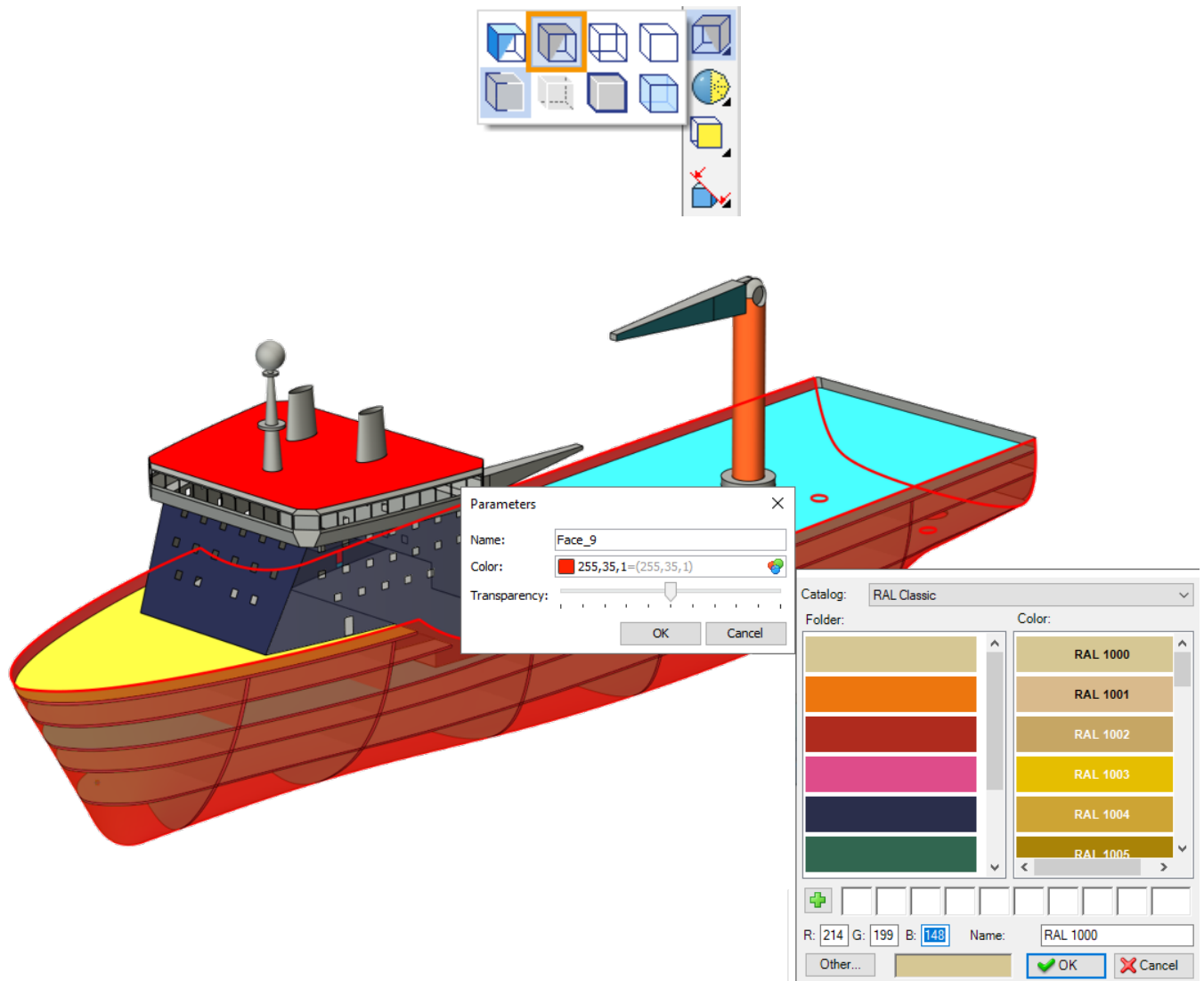


The column allows you to display the operations to which the rollback was performed.

Rollback works according to different principles with different modes of the model tree. When the **In Order of Creation** mode is active, the command rolls back the model according to the creation history, and when the standard mode is active it rolls back operations along the body construction chain.

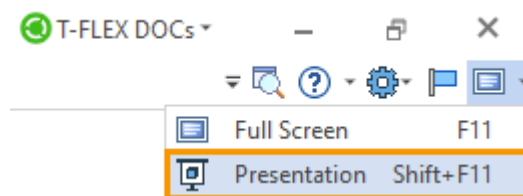
Setting Color and Transparency on Faces

Now it is possible to dynamically change the color and transparency of individual faces in the mode in the **Shading** mode:



Presentation Mode

A presentation mode was added.

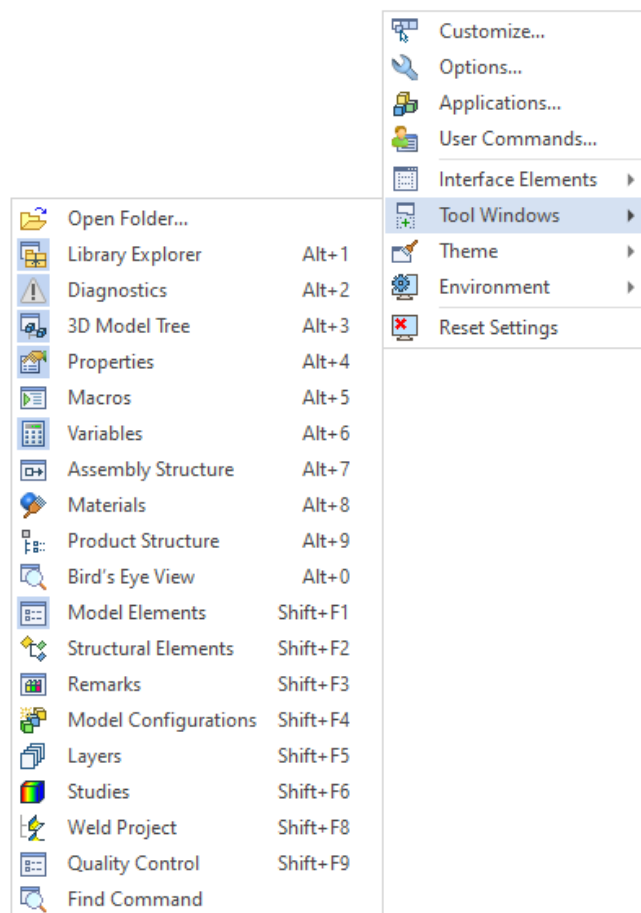


Only 3D scene remains in the presentation mode. 3D model tree is hidden. Ribbon is available in the full screen mode.



Hot Keys to Control Auxiliary Windows Visibility

Hot keys were added for auxiliary windows output commands.

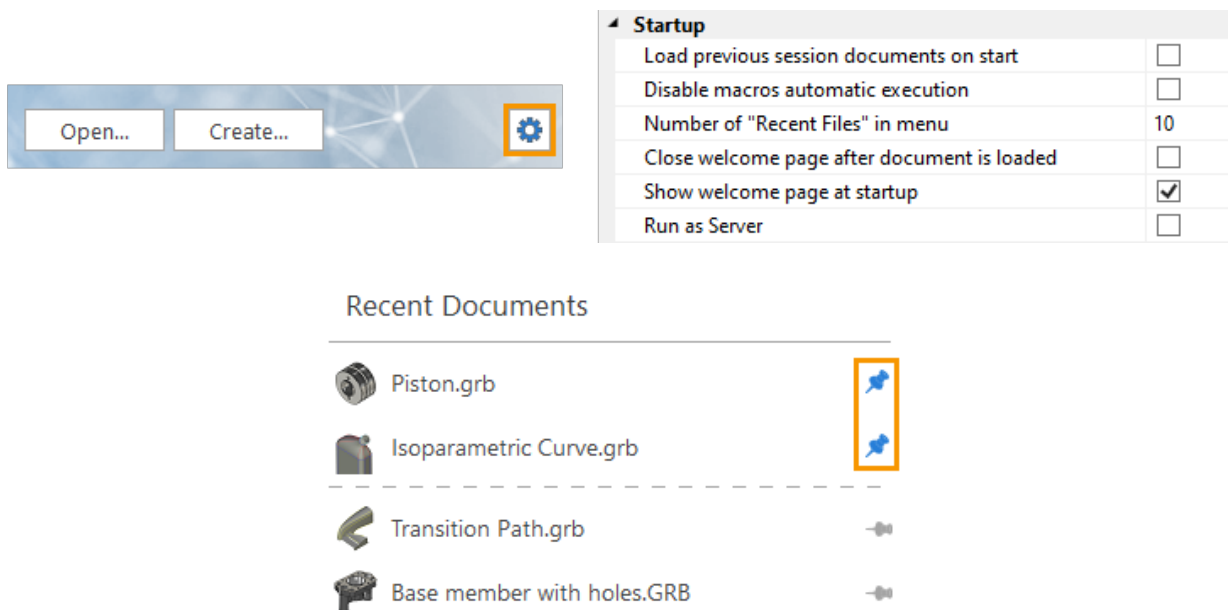


Start Page

Start page information is divided into thematic groups. Now all the necessary information, contacts, files and learning materials are easier to find.

Information	Learning Materials	Resources	Contacts
<ul style="list-style-type: none"> T-FLEX CAD 17 Release Notes T-FLEX Products Overview 	<ul style="list-style-type: none"> Tutorial Manuals (PDF) 	<ul style="list-style-type: none"> T-FLEX Resource Center Design Examples Free T-FLEX Viewer 	<ul style="list-style-type: none"> Official Web Site Official Forum Contact Us Technical Support 

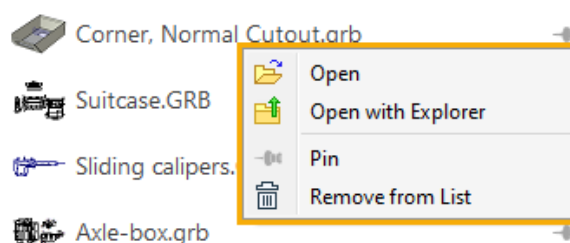
In addition, the appearance of the **Start Page** has been updated, the ability to pin files has been added, settings have been added at startup, and the resource center has been updated.



A context menu for recent documents that allows you to quickly navigate to the location of a file, open a file, remove and pin a file in the list has been added.

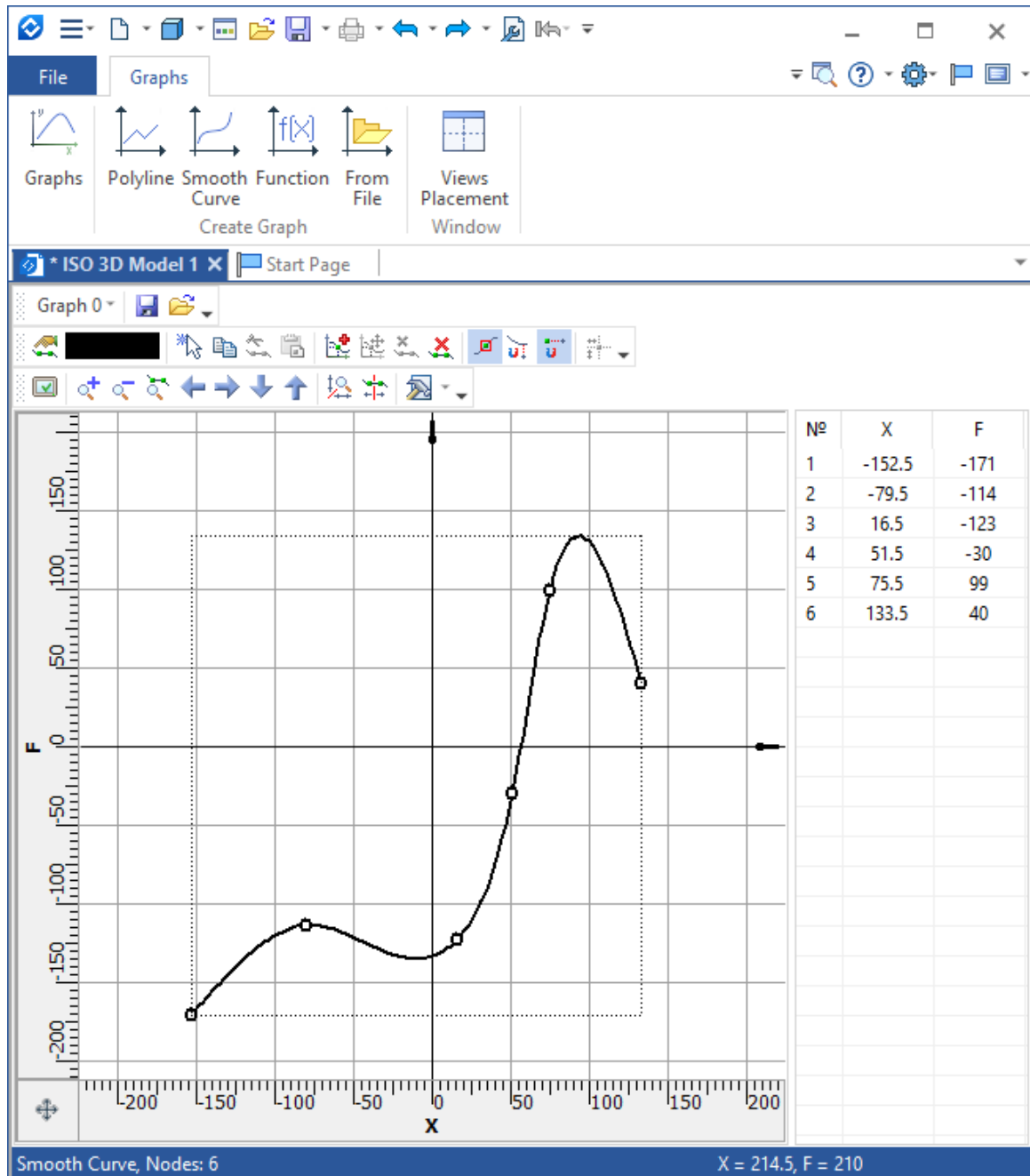
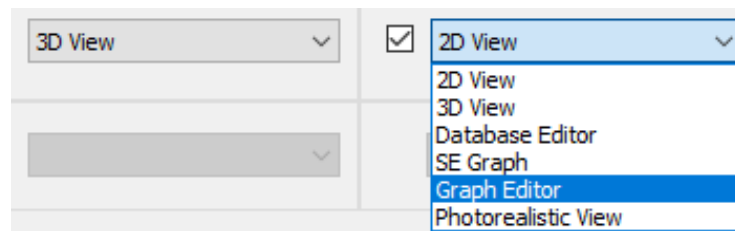


Recent Documents



Views Placement Command

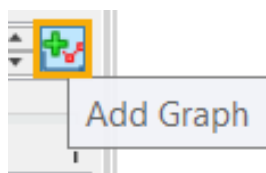
Using the **Views Placement** command, the Graph Editor can now be placed in a separate document window. When working with a graph, a special ribbon tab **Graphs** will be automatically activated.



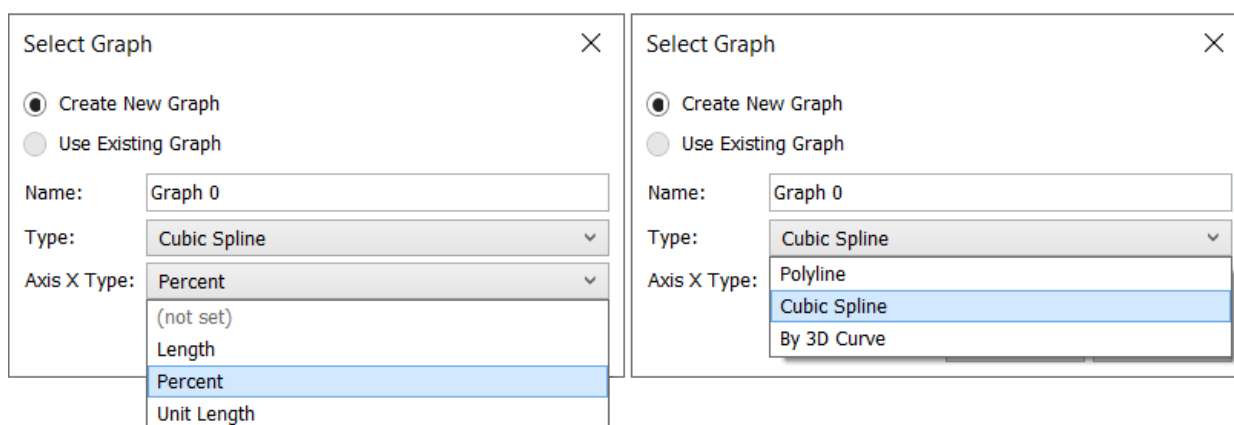
Graphs

Graphs have been improved, becoming a system-wide mechanism for managing various quantities in various operations. For example, by connecting a graph, you can control the magnitude of the continuation of the surface or the radius of the arc, thereby making these values not constant. You can control different values in different operations, for example, in operations **Transition surface** and **Extension by Law**.

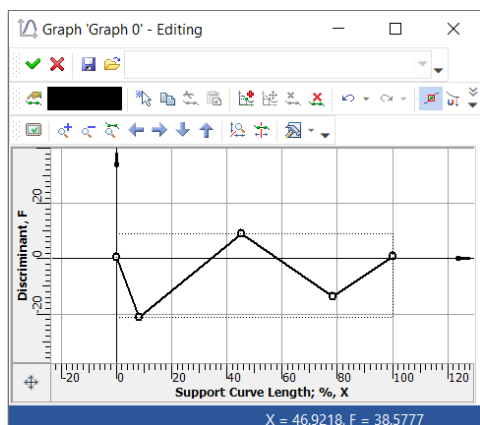
You can call the graph editor from the command dialog in which the creation of a graph is available.



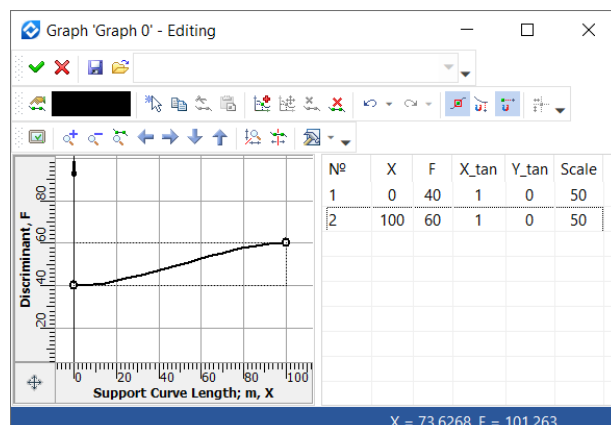
Special modes were added:



The types of graphs **Polyline** and **Cubic Spline** are used to linearly change the parameter and the cubic law of change. At each point of the cubic spline, the parameters of the angle of the tangent and the scale of the vector of the tangent are available. This allows to accurately describe any mathematical or empirical dependence.

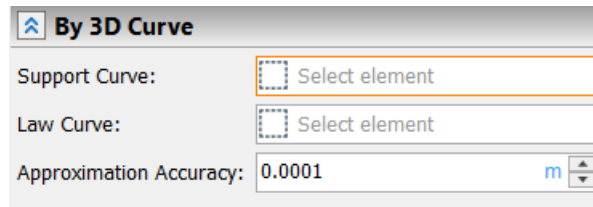


Graph type: **Polyline**

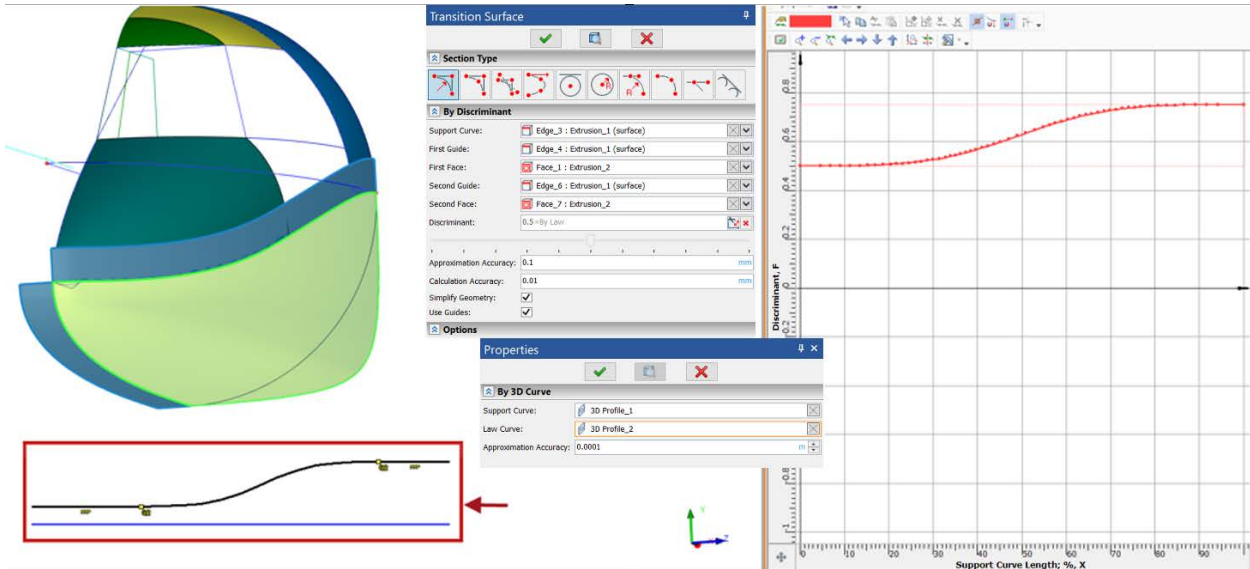


Graph type: **Cubic Spline**

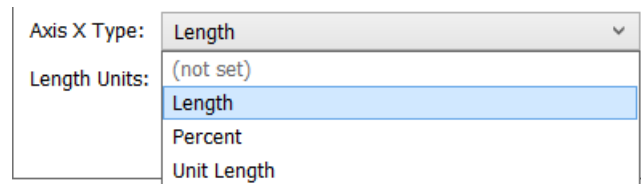
If the option of a graph type **By 3D Curve** was selected, the system will offer to select two curves in the 3D scene.



The first will determine the X axis: **Support Curve**. The second will determine the distance from the support curve, thereby setting the value of the function: **Law Curve**.



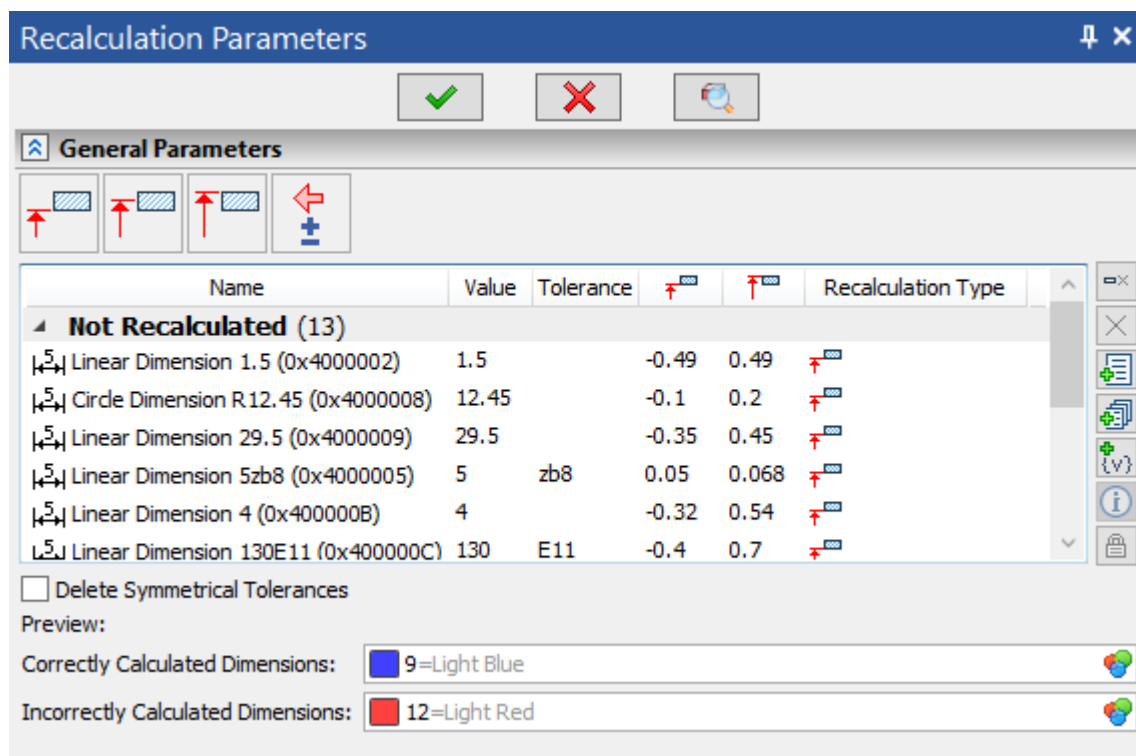
The dialog for creating a graph also allows you to determine the type of quantity along the X axis and the unit of measurement for length, if the option of the X axis as length is selected.



New types of graphs will be added to other commands in future as needed.

Recalculate Tolerances

The command interface was improved. As for other commands, an editable list of selected elements is available in the new interface.



For each selected dimension, you can specify your own recalculation option.

	Recalculate to Lower Limit of Tolerance
	Recalculate Middle of Tolerance
	Recalculate to Upper Limit of Tolerance
	Restore Tolerance Values

The main innovation of the command is the ability to restore tolerance values and geometrical dimensions: in the previous versions it was only possible to restore model dimensions by cancelling the action. Now you can restore the initial dimensions of the model at any design stage with the help of a special command mode. There is also a feature of recalculating to lower and upper limits of tolerance.

Import of Models and Drawings

Added new formats: Revit, FBX.

Added support for formats of new versions of systems: Inventor, SolidWorks, Solid Edge, Parasolid, JT, Revit, NX, ACIS.

Improved AutoCAD import, added support for the latest versions, and also added the ability to import from AutoCAD and export to AutoCAD gradient fills.

For AutoCAD, STEP, IGES, ACIS, SolidWorks, Autodesk Inventor, NX - Unigraphics, Creo – Pro/E, CATIA, Solid Edge, Rhino, Revit, I-DEAS, VDA-FS, JT, PRC, CGR, U3D, FBX formats added import option layers.

For CATIA, SolidWorks, Creo - Pro/E, I-DEAS, JT, NX – Unigraphics, PRC, Solid Edge, STEP formats added the option to import 3D PMI elements (annotations).

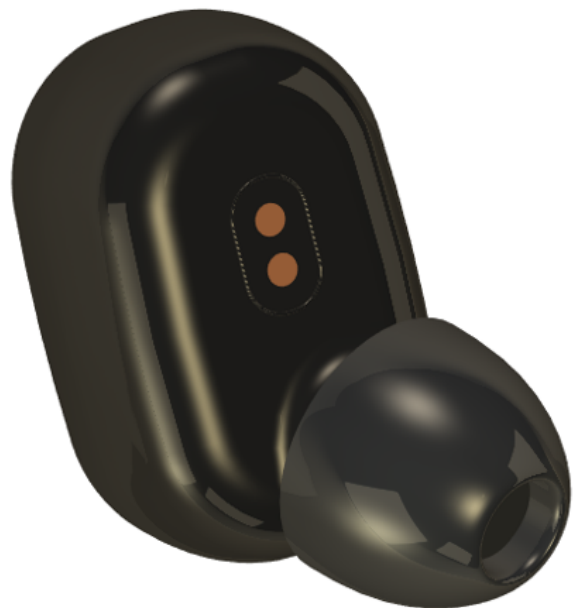
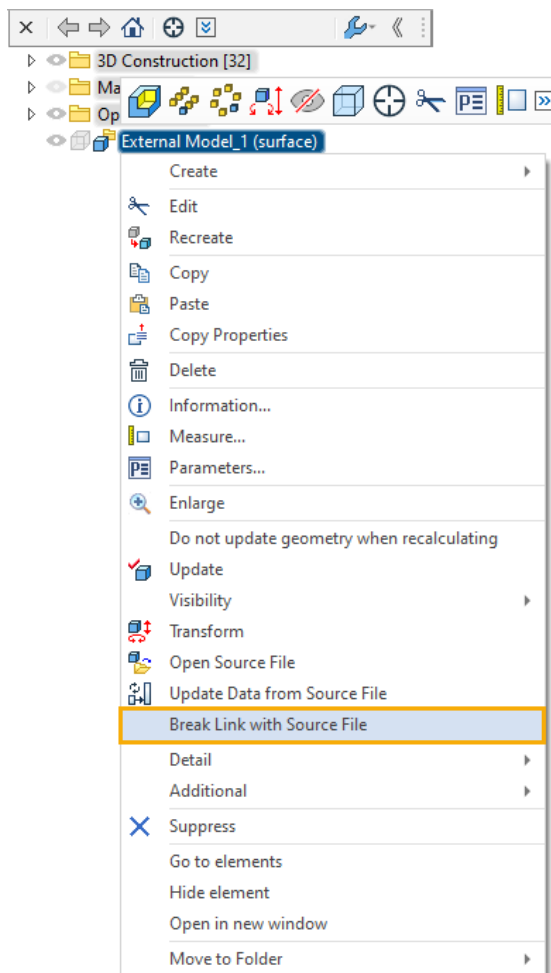
Added the ability to import drawings from KOMPAS v.18/19.

Improved automatic generation of the Product Composition when importing assembly models.

In the **T-FLEX CAD Extended Import** module, the ability to directly read the CATIA V5 format has been added, as well as support for new versions of Creo systems.

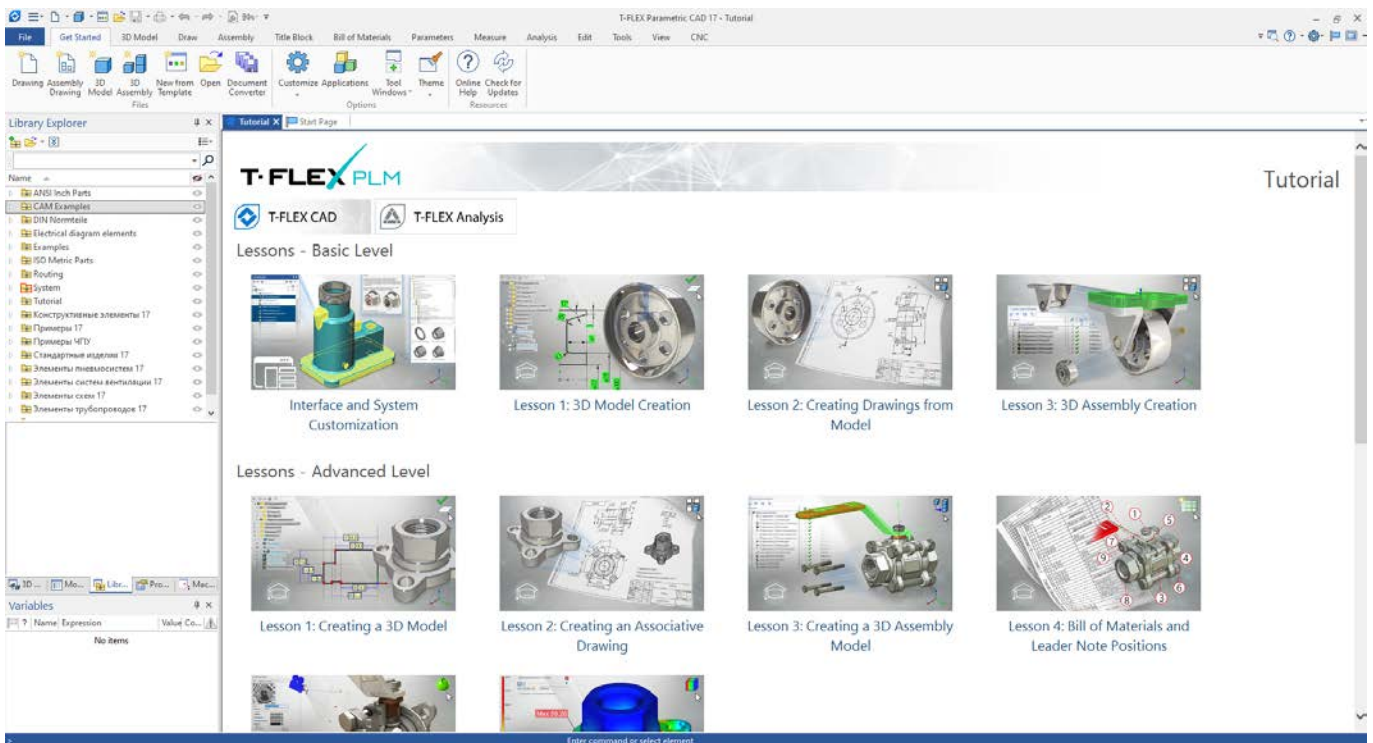
Break Link with Source File

Added the ability to break the link with the source file of the external model. When this option is activated, all parameters associated with the source file will be removed from the operation context menu.



T-FLEX Tutorial 17

The tutorial interface has been updated.

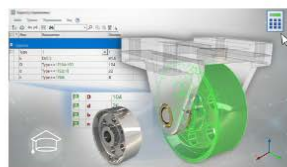


The section on parameterization has been significantly reworked.

Lessons - Parametrization



Lesson 1: Parametric 3D Model Creation



Lesson 2: Parametric 3D Assembly Creation



Lesson 3: Control Dialog Creation



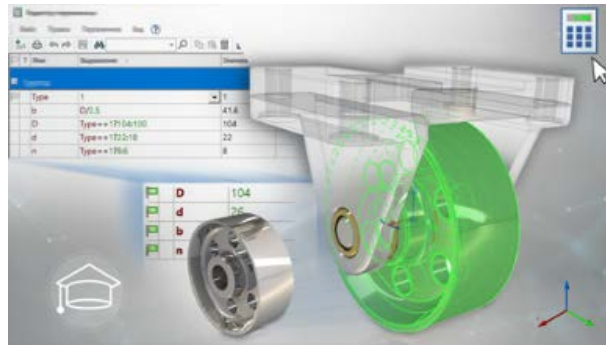
Lesson 4: Library Item Creation

Added a lesson on creating a parametric 3D model.



This lesson describes how to create a parametric model based on two methods of sketching - using construction lines with graphic strokes and using sketch elements, constraints, and driving dimensions. In this lesson, the user will get acquainted in detail with the concepts of a **variable**, an **external variable** and a **variable editor**.

Reworked the lesson on creating a parametric 3D assembly.



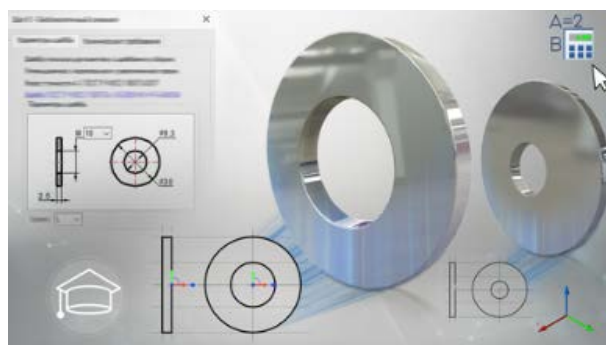
The parametric 3D assembly now uses the 3D assembly described in the basic level of the tutorial, making the tutorial much easier to understand. In this lesson, parametric 3D models are gradually inserted into a pre-prepared 3D assembly, the external variables of which are transferred to the assembly level, after which the 3D assembly becomes parametric.

Reworked the lesson on creating a control dialog.



Now, the roller created in the first lesson of the basic level is used as a model for creating the control dialog. This lesson describes in detail how to create a convenient control dialog with which you can quickly control the main parameters of a part.

Reworked the lesson on creating a library element.

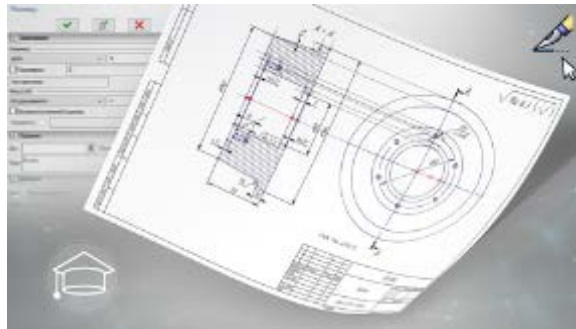


Now the washer **Washer ISO 10673** is used as the model under consideration. This lesson is the final one in the parameterization section and includes all the information learned by the user in the previous lessons. The lesson explores the classic approach of creating a model - the 2D to 3D method. The lesson is divided into the following stages:

- Database Creation
- Creating Control Variables

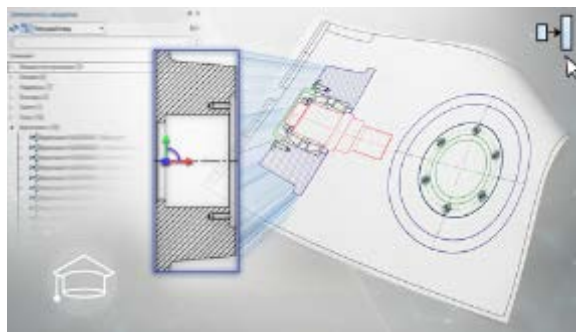
- Parametric 2D Drawing Creation
- Parametric 3D Model Creation
- Fixing Vector and 2D Connector Creation
- LCS and 3D Connector Creation
- Control Dialog Creation
- Setting Data for BOM
- Preview and Icon Creation

Added a lesson on creating a 2D drawing.



This lesson focuses on working with 2D drawings, namely, it describes the creation of a parametric 2D drawing, its design, the creation of layers and reference vectors.

Added a lesson on creating a 2D assembly.

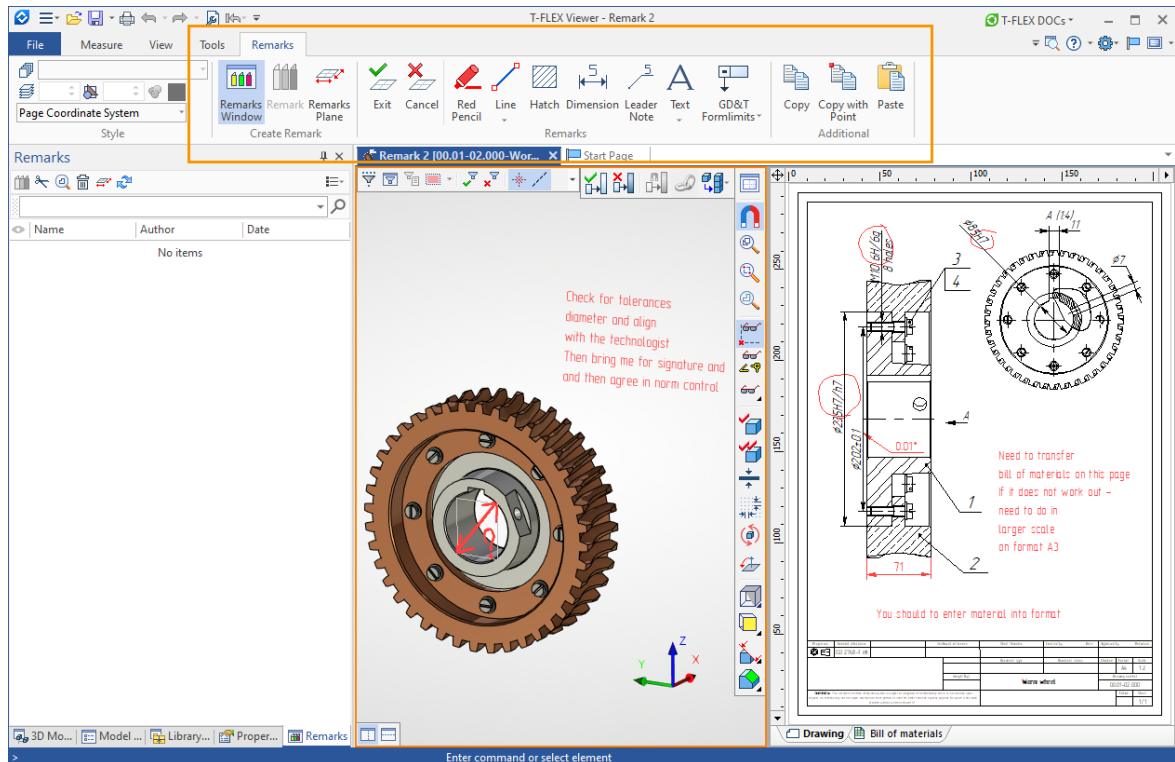


This lesson describes how to work with 2D assemblies. As one fragment, the drawing obtained in the lesson on creating a 2D drawing is used. In the course of the lesson, various fragments are sequentially inserted from the document menu, including standard parts. Next, fixing vectors are created, positions are placed and the bill of materials is filled in.

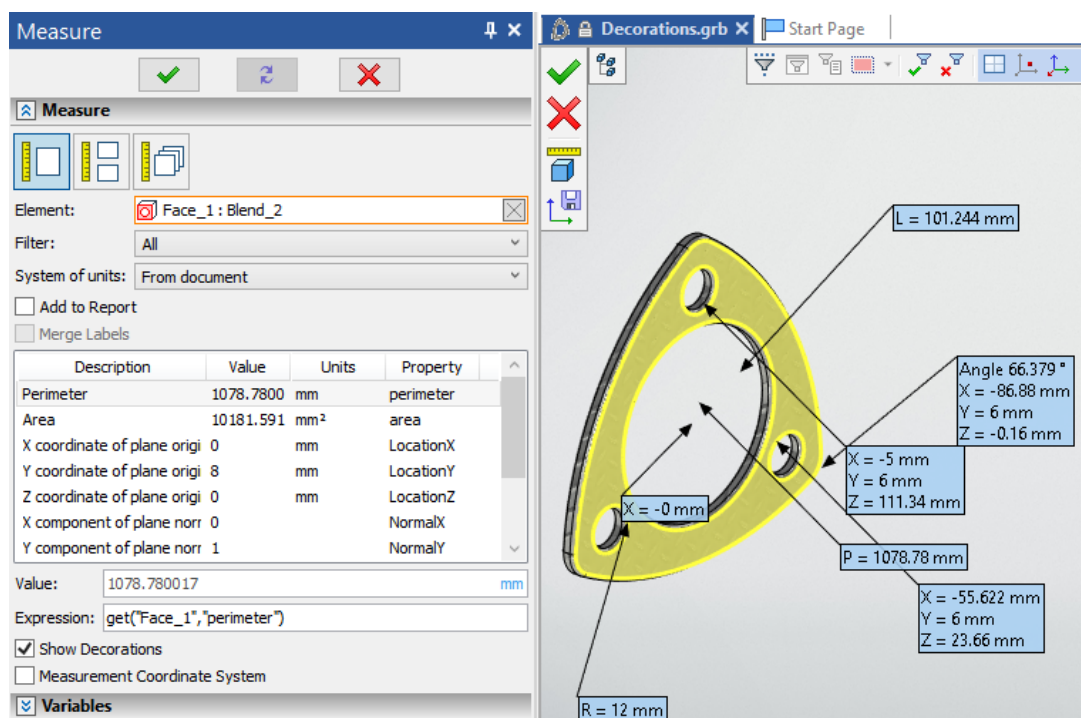
T-FLEX Viewer 17

T-FLEX Viewer has become more convenient tool for quick viewing and monitoring of projects during teamwork.

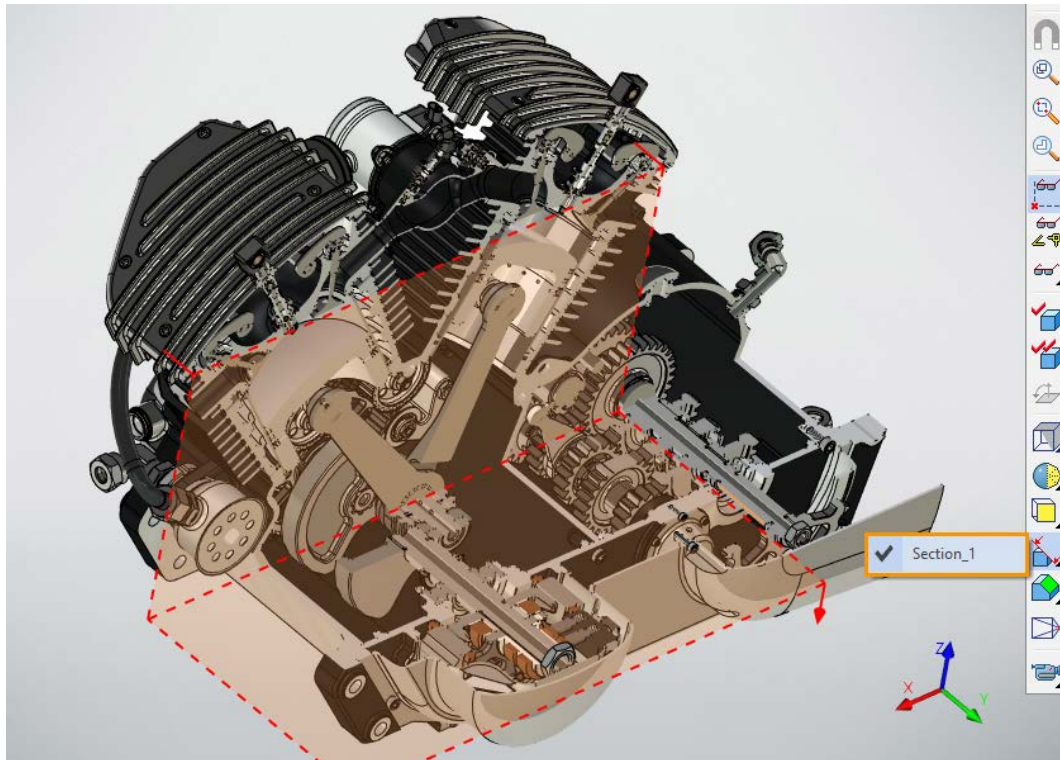
- A new remarks mechanism is available, including work in the **Remarks** window.



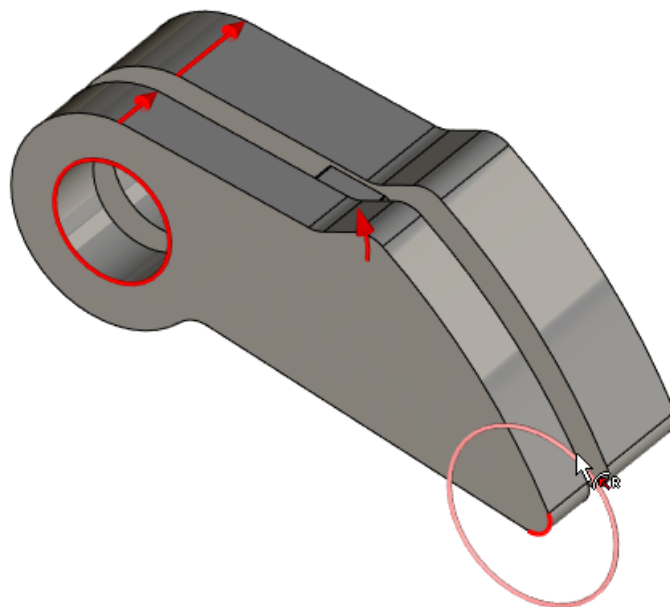
- You can measure models using the **Measure** command and calculate mass-inertial characteristics.



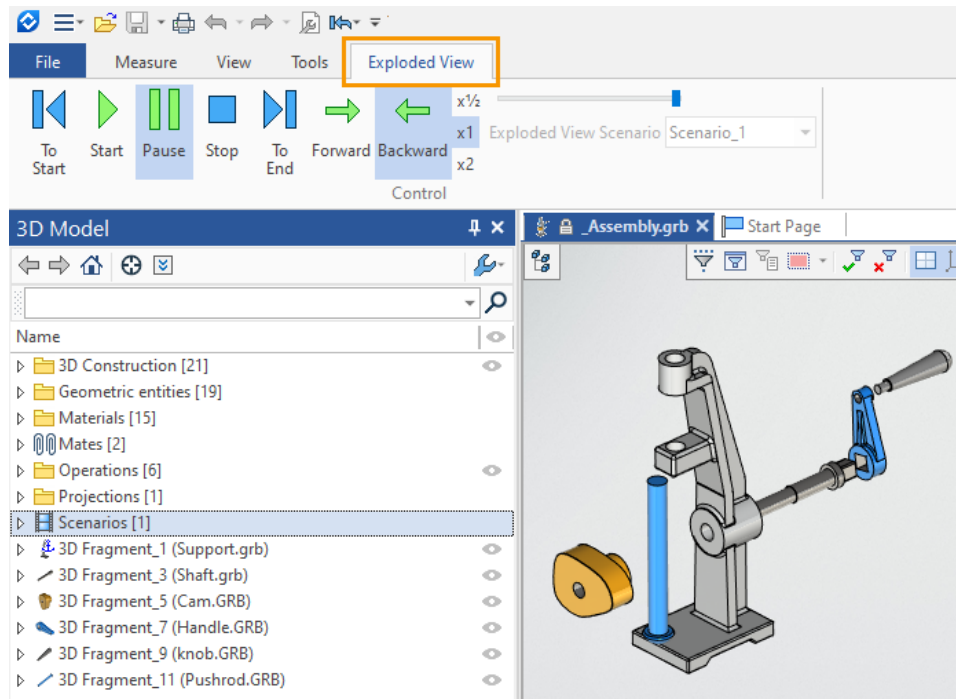
- For a detailed view of the models, commands for using 3D sections and clipping planes are available, in addition, visual display options are available, as well as options for models searching in the 3D scene and the **3D Model** and **Assembly Structure** windows.



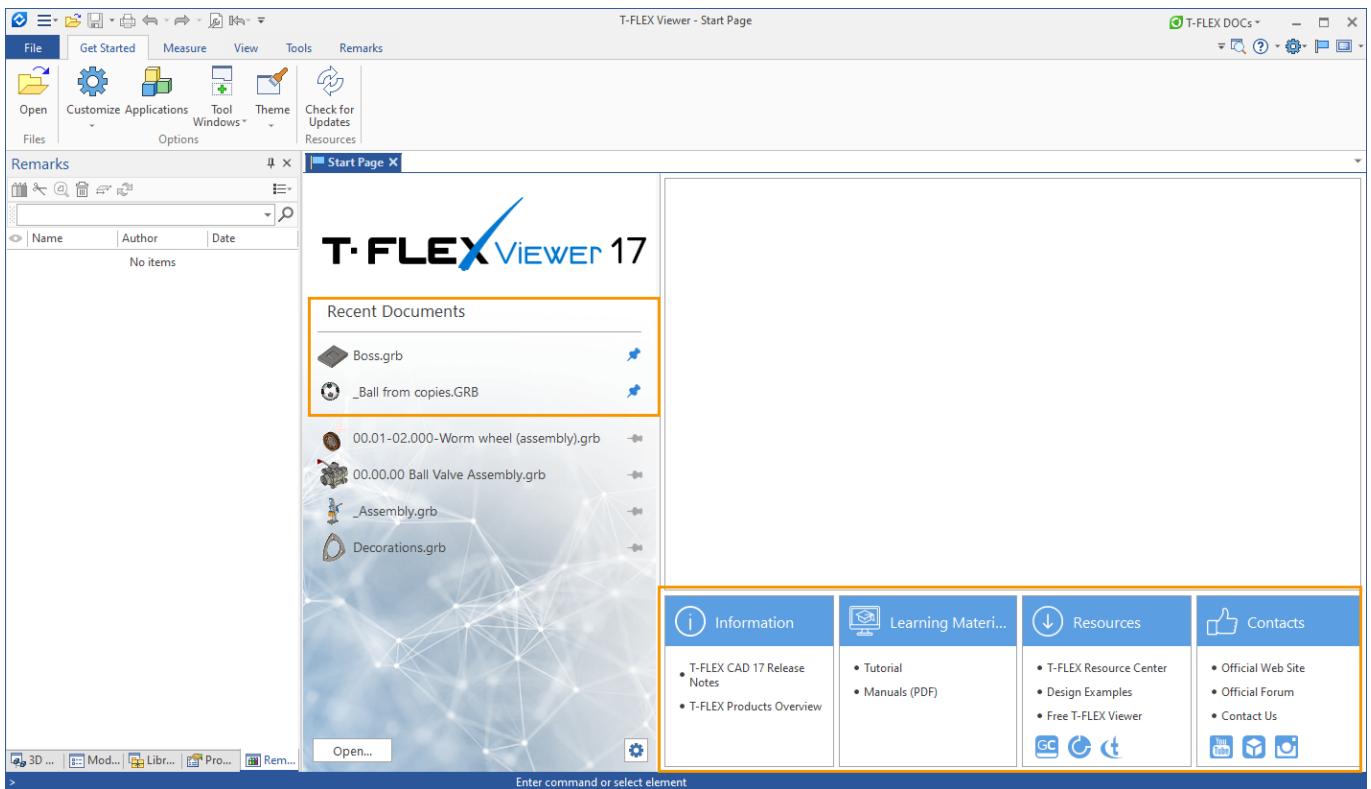
- 3D models can be edited using manipulators and external variables



T-FLEX Viewer supports an exploded/unexploded view scenario player, which makes it possible to use 3D models as instructions for the assembler in production.

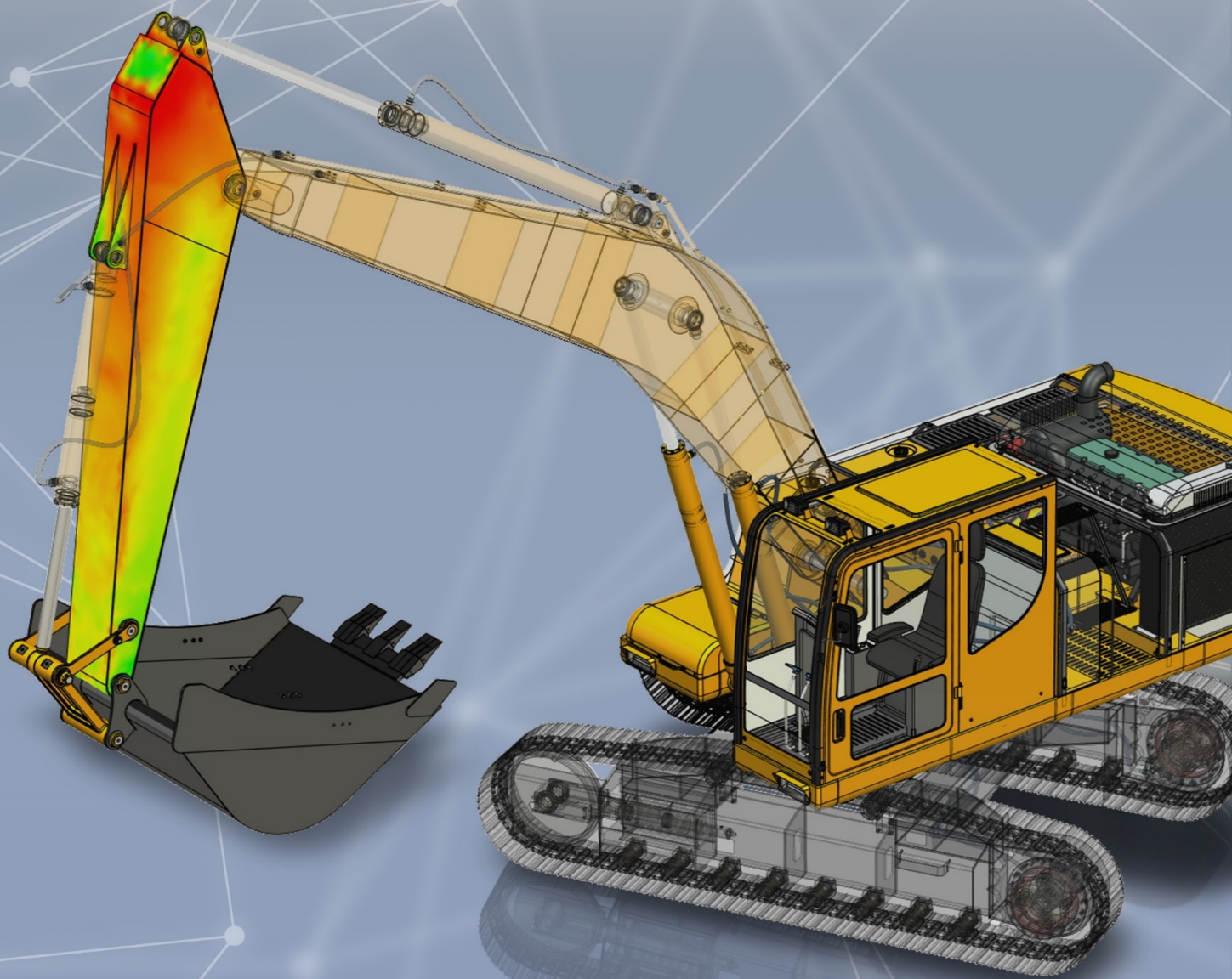


Work in the T-FLEX Viewer now starts with the start page, where, like in T-FLEX CAD, the fixation of important files is available, as well as other new features of the start page described earlier.



T·FLEX Analysis 17

What's New

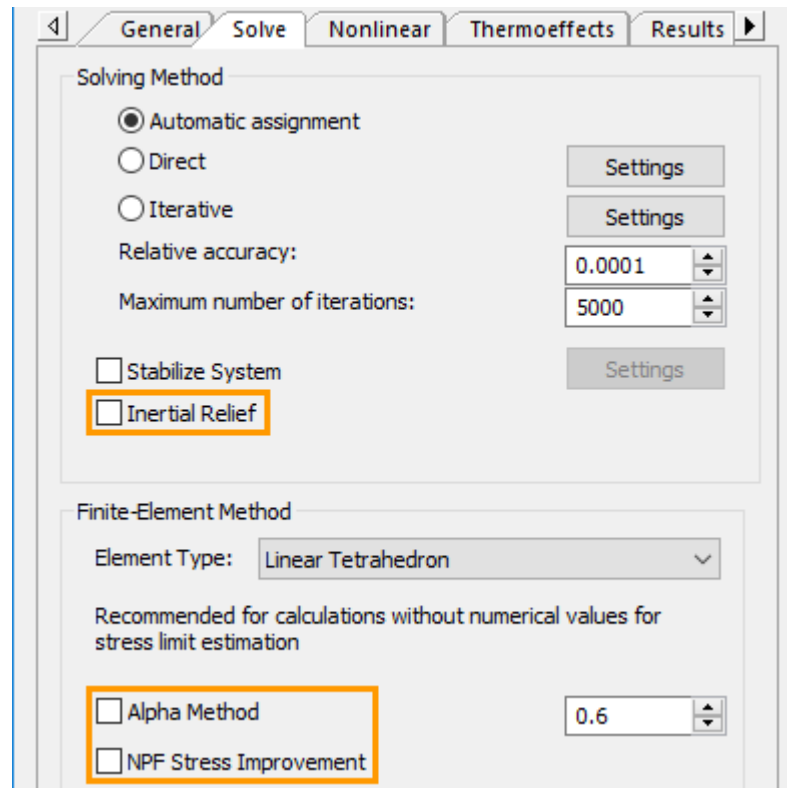


T-FLEX Analysis 17

The Analysis module is developed in following directions: new types of elements, new calculation methods, modeling of new physical effects, interface updates. As a result, Analysis became more convenient, much faster and more stable, and now solves more tasks.

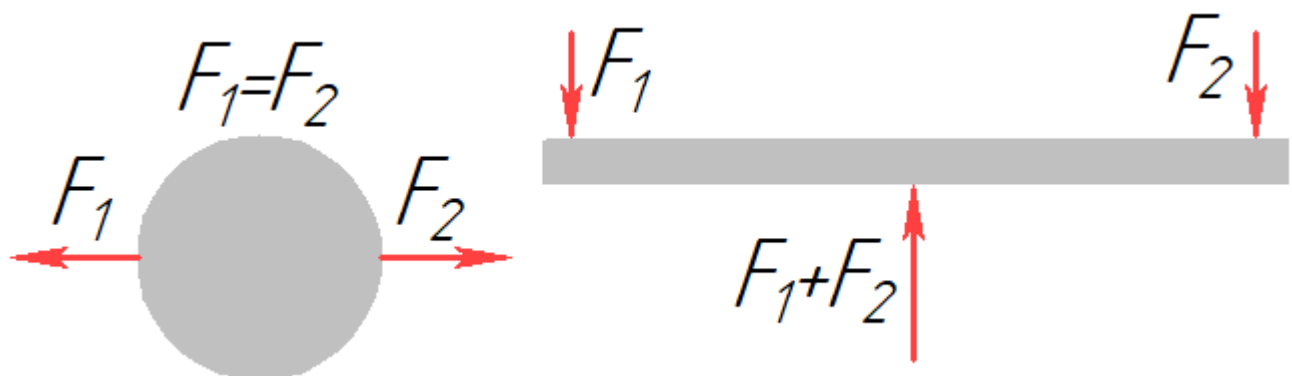
New Algorithms for Solving

New options are available in the task parameters dialog on the **Solve** tab. Calculation on linear tetrahedrons is now more accurate, and you can now stabilize the system in two ways. A new stabilization option has been added to the previously available one: **Inertial Relief**.



Inertial Relief

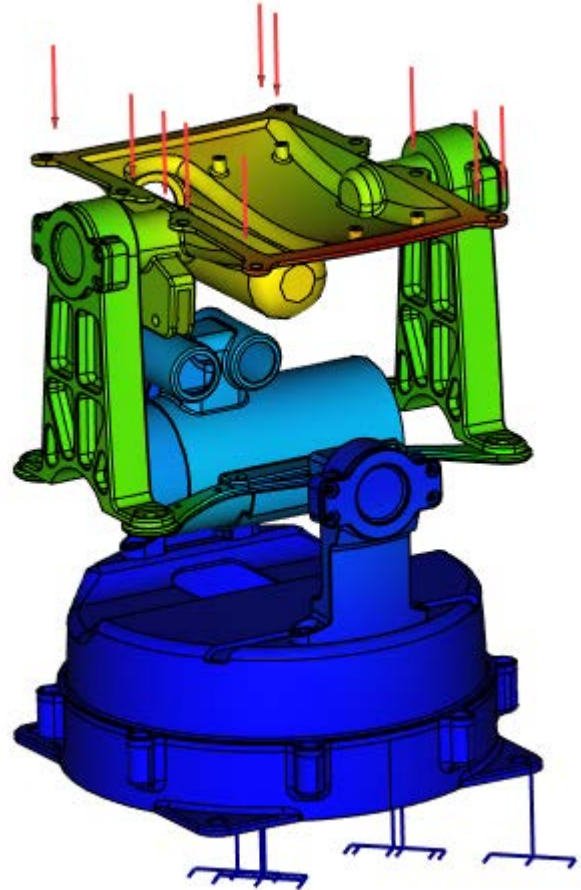
In contrast to stabilization, which is aimed at calculating studies with insufficient restraints, **Inertial Relief** enables you to solve studies where the equilibrium of the system is a consequence of the balance of forces.



Due to some errors in the numerical method for solving idealized studies with force balancing, special methods for stabilizing the calculation model are required. They are implemented by the **Inertial Relief** option.

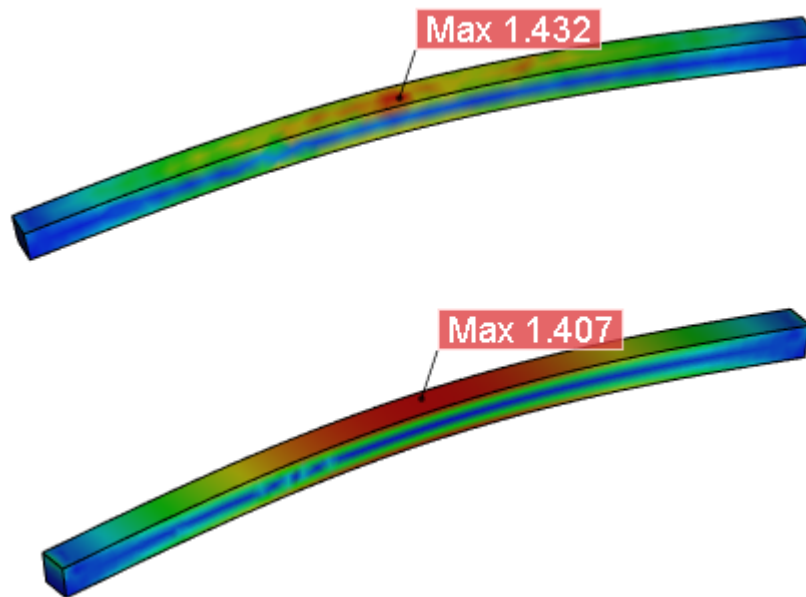
Smoothed FEM

For a linear tetrahedron, the **Alpha Method** option was added. It implements "smoothed FEM" algorithms when solving studies. The algorithms lead to more accurate results for both displacements and stresses. The resulting system of equations is simpler and faster to solve than a quadratic tetrahedron, which makes it rational to use this method on tetrahedral grids with a large number of finite elements. For the shown model, which contains 215198 elements after digitization, the solution time using a linear tetrahedron with the alpha method is 12 seconds. The solution time using a quadratic element is 35 seconds, which is almost 3 times longer.



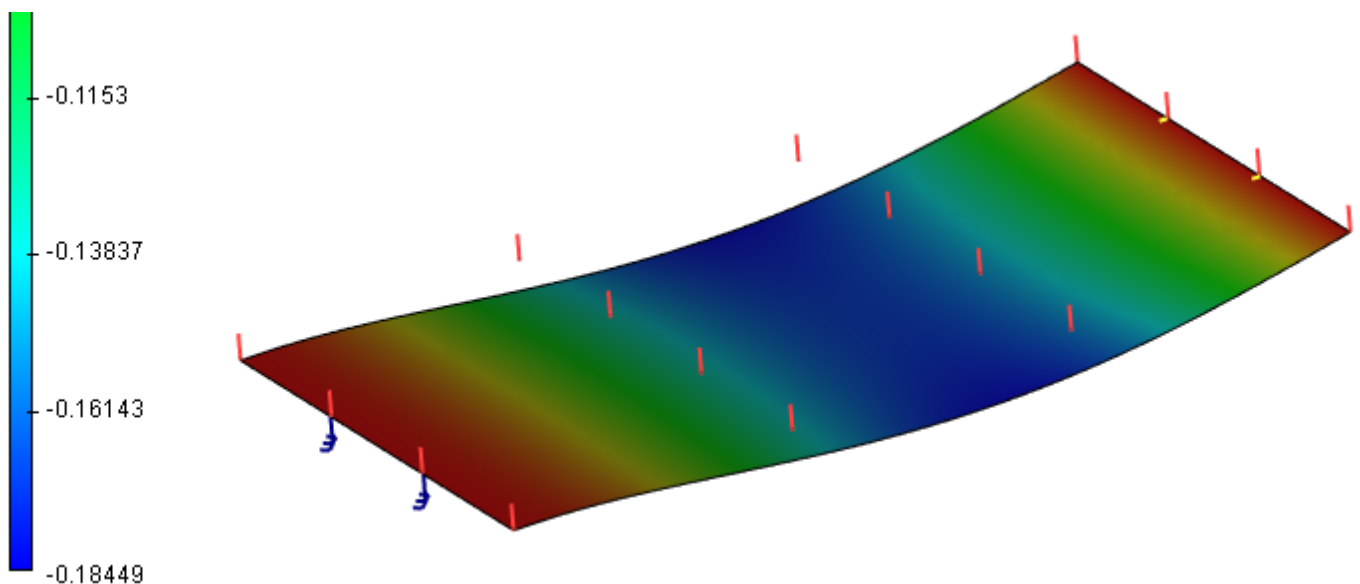
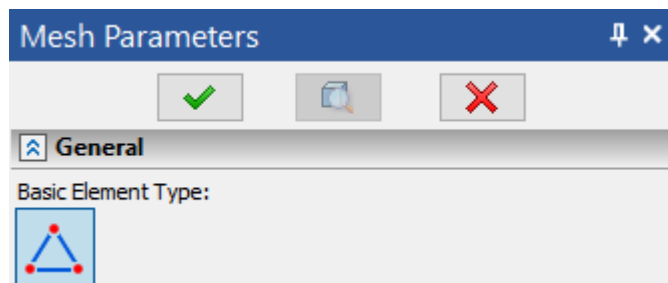
Improved Stress Calculation

Another new option for calculating linear tetrahedrons is **NPF Stress Improvement**. This algorithm makes it possible to obtain a fairly accurate calculation of stresses even on a relatively rough grid of linear tetrahedrons. In comparison with the calculation on a quadratic tetrahedron, the relative error for maximum voltages in the verification example is 1.8%. At the same time, the calculation speed on linear tetrahedrons can be several times faster than on quadratic ones, which depends on the number of elements in the calculation model and its complexity.



Calculation of Shells by a Three-node Element

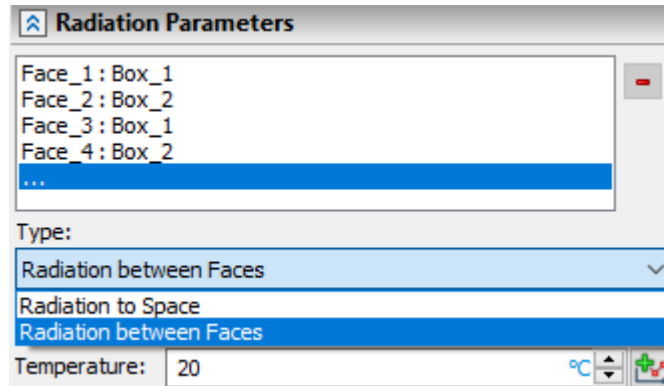
The calculation of the loading of thin-walled parts and structures using shell elements became faster and more accurate. The new three-node shell element enables you to get a solution with accuracy comparable to or higher than the six-node element.



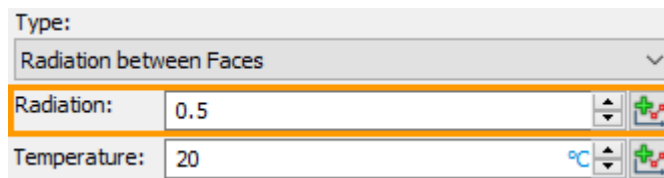
Analytical Calculation of Displacements	Numerical Calculation of Displacements	Error
-0.18455 m	-0.18499 m	0.24 %

New Type of Heat Exchange

In T-FLEX Analysis, you can now calculate the heat exchange by radiation between the surfaces of bodies. You should set the radiation heat exchange conditions in the **Radiation** command in the group of temperature loads and boundary conditions. You can select one of the calculation modes **Radiation to Space** or **Radiation between Faces** in the special list of the command dialog. If you select the **Radiation between Faces** option, the "visibility" of other elements of the radiating surfaces will be calculated for each element of the radiating surface. If there are empty areas of visibility, the radiation heat exchange will be calculated as heat exchange with the environment.

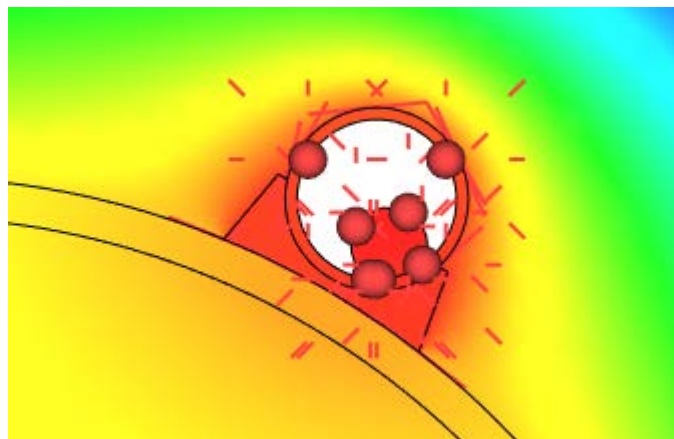


The degree of blackness of surfaces is set by the **Radiation** parameter.



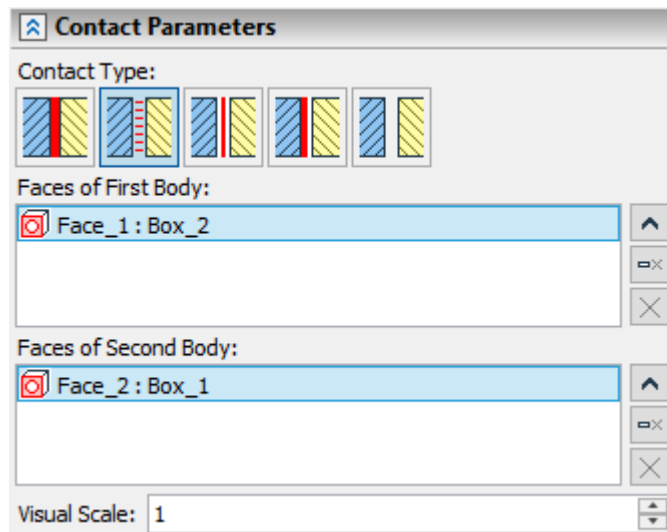
The **Radiation** parameter can be set as a function of time or temperature. **Temperature** can be set as a function of time.

The heat exchange between the selected surfaces of bodies will be calculated by radiation, together with other thermal loads assigned in the study.



Contact

The command dialog has a new interface.



Contact types can be switched using icons.

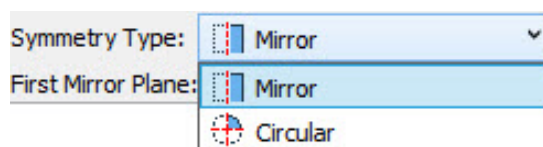
	Rigid Constraint
	Tangency
	Gap
	Rigid Wall
	No Contact

The faces of the first and second bodies are divided into windows. In the windows, you can use the icons and to delete extra faces. You can use the icons and to add new ones.

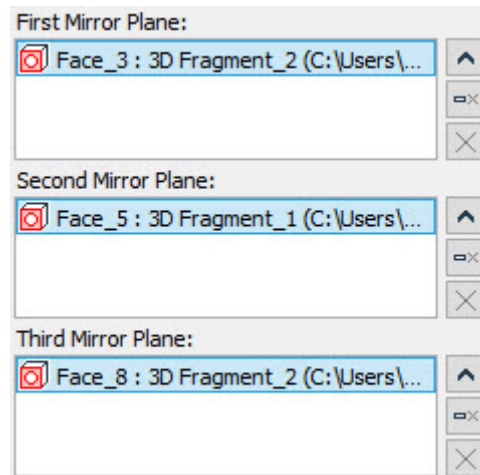
A new type of contact **Gap** allows you to specify faces that in the initial position of the bodies do not contact, but it is necessary to consider the contact in the process of solving, when the faces will touch when the bodies are deformed.


Symmetry

Two types of symmetry are available: **Mirror** and **Circular**.



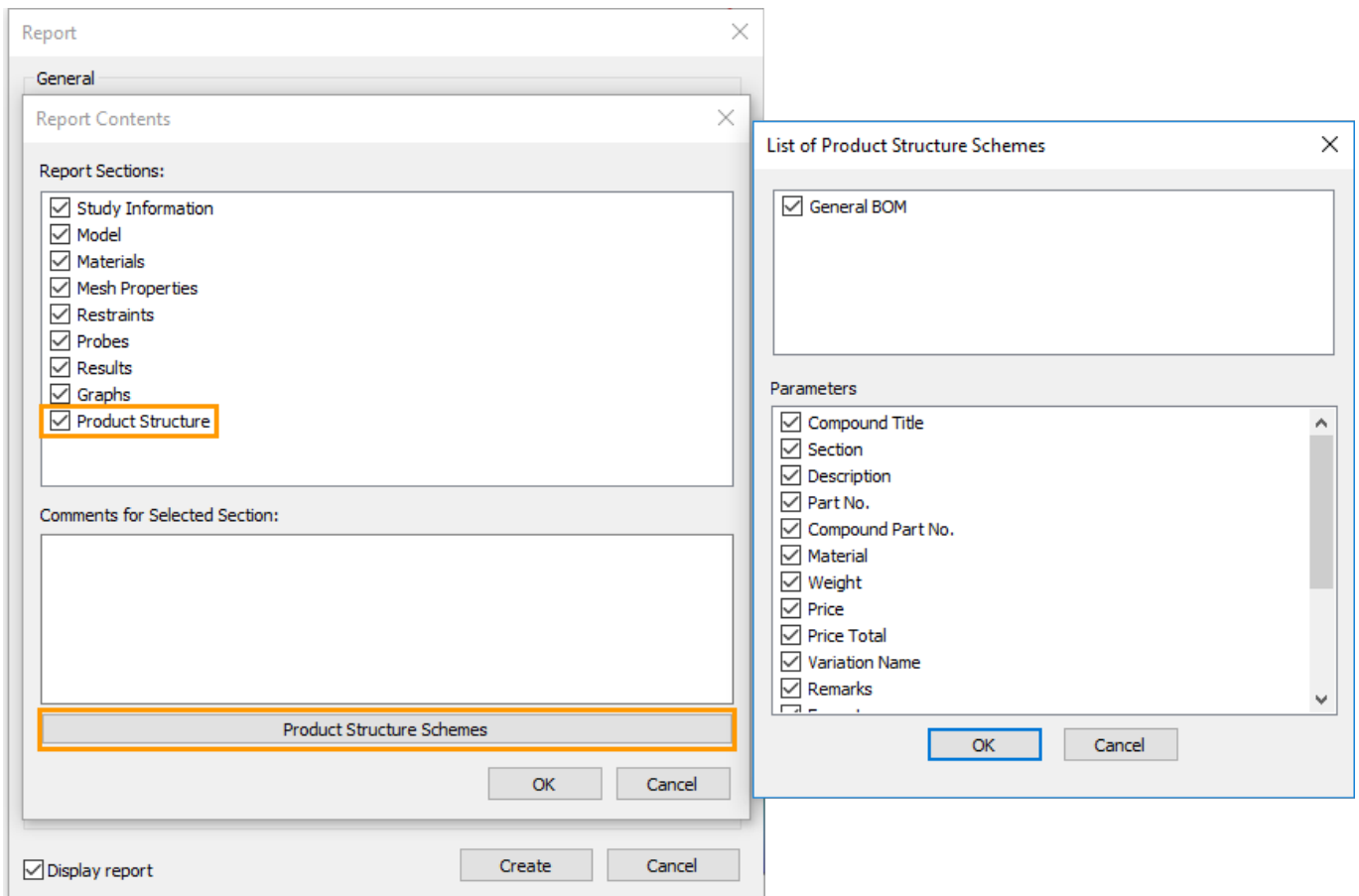
Circular symmetry is helpful when calculating axisymmetric structures. Mirror symmetry allows you to specify symmetry relative to one, two or three planes. Each selected symmetry plane is placed in the editable field.



On all faces lying in the same plane as the selected one, **Symmetry** constraint will be assigned. In addition, you can set **Symmetry** constraint on such faces explicitly using **Select all faces in plane** option .

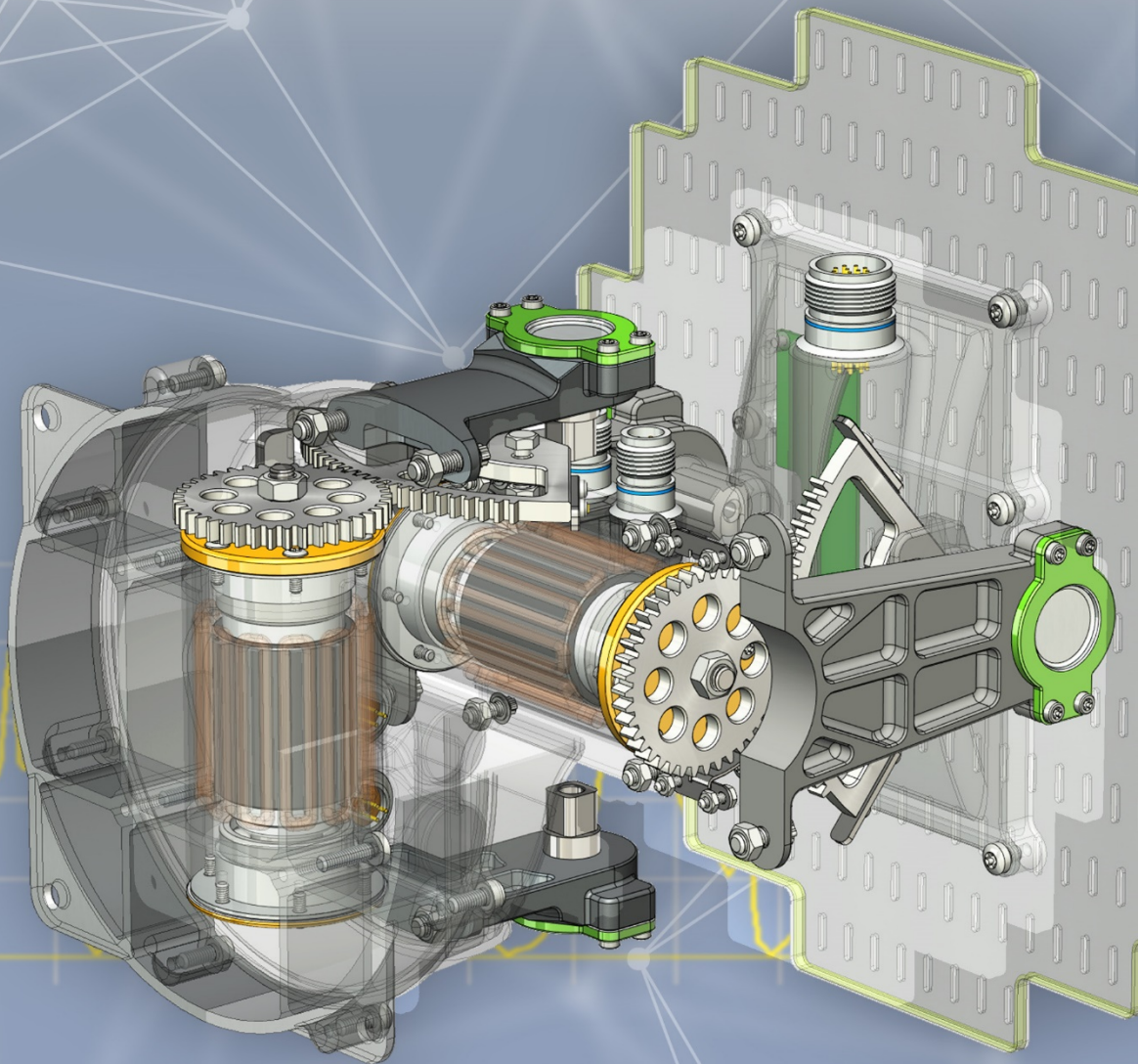
Product Composition in Analysis report

Now, when you create a report in the Analysis, you can output information from the Product Composition. You can configure the number of columns displayed.



T·FLEX Dynamics 17

What's New

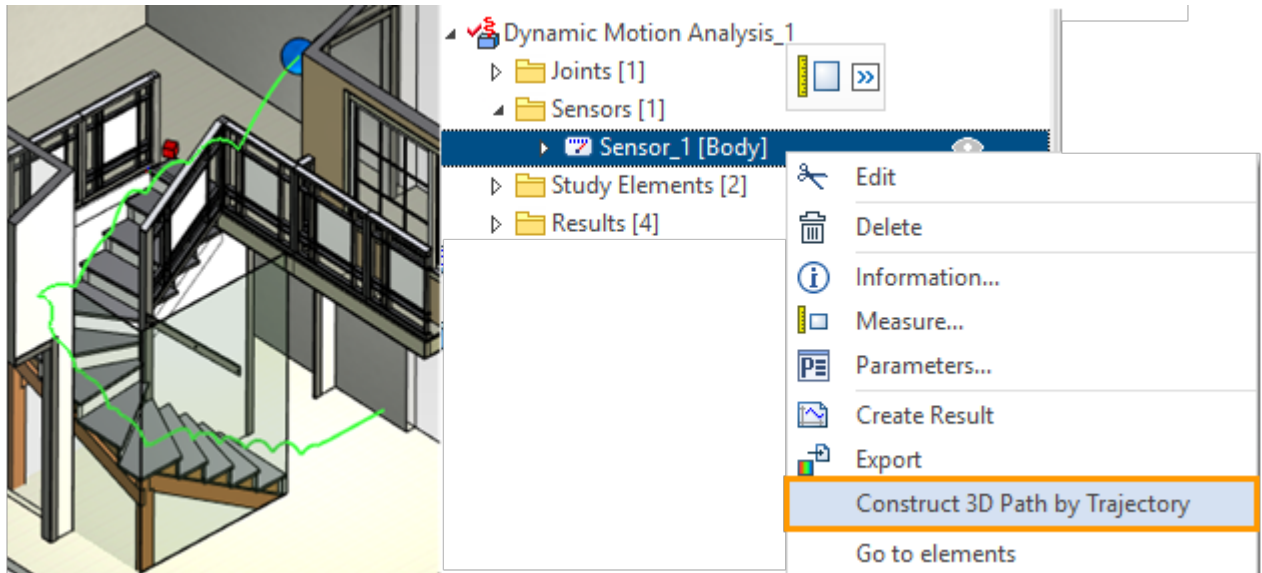


T-FLEX Dynamics 17

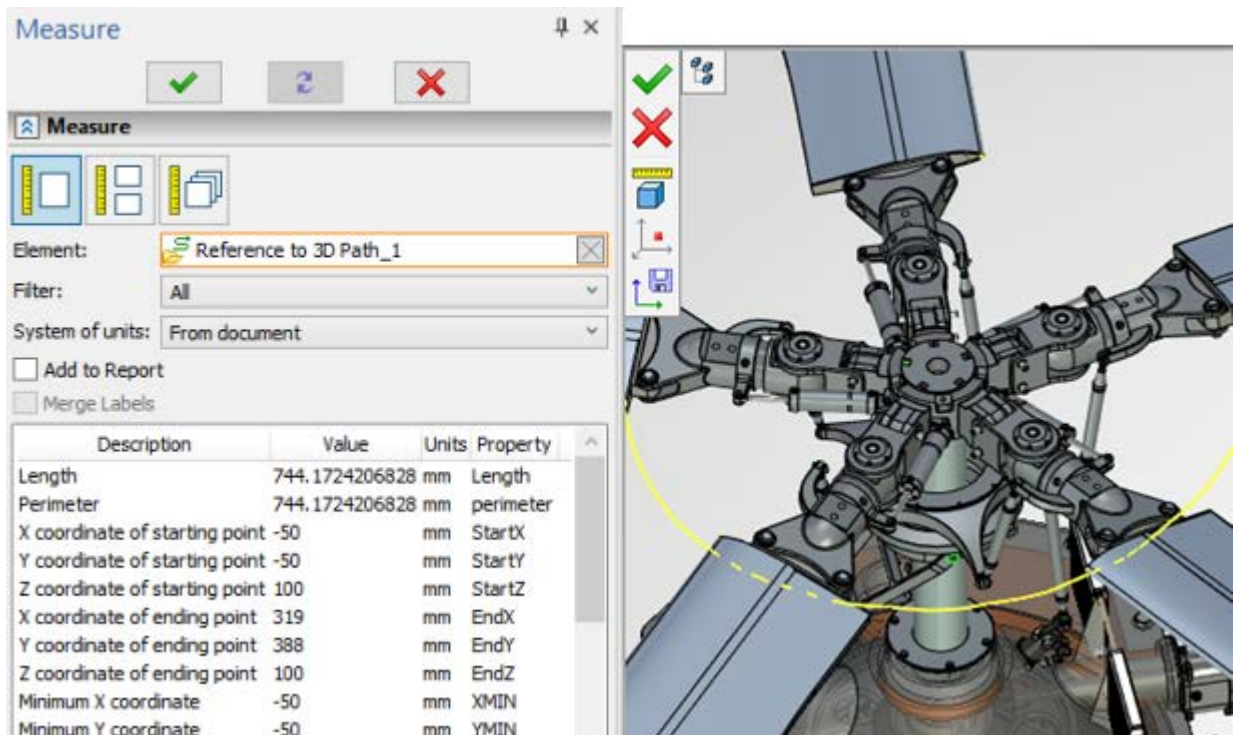
The resulting calculations in Dynamics are now easier to analyze. Any calculated state of the model can be transferred to CAD and T-FLEX Analysis. Calculations of mates, the accuracy of the calculations and the stability of calculations were improved.

Path Along the Movement Trajectory

Now you can create a 3D path along the trajectory of the sensors. This enables you to analyze the trajectory of bodies using CAD measurement tools. You can also use the path as a CAD object to create bodies and construction elements.



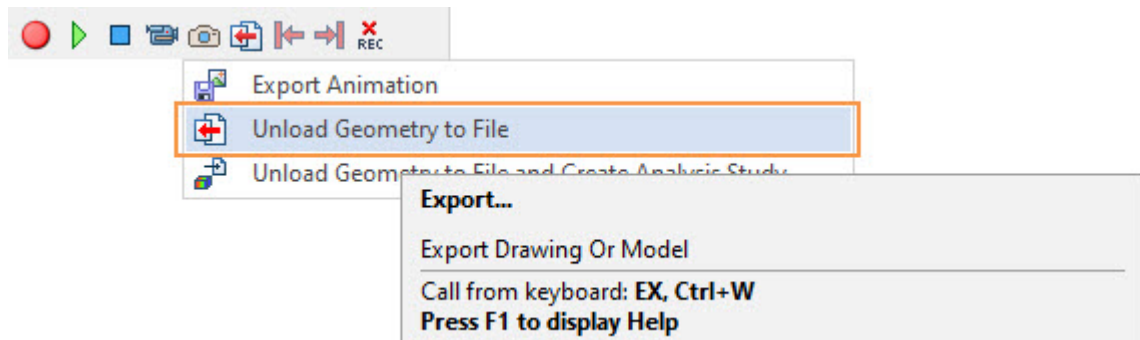
Use the **Measure** command to analyze the path-trajectory.



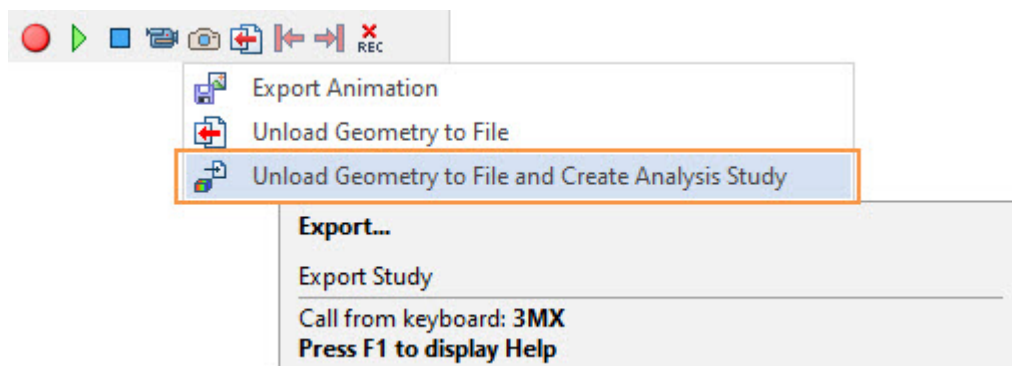
Unloading Model Calculated States

Now you can unload each position of the calculated objects to a separate T-FLEX CAD file. You can automatically create a task in T-FLEX Analysis, and the calculated characteristics of the movement of bodies will be transferred to the task and set as initial loads.

To unload the calculated position of the model, you can use the new command **Upload Geometry to File**, which has corresponding icon in toolbar.

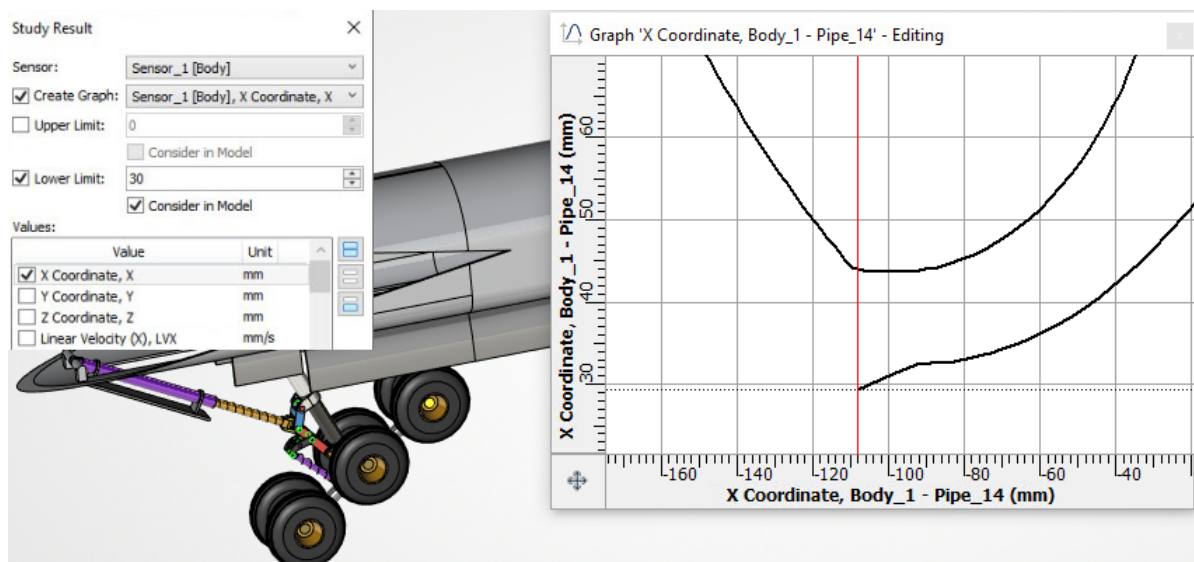


To unload the calculated position of the model, you can use the new command **Upload Geometry to File and Create Analysis Study**, which has corresponding icon in toolbar.



Calculation to given position

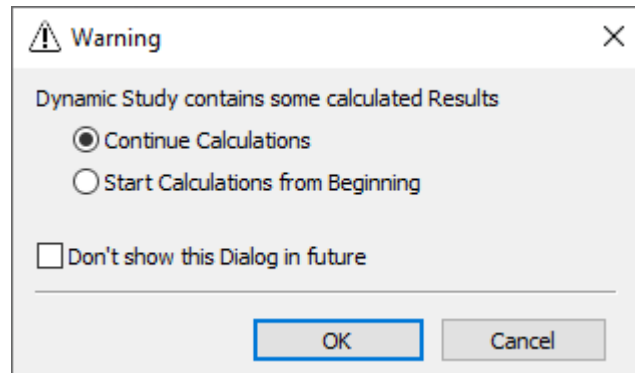
Now it is possible to stop automatically the calculation when the sensor values go beyond established limits. In the settings of the task result, you can specify the limits and set the flag **Consider in model**.



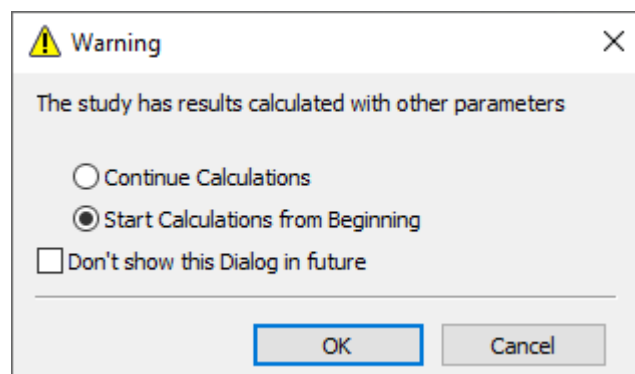
Now you do not need to spend extra time calculating and looking for the desired position of the mechanism manually in the graphs of the results: the movement of the model will be stopped according to user-defined conditions.

Control of Calculation Result Record

Warnings when restarting the calculation are divided into two types. In the first case, the system warns about the presence of calculated results for the current parameters.



In the second case, the system warns of the presence of calculated results for parameters that differ from the current ones.

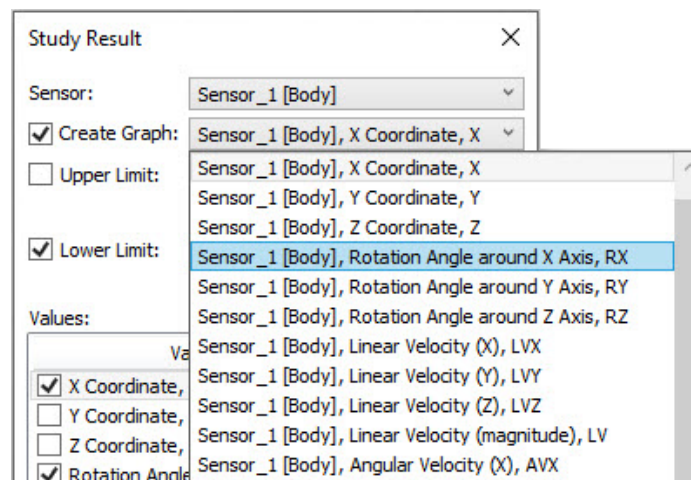


Graphs as Cyclogram

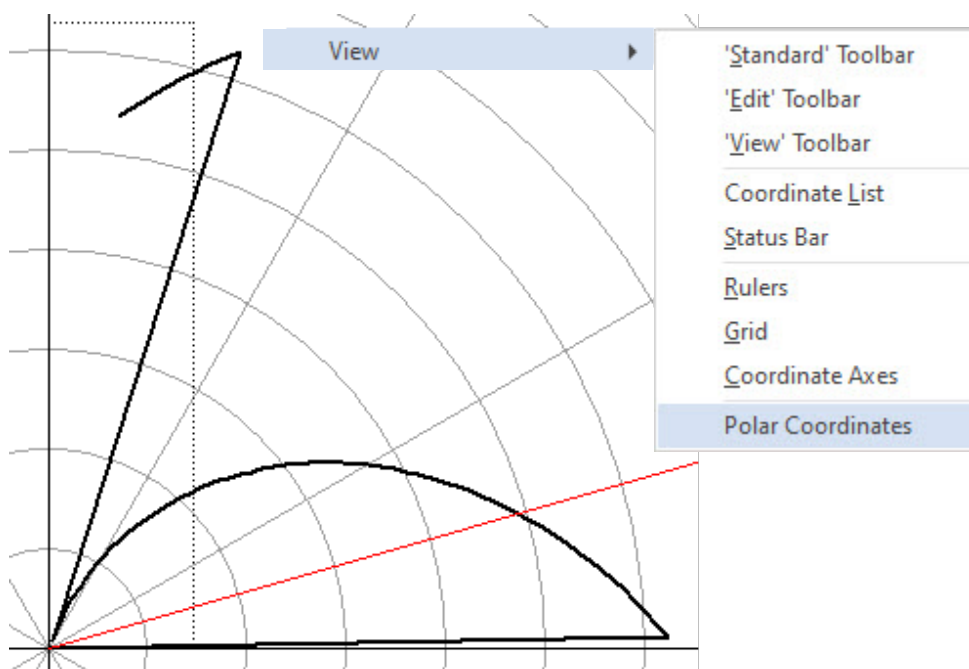
Calculation of mechanism cyclogram is an important and convenient functionality for evaluating the joint operation of the product units under study. As is known, most mechanisms have a rotary drive, and therefore, the cyclograms of the machine components are usually built in the coordinates given by the angle of rotation of the drive shaft. In T-FLEX Dynamics module, you can now build graphs as cyclograms. For this purpose two new features are implemented.

- For **Body** type sensors it is now possible to measure rotation angles around the X, Y, and Z.
- Now graphs can be plotted not only depending on time, but also on any other characteristic of any other sensor. For example, you can specify drive shaft angle of rotation as the abscissa axis.

You can cut off the continuous rotation using the new option to stop the calculation described above when the sensors show the specified values.



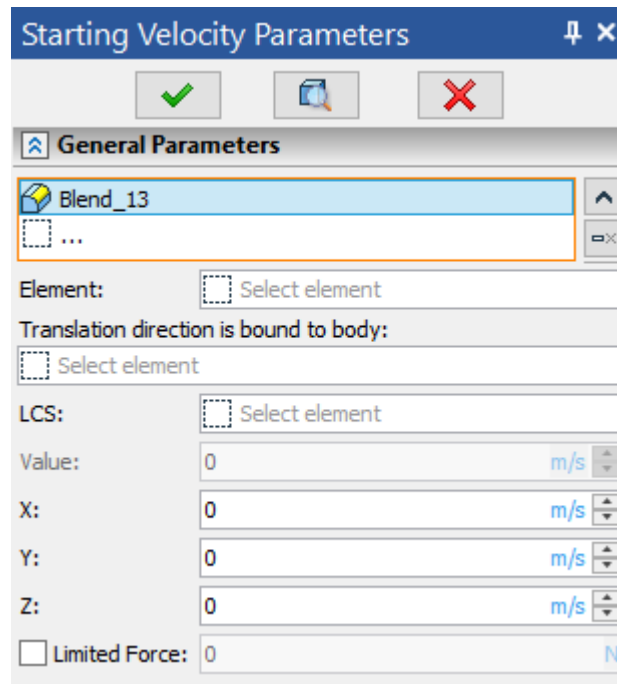
In addition, for analyzing the results with respect to bodies of revolution, now it is possible to display graphs in polar coordinates. For this, there is a corresponding option in the graph context menu.



New Load Types

Translation with Constant speed

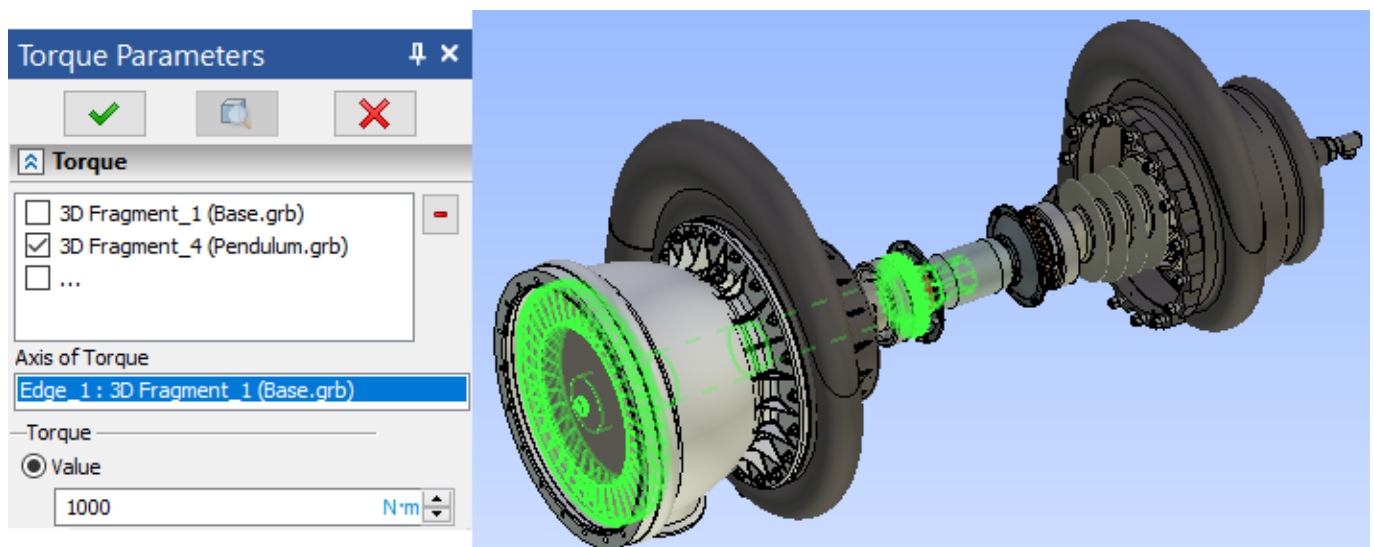
If the driving link of the mechanism has a constant speed and moves rectilinearly in a given direction, you can set the law of its movement using the new **Translation** command.



Translation can be set along the selected element in the given coordinate system, in the direction of the selected body. The speed value of translation can be set different in the corresponding directions of the coordinate system. You can also set a limit on the force reaction on a moving element.

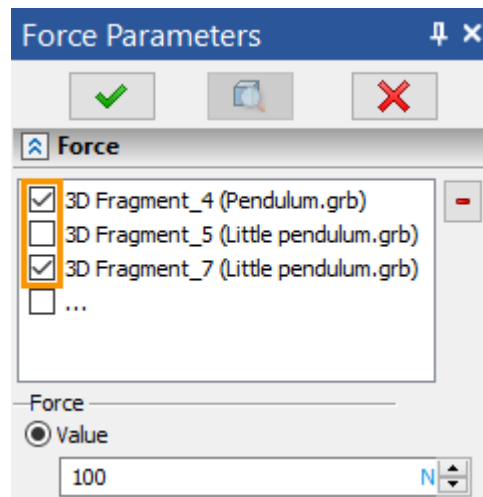
Force Moment between Bodies

For bodies whose rotation depends on each other, a new loading mechanism has added. Now in the **Torque** command, load inversion is available by setting a flag. Thus, by choosing two bodies and changing the direction of the action of the moment on one of them, you can set the action of the moment between the bodies.



Load Inversion

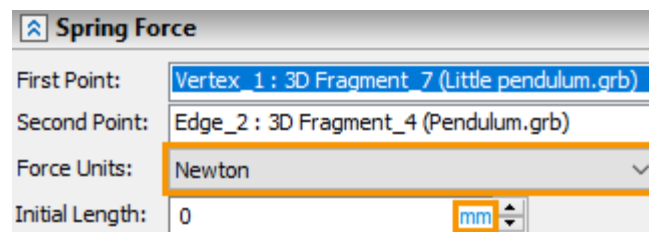
As well as in the **Torque** command, in the **Force** command it is now possible to invert loads by setting flags.



Interface and Visualization

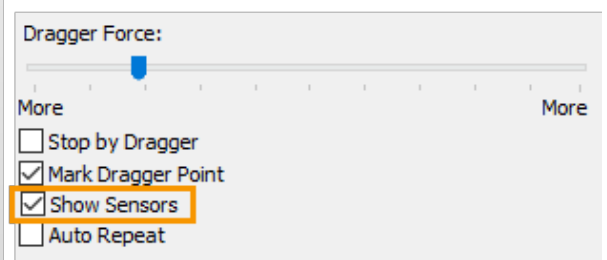
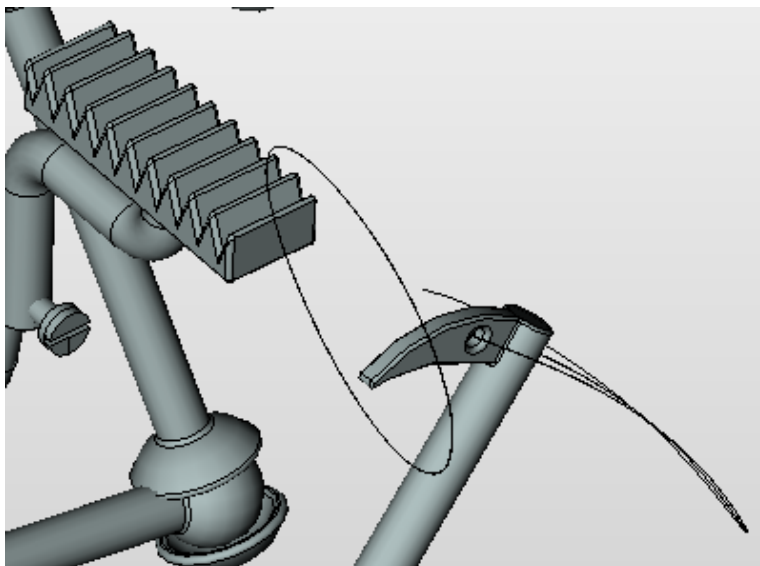
Unit Setting

Now, in all input fields, the selection and conversion of measure units is available.



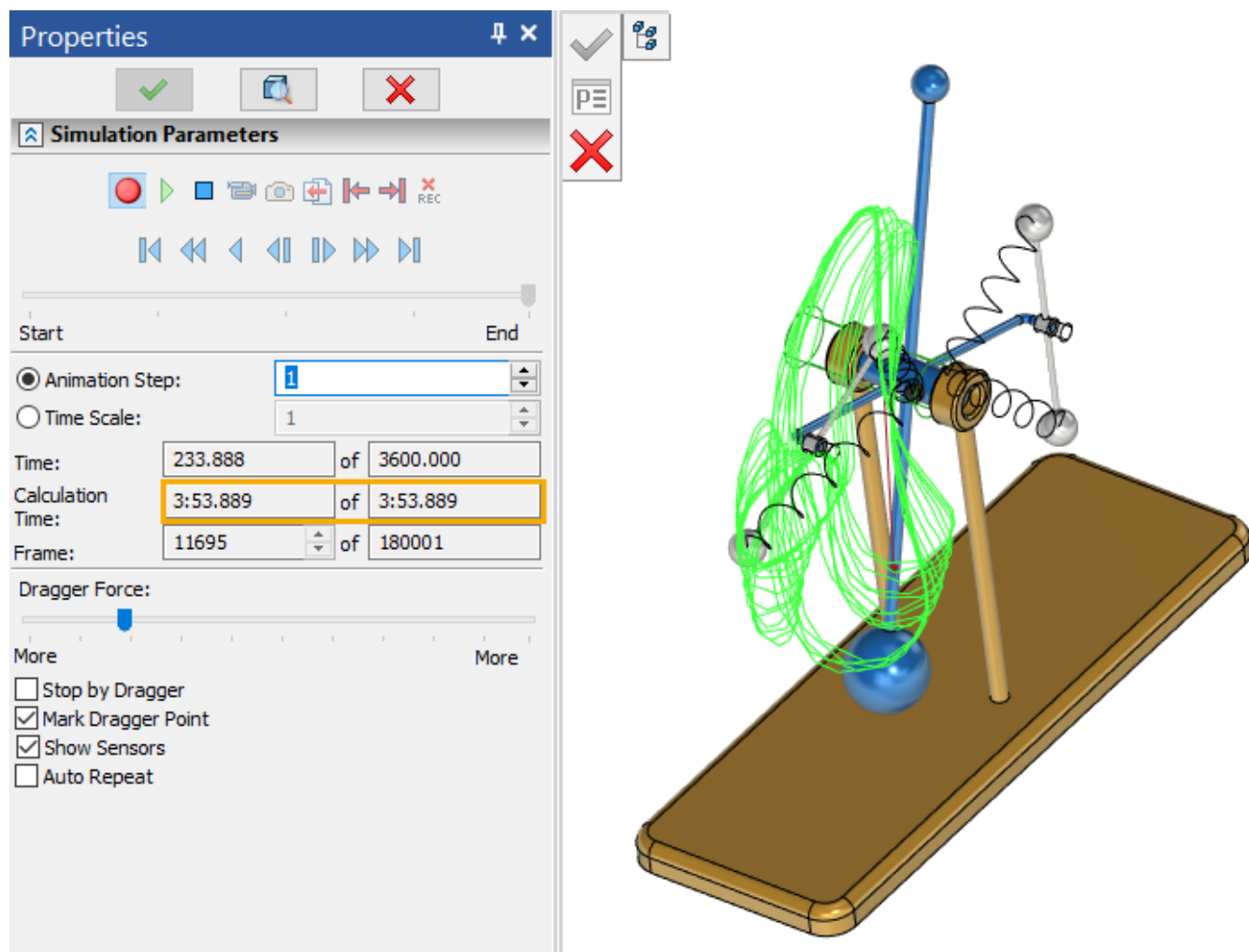
Show/Hide Sensors

The option to hide sensors is now available. If the sensors interfere with analyzing the translation of the mechanism, for example, if the sensors close the path, they can be hidden.



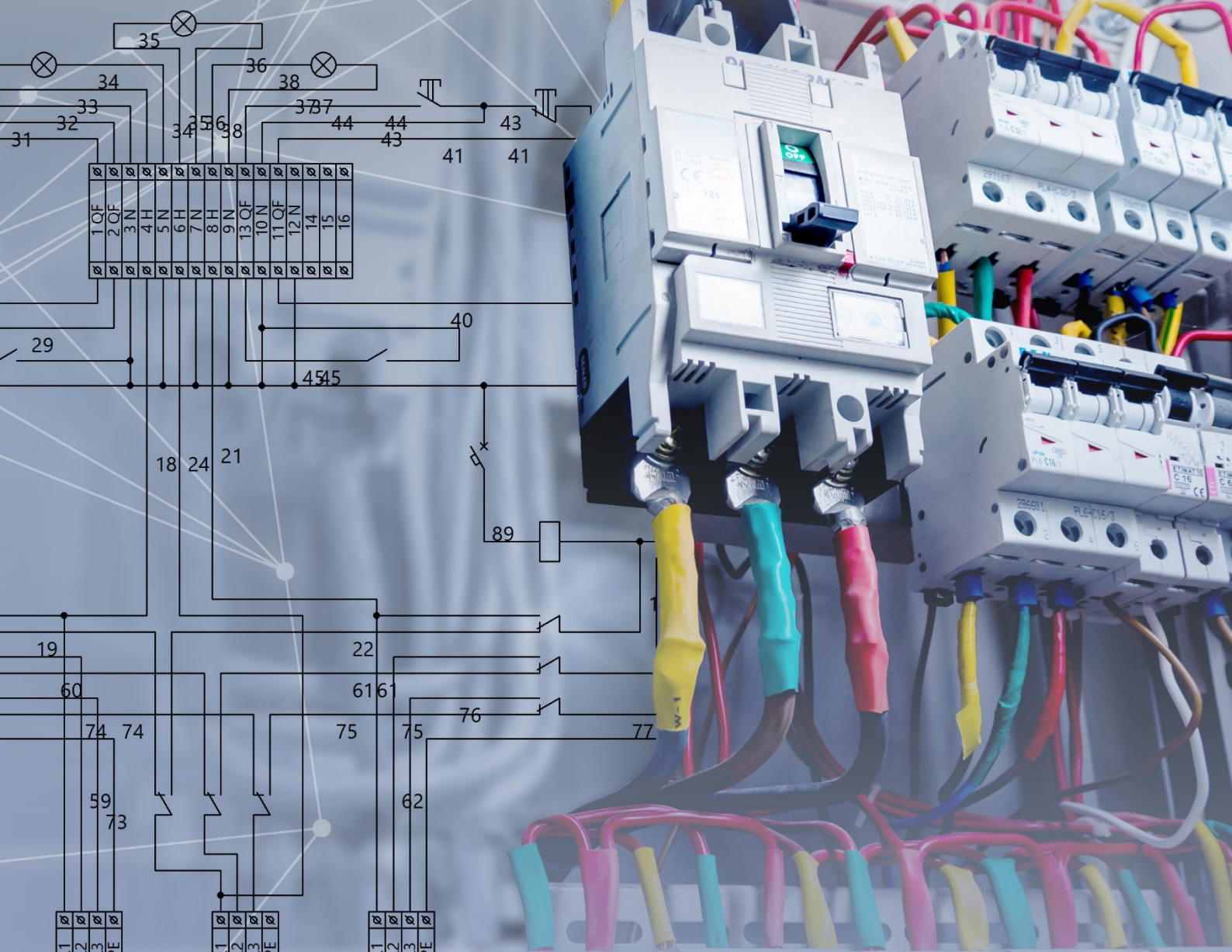
Time Counter for Calculating Studies

A total **Calculation Time** counter has been added to the **Simulation Parameters** tab of the T-FLEX Dynamics module studies calculation command, which makes it possible to predict the time of recalculation or calculation of similar tasks.



T·FLEX Electrical 17

What's New



T-FLEX Electrical 17

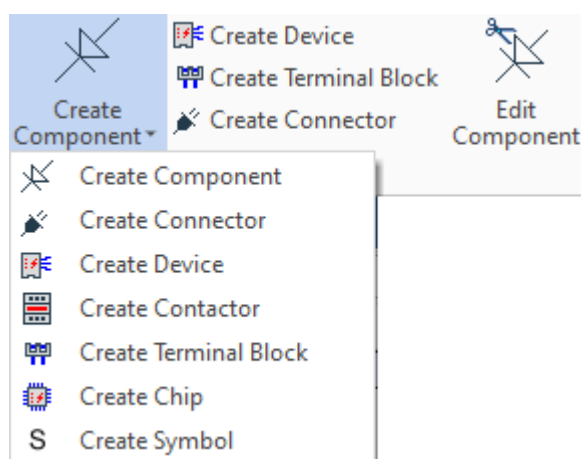
The tools of diagram design and cable components design are expanded in the new version of the T-FLEX Electrical. This made the module even more flexible for solving tasks in the field of electrical engineering.

Diagram Elements

Commands for Creating New Diagram Elements

Previously, you had to select the prototype in the **Start Page** to create a new document and then run the command to create the corresponding element type in it. It was not obvious and required many "clicks".

New commands have been added to the Ribbon for a quick transition to creating new elements of diagrams. When you select one of the commands in Create Component group, the system automatically opens a document prototype of the corresponding type and activates the component creation command. The number of commands for creating elements and the number of document prototypes has also been minimized to simplify and unify user actions.



Commands for editing all elements now have one common button - Edit Component. It automatically determines element type and launches the corresponding command interface.

Component Structure and Tree

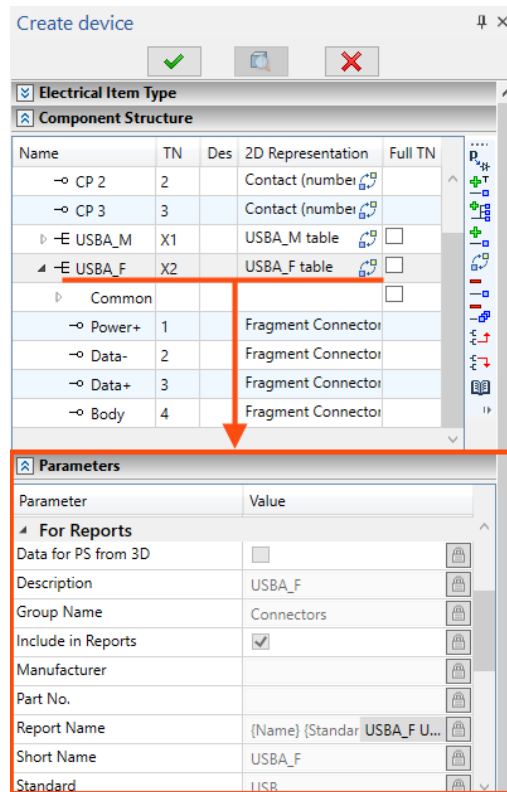
Component Structure block was redesigned in all commands for creating / editing elements. Now it has an explicit tabular view with a hierarchical tree of constituent parts.

Tag names (TN) of component parts of all nesting levels are displayed in a separate column now.

A button to replace graphics with an alternative one from the library is located next to each constituent element In the same window, in the 2D Representation column.

Full TN column and corresponding parameter **Full Tag Name** were added (see [Full/Address Tag Name and Local Tag Name](#)). The check-box opposite each independent element in this column allows the designer to choose whether the full tag name will be displayed in the diagram or the element will be displayed with the local tag name.

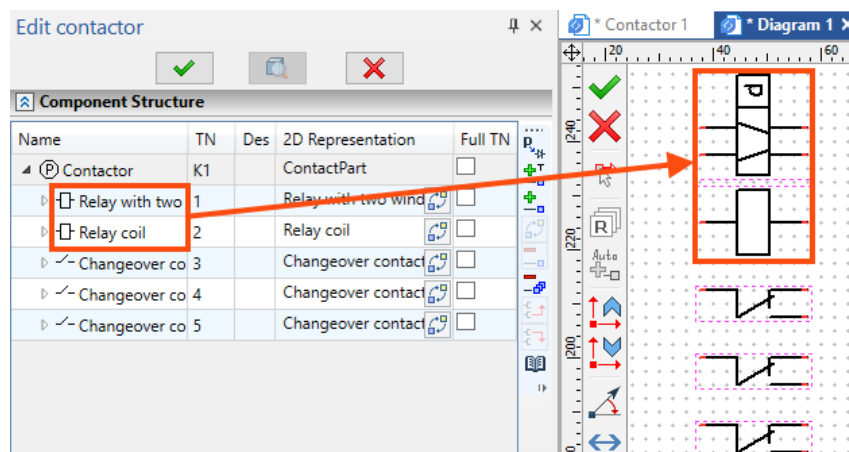
The **Parameters** block is now context sensitive, i.e. its content varies depending on the selected object in the **Component Structure** block.



New Types of Multicomponent Diagram Elements

The tools for creating new types of library components - **Contactors**, **Terminal blocks** - were added.

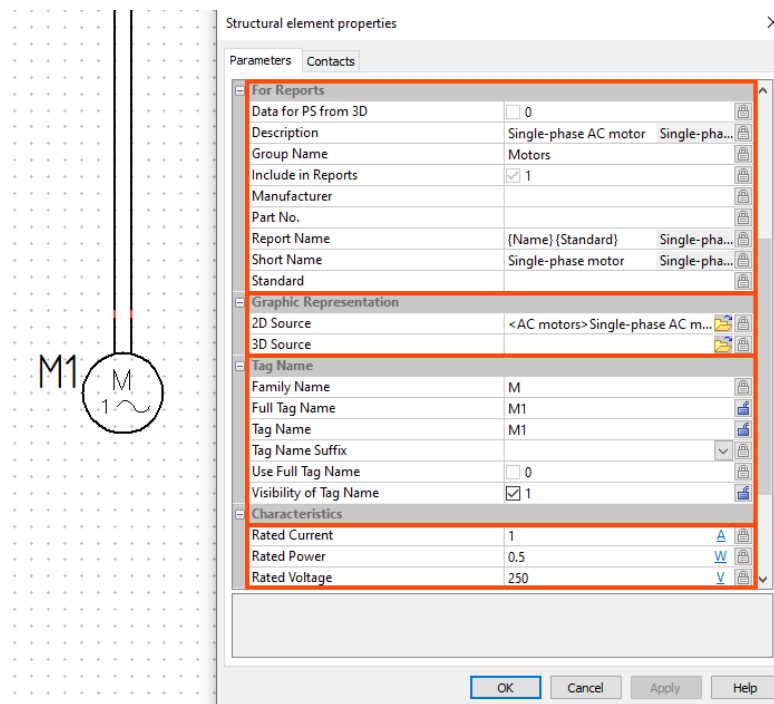
Now the command for creating a relay is included in one common tool - **Create Contactor**, where, in addition to a relay, it is possible to create a contactor with mechanical effect on the contacts. The structural element **Contactor** is created "at the head" of any component that acts as a contact device. It includes relay coils and controlled contacts. Also it became possible to create relays with two or more control coils.



A proprietary component type is also available for the terminal blocks. You can also create prototypes of **Terminal block** in **Component Editor**.

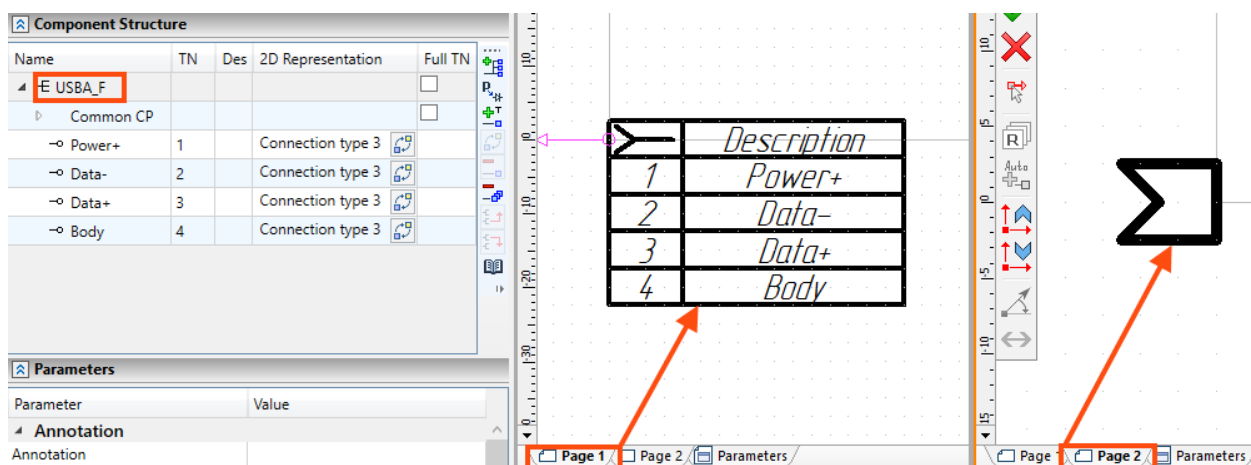
Grouping Component Parameters by Purpose

Parameter categories have been added for all diagram elements. This makes it easier to find the desired element information for the user. The following categories of parameters act as the main groupings: **For Reports** (parameters that directly affect the element information included in a text report), **Tag Name** (control of the content and method of outputting tag names to the diagram, as well as in report), **Connection Points** (management of textual information about connection points of elements) and **Characteristics** (parameters of elements that determine their operating characteristics and features), etc.

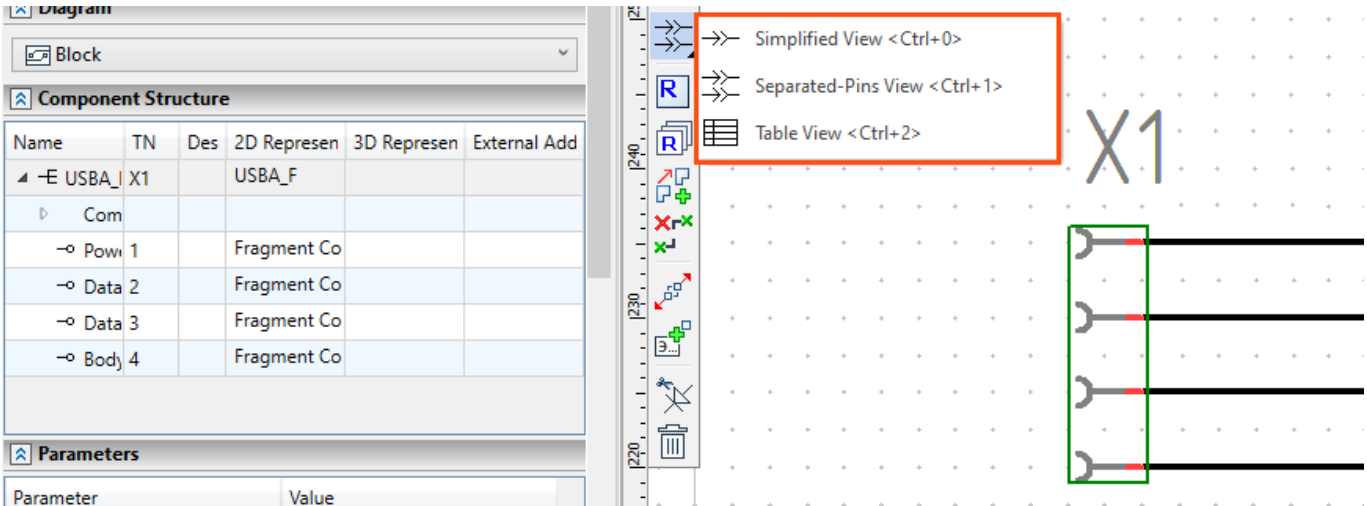


Revised Create Connector Command

Connectors have few pre-configured graphics for diagrams in the new version of Electrical module. One symbol - a simplified one - is intended for the use in general diagrams, as well as connection diagrams saturated with switching elements. Another symbol is presented in the form of a table with a customizable set of columns, where each contact has its own cell. Now both of these graphic types are stored in one file and, by default, are generated by the system automatically when the user is working on the structure of the future connector.

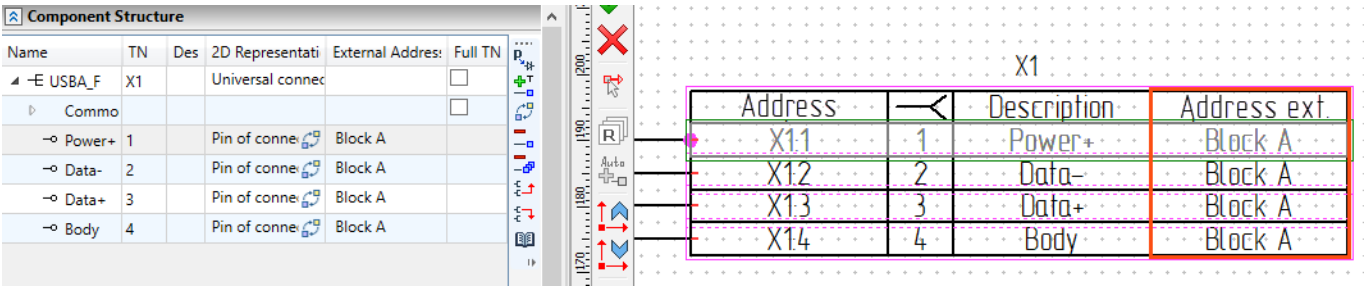


A separated-pins representation of the connector is also available, in which each contact is depicted as its own symbol.



Switching between the types of graphics is available at all stages of working with the component: both when creating a connector and when inserting a finished element into the diagram.

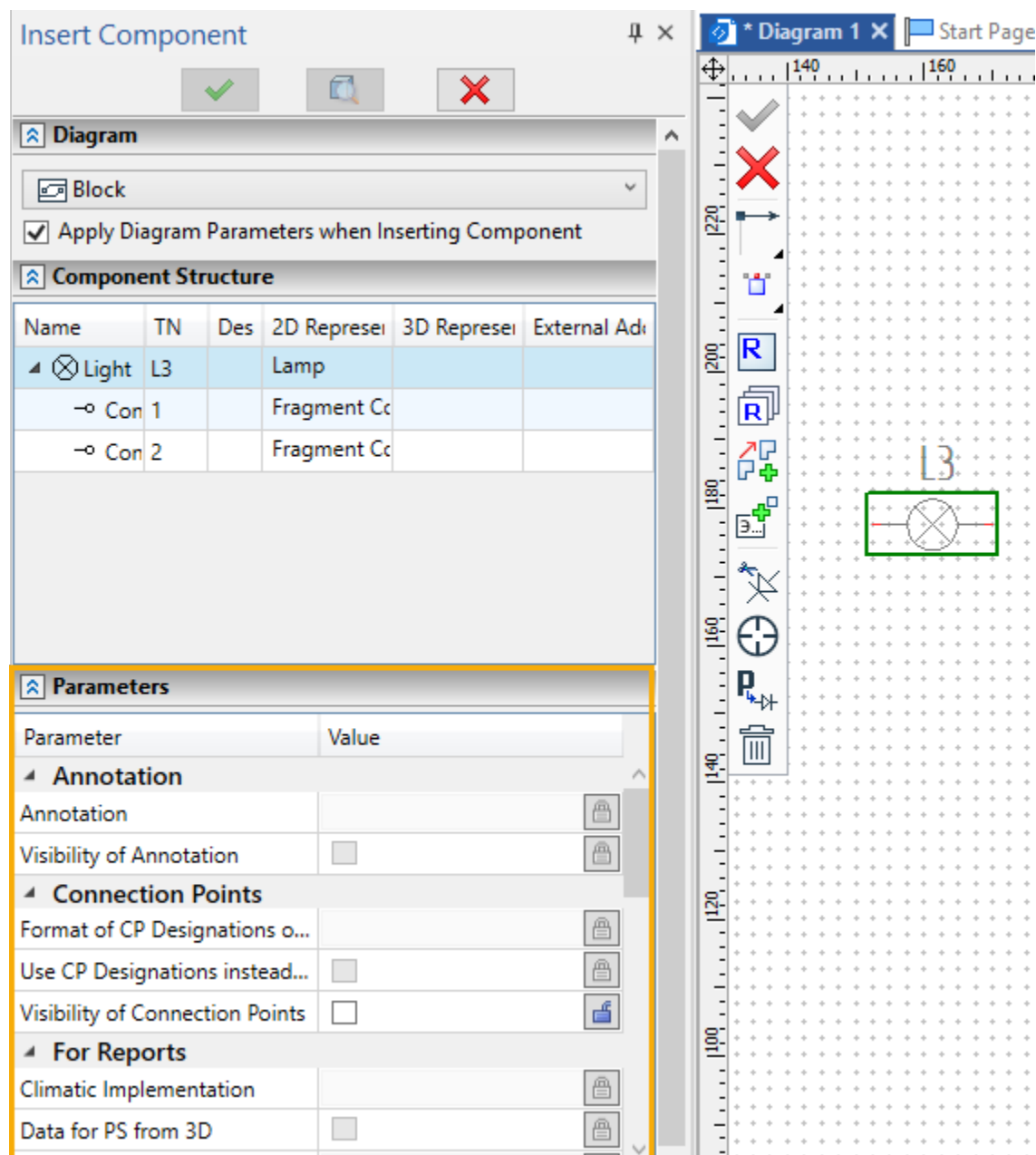
A new column, **External Address**, is added to the symbol of connectors in the table view, it is designed to indicate the addresses of connections that are outside the current block. Its filling is done manually in the connector edit mode.



Component Editor

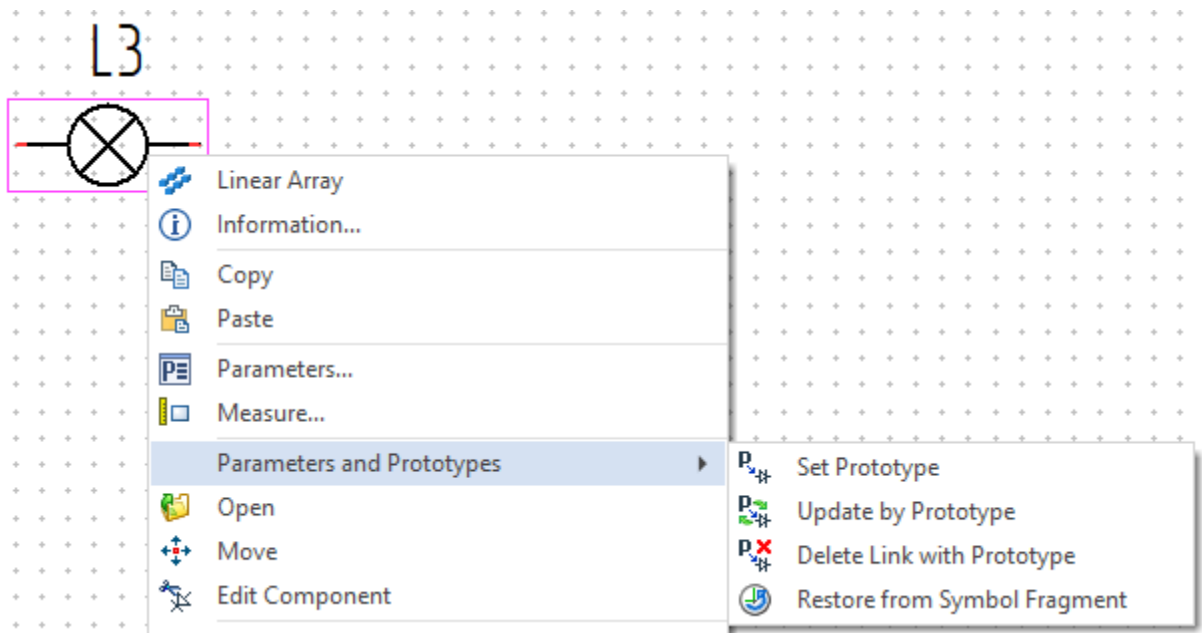
Storing Purchased Components Data in the Component Editor

The traditional way of working with purchased components in **T-FLEX Electrical** presume storing all data of a component in a file of its 2D representation. This is inconvenient when working with a large number of purchased components, which use the same 2D representation.



Now all purchased components data can be stored in the **Component Editor**. 2D representation's file is only required to store a symbol of a component in a diagram. The rest of the component's data stored in 2D representation's file is used as default parameters of the component, upon its insertion into a diagram. Then, you can replace default values taken from the 2D representation's file by data of a certain purchased component stored in the **Editor**. Purchased components library is filled by creating prototypes in the **Component Editor**, using standard tools familiar from previous versions of the application. Additionally, a data can be imported from external sources (*for import please contact the Technical support support@topsystems.ru*).

The **Parameters and Prototypes** item has been added to contextual menus of all components' symbols in a diagram. It contains following commands for working with **Component Editor** data:




Set Prototype

This command is also available in the automenu, when inserting a component's symbol into a diagram. It invokes the **prototype selection window**, where you should select a prototype, whose data will be assigned to the current component of the diagram. The command is used for initial selection of a prototype and for re-selection of another prototype instead of the previously selected one. The selection process is described below.


Update by Prototype

This command is available, only if a prototype is selected. It reads data of the selected prototype from the **Component Editor** and re-assigns it to the current component. Thus, you can update component's data in a diagram after editing a prototype in the **Editor**.




Delete Link with Prototype

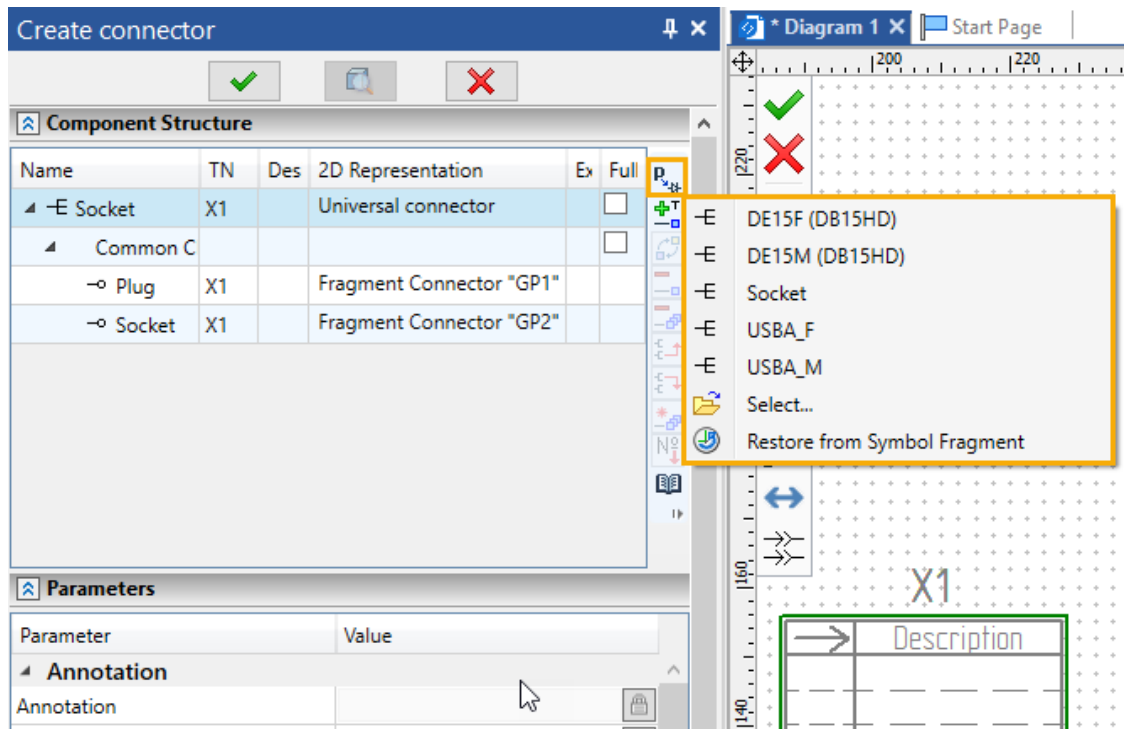
This command is available, only if a prototype is selected. It breaks link between data of the current component in the diagram and the prototype data. In result, the component retains data previously taken from the prototype, but this data can no longer be  **Updated by Prototype**.



Restore from Symbol Fragment

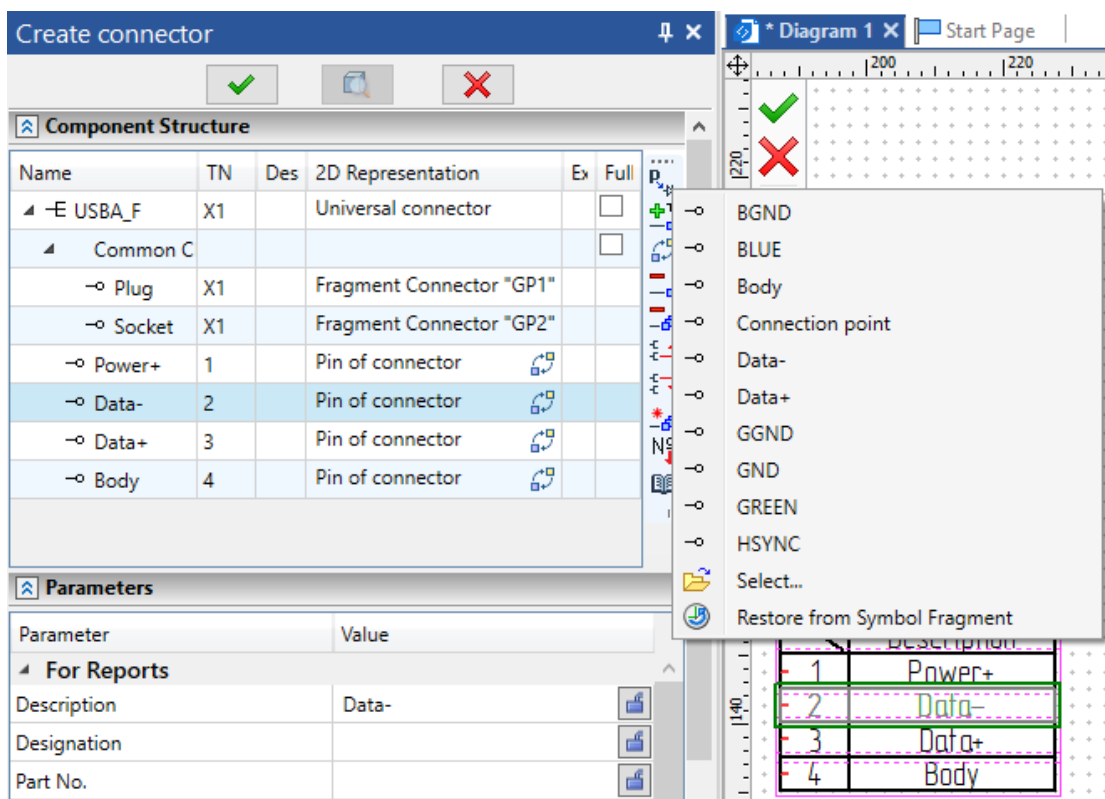
The command reads component's data from the 2D representation's file. However, it doesn't break the link between the component and the prototype, so the data read from the representation can be later replaced by data from the prototype using the  **Update by Prototype** command.

The aforementioned commands available in contextual menu and in automenu work with data displayed in the **Parameters** section of the **Parameters** tool window, when inserting and editing a component. They do not affect the **Component Structure**.


Moreover, the  **Set Prototype** and  **Restore from Symbol Fragment** commands are available in the **Component Structure** section of the parameters tool window, upon placing a contextual compound component (connector, contactor, terminal block, chip) on a diagram. In this case, in addition to assigning data described above, these command copy the components structure from prototype or from 2D representation's file correspondingly. A desired prototype can be selected by name from the drop-down menu. The  **Select...** command in the menu opens the **prototype selection window** described below

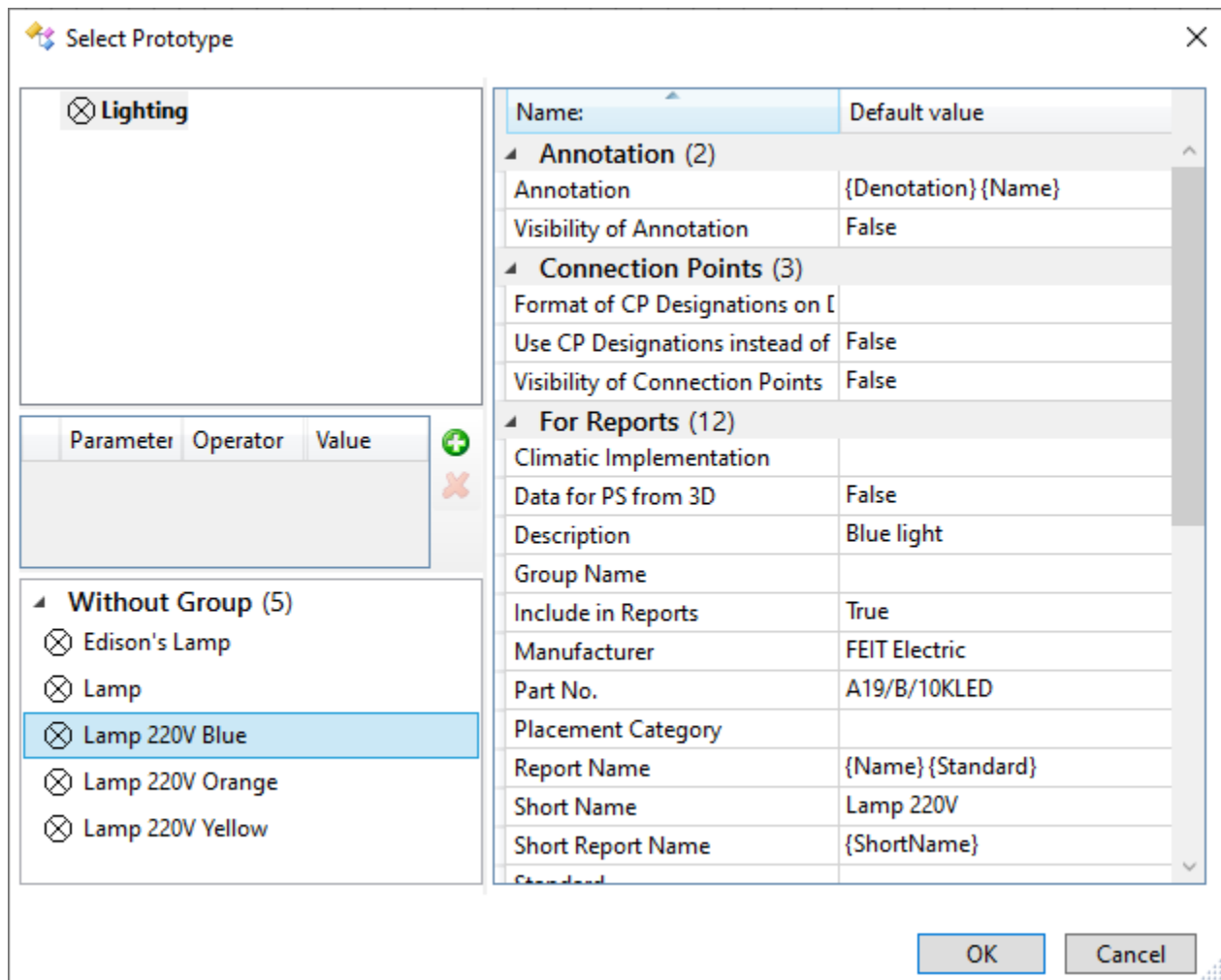


 **Set Prototype** and  **Restore from Symbol Fragment** commands in the **Component Structure** section are applied to the currently selected level of the structure. Thus, you can, for instance, copy a connector's structure from a prototype first, and then set prototypes for its particular pins individually.




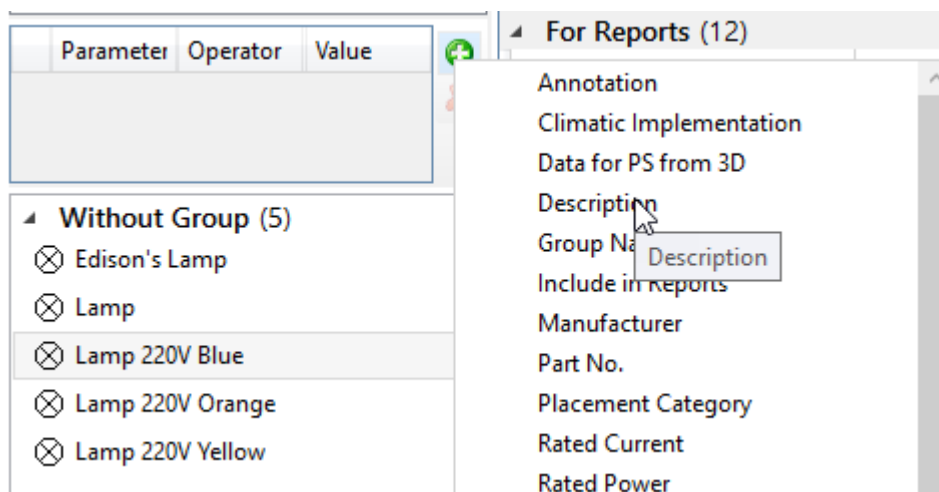
Prototype Selection

Upon calling the  **Set Prototype** command, the **Select Prototype** window appears:



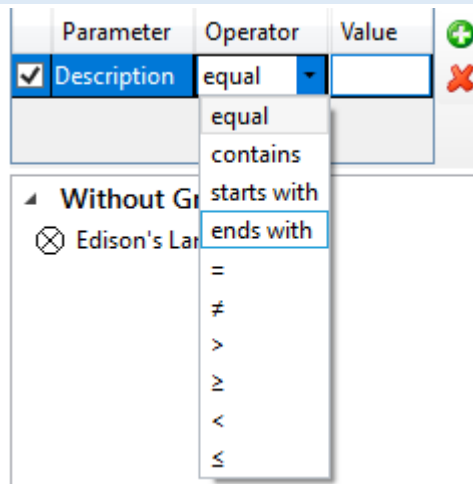
Upper left section of the window displays the type of the current component. Lower left section of the window displays the list of prototypes available for components of this type. Right section of the window displays the data of the selected prototype. Select a desired prototype in the list and press **OK**. In result, the data of the selected prototype will be applied to the current component.

The list of filters is located above the list of prototypes. It allows you to filter the list of prototypes by values of a particular parameter. In order to add a filter, press the  **New parameter** button and select the desired parameter (e.g.: **Description** or **Rated Power**) in the drop-down list



The new filter will appear in the list of filters. In addition to the **Parameter** selected on the previous step, the filter requires specify a logical **Operator**, i.e. filtration criteria, and a **Value**. Values of the selected **Parameter** of prototypes will be compared to the specified **Value** using the specified **Operator**.

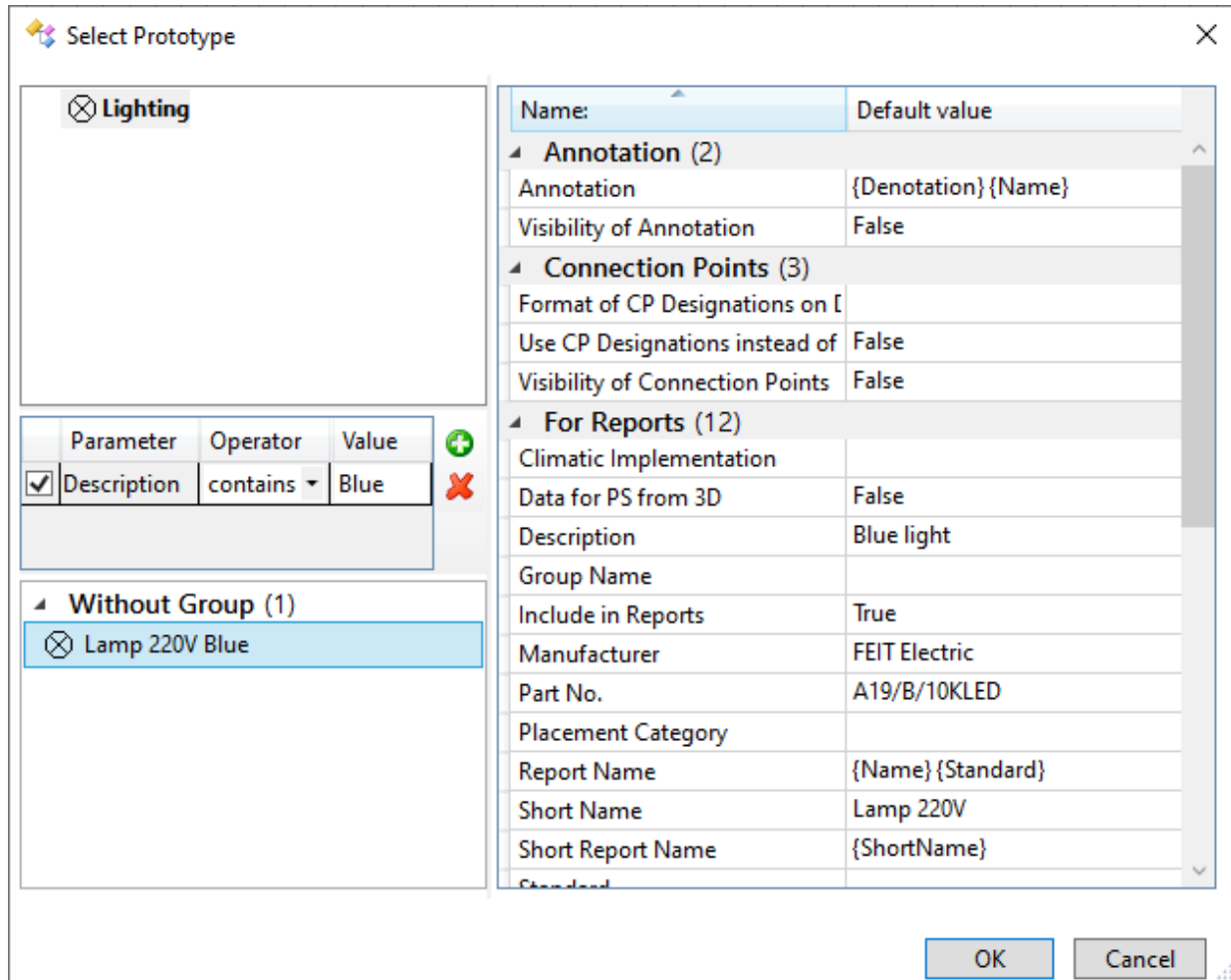
Attention: first four operators in the list work with textual values only, the rest of the operators work with numerical values only.



In result, the list of prototypes will only contain prototypes, which pass the specified filter.

When applying a filter, the checkbox gets enabled in the left side of filter's row. Then, you can disable or enable this checkbox manually, in order to misapply or apply the filter without deleting it and creating again. In order to delete a filter, select it in the list and press the **Delete parameter** button.

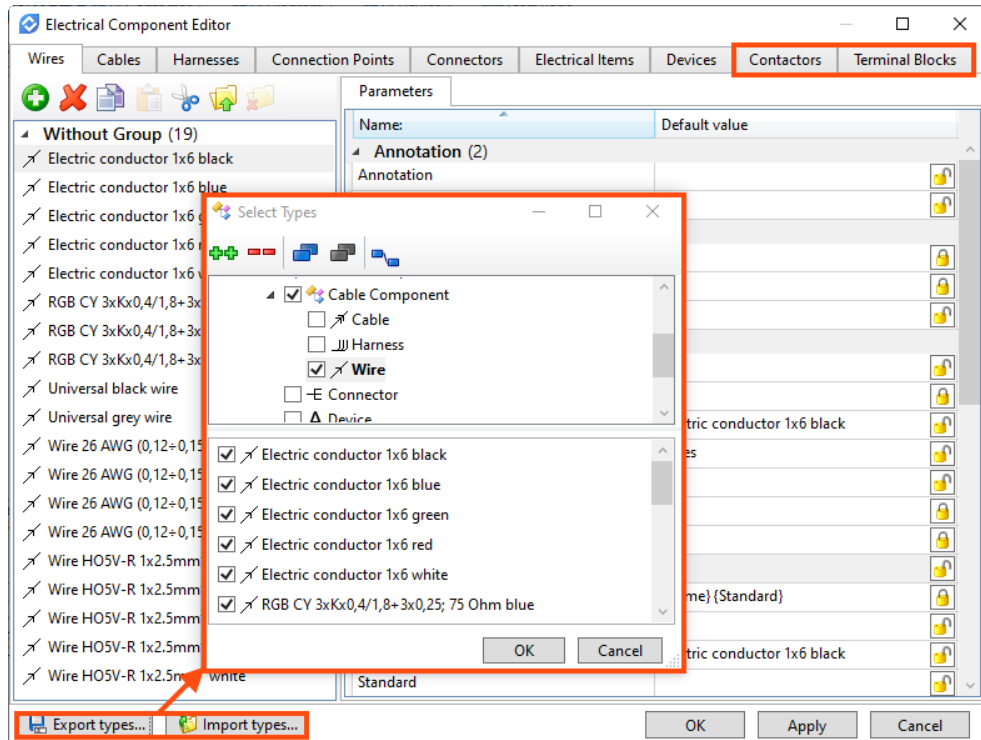
Multiple filters can be applied simultaneously. In such case, the list of prototypes will only contain prototypes, which pass all applied filters.



New Tabs and Tools to Export / Import Prototypes

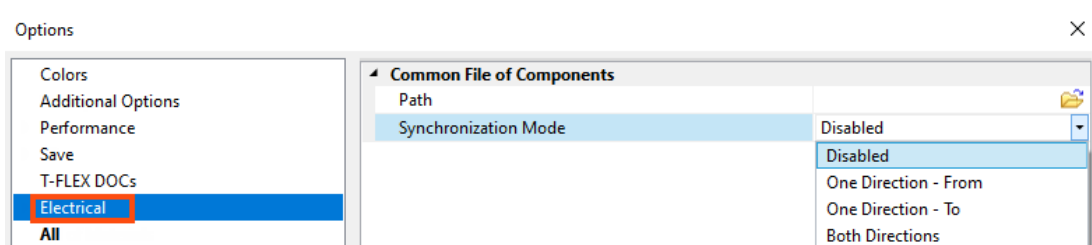
Changes in the new version of the module also affected the **Component Editor** of electrical products. It got two new tabs, one for each new type of element (**Contactors** and **Terminal Blocks**).

Among other things, **Component Editor** has tools for unloading / loading the prototypes via an external file. Due to this, the base of prototypes of elements can be transferred between workstations or even made it networked.



Collaboration in Component Editor and Network Data Storage

Electrical section has been added to the **Options** command for the network mode of working with **Component Editor**. It contains parameters that determine the path where the editor's network file is located and the mode of synchronization with it.



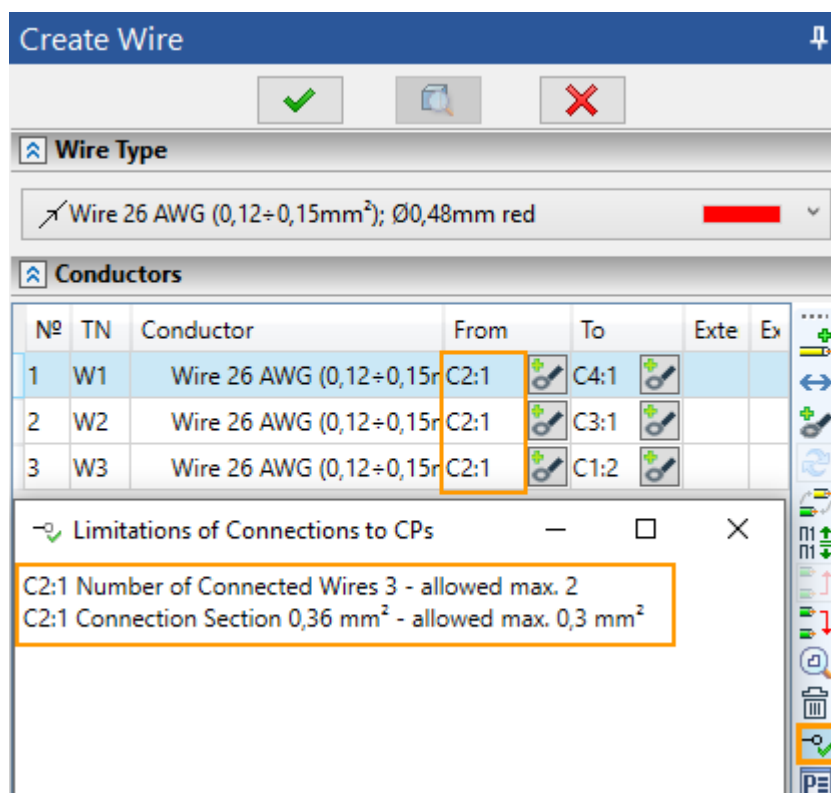
Now a file (an XML file that can be copied from a system location or downloaded from a workplace) with the composition of **Component Editor** can be located in any user-specified location, whether it is a network location or some folder on the current computer. Setting the **Synchronization Mode** will help you choose a suitable option for interacting with an XML file - with transferring in **One Direction** (changes in the editor will be transferred only from the user to a network file or vice versa, i.e. **From** or **To**) or in **Both Directions** (changes will be transferred in two-way mode).

Limitations for Connection Points

Connection Points now have new parameters, which limit number and sections of wires attached to this CP.

Limitations		
Allowance for Wire Length	0	mm
Connection Section, max.	0.3	mm²
Connection Section, min.	0	mm²
Contact Size	0	mm
Max. Number of Connections	2	

If the parameters are set, then, upon assigning cable components, the user can check the correctness of connections using the **Check limitations** options. It is most useful for branched connections.

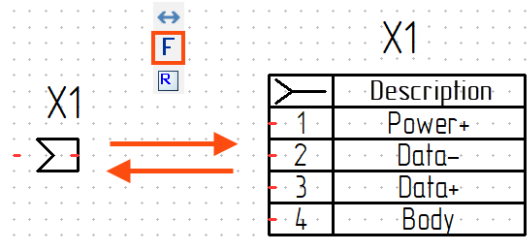


Limitations of connection points may also be checked in the **Check Diagram** command.

Development of Electrical Diagrams

Converting Connector Graphics on the Diagram

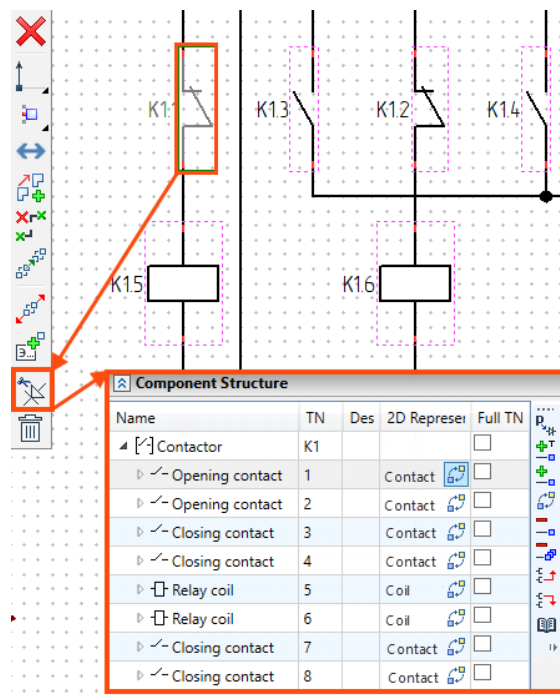
After the work carried out on the connector data model and expanding its capabilities for storing additional graphics, the user can switch the connector symbol from simplified to full (tabular) directly in the diagram context at any time (F button in the automenu of connector).



Viewing the Component Structure in the Context of Diagram

Due to the **Component Structure** block has been redesigned (see [Diagram Elements](#)) and the structure of each element looks like a tree expanding to the entire nesting depth, it is possible to see the information about the selected element on the diagram. The dialog, in which the component structure is presented, has an identical interface and is also designed as a block. There is only one difference - the user cannot access the buttons for editing the composition itself. At the same time, you can edit the parameters of elements, some of which are placed in the columns of **Component Structure** (TN/tag name/, **Name**, **Designation**). This has greatly improved the convenience of inputting basic data.

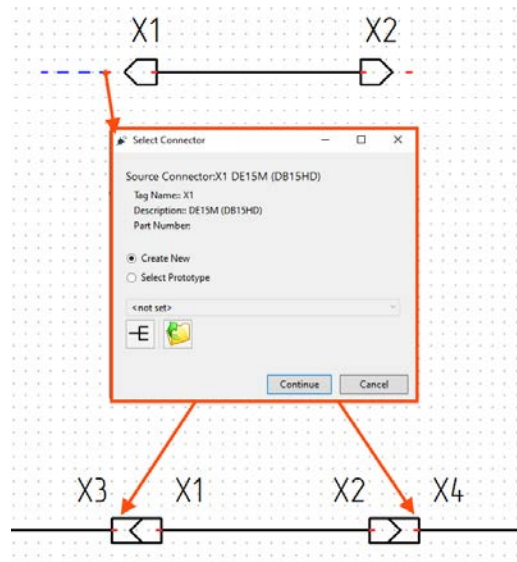
Viewing the complete composition is especially relevant for elements that have a [splitted representation](#), because sometimes it is difficult to find all parts of them on the diagram quickly.



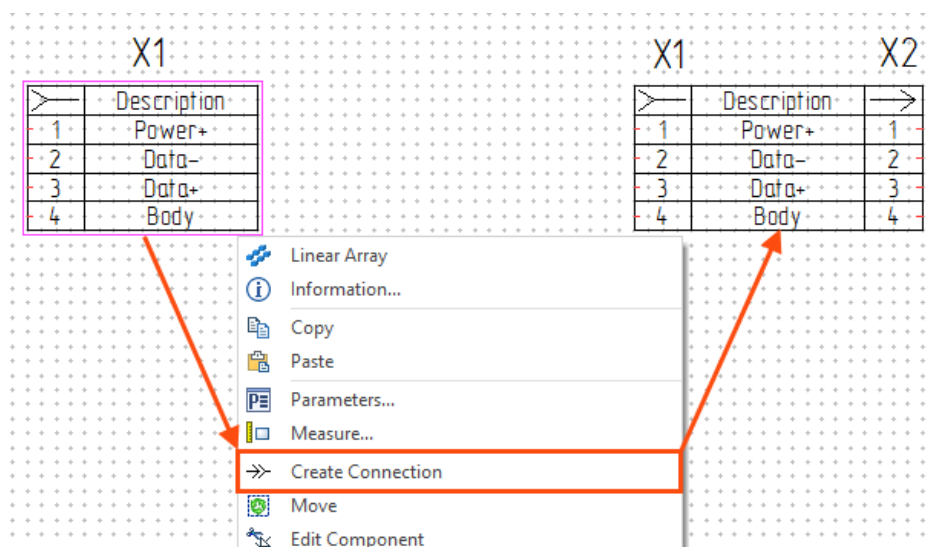
Selection of Mating Compatible Connectors

In previous versions, selecting and inserting a mating connector run when a cable component was assigned to a connection line "directly" connected to a connector in the device. In this case, only the tag name of the mating connector without symbol appeared on the diagram. The selection of the mating connector was carried out according to **Relations** parameter of connector.

Now, to call this function, it is enough to simply connect the connection line to any connector, regardless of its entry into the device. In this case, not only tag name is now displayed on the diagram, but also the symbol of the mating connector. The principle of the counterpart selection has not changed.



Create Connection function has been added for independent connectors. It launches the same dialog for selecting a mating connector. As a result, a mating connector with its own symbol and tag name is connected to the original one on the diagram.



The function is available for connectors in simplified and full (tabular) representation.

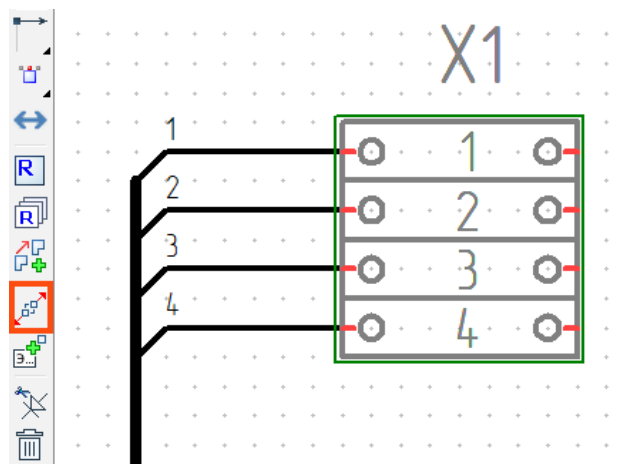
Splitted Representation for All Types of Compound Elements

Previously, **Terminal Blocks** and **Contactors**, created in context of diagram, could be "splitted", that is, the elements included in them could be placed in arbitrary places on the diagram.

You can now split any multicomponent (compound) elements, including coupled connectors and functional groups (see [Functional Blocks](#)).

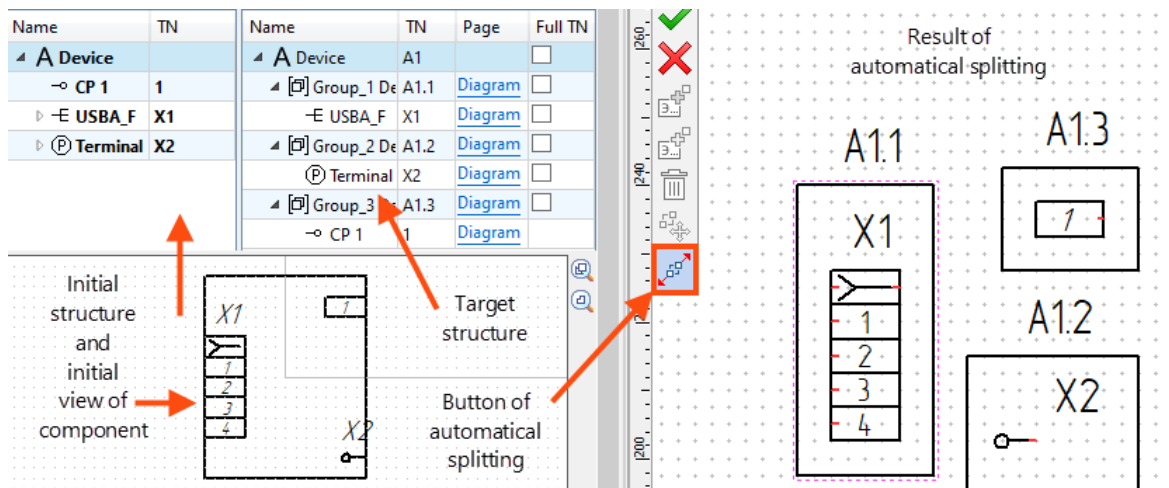
- ❗ Note: the condition for accessing the command is that the element must be created in a separate document (not in context of diagram) and inserted into the diagram.

A separate **Splitted Representation** command has been created for splitting elements. It runs from the automenu of the selected item.



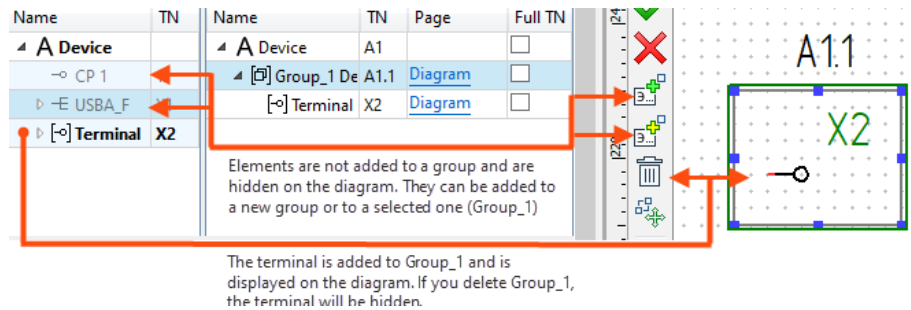
The dialog always shows the initial view and structure of component being splitted. The parts of the component into which it is divided are called **Groups**.

There are two splitting modes available in the command - automatic and manual.



Automatic splitting is activated in the automenu of command, one element is added to each group. Thanks to this, each element of the product can be placed in an arbitrary place in the diagram. You can manually change the result of automatic exploding. You can also not use auto-division at all and divide the product into groups at your discretion.

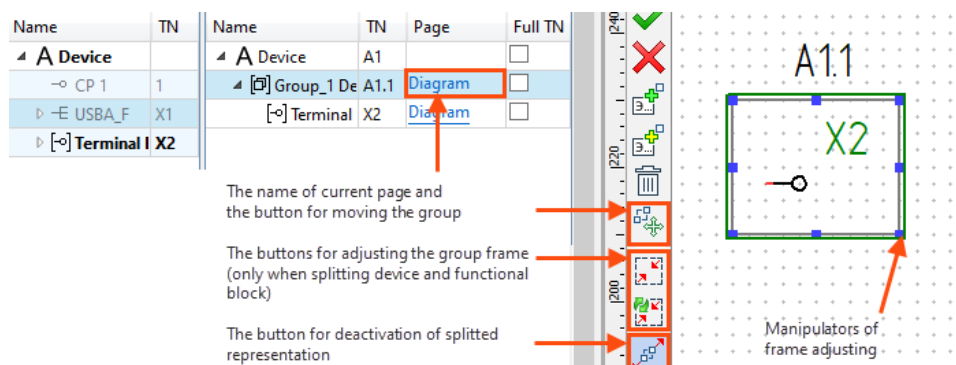
When exploding manually, you can **Add Element to New Group** (the group will be created automatically) or **to Selected Group** (the element will be added to one of the existing groups; the group must be preselected). The element included in the group is marked in bold. For such an element, the commands of adding to groups become inactive.



You can **Delete** the group, then its elements will be hidden in the diagram, and they can be added to other groups.

Some elements of the component may not be added to groups at all. This will allow you to display the component on the diagram not completely, but partially.

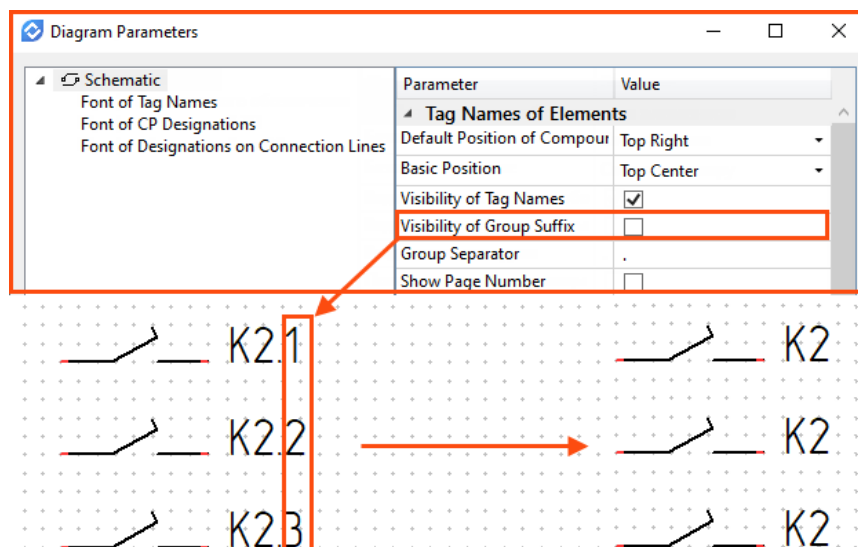
You can **Move Group** to any other page in the diagram. There is a special button for this in the automenu. It also allows you to move the group around the current page without quitting the command. The name of the current **Page** where the group is located is displayed in the dialog.



Device and **Functional Block** (FB) are framed by default. As a result, when splitting them, groups are also outlined by frames, the size of which is calculated automatically. You can manually resize the frame if necessary using the dot manipulators. If the **Device** or **FG** group is selected, buttons for contour control will be available in the automenu.

Pressing the **Splitted Representation** symbol again will disable the splitting and return the component to its initial view.

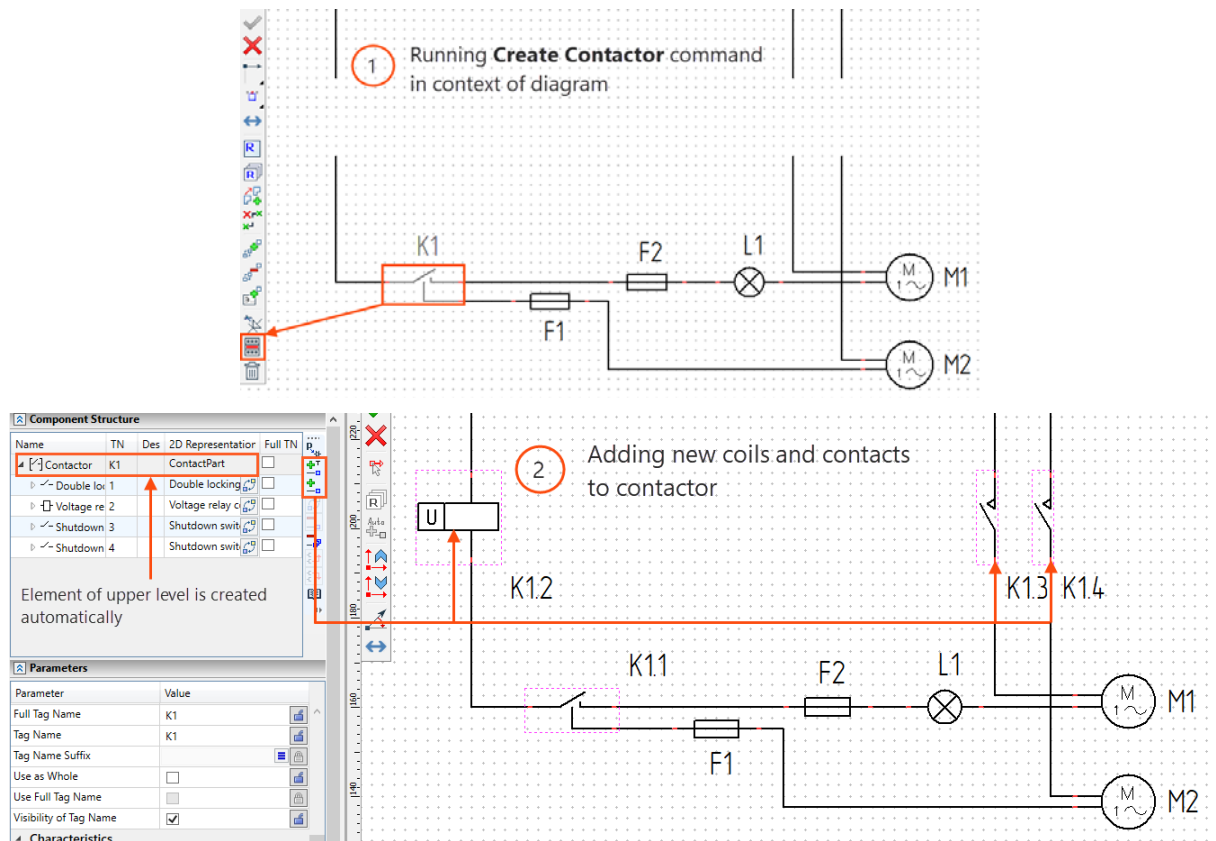
By default, the tag name of group consists of tag name of the initial component, the **Separator** and the **Group Suffix**. In the **Diagram Parameters** window, you can hide the group suffixes in all splitted components in the diagram.



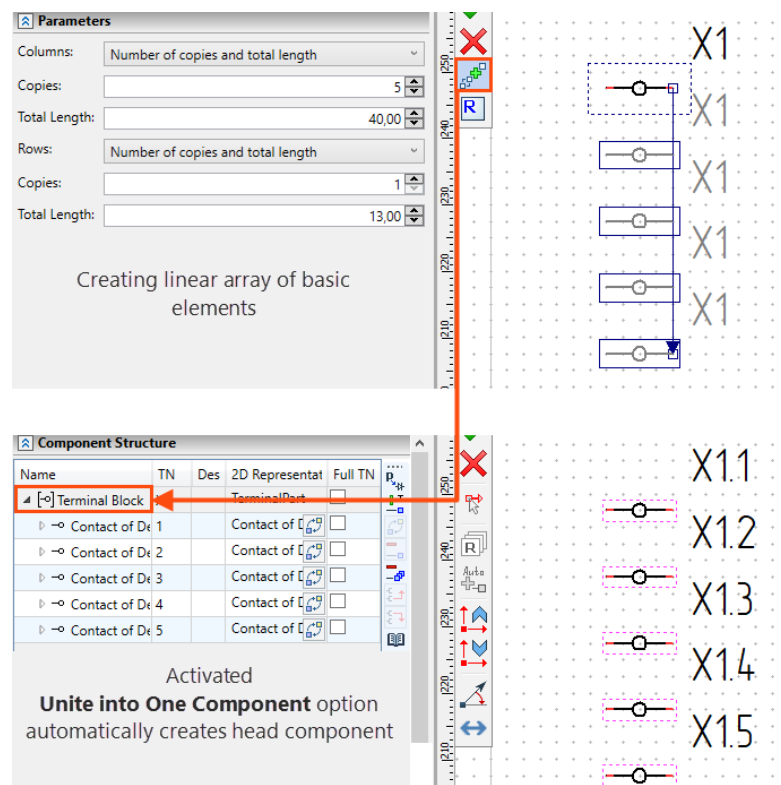
You can also change the separator character, for example, by replacing point with a colon.

Creating Elements in Context of Diagram with Saving to the Library

Previously, all elements of the diagram could only be created by generating a separate file based on a special template. Then the file was proposed to be inserted into the diagram. Now users have access to the option of contextual creation of diagram elements belonging to the class of multicomponent (compound). Elements created in this way have the ability to edit not only the values of their parameters, but also the structure.



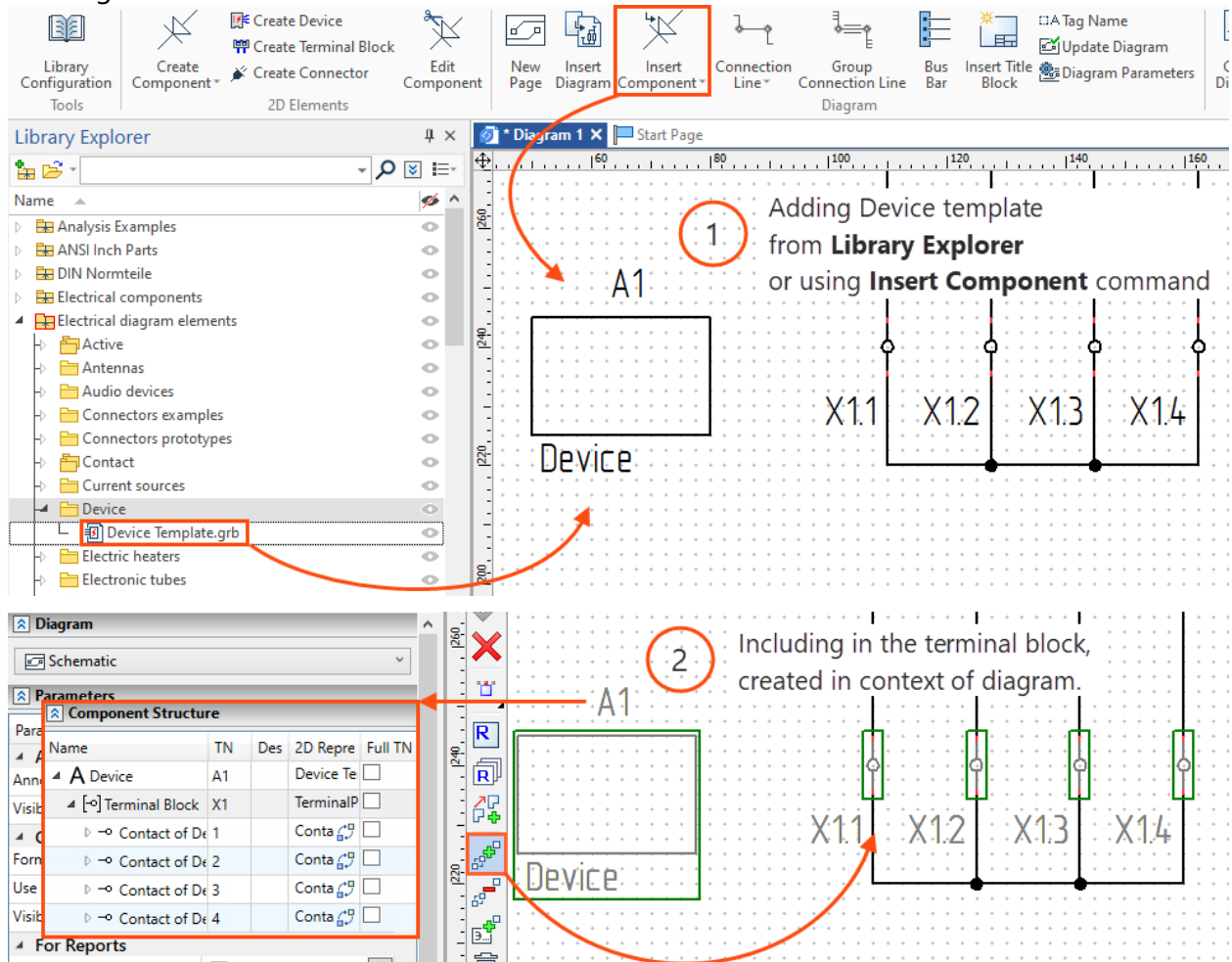
In addition, the option to auto-form a compound element from electric items is now available in the **Linear Array** command.



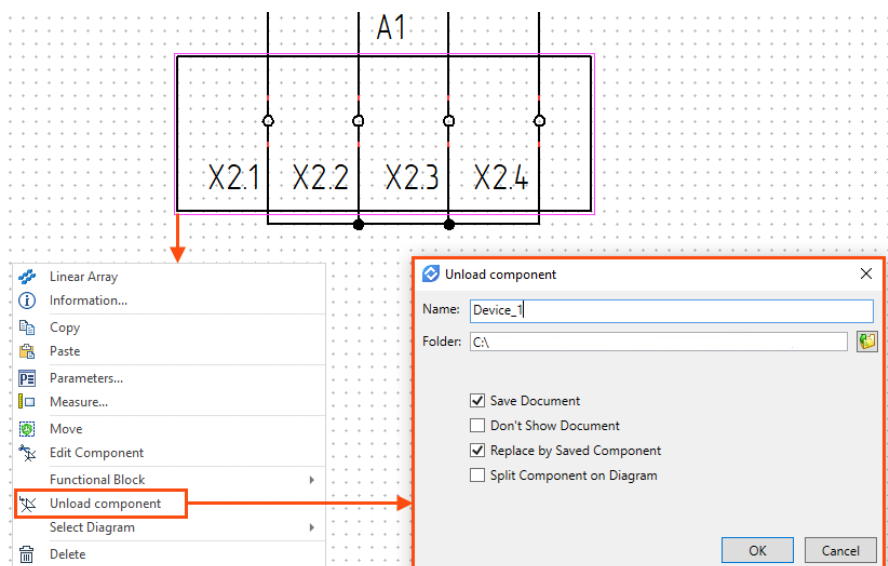
In addition to the previous two methods of contextual creation of elements, the system also provides a mode of combining several elements into a compound one. According to this principle, it is possible to create both switching elements (**Contactors** and **Terminal Blocks**), and **Devices** with **Connectors**.

To use this mode, the user needs to add all "parts" of compound element to the diagram (device can be inserted without filling the structure). When you select device, terminal, relay coil, contact or chip, special commands **Add Link with Element** and **Delete Link with Element** will appear in the automenu.

The commands allow including an element of a suitable type in the structure of the current component and deleting it.

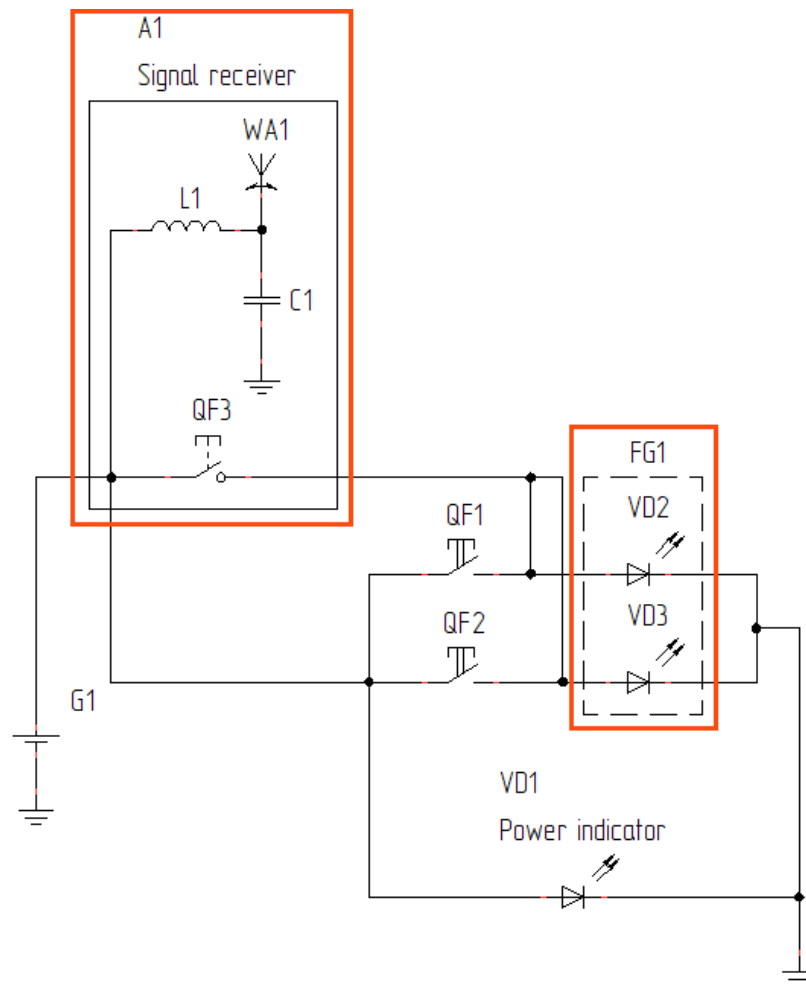


Each element created in any of the above ways can be saved in the library using **Unload Component** command. This means that it has become even easier to create elements and now these actions can be performed without interrupting the process of diagram design.

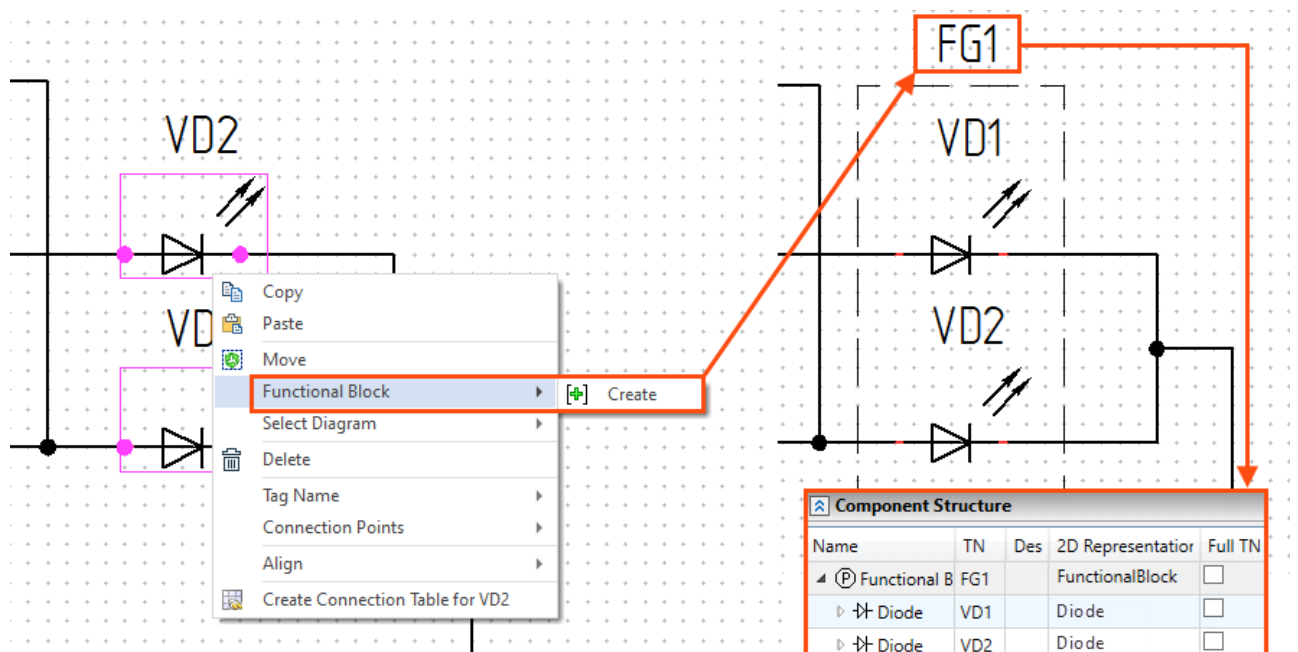


Functional Blocks

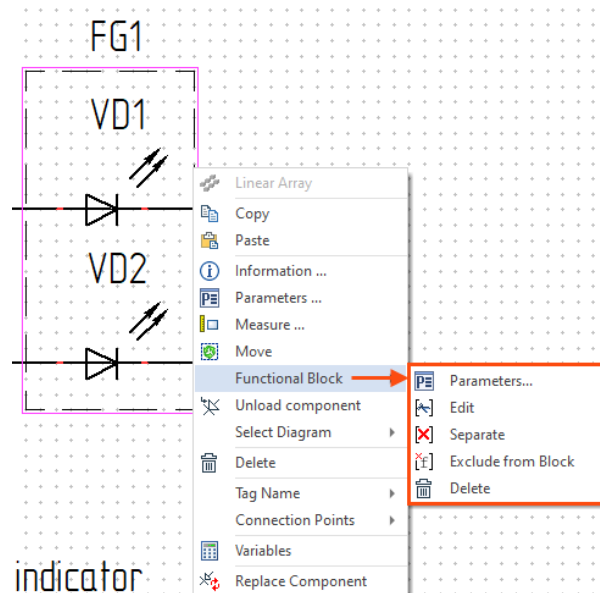
A new tool has been added that allows you to combine any diagram elements into a single group, which can be assigned its own name and tag name, as well as moved simultaneously on the diagram field.



Functional blocks (FB) are created in the context of diagram. To do this, select several elements on the diagram, open the context menu and run **Functional Block - Create** command. A separate structural element for the created FG will be added to the model tree.



The main options of the command are controlled through the context menu called by the RMB when the cursor is pointed at the element.

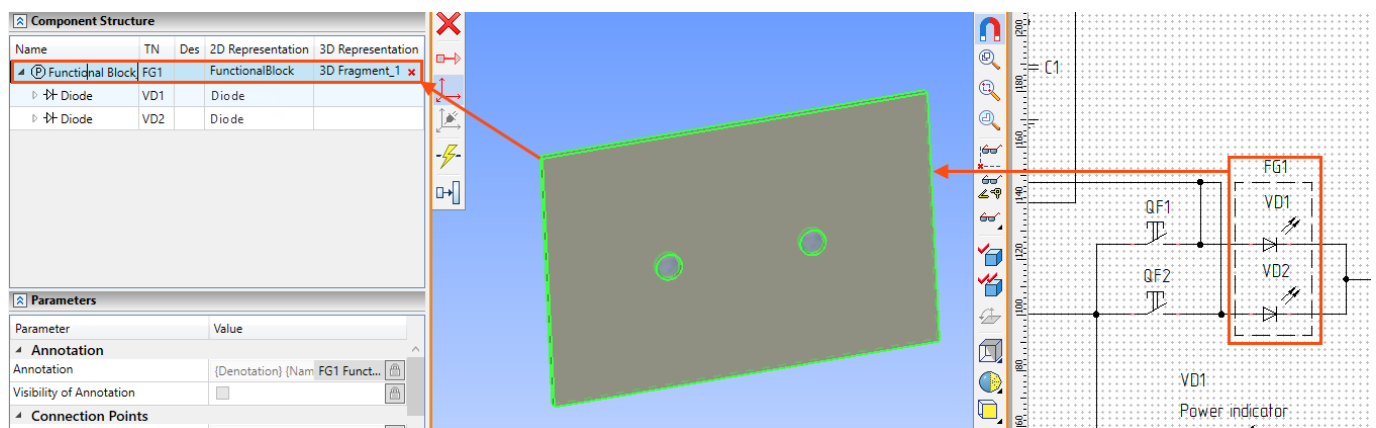


A functional block can be displayed with or without an outline (solid or dashed). In the latter case, the system will automatically turn on the display of the Full Tag Name (see [Full/Address Tag Name and Local Tag Name](#)) for elements included in the FG.

Splitted Representation command is available for functional blocks (see [Splitted Representation for All Types of Compound Elements](#)).

The functional block can be unloaded to a separate file. To do this, use **Unload Component** command (see [Creating Elements in Context of Diagram with Saving to the Library](#)).

Functional grouping also allows you to link multiple elements to a single 3D representation in the product model.



It is possible to configure the representation of functional blocks in reports:

- Displaying a functional group with its own tag name, name and included elements,
- Displaying only included elements with the tag name of the functional block,
- Displaying only functional block.

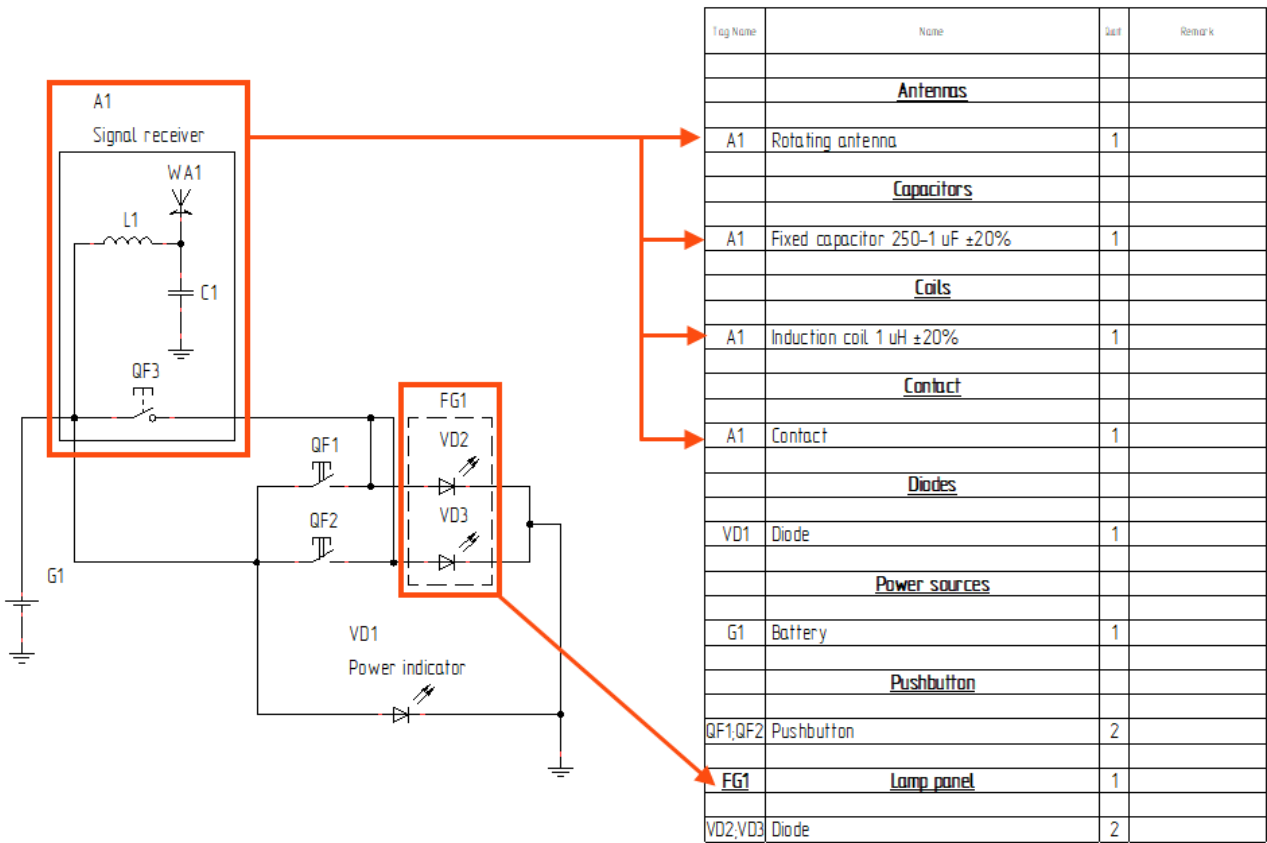


Diagram Parameters

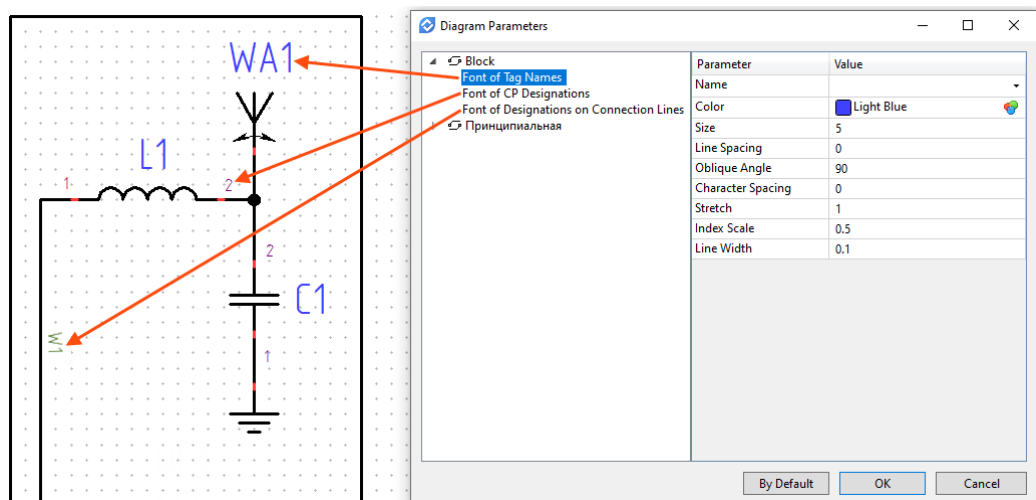
Diagram Parameters command has been significantly improved.

Diagram parameters are stored in the document, that is, each document can have its own parameters. If new diagram need to be created with parameters different from those set in the system by default, you can create your own template of Diagram document, having previously specified the required parameters in it.

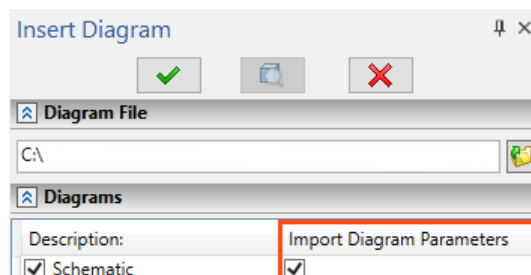
The diagram parameters are the main settings in the document: all new elements inserted into the diagram and created in it "obey" them. An element on the diagram can be set to its own settings, different from the parameters of the diagram. However, after clicking the OK button in **Diagram Parameters** window, all local element settings will be overwritten.

Each type of diagram can be assigned its own parameters in the command.

Fonts. Each type of diagram has its own font settings. Now the **Font of Tag Names**, the **Font of CP (connection points) Designations** and the **Font of Designations on Connection Lines** are separately configured. Separate color adjustment is also available.



The assigned diagram parameters can be saved in the target file after import.

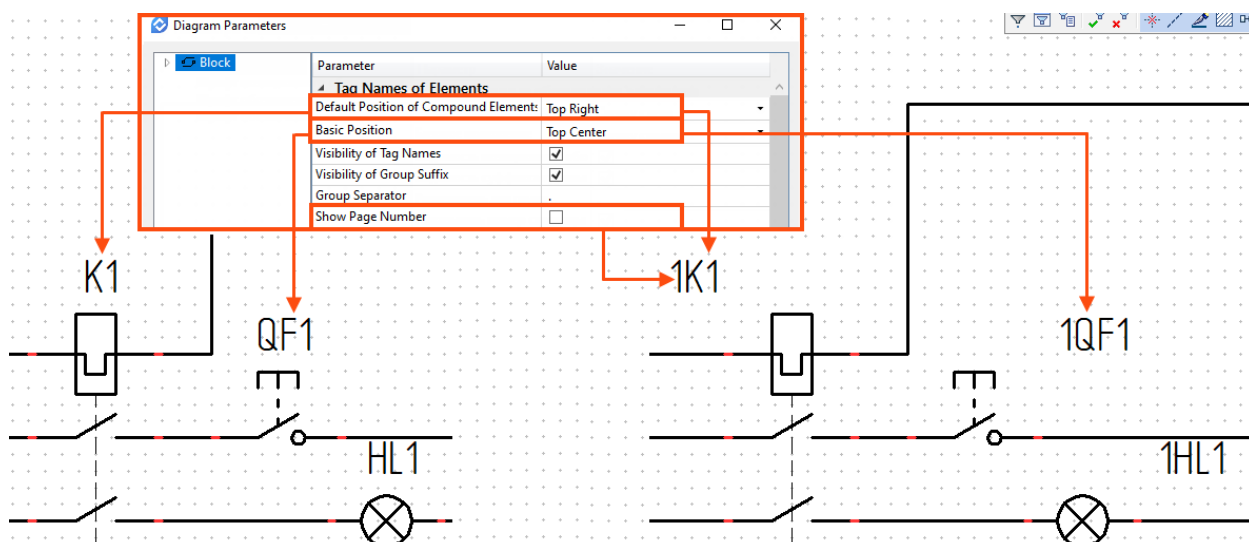


For this, a special checkbox has been added to the **Insert Diagram** command.

The diagram parameters are divided into 4 blocks.

Tag Names of Elements

The position of tag name (TN) for compound and ordinary elements is configured in the block. The position of the TN is set relative to the symbol of element. You can also disable **Visibility of Tag Names** of all diagram elements and **Show Page Number** in all tag names of them. The latter function is convenient when working with a multi-page document.

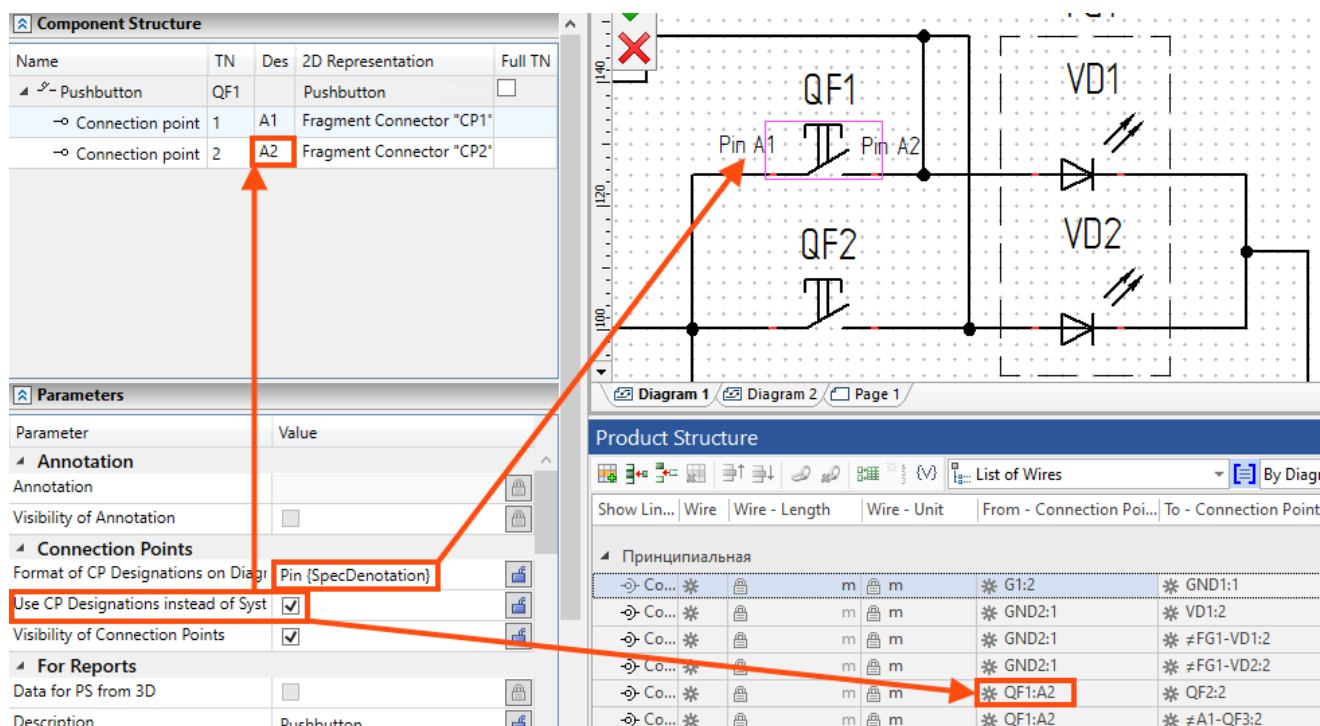


Visibility of Group Suffix and **Group Separator** options are designed to work in splitted representation and are described above (see [Splitted Representation for All Types of Compound Elements](#)).

Connection Point Denotations

The block contains parameters that control connection point (CP) signatures. The **Visibility** checkbox hides all designations of CPs in the diagram.

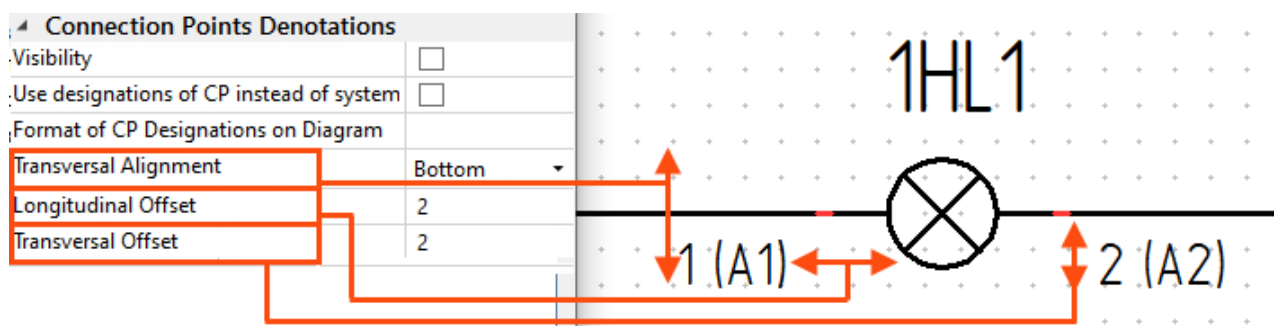
- i** Commentary on the illustration below: the parameters **Use CP Designations instead of System Values** and **Format of CP Designations on Diagram** are available both in the parameters of individual elements and in Diagram Parameters. For ease of understanding, the operation of these parameters is explained using one element as an example. They are applied simultaneously to all elements in the diagram, if set in Diagram Parameters.



Use CP Designations instead of System Values checkbox replaces the system designations of CPs with custom ones (in the connection table). The custom value will be taken from **Designation** parameter of connection point.

Format of CP Designations on Diagram field is intended for entering a "rule" that controls signatures for all connection points of an element. In the screenshot above, SpecDenotation is a system synonym for **Designation** parameter.

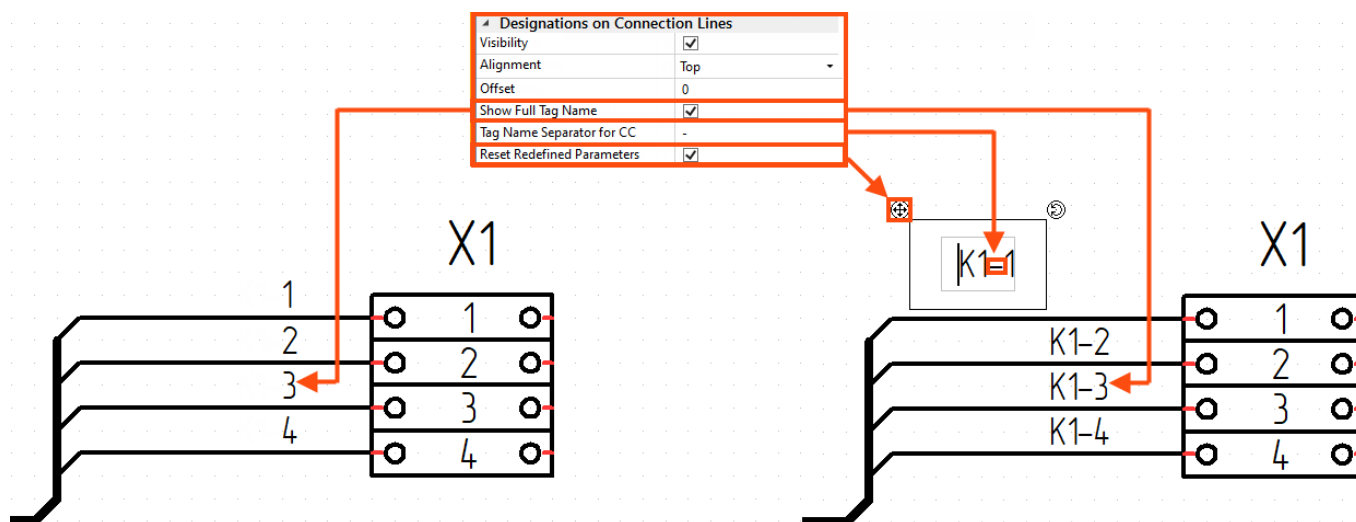
Longitudinal Alignment and **Longitudinal Offset** parameters control the position of the CP signatures relative to the connection lines. The **Transversal Offset** parameter controls the position of CP signatures relative to 2D connectors.



Designations on Connection Lines

The block contains parameters that control signatures to connection lines (CL): tag name, name and other parameters of cable components (CC). The **Visibility** checkbox hides all designations of the CLs on the diagram. The **Alignment** and **Offset** options set the position of the text relative to the CL.

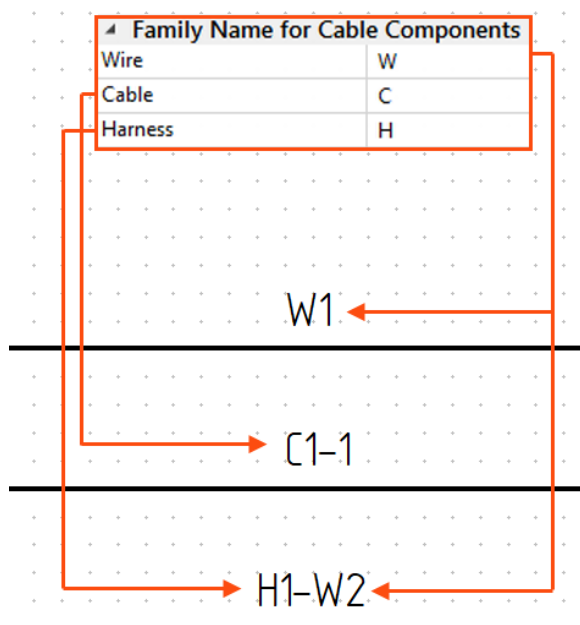
The **Show Full Tag Name** checkbox allows displaying the complete tag name of cable components on the CL. The **Tag Name Separator for CC** parameter sets the character to be inserted between the parts of the full tag name of CC.



The **Reset Redefined Parameters** checkbox prohibits and allows arbitrary placement of text blocks related with connection lines. With the active flag, "manual" movements will be reset, with the unchecked - saved.

Family Name for Cable Components

The block contains fields for specifying prefixes used in the formation of tag names of conductors.



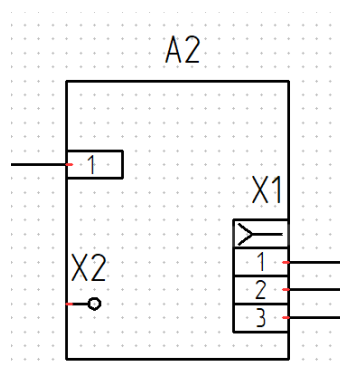
Thus, all cable components will be assigned the tag names using a unified "mask" specified in the diagram parameters.

Full/Address Tag Name and Local Tag Name

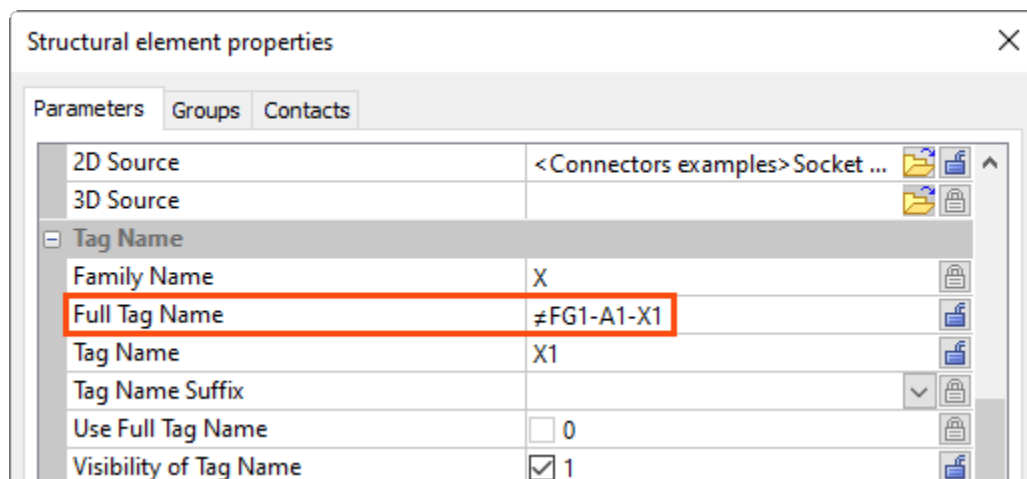
In previous versions, the **Tag Name** parameter was used to uniquely identify an element on a diagram. Typically, tag name (TN) consists of a prefix indicating belonging to a group of elements, and the serial number of the element on the diagram.

All elements have a new parameter **Full Tag Name**, which makes it easier to identify the element. The full TN consists of TN of the element itself and the TN of the "higher" components, of which it is included. Now, when it comes to an element of a compound product, it is possible to trace the entire "affiliation chain" of the element with the help of full TN.

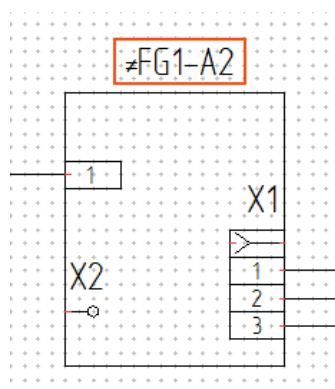
For example, we see X1 connector in A2 device on the diagram.



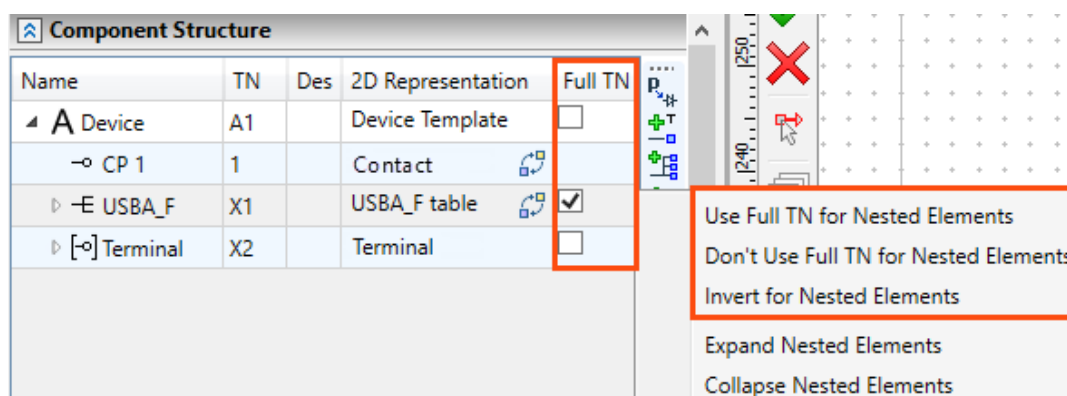
To check the affiliation of the connector, we turn to its parameters and see in **Full Tag Name** line that A2 device, in turn, is part of FG1 functional block.



In order not to forget about this (the tag name and the frame of functional block are hidden in our example), we display full TN of the device instead of the local one checking **Use Full Tag Name** in device parameters.



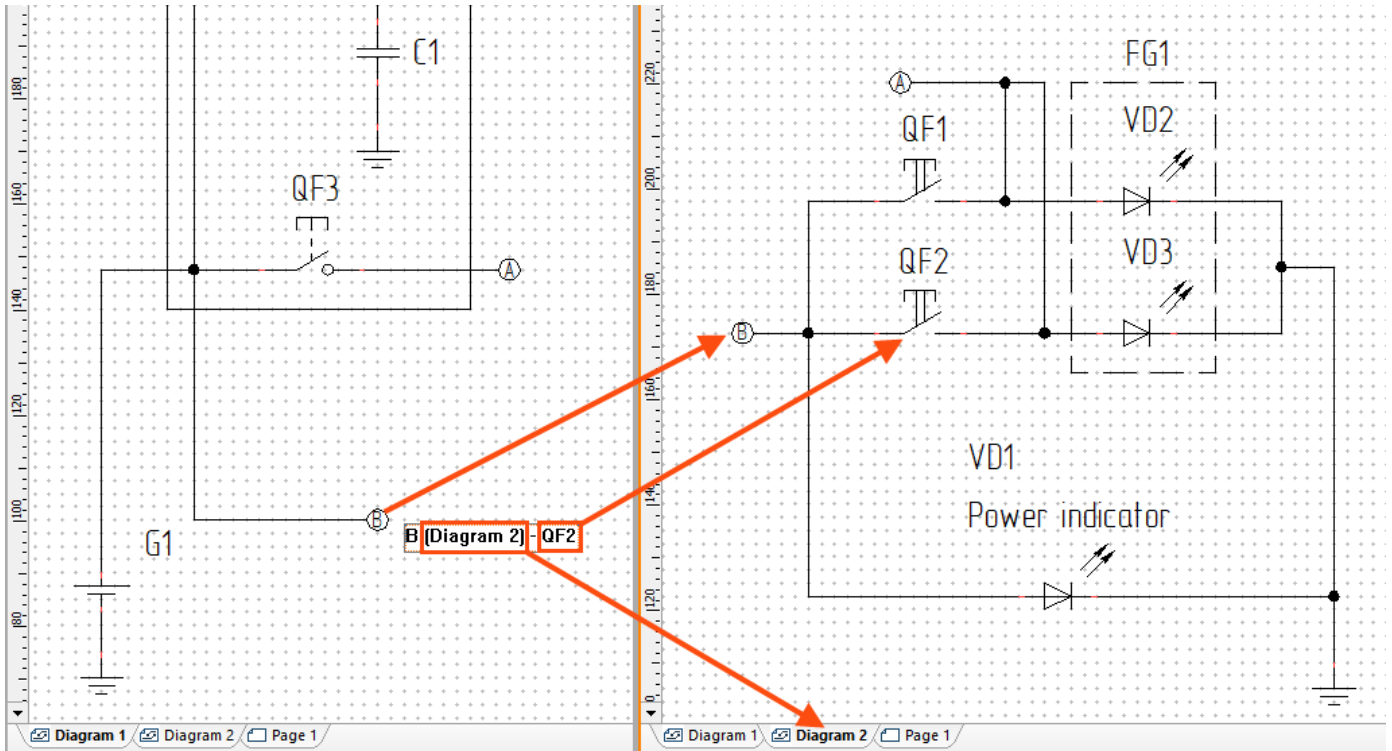
Quick access to **Use Full Tag Name** checkbox is provided in all component editing dialogs at all nesting levels (**Full TN** column).



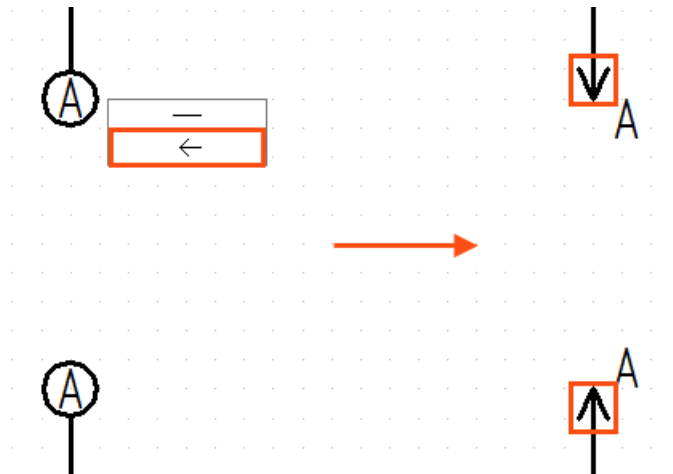
In addition, the commands for group setting of the full TN are available in the drop-down list by clicking on the RMB.

Hyperlinks for Breaks of Connection Lines

The ability to switch between the break points on connection line and group connection line was added. To do this, just point the mouse cursor over the break point and press the button to call the drop-down list. After click on the address, the diagram area with the associated break point will be displayed on the screen. If the break points are created on different pages of the diagram, then clicking on the hyperlink will move the user to another page.

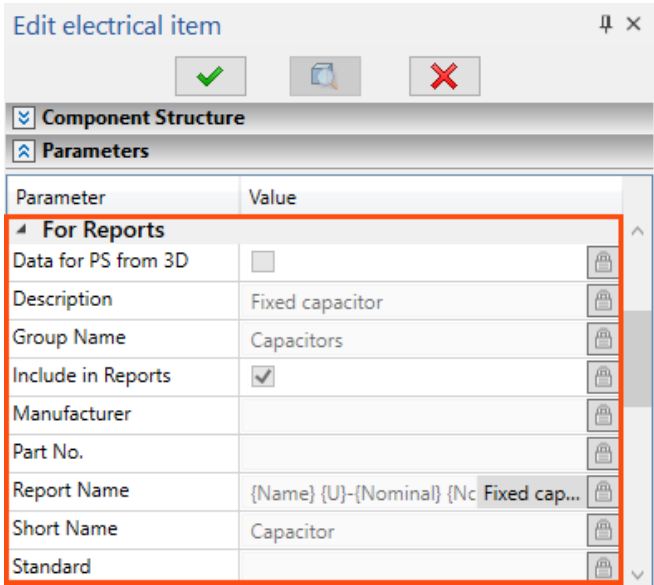


Terminations of breaks in connection lines can be indicated for different options, with a synchronous change within the group of one break.



Setting Report Information about Elements from Context of Diagram

The new version of the module has an opportunity for flexible customization of information displayed in reports. Parameters are edited in context of diagram using **Edit Component** command, which is running from the automenu of selected element.



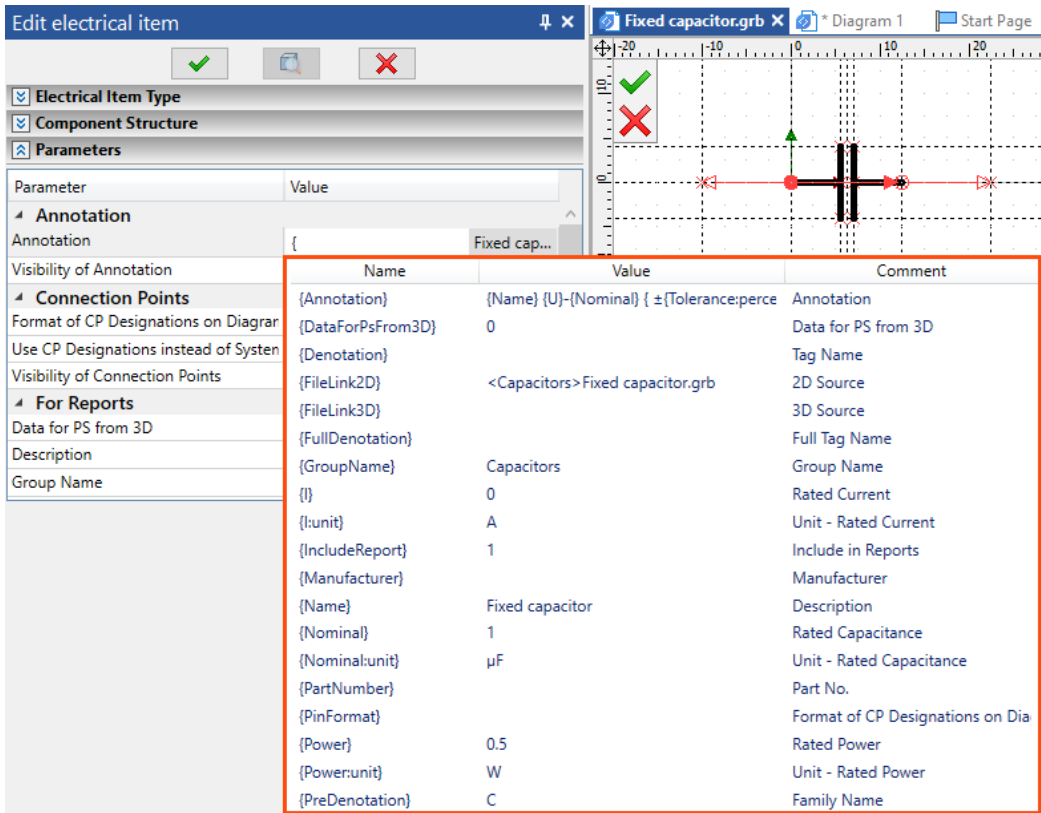
In the command window, **For Reports** section is available in the **Parameters** block.

Note: to change the parameter, the lock on the right edge of the line must be "open": . "Opening" is performing by clicking LMB on the icon: → . Clicking on the "open" lock will "close" it and restore the default value.

New parameters – **Report Name** and **Short Report Name** are intended for specifying the information that will be displayed in the report, or for the "mask" of output to the report. These parameters can be configured both in **Components Editor**, even before the element is created, and for an element already inserted into the diagram.


Parameters Selection When Entering a Value

When editing some component parameters (**Description**, **Report Name**, **Short Name**, etc.) in **Parameters** block, you can use the values from other columns. In this case, you can use sequentially as many previously filled parameters as you like and alternate them, if necessary, with arbitrary symbols. To call the list of available parameters, enter the curly bracket symbol "{" into the string.

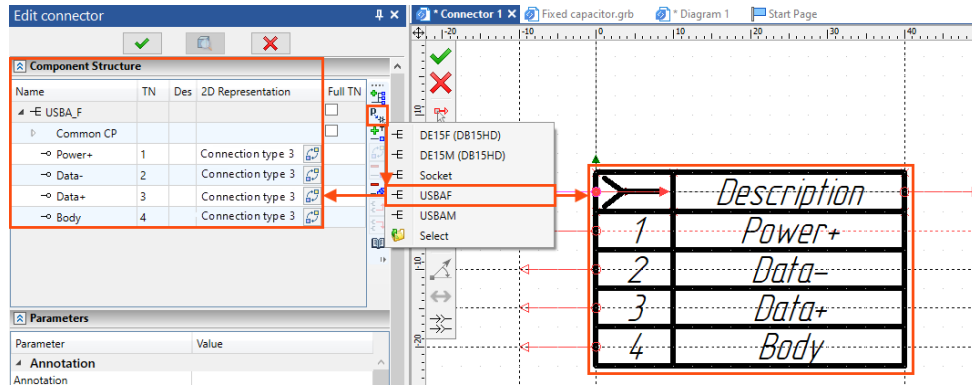


Parameter values are entered as system synonyms. This allows you to form the value of the "composite" parameter from the values of the parameters "included" in it and automatically update "composite" parameter if the "included" parameters were changed.

Replacement of a Prototype in an Existing Component

When creating the elements **Device**, **Terminal Block**, **Contactor**, **Chip** and **Connector**, it became possible to assign and replace a prototype from **Component Editor** using the **Set Prototype** command. The button  is located in the dialog of **Create <Component>** and **Edit Component** commands.

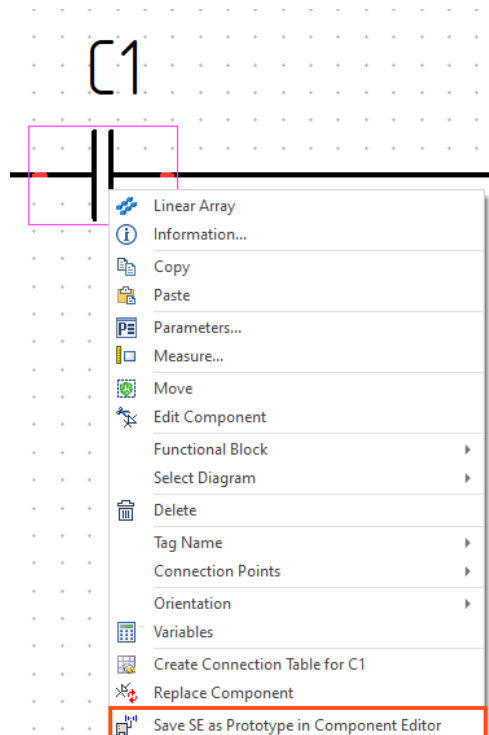
The command is available when creating the compound components in a separate file and in context of diagram.



After clicking on the button in the dialog, you need to select a new prototype of the component. For connectors, when changing the prototype, the system will also suggest changing the composition of the contacts.

Saving Element to Component Editor as Prototype

For elements created in context of diagram or received from another user with diagram (not explicitly saved in the library), it became possible to save the element data as a prototype of the **Component Editor**. This can be done from the context menu of the element.



This function is also useful if parameters of a library element have been changed in the diagram, and a prototype with such parameters may be useful in the future.

3D Cables and Harnesses

Splices

By default, when creating a cable component by a group connection (connection of more than two connection points), the system creates multiple wires, where each wire directly connects to each other two of the connection points. In result, some segments of a harness contain multiple parallel wires, which increases materials consumption and weight of a product.

Splices allow to connect single wires to each of the connection points and connect these wires to each other in selected points on a harness.

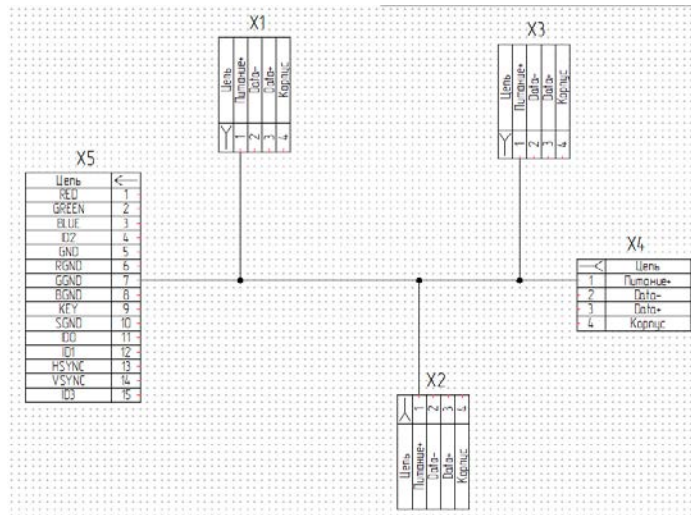
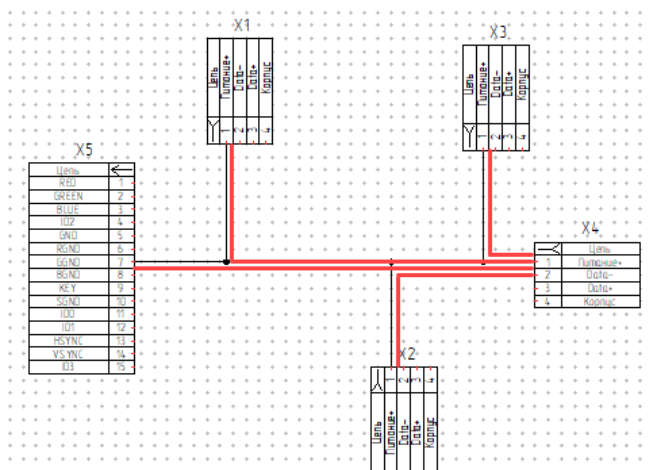
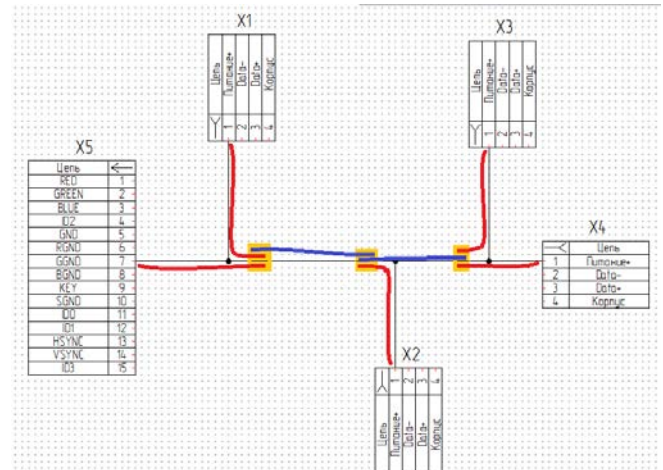




Diagram with group connection



Wires without splices

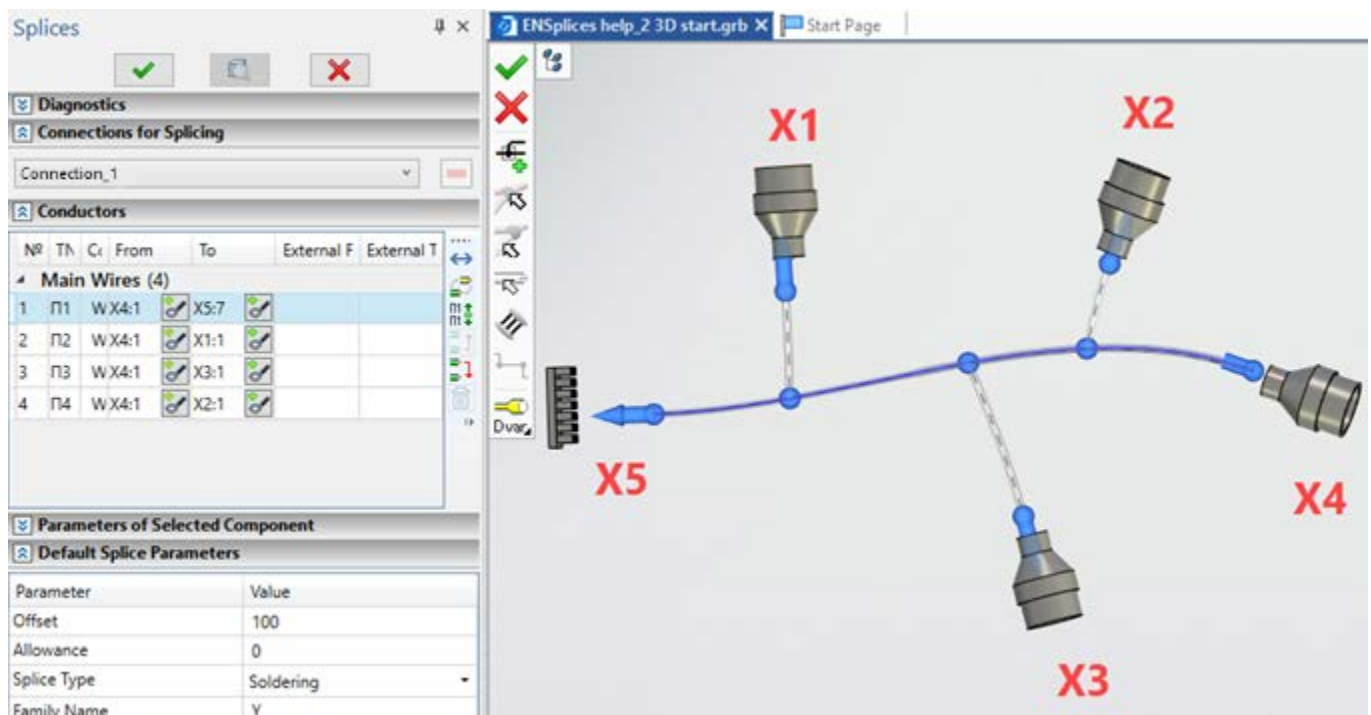
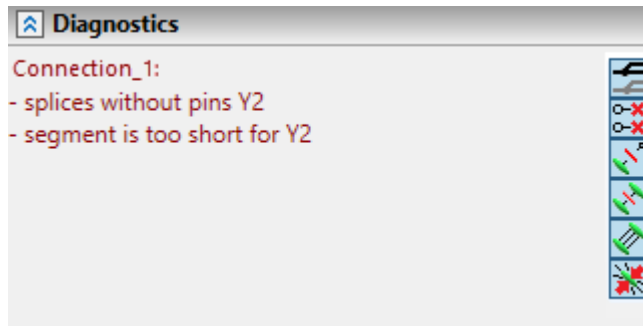


Wires with splices

Splices are created using the new command  **Create Splices**, which is available in contextual menu of a 3D harness containing a group connection. If a harness already contains splices, this command changes name to  **Edit Splices** without changes in functionality.

Command's parameters window is split into following sections:

- **Diagnostics**
Errors found by the system are displayed here. Buttons in the right side of the window allow you to enable/disable display of particular types of errors.
- **Connections for Splicing**
If a harness contains multiple group connections, here you can select a connection to work with. You only work with a single connection at a time.
- **Conductors**
All wires of the selected connection are listed here. Right side of the window contains buttons for working with wires, same as in the harness creation and editing commands.
- **Parameters of Selected Component**
Parameters of a selected wire are displayed here, same as in the harness creation and editing commands.
- **Default Splice Parameters**
Here you can customize parameters assigned by default to new splice points. Parameters of an already created point can be edited using the Parameters command available in point's contextual menu.



Splicing creation starts with creating splice points, i.e. points, where wires are connected to each other. Points are created using the following automenu option:



<N> Create New Splice Point

Upon activating this option, another option is activated automatically:



<T> Select harness branch marker to snap splice

Select in 3D window a harness branch marker. In result, another option is activated automatically:

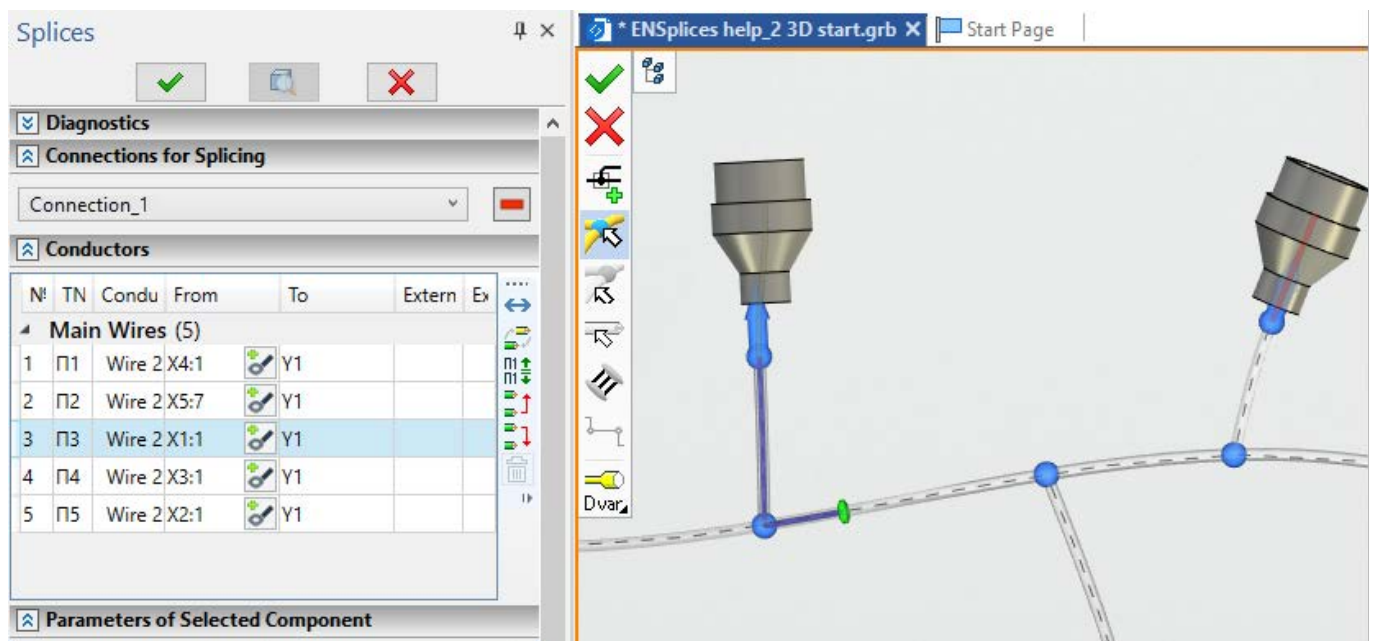


<S> Select harness segment to place splice

Select in 3D window a harness segment, where you want to place a splice point. The segment should be directly connected to the selected harness branch point. In result, a splice point will be created on the selected segment, and initial wires will be replaced by new ones, where each new wire connects one of the connection points to the created splice point. By default, tag names of splice points start with Y letter followed by the order number of a point (Y1, Y2, and so on). In 3D window splice point is indicated by a toroidal marker enclosing the harness. Marker may have one of the following colors:

- **Green**, if the point is connected to at least one of the connection points of components and either there are no other splice points in the current connection, or the point is additionally connected to another splice point.
- **Yellow**, if the point is connected to at least one splice point, but isn't connected to any connection points of components.
- **Red**, if there are other splice points in the current connection, and the current point is not connected to any of them.
- **Gray**, if the harness contains multiple connections, and the current point belongs to an inactive connection (i.e. connection other than the one being currently edited).

The position of the splice point on the harness is defined by default by the **Offset** from the selected harness branch point. You can drag a splice point along the harness with mouse, if necessary.



Upon creating all necessary splice points, you have to connect wires to them. In order to connect wires to a splice point, select the point's marker in 3D window and activate the following automenu option:



<W> Select wires to connect to splice

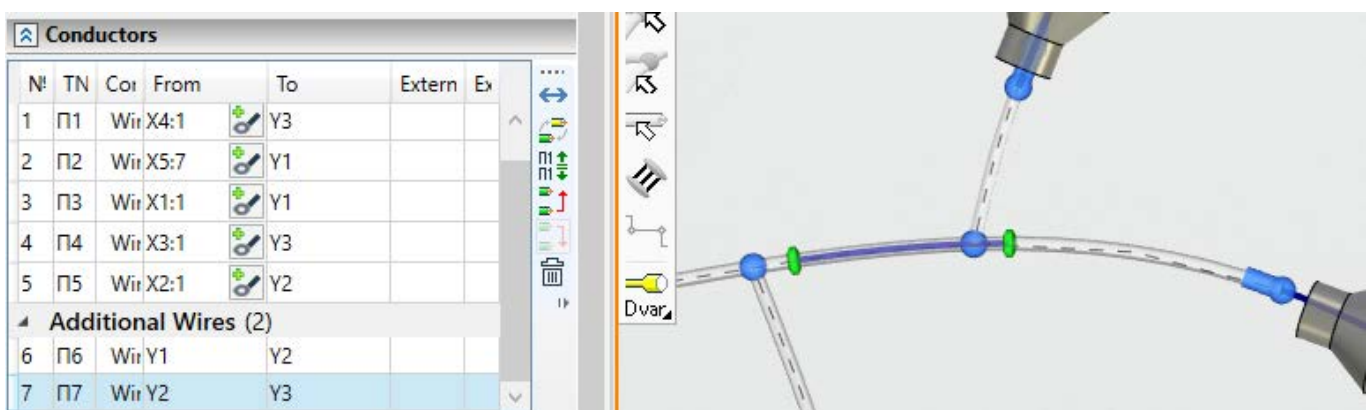
Upon activating this option, selecting wires in 3D window or in the parameters window connects them to the selected splice point.

If a harness contains more than one splice points, then after connecting existing wires to splice points, you have to connect splice points to each other by new wires. To do so, select a splice point's marker in 3D window and activate the following automenu option:



<C> Connect to Splice

Upon activating this option, selecting another splice point in 3D window creates a new wire, which connects the current point to the selected one. The resulting wires are put into the **Additional Wires** group in the list of wires.



There are several methods of modelling a harness containing splices in 3D scene. Following automenu options are used for switching between these methods:



Dmax <0> Create Harness with Maximum Diameter

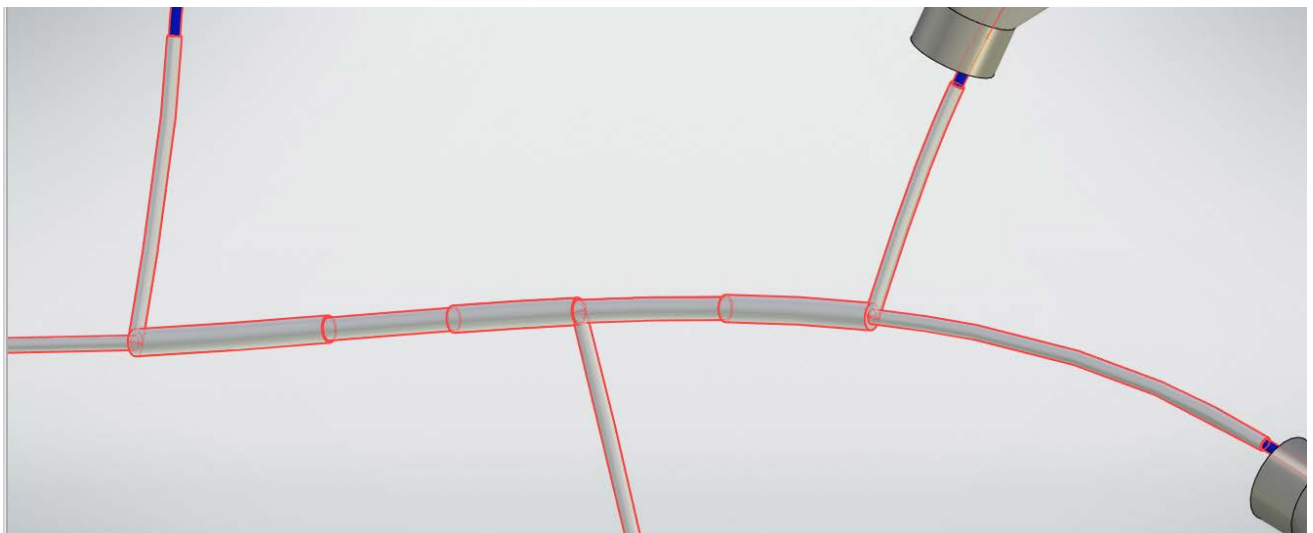


Dmin <1> Create Harness with Minimum Diameter



Dvar <2> Create Harness with Actual Diameter

The actual diameter option is selected by default, so different segments of a harness have different diameters:

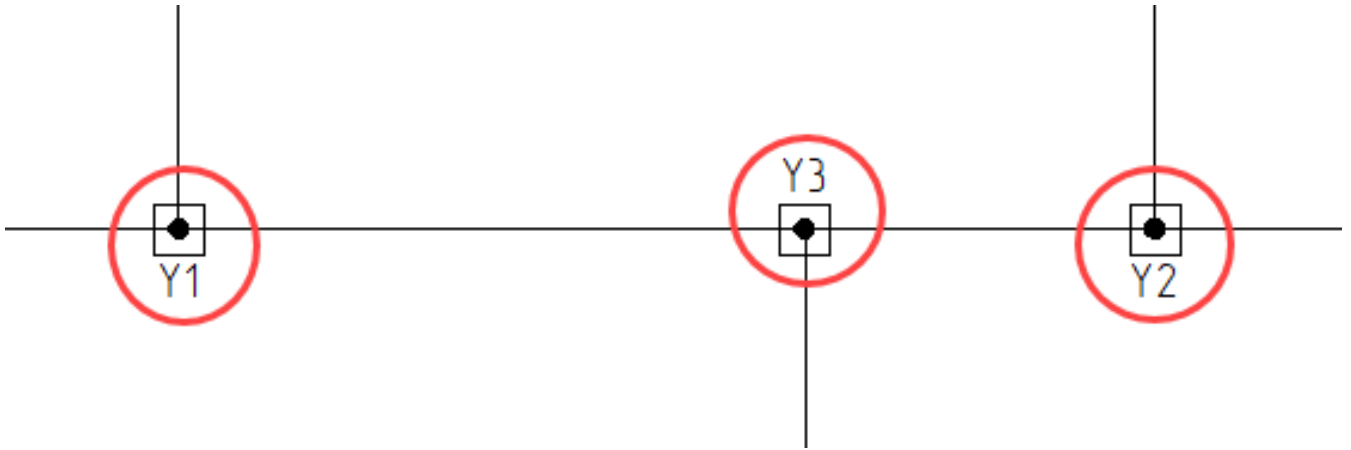


In order to place a splice point's tag name on a diagram, select a splice point's marker in 3D scene and activate the following automenu option:

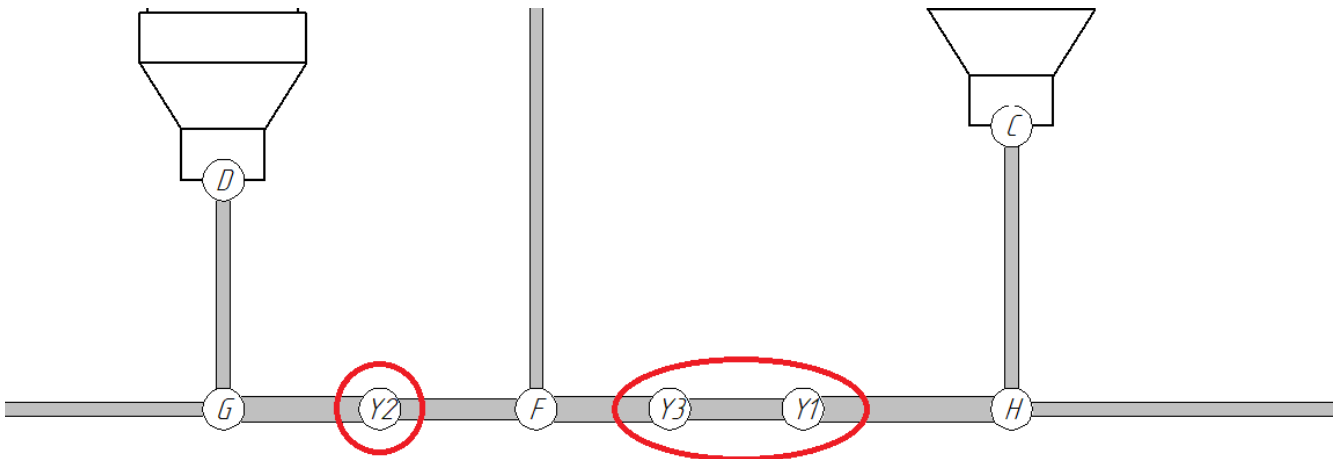


<R> Select point on connection line for splicing

Upon activating this option, you have to select in a diagram a branch point of connection line corresponding to the selected 3D splice point. In result, the point in the diagram will change its appearance to the one representing splice and the tag name of the splice point will appear nearby.

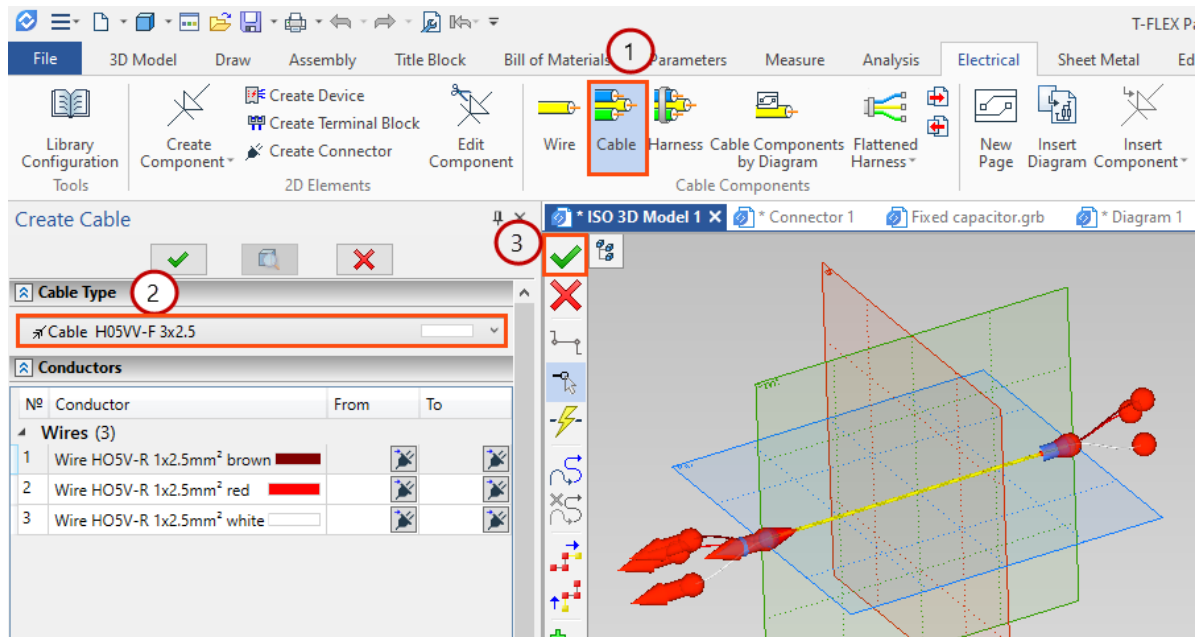


Splice points are automatically indicated in assembly drawings of cable components.



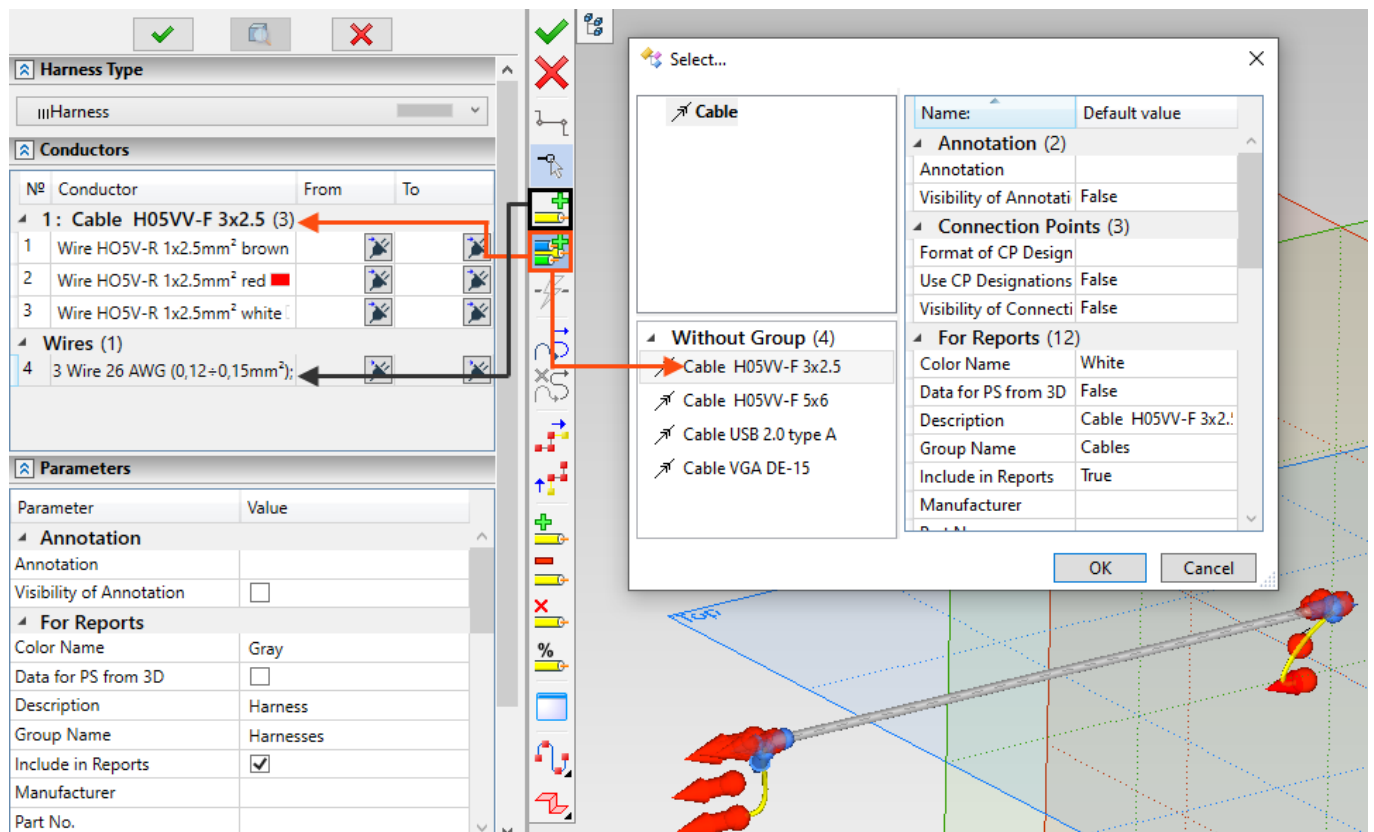
Creating Cable Components without Diagram

Now Cable Components (CC) can be created in a 3D scene without referring to the diagram at all. The **Cable** and **Harness** commands create the corresponding objects in the 3D scene after selecting a prototype and confirming the operation. If the association with the diagram is performed, the geometry of the CC will be rebuilt after finding the specified connectors.

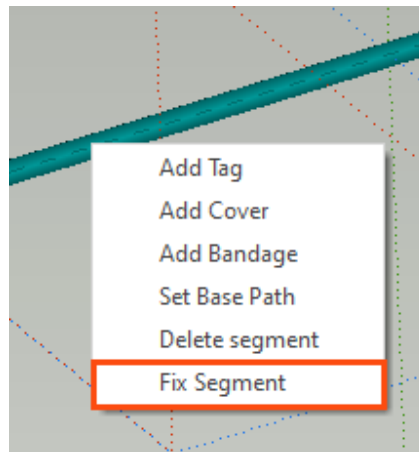


New Functions for Managing Topology and Composition of Cable Components

A harness can now be filled with new wires and cables directly in the **Harness** command, without first adding them to the scene using separate commands. To do this, buttons have been added to the automenu of the command. After clicking on one of them in the dialog, you need to select a prototype of a wire or cable that will be added to the harness.

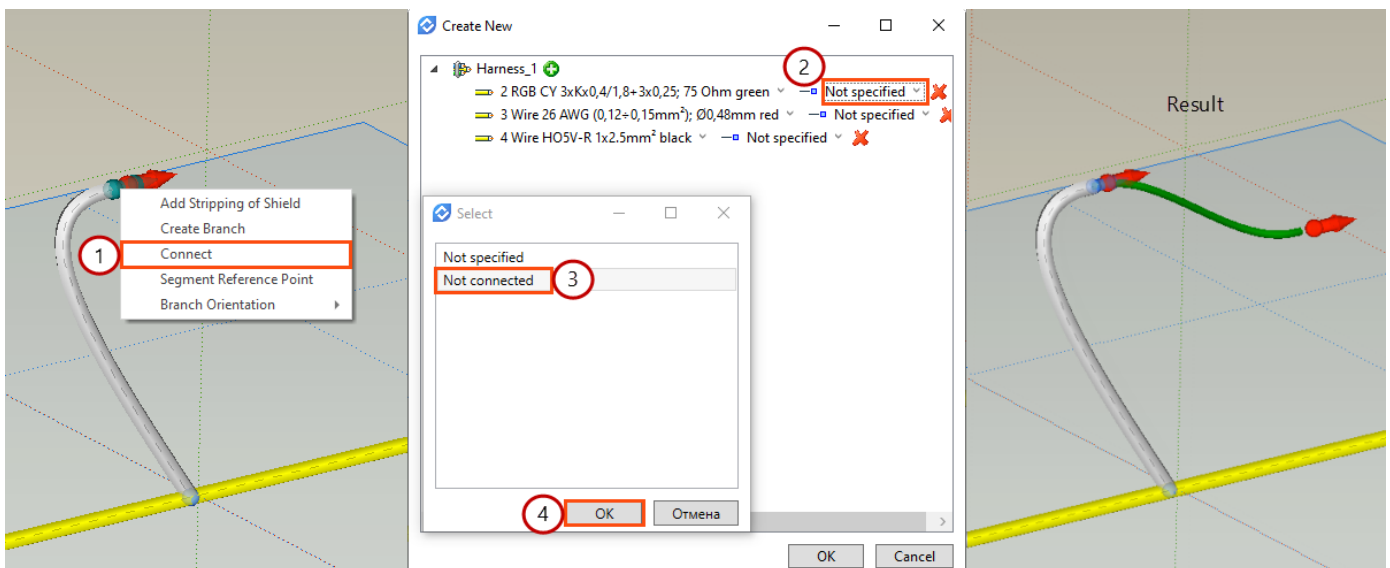


Individual segments of the harness can now be fixed. If the harness geometry is updated, the fixed segment will not change (it will retain its position and geometry). Also, the **Optimize Conductors** option will not affect the geometry of the fixed segments as they are excluded from the optimization calculation. To fix a segment, select it in the harness editing mode and activate the **Fix Segment** function.



Setting Points of Entry and Exit of Wires into Harness

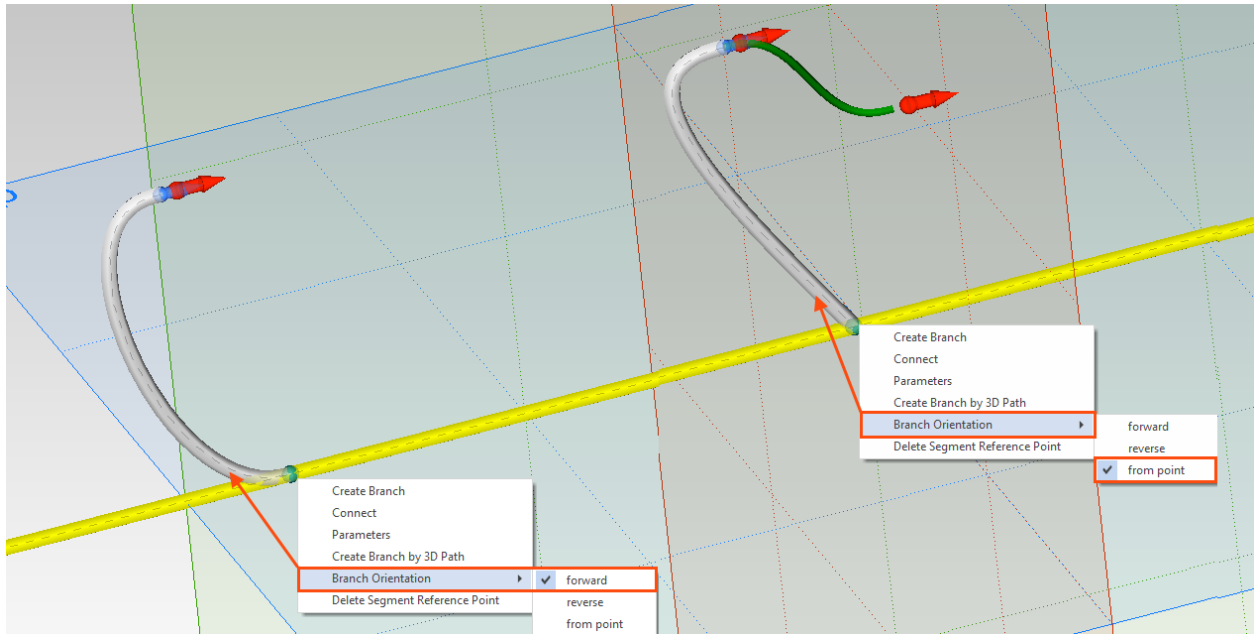
The entry and exit points for wires into and out of the harness can now be set manually. To do this, use the **Connect** command. The command is run from the context menu of endpoints and branch points of the harness.



To remove the wire from the selected point, select the **Not Connected** option next to the wire name in the opened dialog, and **Not Specified** option in order to return the wire inside the harness sheath.

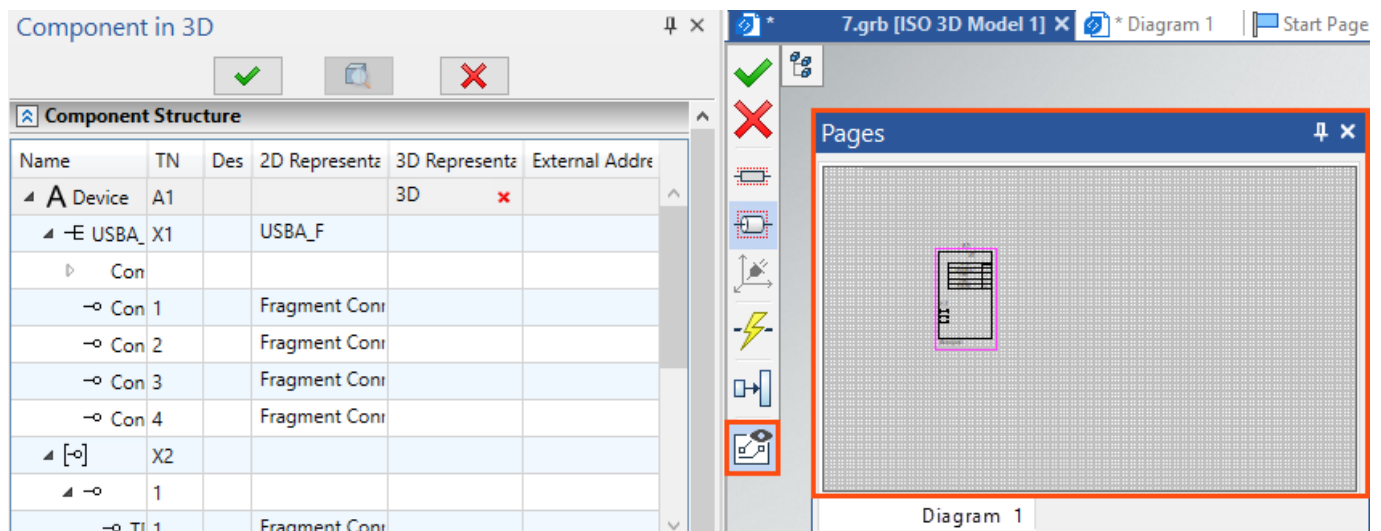
New Forms of Harness Branches

Harness branches can now have a T-shape. To do this, open the context menu of the branch point and select the appropriate option for **Branch Orientation**.



Designing a harness in the context of an assembly

A harness that is part of a subassembly can now be designed with snapping to assembly elements, since **Component in 3D** command now has access to the subassembly diagram. Access appears if the subassembly is open in context of assembly.



To accomplish the task, you need to open a harness fragment in context of assembly, run **Component in 3D** command, select an element in the harness diagram, and then select 3D connectors from the top-level assembly. The connectors will come to the subassembly as referenced elements.

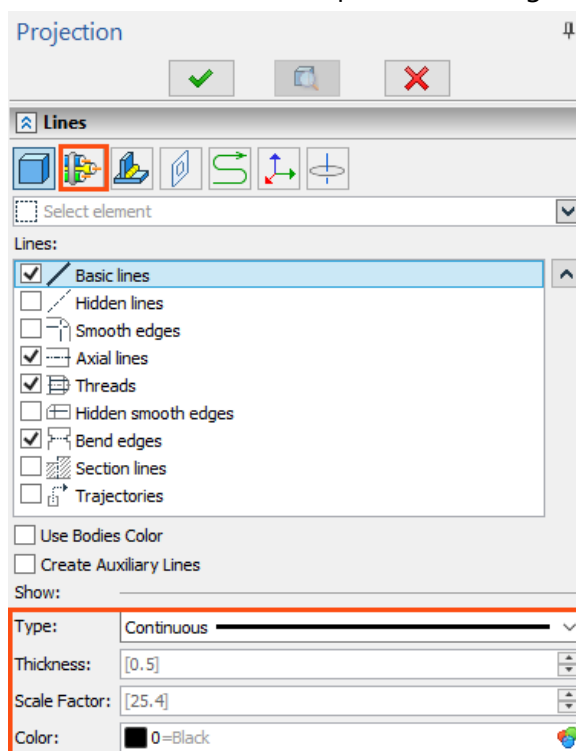
It is important to remember that in order to select assembly objects, **Select Assembly Elements** option must be enabled in the menu in the upper right corner.



Reports and Drawings

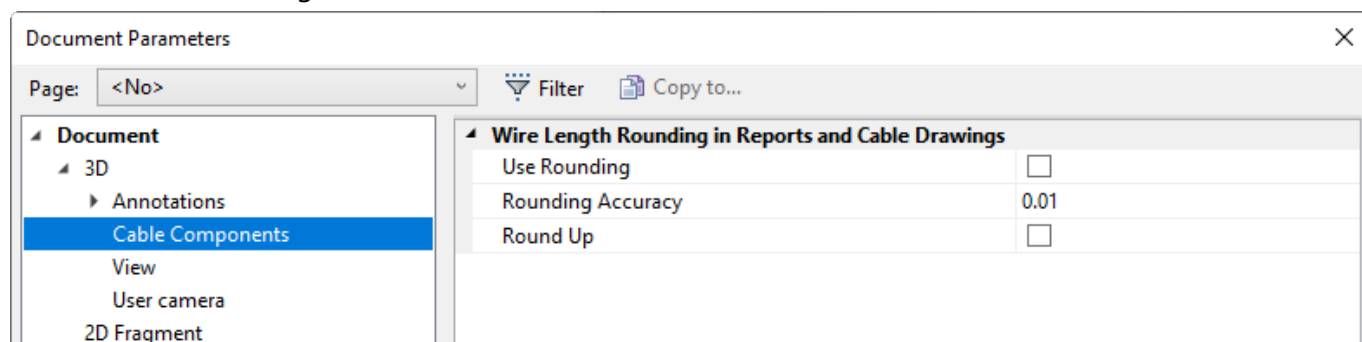
Drawings of Cable Components

A block of settings for cable components has been added to **Projection** command in **Lines** section. Here you can set your own settings for the lines of cable component drawing.



Automatic rounding of CC lengths in reports

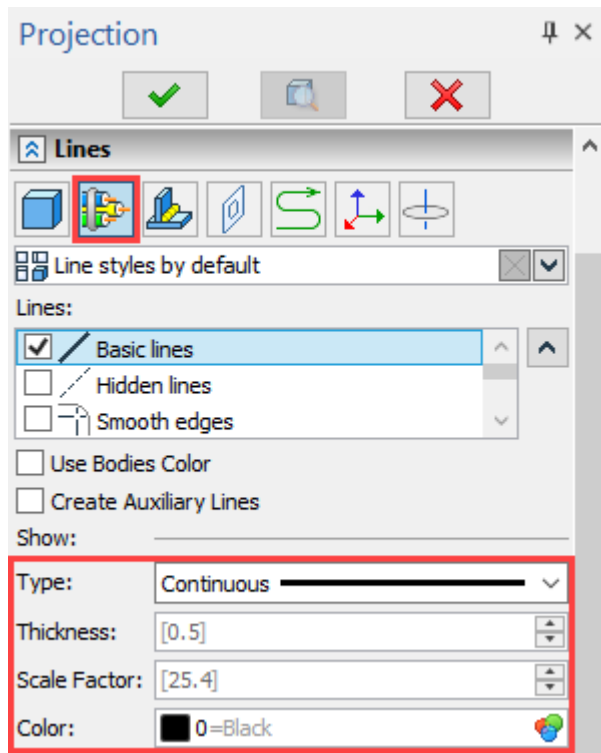
Automatic rounding of lengths of harness segments and CCs in reports and drawings has become available. For this, a new **Cable Components** group of parameters has been added to **Document Parameters** command in 3D settings block.



Rounding is performed with a user-selectable accuracy (in meters), rounding to stock is possible (**Round Up**), length rounding in documents of all types is strictly coordinated.

Projection of CC

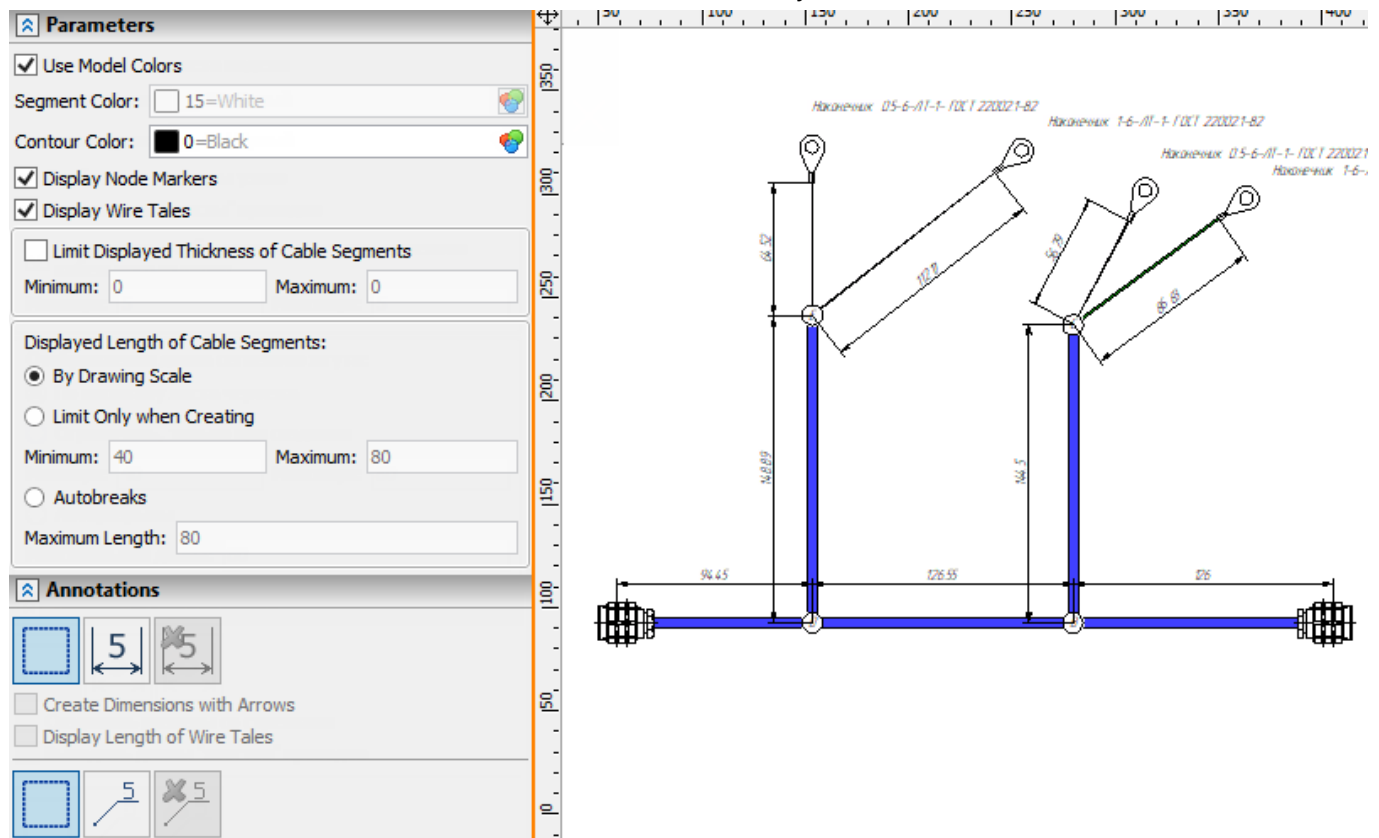
Lines tab of the 2D projection command's parameters now has the cable components section, where you can customize cable components' lines parameters.



Assembly Drawing of Cable Component

The **Assembly Drawing of Cable Component** command has been reworked.

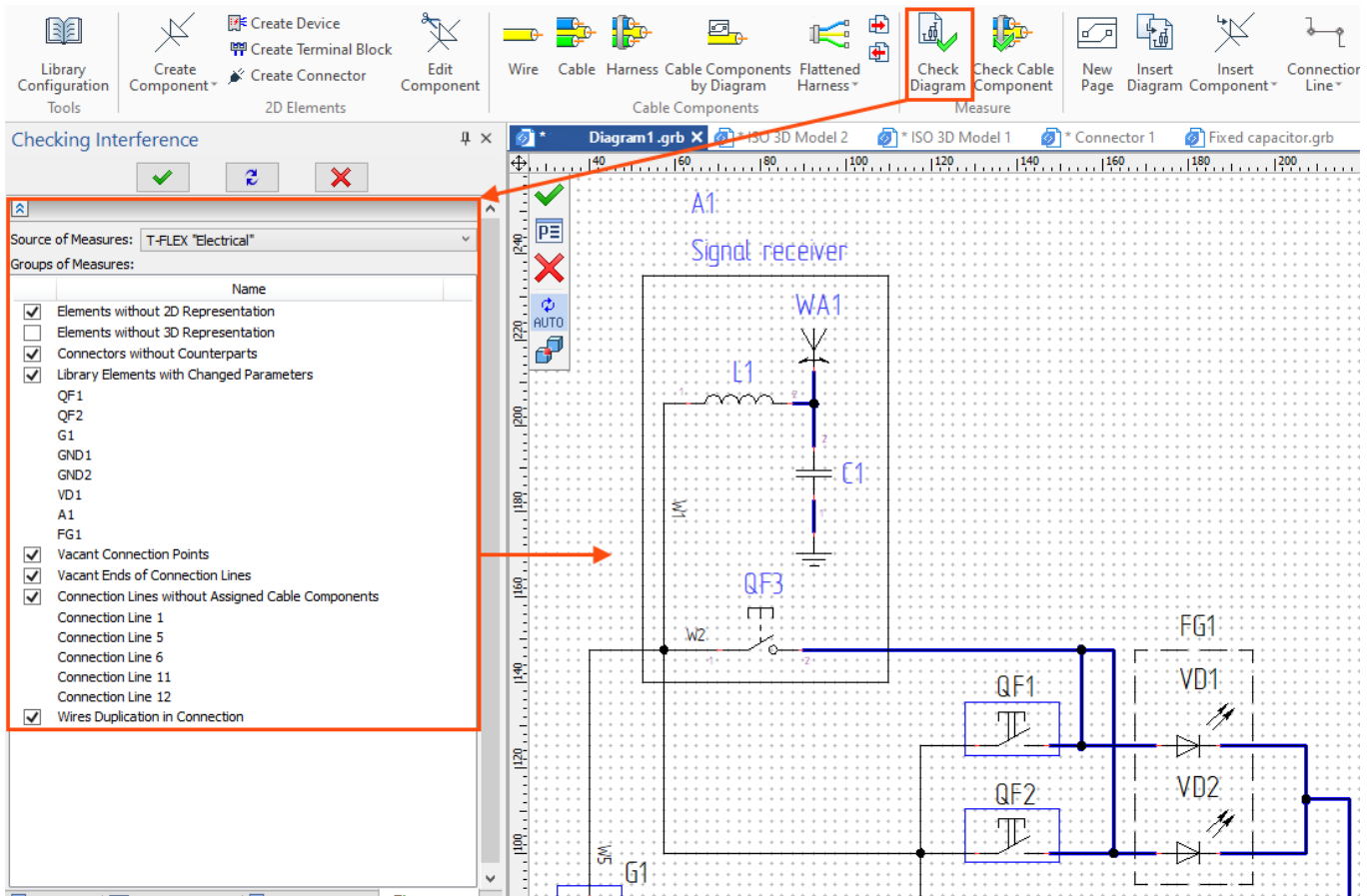
Added customization of colours, displayed length and thickness of segments, node markers. Dimensions and leader notes can now be added automatically.



Functions of Control over Designer's Actions

Diagram Data Analysis Tools

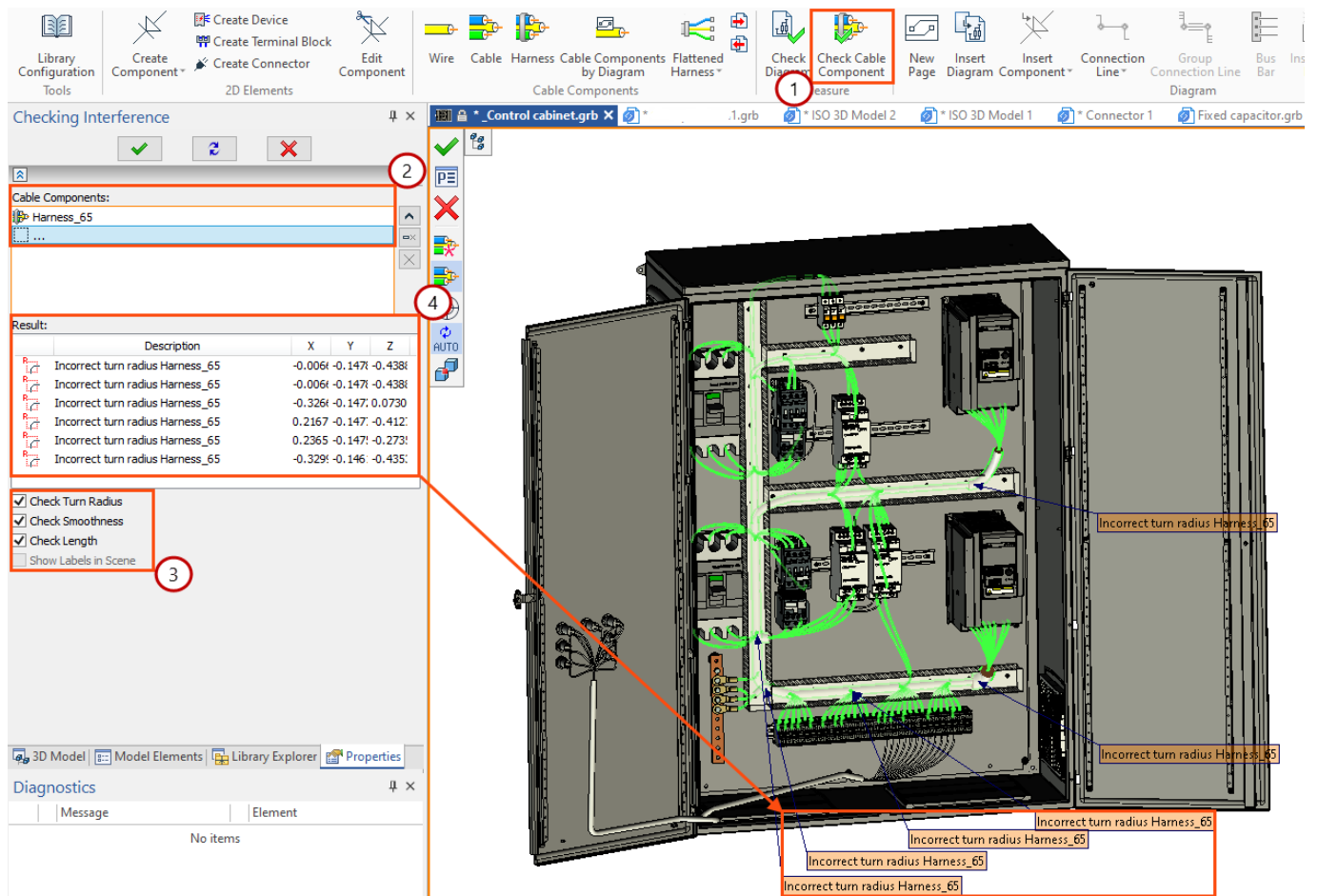
In the new version of T-FLEX Electrical module, the user can check the developed diagrams for errors and design inaccuracies using the **Check Diagram** command located in the **Measure** group of **Electrical** tab. The inspection result is automatically displayed in the corresponding service window when the checking parameter is selected. When errors are detected, the system highlights the results in the 2D scene.



The result of the check can be saved as a permanent set of labels in the scene, which is provided by a special option in the automenu of **Check Diagram** command.

Geometry Analysis Tools for 3D models of Cable Components

Now it is possible to check the geometry of cable components when designing harnesses in 3D. It's performed in **Check Cable Component** command, ran from the **Measure** block on **Electrical** tab of the Ribbon. After the selection of cable components for checking, as well as the type of necessary inspections, the system displays a report on the result and, in case of errors, indicates them in the 3D scene.



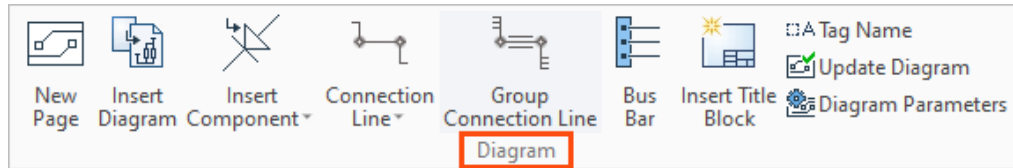
The result of the check can be saved as a permanent set of labels in the 3D scene, which is provided by a special option in the automenu of **Check Cable Component** command.

Interface

Redesigned Ribbon

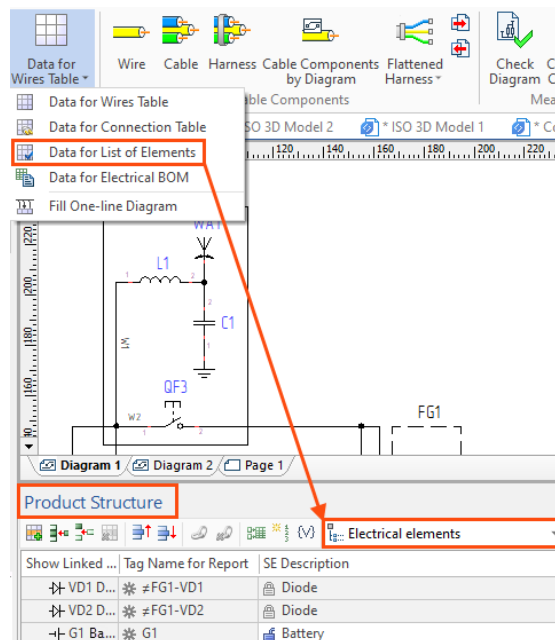
The **Electrical** tab of the Ribbon panel has been redesigned.

The grouping of commands has changed, the order of the groups has also been changed. For example, all commands for working with a diagram are now collected in the Diagram group.



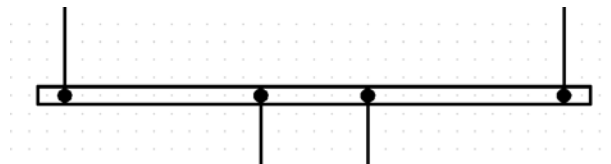
New Commands

New commands were added to **PS (Product Composition) data** group.



With their help, you can create a product structure of a given type in a document with one click, and the system will automatically open it in the **Product Composition** window.

The **Bus Bar** command has been added to the **Diagram** group to display electrical buses. Buses are drawn similarly to connection lines.



The **Create 3D <Component>** commands have been added to the **3D Elements** group.

T·FLEX CAM17

What's New

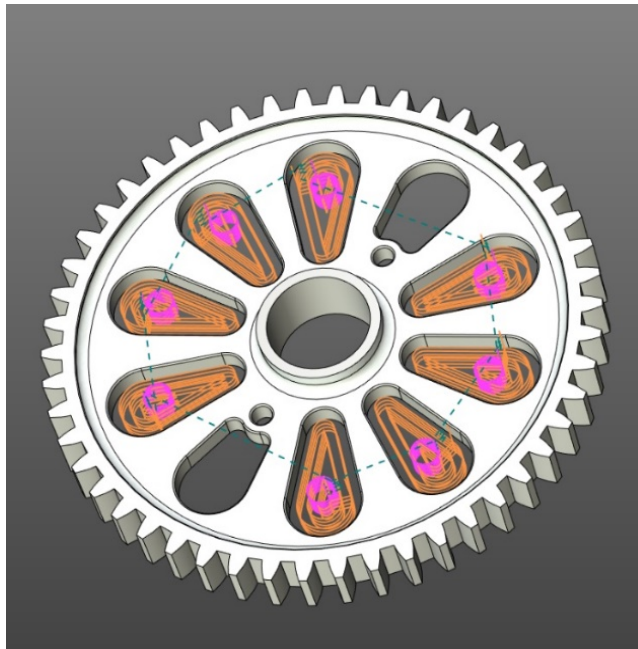


T-FLEX CAM 17

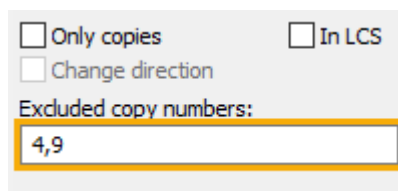
A significant number of changes have been added to the T-FLEX CAM 17 module, which include functionality for working with toolpaths arrays, the ability to change the colors of toolpaths sections, new possibilities in 3D milling, engraving. The machining manager window has been redesigned, the ability to form installations with different positions, configurations and set of tools has been added, and much more.

Array of Toolpaths

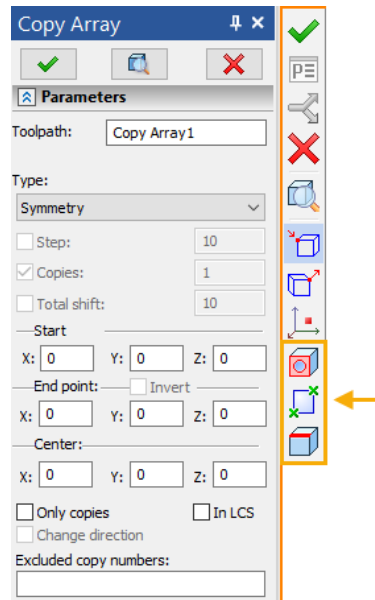
Significant work has been done with the array of toolpath.



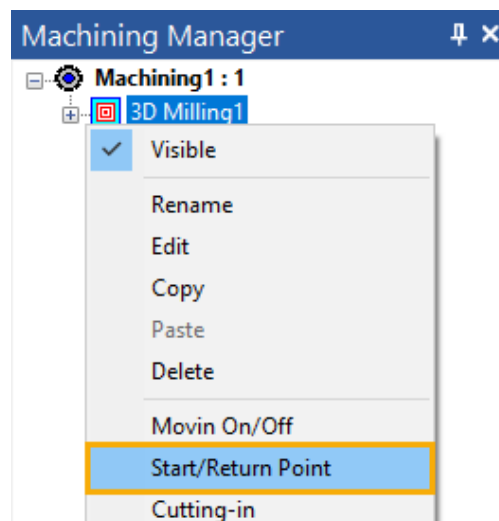
The ability to exclude an element of the toolpath array by its ordinal number has been added. You can specify both numbers and the range of elements (for example, 3-5).



The ability to use the workplane as an orientation for arrays of the **Symmetry** and **Circular** types has been added, as well as the ability to use an edge, face and workplane to set the direction of a linear array has been added.

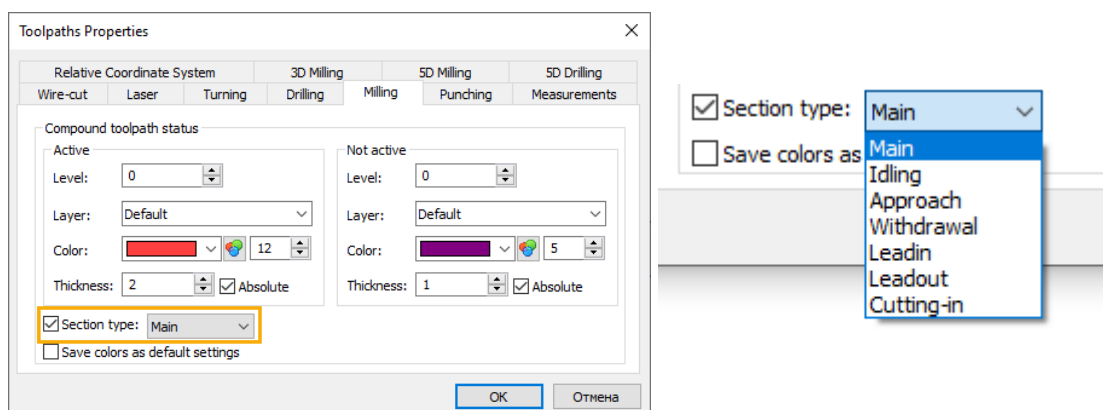


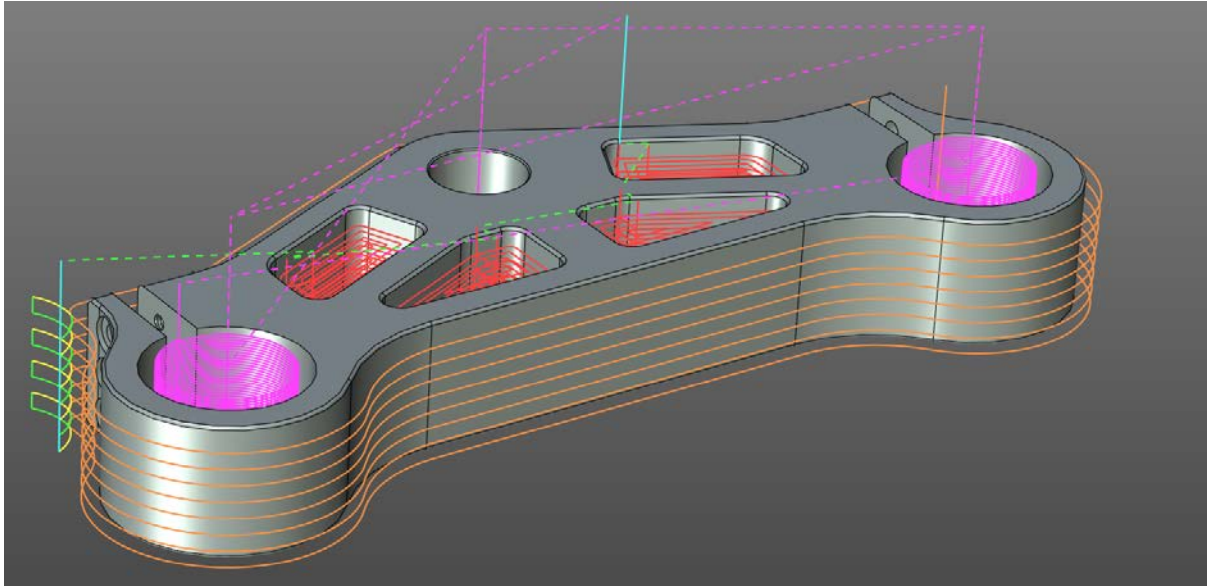
Start/Return Point has been added to override the start of the first and return of the last toolpath of the array.



Toolpath Color Editing

Added the ability to change the colors of all toolpath sections. The **Section type** option has been added to the **Toolpath Properties** menu. Upon its activation, a drop-down list appears, in which you can select colors for the toolpath sections.



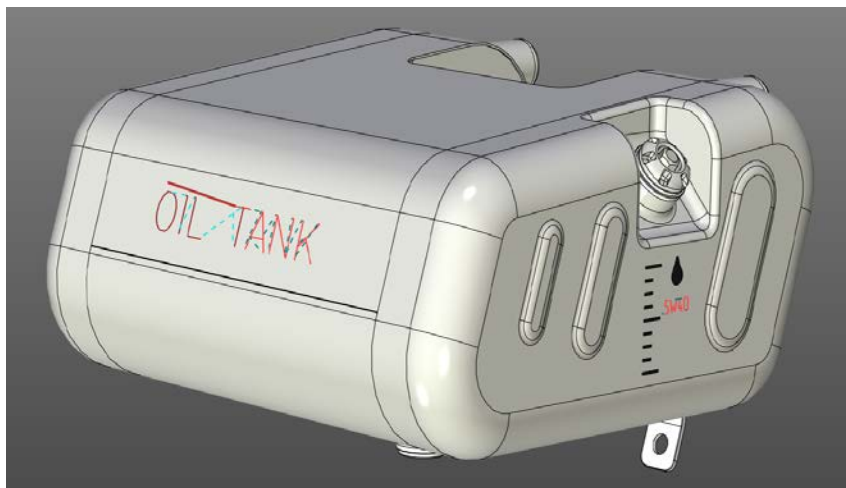


Engraving

Added the ability to use engraving text in a 3D scene for 3D and 3+2 machining. The key feature is that you can create the toolpath from fonts that have no thickness.



The engraving toolpath is drawn perpendicular to the face on which the text was created. NC for toolpath engraving can be formed in 2 axes, the text in the plane must be perpendicular to the Z axis. Also, the NC can be formed in 3 + 2 axes - positional machining.

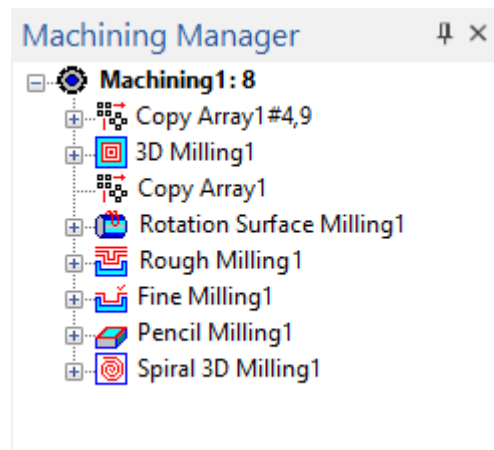


User Interface

Machining Manager

Updating icons and changing the default toolpath generation principle

In T-FLEX CAM 17 in the machining manager, the toolpath icons correspond to the selected type of machining. By default, the name of the toolpath also corresponds to the name of the selected machining type.



Multiple selection of 3D/5D toolpaths when specifying parameters for moving on/off, start/return point and cutting-in.

This feature allows you to set the parameters of the specified functions in one step - for example, cutting-in - for several toolpaths at once, if these parameters are the same.

To do this, using the "Shift" button, select a group of consecutive toolpaths in the machining manager tree and press the right mouse button.

In the context menu that opens, click on the desired function - for example, "cutting-in" - in the dialog that opens, set the required parameters and complete the input.

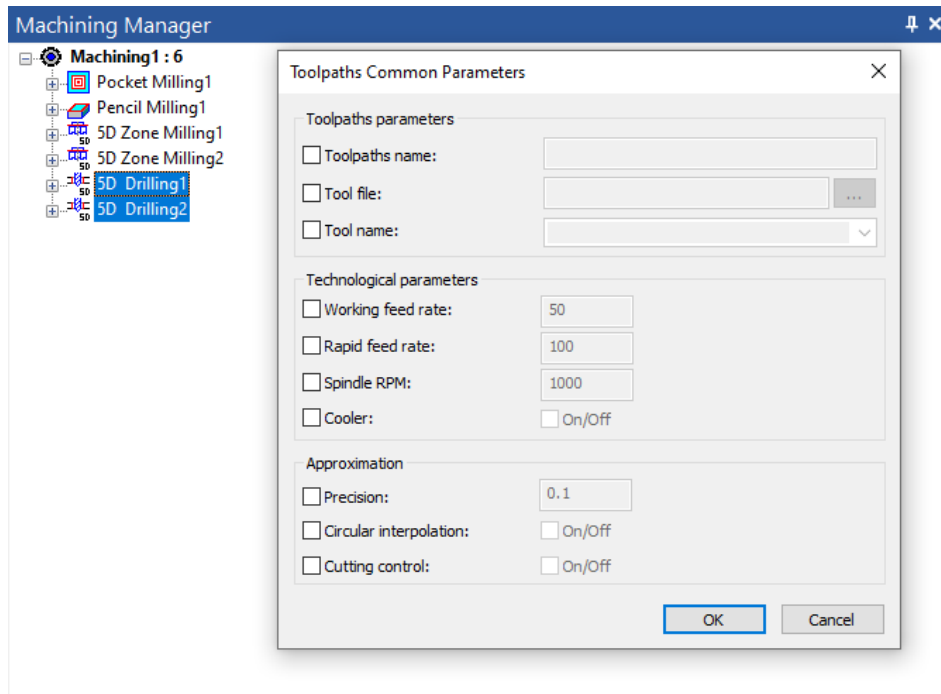
Multiple assignment is possible only for active toolpaths of the same type - for example, rough boundary 3D milling. Otherwise, an error message is displayed. Inactive toolpaths are ignored.

The approach to the creation of machinings has been reworked, namely the ability to form installations with different positions, configurations and a set of tools has been added.

Multiple selection of 3D/5D toolpaths when setting / changing their parameters.

To do this, select the required toolpaths in the machining manager tree, press the right button and press **Edit** in the context menu that opens.

This will open a dialog (see the figure below), which allows you to set some common parameters of the toolpaths.



To set other parameters that are not in the dialog, you must, without setting any flags, just close the dialog by clicking **OK** or **Cancel**, in the opened dialog of the first selected toolpath set the necessary parameters and click the **Finish Input** button.

All changed parameters will be automatically entered into the remaining selected toolpaths and recalculated along with the first toolpath. Inactive toolpaths and toolpaths of a different type than first are ignored.

Unchanged parameters remain unchanged in the other toolpaths, even if they differ from their values in the first of them.

Activation of machining by double-clicking LMB

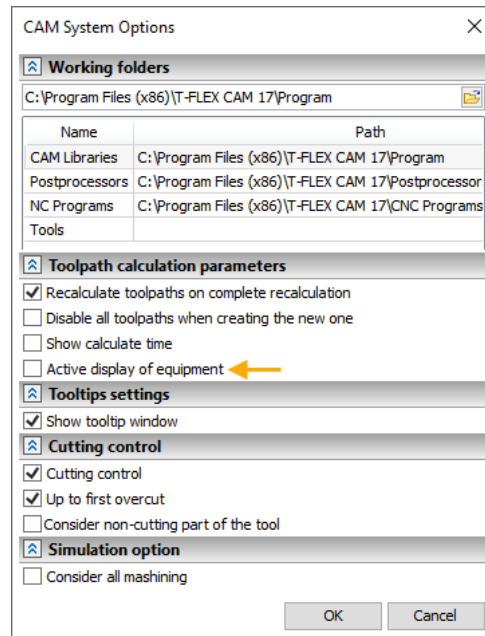
Now, when you double-click on an inactive processing, it is activated. In turn, when you double-click on the active machining, as before, a full recalculation is started.

Selecting multiple paths while holding down the <Ctrl> key

Now multiple selection of toolpaths using the <ctrl> key works for the same functions as the <shift> key, i.e. for color marking of toolpaths in the scene and context menu functions - visibility, delete, copy/paste and drag and drop, recalculation, moving on/off, start/return point, cutting in, change, statistics. For an array, it is necessary that the toolpaths go in a row. In the latter case, to set the toolpaths of the array by <ctrl>, it is enough to set the first and last. Intermediate toolpaths are optional.

Displaying equipment bodies in the scene only for active machining

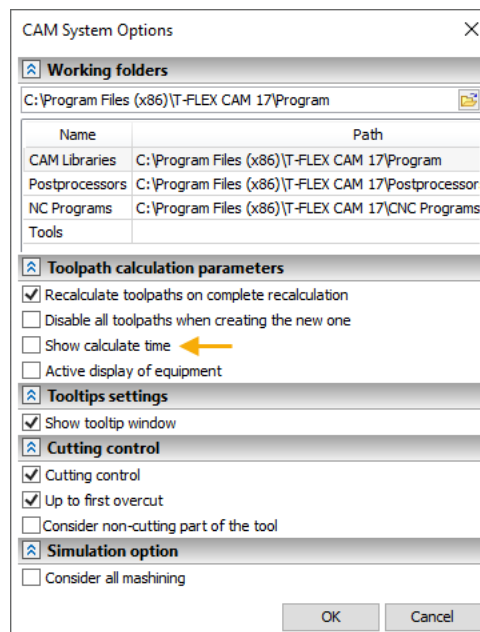
Added the ability to display equipment bodies in the scene only for active machining. For this, in the CAM Options, the **Active display of equipment** flag has been added.



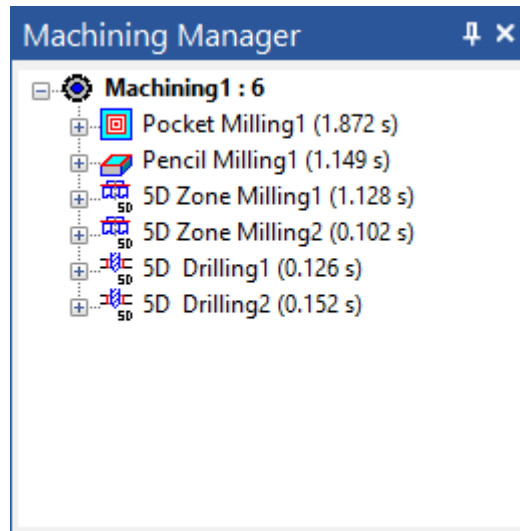
If the flag is cleared, then the scene displays the equipment bodies of all machining, and if set, only the active machining. To activate the option after changing the state of the flag, double-click on any of the machining in the machining manager tree or on the **Update Model** button. In this case, if machining is not active, it is activated, and if it is active, its full recalculation is started. If the flag is set, then the visibility of inactive machining bodies cannot be controlled from the model tree, but the visibility of active machining bodies can be controlled.

Show calculate time Option

Added the ability to display the calculation time of toolpath. The flag for activating this option is located in the **CAM System Options**.



If the flag is active, then when recalculating the entire machining or individual toolpaths, the calculation time in seconds is added to the name of the active toolpaths in brackets.

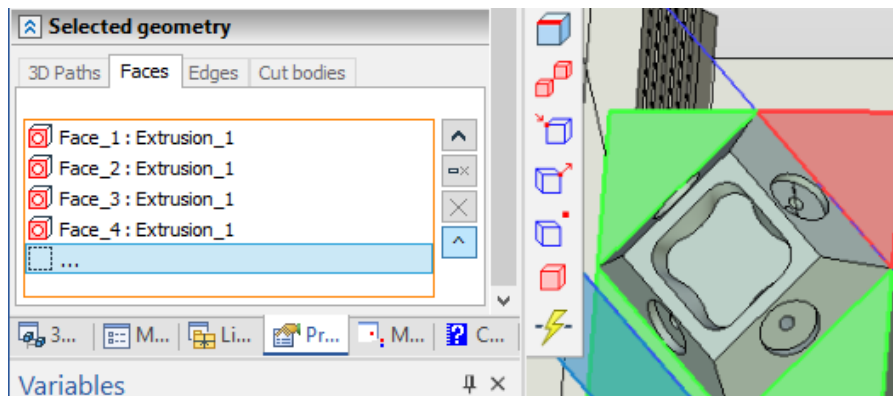



The calculation time is not displayed in inactive toolpaths. To remove the calculation time, you need to clear the flag and recalculate any trajectory by RMB.


“Selected Geometry” Tab

Selected Geometry tab has been added to dialogs with multiple selection of objects.

The tab has been added to all dialogs of 3D and 5D zone machining, 5D drilling and toolpaths of the **Machining Parameters**, i.e. wherever there is a multiple selection of objects. For example, for pencil milling, the tab looks like this:

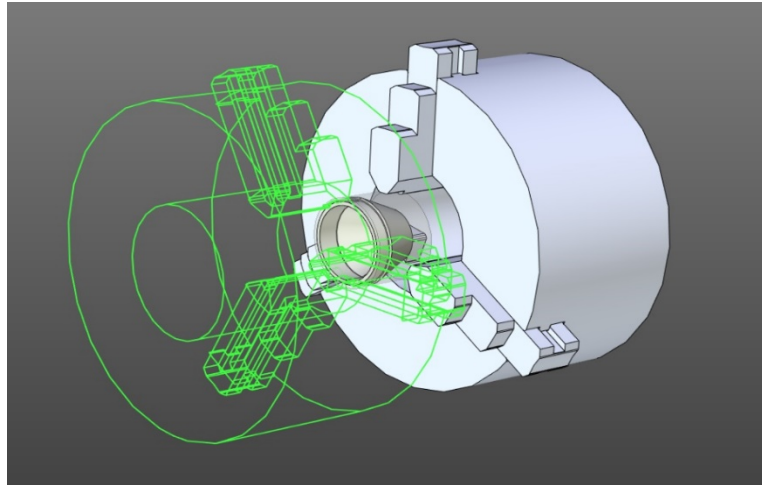
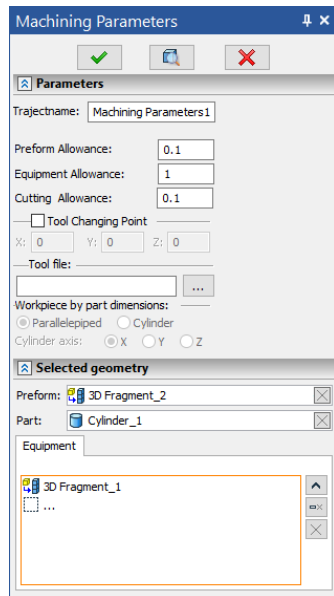


The tab allows more flexible control over the specified geometry. For example, earlier, in order to redo any of the selected faces, it was necessary to delete everything (automenu button ) and redo everything again. Now you can simply delete the unneeded object and, if necessary, replace it with a new one. The selected list items are displayed in a different color in the scene, which allows them to be separated from the rest.

In addition, it became possible to use the button  to change the position of elements in the list by moving the selected element up in the circle. Thus, it is possible to change the machining sequence of individual elements when calculating the toolpath. This button is only available for faces and toolpaths where changing the position of objects makes sense. Objects that do not have multiple selection, i.e. 3D profile, tool, etc., are displayed when selected.

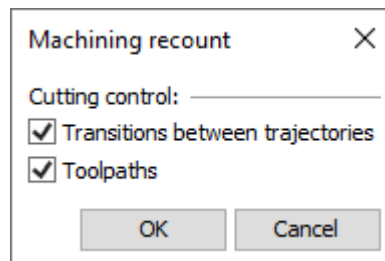
Formation of various installations

The approach to the creation of millings has been reworked, namely the ability to form installations with different positions, configurations and a set of tools has been added.



Machining recount window

Now when you click on the "Rebuild" button in the processing context menu, the modal dialog "Machining recount" (see the figure below) is displayed, which allows you to set the options for cuttings control during the recounting process.



If both flags are cleared, the recounting is performed without cutting control, which can significantly reduce the total recounting time for complex machining. By default, when the dialog is first opened, both flags are active. When you exit by **OK**, the new values of the flags are saved within the current session or until the next change. When you exit by **Cancel**, the old values are retained. After that, recounting starts taking into account the set flags. You can also start a full recounting of active machining trajectories by double-clicking on it. But the window is not displayed and the previously set flag states are in effect. To make the machining active, now it is enough to simply double-click on it.

Rough Milling

Reducing toolpath calculation time

The time for calculating the toolpath with spiral passes has been reduced several times.

Recycling of types of spiral passages

Spiral by Contour has been renamed **Spiral Outward**. **Spiral against Contour** reworked and renamed **Spiral Inward**.

New options

New options have been added: **Use part dimensions testing**, **Fix direction**, **Allow out of workpiece**, **Cut by layers**.

Additional Parameters

☐ Boundary Allowance: 0

☐ Fillet Radius: 0

☐ Fix direction

☐ Use part dimensions testing

☐ Cut by layers

Type of

☒ Rough ☐ Semifinished

Previous machining parameters

Diameter: 5

Number of layers: 20

☐ Use the cutter fillet radius

☐ Undercuts exception

☐ Add layers of flat faces

☐ Allow out of workpiece

Fix direction

Now, within the limits of the layer, it is possible to change the direction of bypassing the contours, if the part has a geometry of the "pocket with a boss" type, since in order to comply with the established mode, passing/countering near the shaping contours, it is necessary to change the direction when moving from the walls of the boss (i.e., the protrusion) to the walls of the pocket. This option allows you to rigidly fix one direction of bypassing the contours, determined by the switch passing/countering.

Use part dimension testing

Previously, in the **Incremental Lift Tool** mode when determining the transition height on rapid traverse, collisions with the part were not checked, which led to the need for a thorough subsequent check in the simulator to avoid cuttings. This option takes into account the possibility of collision with the part when calculating the Z-level of the transition. If a collision is detected, then the Z offset is not made from the last point of the toolpath, but from the upper dimension of the part.

Cut by layers

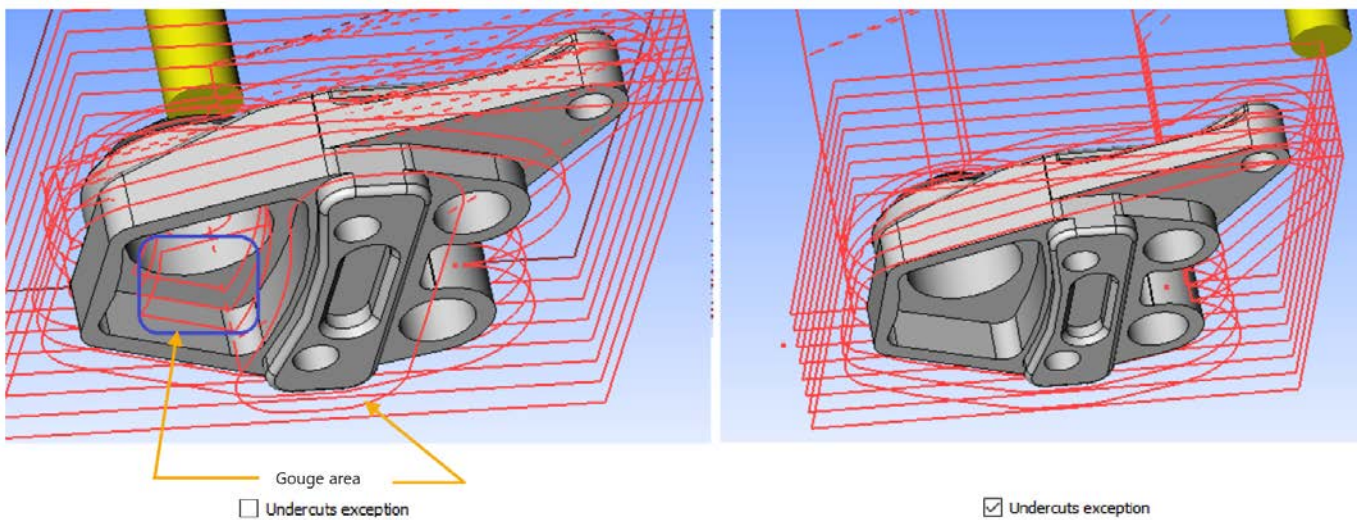
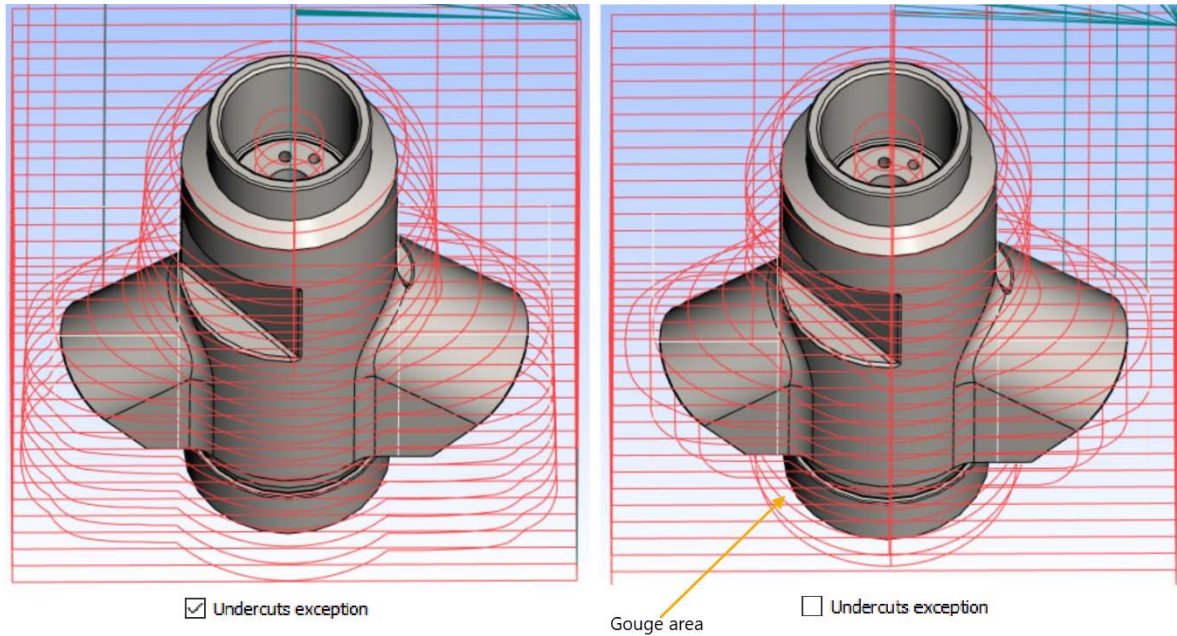
Previously, machining for some types of passes was performed not in layers, but in local zones ("wells"), where possible, in order to reduce the number of transitions at idle speed. In some cases, this can lead to an incorrect toolpath, in which the tool cuts immediately along the bottom layer bypassing the top ones, which leads to tool breakage. This option avoids the above.

Undercuts exception

Previously, roughing was calculated without taking into account undercuts (or internal voids) of the part. Direct calculation of such a part led to cuttings, since in the process of creating sections, all contours were taken into account, including parasitic ones. This required creating of special technological models before calculating, in which all undercuts were removed, or the selection of a special limiting body, which is not always convenient.

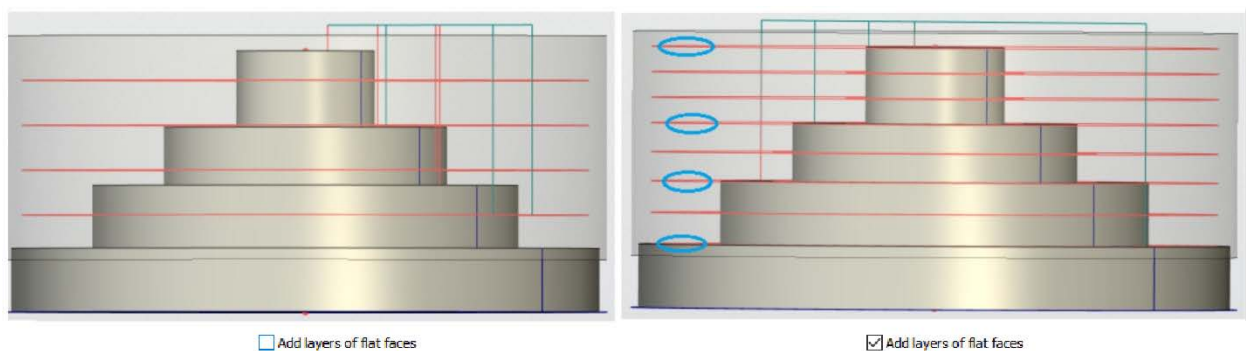
Now, to take into account undercuts, it is enough to set the corresponding flag in the **Additional Parameters** tab. In this case, each subsequent section is created taking into account the machined parts of all upper sections, which excludes the cuttings associated with undercuts.

This functionality allows in many cases to select for the calculation only the body of the part itself. In this case, you may also need to set the positive **boundary allowance** slightly larger than the tool radius. In this case, the creation of labor-intensive technological models and limiting bodies becomes unnecessary.



Add layers of flat faces

When this option is activated, the system recognizes flat faces perpendicular to the tool axis and forms additional pass layers on them that are not multiples of the specified pass step.



Allow out of workpiece

Previously, passes in each section were created taking into account the equidistant offset on the tool radius from all shaping contours, including the workpiece contour. This made it difficult to obtain the desired toolpath for the case when the part is open from the side of the workpiece contour and an indent from it

is not needed. This option excludes the indentation from the workpiece contours when forming subsequent contours.

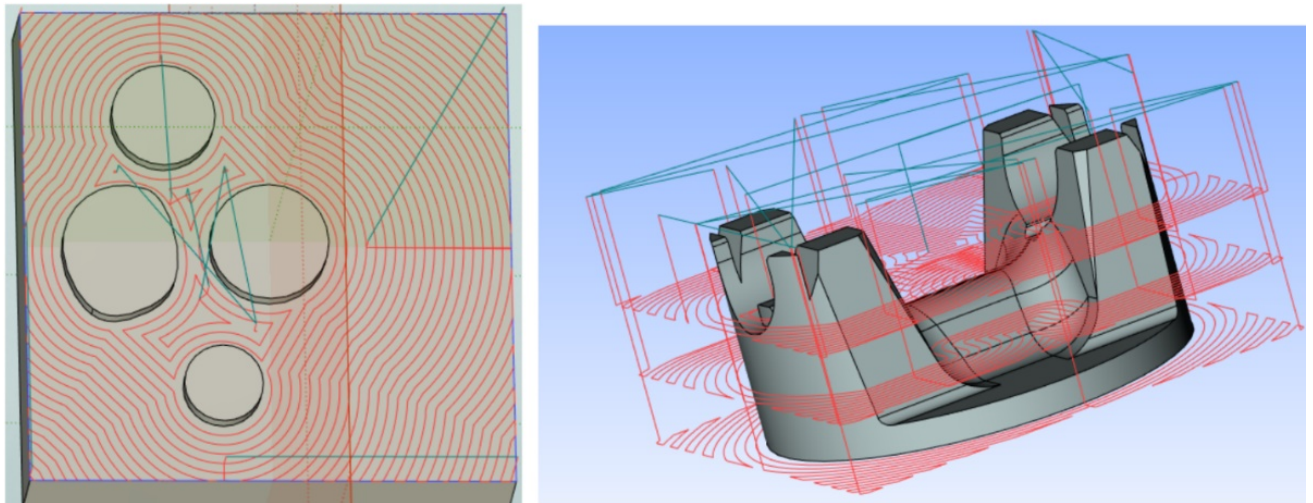
Expansion of functionality for certain types of passages

Cutter fillet radius for passes of the "Spiral" and "Equidistant Zigzag" type

This will ensure smoother tool movement and exclusion of cuttings associated with sudden changes in the direction of the cutter, especially at high feed rates. Previously, this option was only active for pocket milling.

Several contours in the section for passes of the "Equidistant Zigzag" type

Previously, only one internal shaping contour per section was allowed. Otherwise, the toolpath was not create. Now it has become possible.



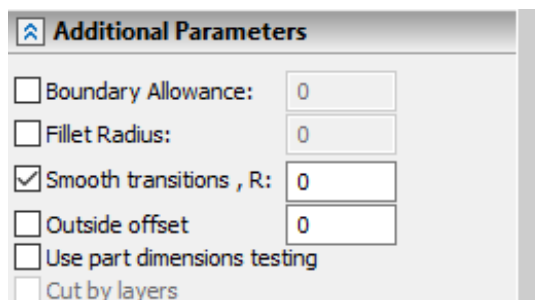
For this type of passes, the "Outside offset" option has also been added, which allows you to extend the toolpaths outside the workpiece.

"Smooth transitions" option

The **Smooth transitions** option has been added.

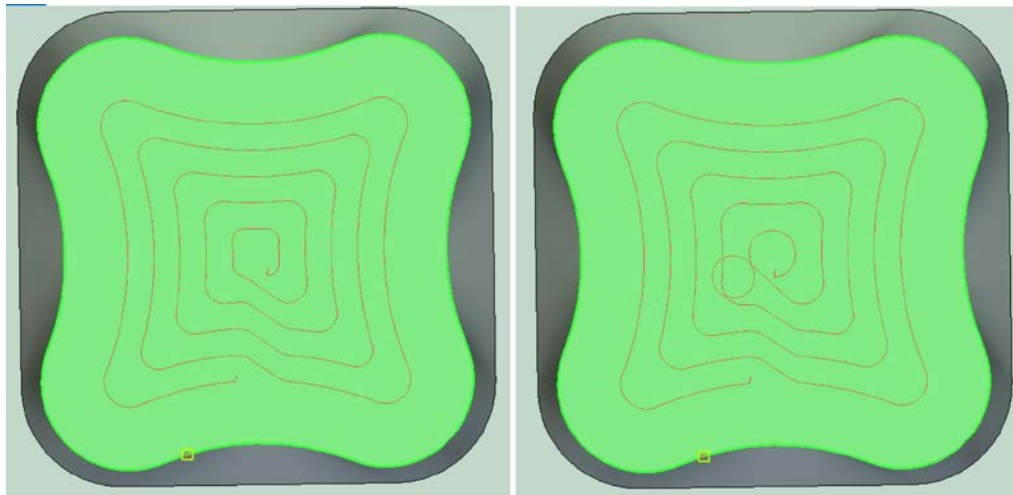
This option is necessary to reduce the inertial load on the machine's actuators. This allows you to increase the machining modes and surface roughness, and hence reduce the manufacturing time.

The option, like the **Fillet Radius**, is effective only for spiral types of passes and is activated when the corresponding flag with the field for specifying the radius of the transition arcs R in the **Additional parameters** tab is enabled.



In this case, a smooth transition is inserted between adjacent passes, consisting of two arcs and a mating segment tangent to them with a radius of arcs equal to $r = \min(R, \text{step})$, where step is the step of passes (see figure). In some cases, smooth mating is possible only with the formation of loops (Fig. b), which can be eliminated by decreasing r (Fig. a) or changing the position of the starting point of the toolpath. For

Outward Spiral/Inward Spiral, a loop, as a rule, forms at the very beginning/end of the spiral during the transition from the first circuit to the next or from the penultimate circuit to the last and usually does not pose a danger, since it is located at a sufficient distance from the walls of the pocket.

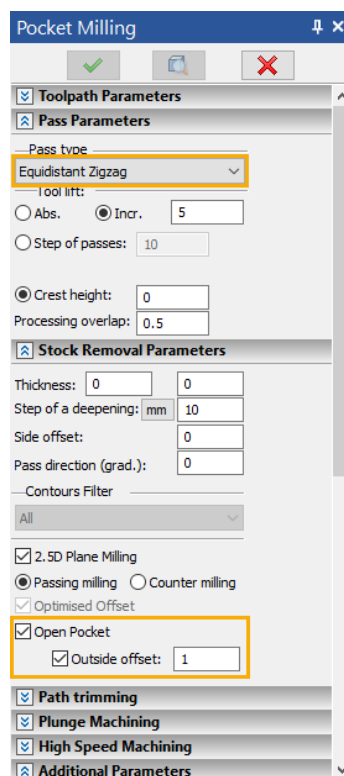
a) $r=1.5$ b) $r=2$

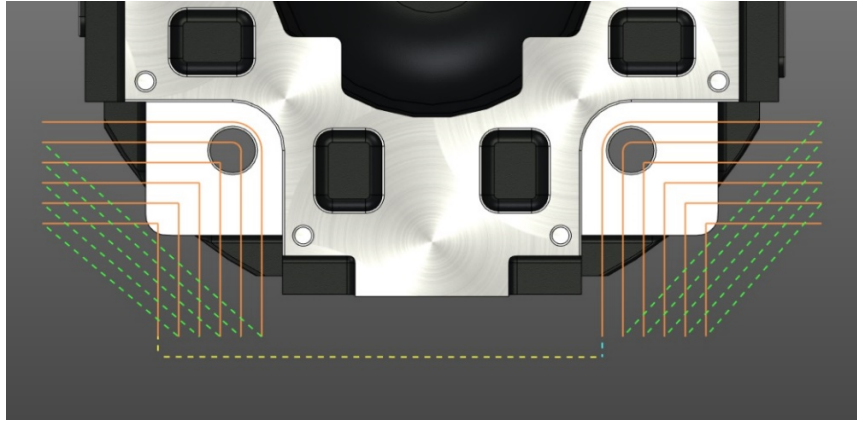
Note: This option has also been added to pocket milling.

Pocket Milling

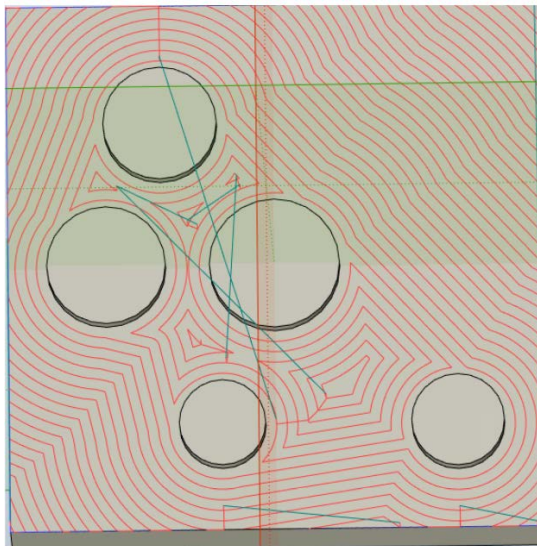
Open Pocket

In the boundary 3D milling type **Pocket Milling**, the **Open Pocket** option has been modified. In particular, for the **Equidistant Zigzag** pass, an **Outside Offset** was added, and for a pocket with a closed outer contour, the case of parts with several inner protrusions or shaping contours was taken into account. Previously, only one was possible.



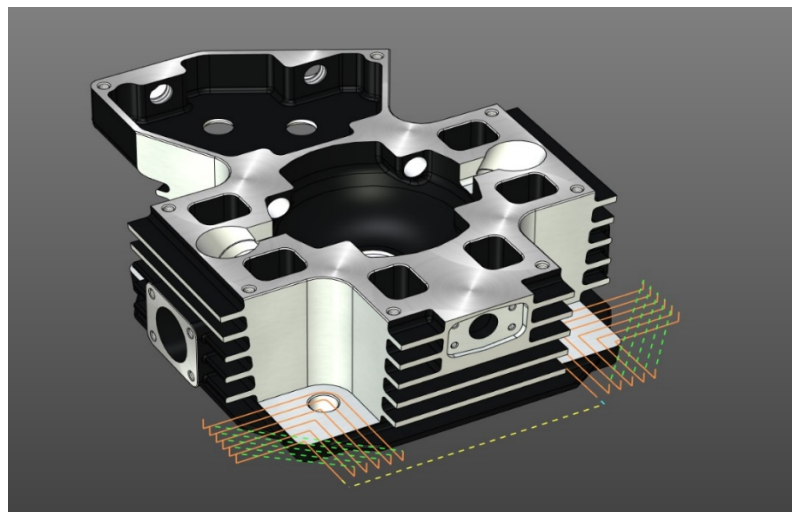


This option allows you to process open pockets using different strategies. Pockets can have a variety of configurations of two main types: with one or two open sides without internal protrusions (see figure above) and fully open, bounded by a closed loop with one or more internal protrusions.



The type selection is determined by the way the geometry is set. In the first case, a pocket face or two open contours are specified that define the boundaries of the open and closed parts. In the second, the outer closed contour and the edges of the protrusions.

Using this option will reduce the machining time due to rapid translations between passes, and will also allow you to form optimal machining paths using the entire cutting part of the tool.



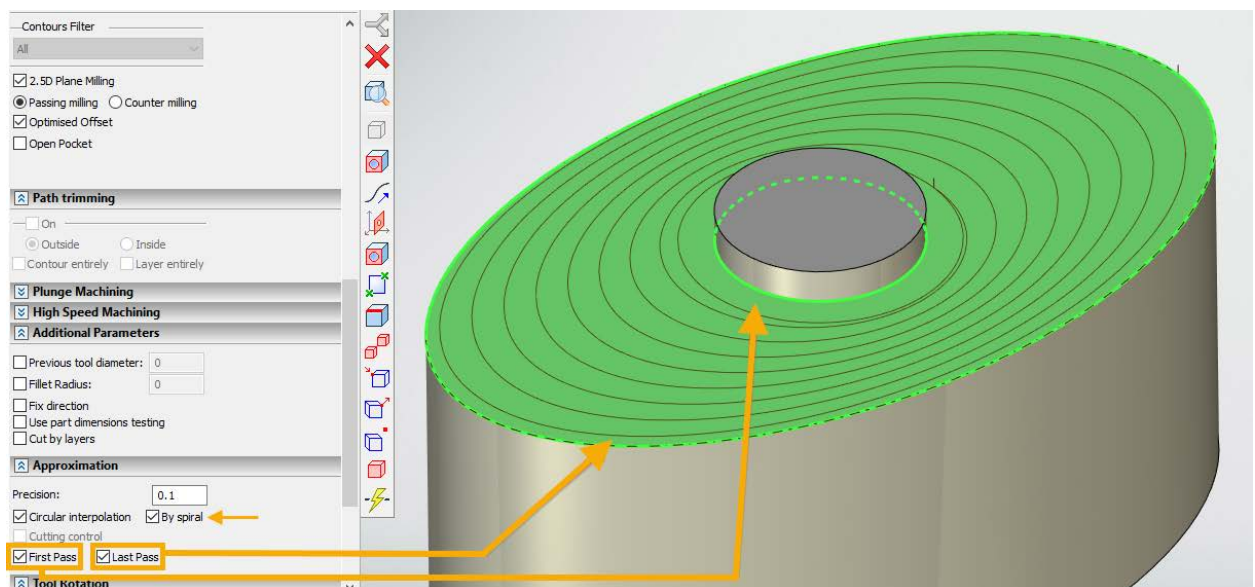
Setting with one face for a fully "open pocket" with internal ridges

For a completely "open pocket" with internal protrusions, added the ability to define using one face. Previously, to define this case, it was necessary to specify the faces of all protrusions and the outer closed contour. In addition, in the **2.5D plane** mode, it was necessary to set a negative allowance equal to the height of the protrusions. Now it is enough to select just one multi-contour edge of the pocket.

The contour filter must be set to the **All** position. The contour filter is now always active - it was previously inactive in active pocket mode. This is done to avoid conflict with the case of a partially open pocket (without protrusions) with a single edge selection. In the latter case, the selected multi-contour face can have indentations that can be ignored if you set the contour filter to **Outer** only.

"By spiral" Option

An option that allows you to machine pocket faces by spiral has been added. Spiral toolpaths, spirals with the first pass along the contour, spirals with an outer pass along the contour are formed.

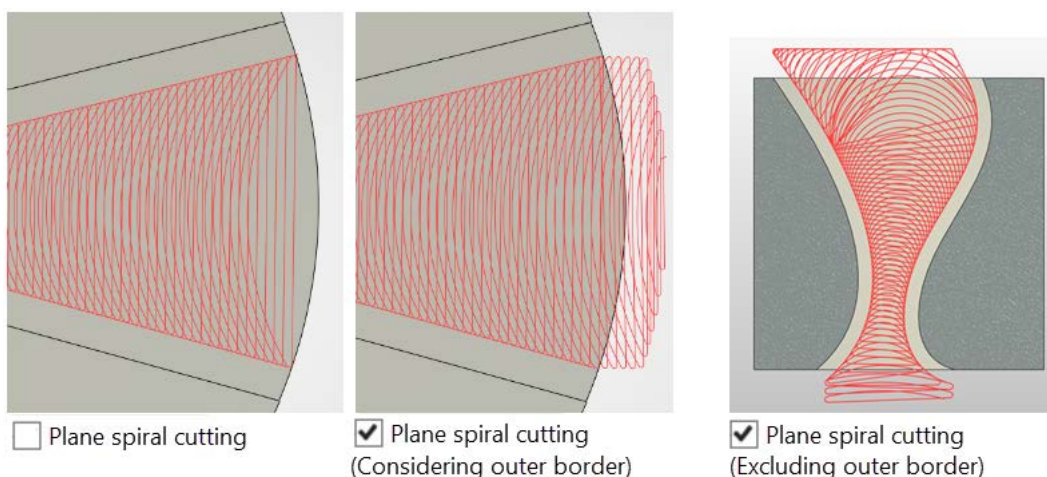


Pencil Milling

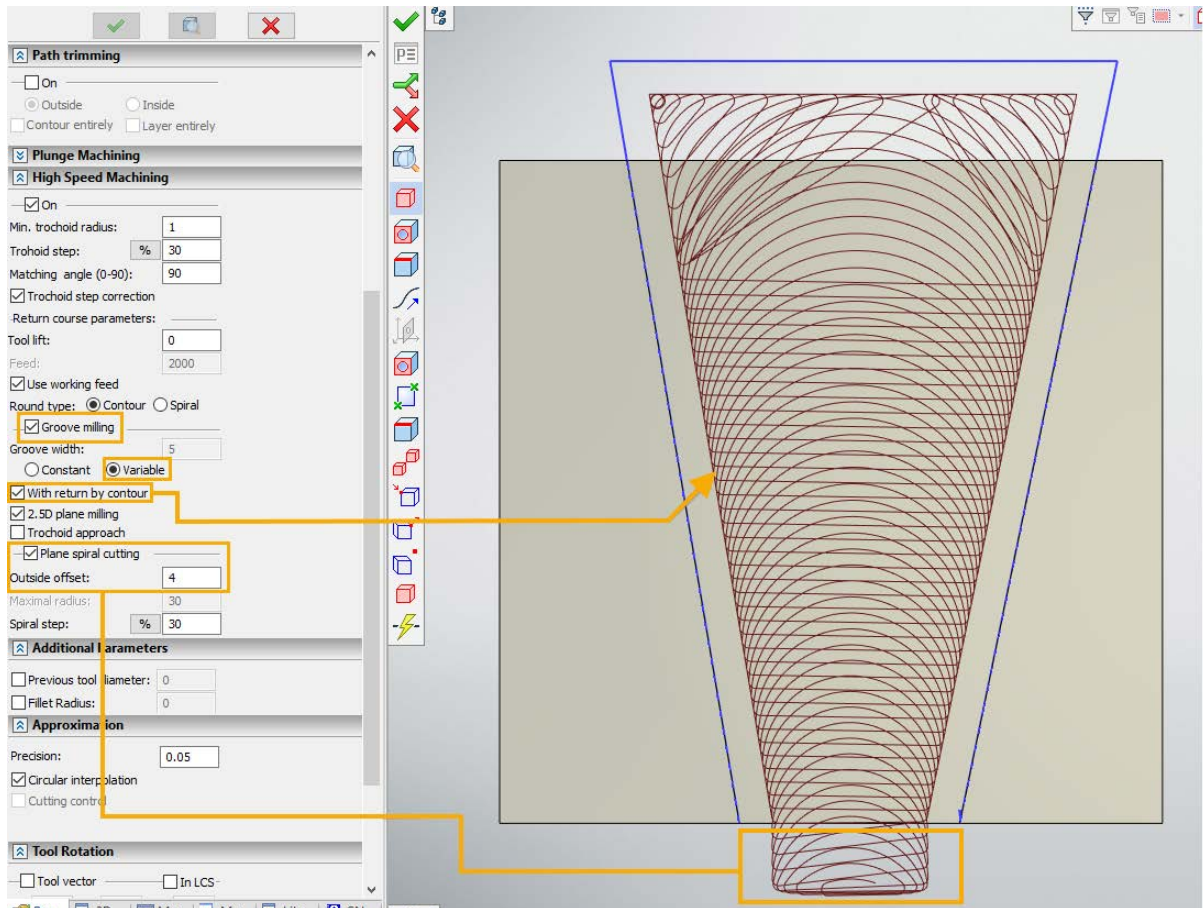
High Speed Machining

Options to control slot milling using high speed machining in pencil milling have been added. Now it is possible to set the cutting-in with a flat spiral, control its parameters and generate a return along the contour, trimming the residues after tool passes.

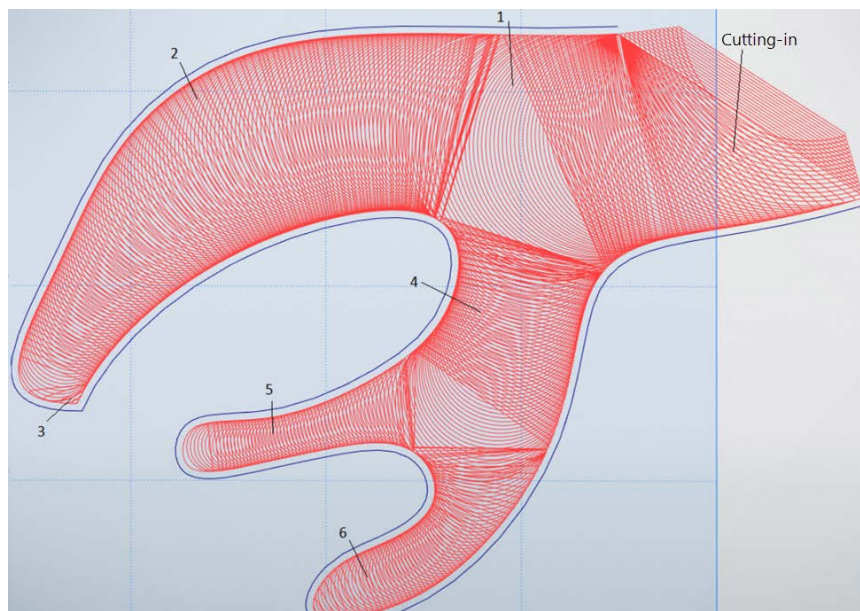
In the "cutting with a plane spiral" mode, the ability to set the outer boundary has been added.



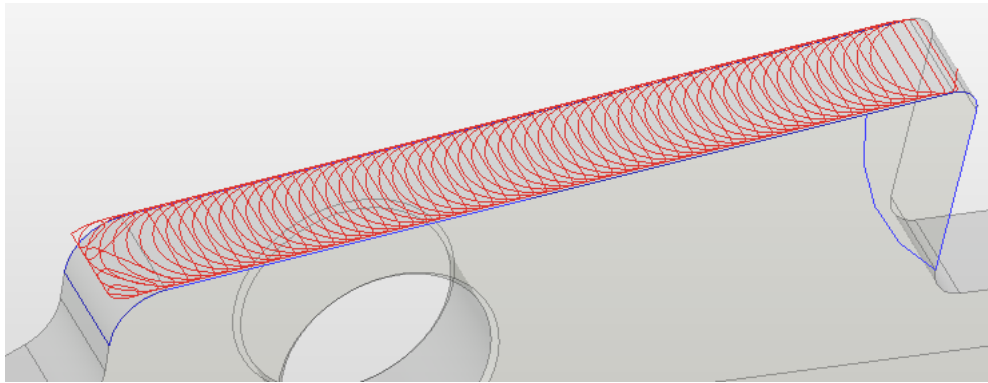
To do this, after defining the main directing open contour in any way (face, set of edges, 3D path, profile), define a closed contour of the outer boundary (there can be several such pairs of the main contour - outer boundary). The calculation of the turns of a flat spiral is made in such a way that at the starting point the cutter only touches this outer boundary (this is achieved by equidistant displacement outward by the value $R_{instr} + dR$, which ensures a smooth exit to the first turn of the main trochoid, where dR is the "Outside offset", which allows additionally control the position of the start point of the lead-in spiral. If you do not specify the outer boundary, then it is formed automatically in the form of a linear segment, the start and end points of which coincide with the start and end points of the specifying 3D path.



Provides the ability to work with a complex guide path that has multiple branches, as shown in the figure.

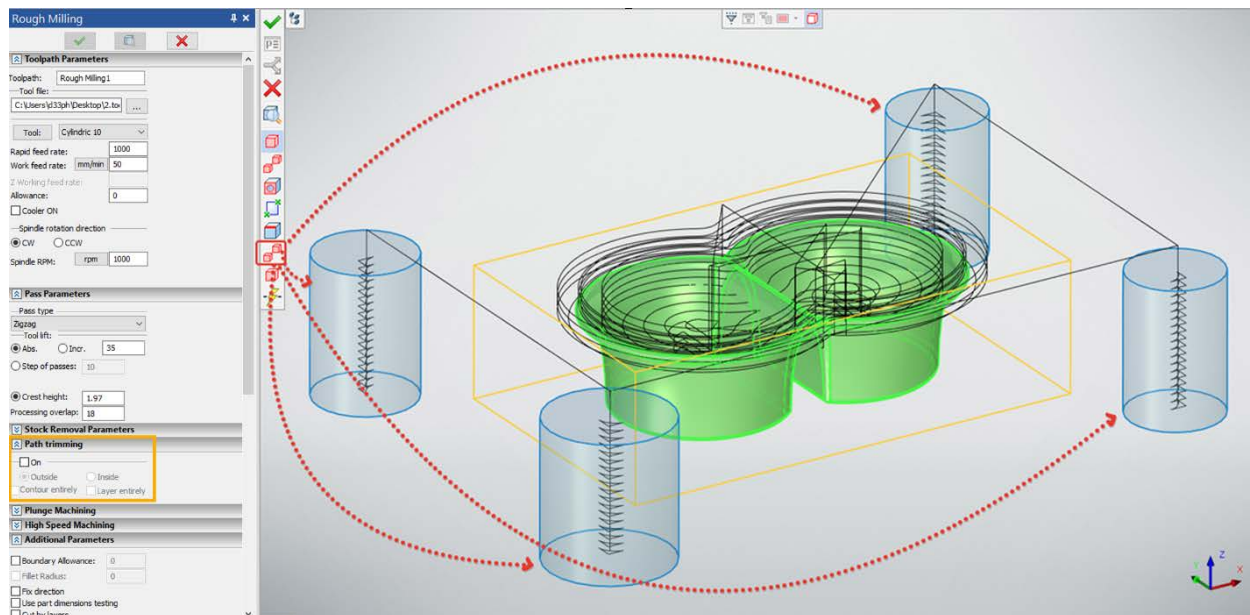


Resetting the **2.5D plane** flag allows you to project with collision control (by analogy with Pocket Milling) the high speed milling toolpath onto an inclined plane or curved surface.



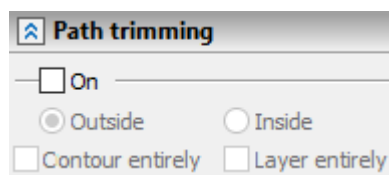
Path Trimming for Boundary 3D Milling

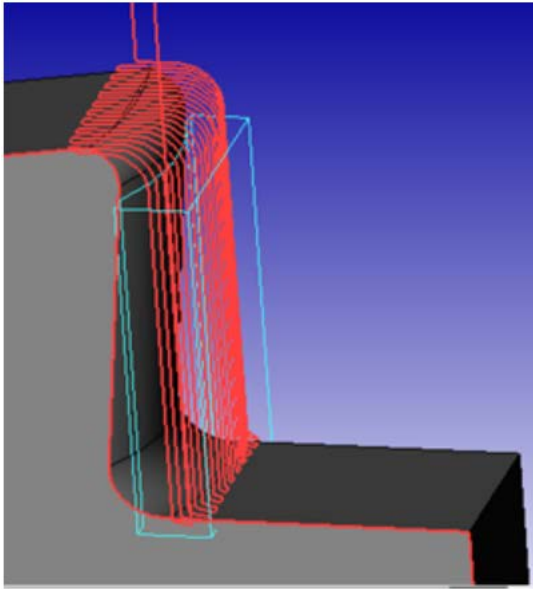
The ability to trim paths with bodies with several options has been added. To trim paths, select the bodies that will be used to trim the parts, and also enable the **Path trimming** option.



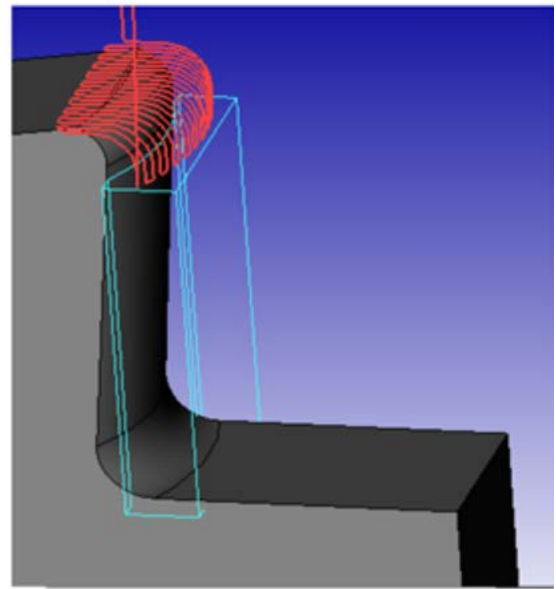
Optionally, you can control the trimming parameters. The inside/outside options trim the paths inside or outside the selected trim body.

The path and layer options allow you to remove both the path inside the trim body and the entire layer. The set of options may differ for different types of machining.

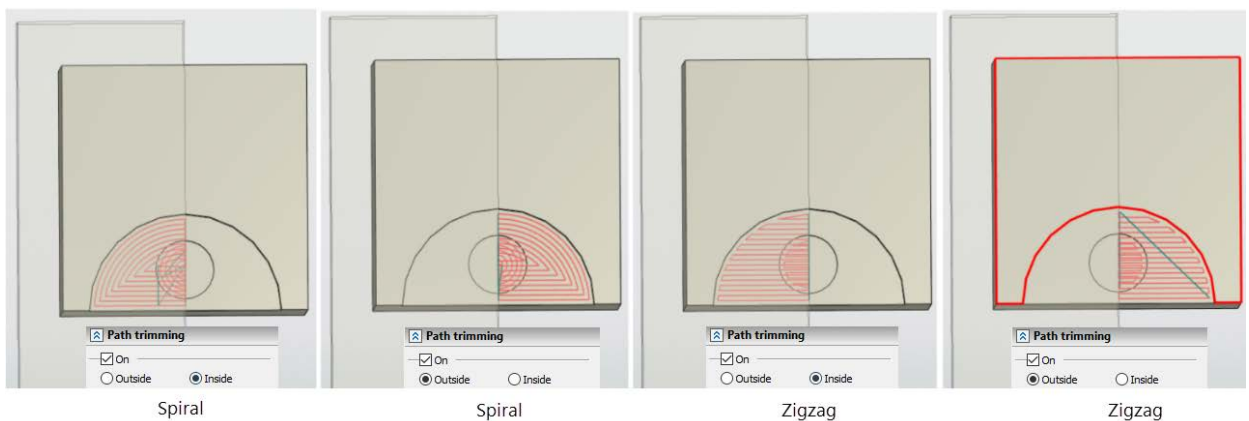
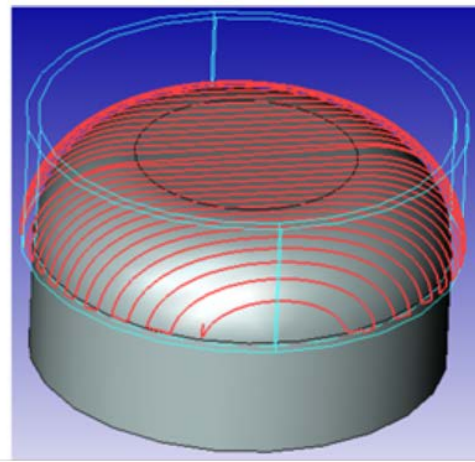
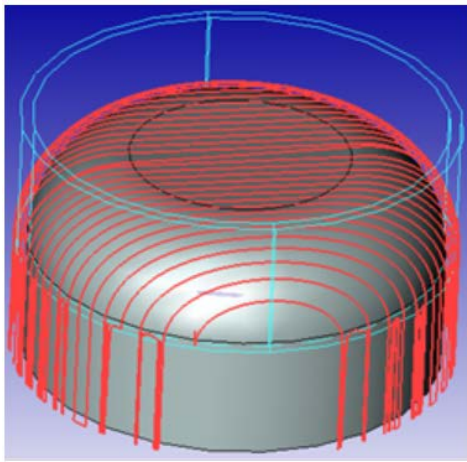




Without trimming



With trimming



Part body as a parametric 3D fragment

Added the ability to connect to a 3D/5D zone toolpath of a part body in the form of a parametric 3D fragment.

Basic variant description

Any number of toolpaths can be connected to one 3D fragment, and each of them can store its own set of external parameters of the fragment that control its configuration. A fragment is connected to the toolpath automatically when it is selected as a part body using the **Select Body** button in the automenu of the

toolpath dialog. When you click on the body of the fragment, a dialog of the external parameters of the fragment appears, where you can set the required values, after which, when you click OK, the body of the fragment is automatically recalculated with this set of parameters. Each such set corresponds to one layer or processing pass made in the form of one toolpath. It is also possible to combine several such layers (clones) at once in one toolpath (see below). The rest of the toolpath creation process remains the same. If a toolpath with a previously connected fragment is opened for editing, then when the body of the fragment is selected, its dialog will display the current values of the parameters previously saved in the toolpath, and they can be changed. This is used when creating clones of a toolpath with different fragment parameters. Each of these clones creates its own layer or processing pass.

When recalculating the toolpath from the machining manager, before the start of the calculation, the external parameters of the fragment are automatically reset to the values of this particular toolpath and recalculated. At the same time, all the elements selected on it are recalculated: faces, 3D paths, 3D nodes, etc. As a result, the toolpath is calculated with its own clone of the fragment body, forming some part of the general processing.

The approximate procedure for using this feature is as follows:

- 1) An additional technological model is created in the form of a Boolean addition of the part body and an auxiliary parametric body (for example, a secant cylinder with some variable D as the diameter).
- 2) The technological model is saved as a 3D fragment in a separate file.
- 3) To create toolpaths, a new 3D model is created and a file of the previously created 3D fragment is loaded into it as a part body.
- 4) The necessary defining geometric elements (for example, 3D paths) are created, attached to the body of the fragment and therefore recalculated along with it.
- 5) The first (base) toolpath is created in the usual way. A previously loaded 3D fragment is selected as a part body and the required combination of fragment variables (D) is set, corresponding to a certain processing layer. Defining geometric elements (faces, 3D paths, etc.) are selected that are attached to the body of the fragment, etc.
- 6) A set of clones of this toolpath is created with different parameter values (D) and forming subsequent layers. In the process of their creation, it is possible to correct their parameters in order to provide the required toolpath shape (to exclude unnecessary rises, gouges, etc.).
- 7) The next basic toolpath is created (it differs from the previous one by a different set of geometric definition elements), its clones, and so on.

Combining several of its clones in one toolpath

In the initial version of this proposal, each processing layer obtained as a result of changing the body of a parametric 3D fragment was associated with one toolpath, which, with a large number of layers, is very inconvenient.

One of the significant improvements was to combine all clones of the base toolpath into one, instead of producing many toolpaths of separate layers. This is especially important for finishing operations where the layer spacing can be quite small.

By "clone" in this case, we mean a copy of the base toolpath, which differs from it only in the value of the parameter of the 3D fragment connected to it, which controls its geometry. For example, the diameter or height of the secant cylinder, etc.

To start this mode, three system (with a fixed name) variables are entered into the external parameters of the fragment: **m_D** - control of the position of the layers, initially the minimum value Dmin corresponding to the base toolpath is written into it, **m_depth** is the total thickness Dmax - Dmin, those are the range of variation of the parameter **m_D**, and **m_step** is the layer step, more precisely, the step of changing the **m_D** parameter from Dmax to Dmin. The number of layers Nc and the current value of **m_Di** is calculated as follows:

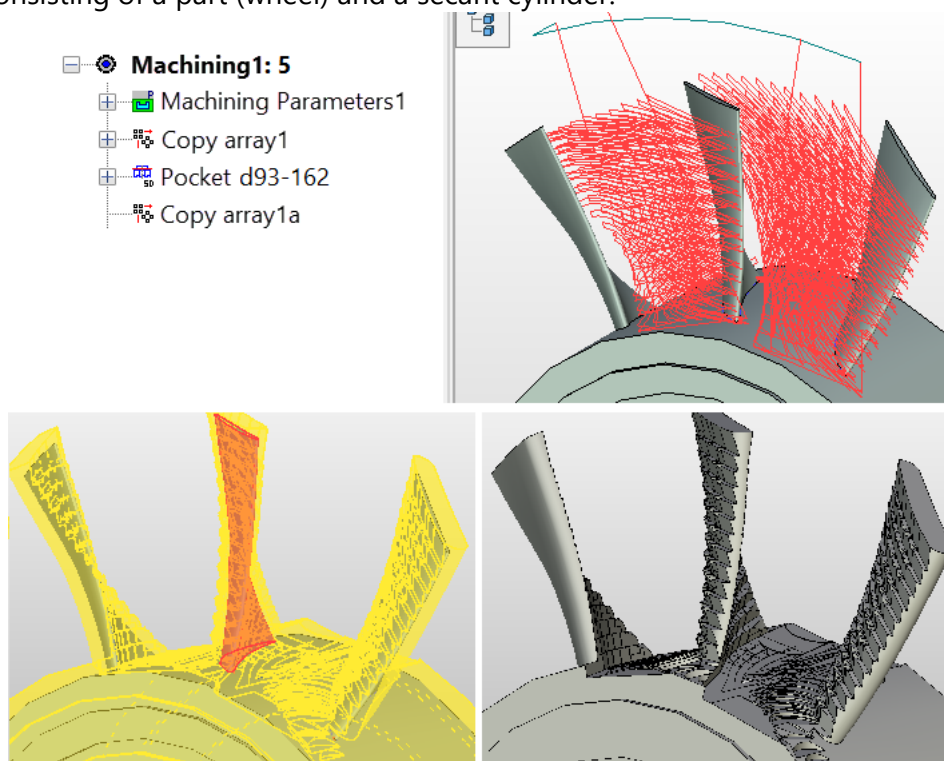
$$Nc = (\text{int}) \text{ m_depth} / \text{ m_step} + 1,$$

$$\text{m_Di} = \text{m_D} + (Nc - i - 1) * \text{m_step}, \text{ where } i \text{ ranges from } 0 \text{ to } Nc-1.$$

Recommended general procedure:

- 1) First, basic toolpaths are created (with **m_depth** = 0), covering the entire processing area, each of which can have its own set of parameters and geometry defining elements.
- 2) Then, for each base toolpath, the upper limit of the parameter **m_D** = Dmax is empirically determined, for which cloning is still possible, i.e. no correction of the toolpath and geometry parameters is required, for example, to avoid unnecessary lifts and gouges.
- 3) Change the value of the **m_depth** parameter to Dmax-Dmin. To do this, open the toolpath dialog, select **Select Body** in the automenu, click on the fragment's body, in the fragment's variables panel that opens, enter the resulting value for **m_depth** and click OK. After that, we start calculating the toolpath together with its clones by clicking the **Finish Input** button. During the calculation, you can observe how the body of the fragment changes and the toolpaths are generated when **m_D** is automatically changed from Dmax to Dmin.
- 4) For convenience, at the end of the toolpath name, you can specify the depth range it covers (changes in the **m_D** parameter), for example, in the form d = 40-85.

The figure shows an example of rough machining of a wheel using this feature, obtained using only one path of the 5D "pocket" type with a choice of 2 paths by automatically changing the body of a parametric 3D fragment, consisting of a part (wheel) and a secant cylinder.



Tool axis rotation in "Rotation Surface Milling"

"Rotation Surface Milling" is well combined with the ability to connect a 3D fragment, because it is convenient to use bodies of rotation (for example, a cylinder) as an auxiliary (secant) parametric body of a fragment.

In the basic version of this milling, the tool axis is always directed normal to the surface to be milled. The need for additional rotation may arise when milling parts with walls inclined relative to the current normal (for example, all kinds of wheels with inclined blades, etc.) to avoid cuts (if there are undercuts) and to reduce the maximum gap between the side surface of the cutter and the part wall to be processed.

To do this, in the parameters of the 3D fragment used, add the parameter **m_A** (rotation angle in degrees) and **m_4D** (type of tool axis rotation): 0-around the current displacement vector (for 5-axis machines), 1-rotation around the projection of the current displacement vector on the Z-axis (for 4x-axis machines), -1 (<0) without rotation (the tool axis coincides with the local normal to the surface of the secant cylinder).

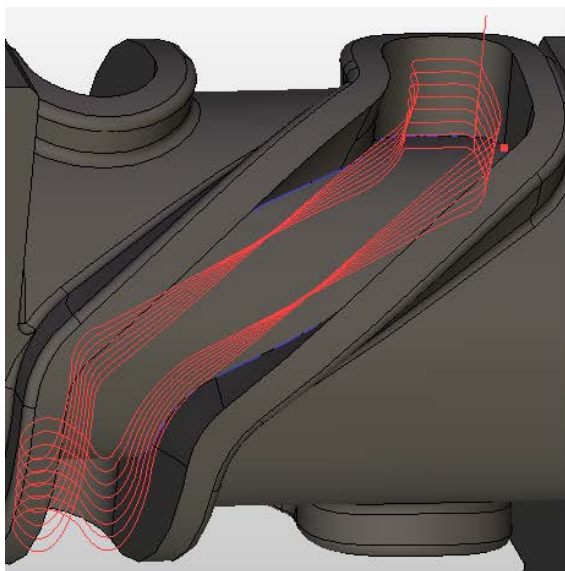
In general, the slope of the wall of a part can vary from point to point. The task of the minimum deviation from the machined curved walls makes it necessary to use the mode of automatic determination of the

optimal local vector of the tool for each point of the toolpath, which will allow in some cases to perform semi-finishing and even finishing.

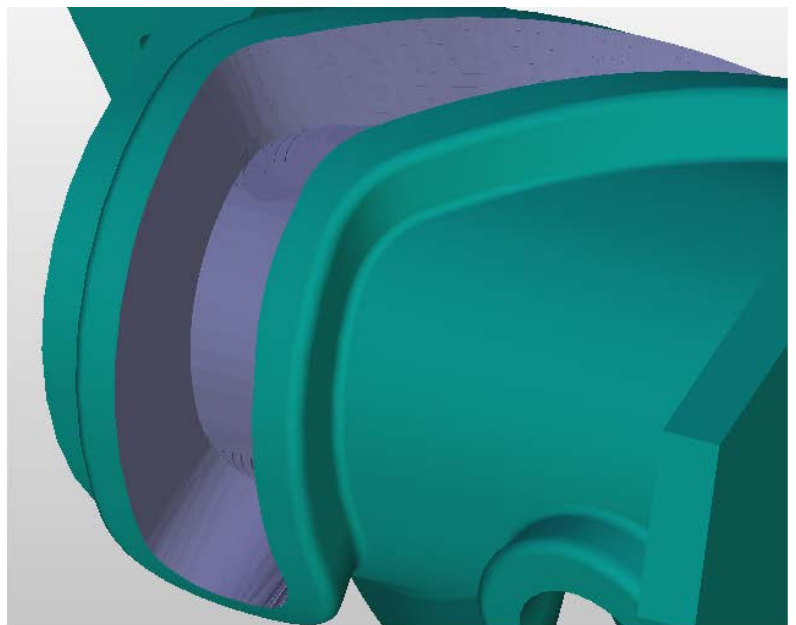
This mode is enabled if the fragment variables $m_A = 0$ and $m_4D > 0$.

When $m_A \neq 0$ and $m_4D > 0$, the tool axis is rotated by the specified angle m_A .

The figure below shows an example of a part processed using this mode, and only one toolpath with a choice of a face (bottom) with 8 clones obtained by automatically changing the variable $m_D = 30 \dots 47.5$ (diameter of the secant cylinder) of a parametric 3D fragment c with a step $m_step = 2.5$. The angle of inclination of the processed side wall for this part varies from 0 to -15 degrees (towards the bottom with overhanging over it). Only the side walls are actually processed. The bottom (the surface of the secant cylinder) is used only for creating the toolpath (it is not in the actual part itself). In order for the side surface of the cutter to machine the entire wall, the algorithm provides not only the optimal orientation of the tool axis at each point of the toolpath, but also displacement along the tool axis towards the bottom by the amount of the cutter rounding radius. For additional correction, you can also use the **Side Offset** and the **Allowance** from the toolpath parameters dialog.



Rotation Surface Milling d=30-47.5 (Face)

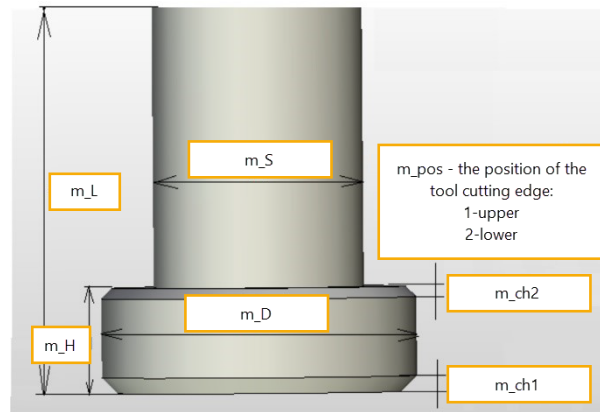


Parametric 3D Tool Fragment

Implemented a mechanism for using a parametric 3D tool fragment to work with the upper cutting edge of the tool.

1. Chamfering - implemented in 5D zone for 3D spiral.

Since the tool editor (TE) does not have a tool with an upper cutting edge and generally with chamfers, it is proposed to use a parametric 3D tool fragment (PTF) with upper and lower chamfers (see figure). The position of the cutting edge is specified by the m_pos parameter (1-bottom 2 top).



System parameters of a parametric tool fragment for chamfering

In the tool editor, select a cylindrical tool without fillets of the required diameter of the cutting (D) and non-cutting (S) part, and the height of the cutting part (H) and the cutter as a whole (L). All these parameters after connecting the PTF are automatically synchronized with the PTF and can be set both from the RI and from the PTF panel.

In the dialog of 5D zoned toolpath, the **Local stock** checkbox should be enabled, setting it to zero if no additional stock is required. Position of the switch for the direction of the local allowance **Along the tool axis / Normal affects** the starting position of the cutter, which can be used. The **normal** position is intended for a chamfer with an angle of inclination $A = 45$ degrees. For the general case, manual correction of the value of the effective chamfer height by a factor of $k = (2 / (1 + \tan^2 A))^{0.5}$ is required.

2. Top rounding - implemented for 5D zone positional spiral in 3D mode, including with the "pocket" option and 3D spirals.

Instead of the PTF with bottom and top chamfers discussed above, you can use a similar PTF with radii (fillets). The lower radius is set by the m_CR system variable (it is synchronized with the CR RI parameter), and the upper m_CR2 . The parameters m_ch1 and m_ch2 , which previously set the chamfer heights, can be excluded in this case. The rest of the parameters remained unchanged.

The possibilities considered are not implemented in all millings, but only where their use is really justified. Other millings will be added as needed.

Cutting Control

In the **Cutting control** option, it is possible to use a parametric 3D fragment as a tool.

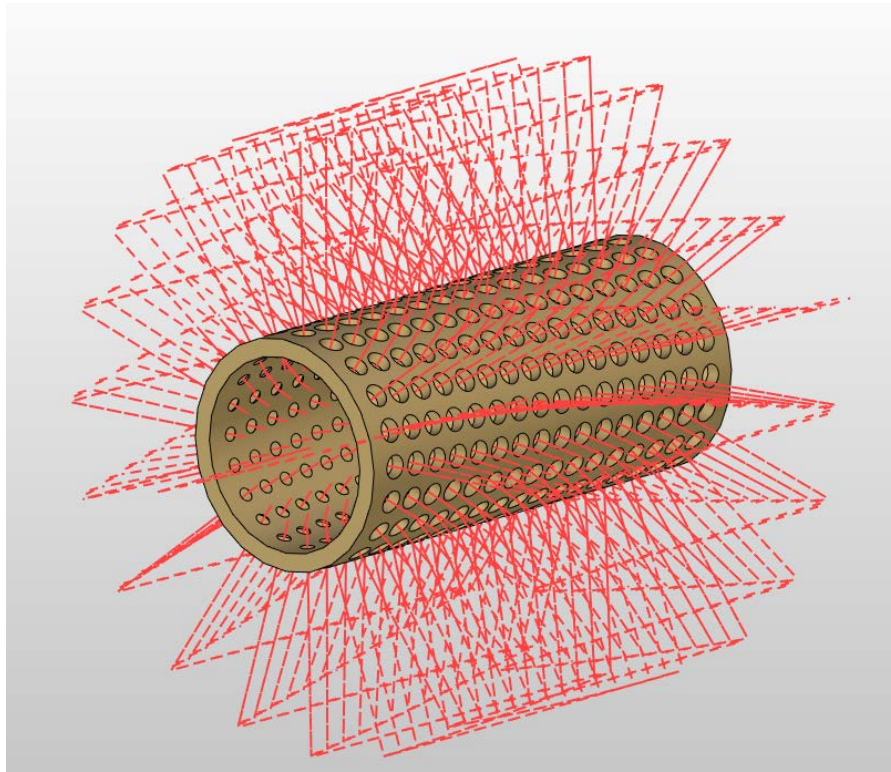
Previously, a testing tool was created only by parameters from the tools file.

Tools based on a parametric 3D fragment (PF) are used when using a shaped tool, as well as in cases where the real tool differs from the one specified or is absent in the tool editor, for example, when using the upper cutting edge, milling chamfers, etc. The **Cutting Control** option allows promptly and quickly (without even leaving the toolpath dialog) to carry out a preliminary check of this toolpath for a cut. Both the cutting part of the tool and the non-cutting part are taken into account - if the corresponding flag is set in the **CAM System Options**. In this case, the numbers of defective frames are remembered (then they can be viewed in the simulator), defective sections of the toolpath are marked in black and in the **Before first cutting** mode the exact cutting value is displayed (but not more than 1). The **Cutting tolerance** is set in the toolpath of the machining parameters. The effect of the option can be checked, for example, by changing the **Approximation** accuracy parameter and observing the result.

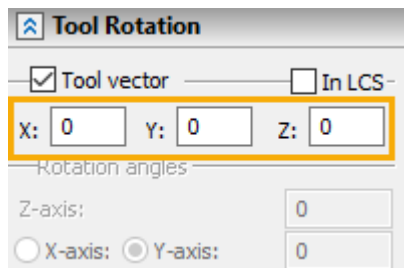
5D Drilling

Automatic recognition of holes with different orientations has been added.

Added the ability to automatically search for all holes on the body, which allows you to significantly save time for generating toolpaths for drilling, boring, threading, etc.



To do this, set the Tool vector flag and set the position of the X, Y, Z axes to 0.



In the **Auto** mode, the ability to automatically detect the depth of through holes has been added, and the depth limitation set in the drilling depth window has been removed. The window itself is now locked. If the drilling depth of one of the holes is greater than the tool length, a warning is issued.

The **Auto** mode determination of the depth of through holes when selecting their cylindrical faces has been improved.

When forming the NC, the case of drilling holes of different orientations in one toolpath was taken into account.

Selecting one flat face uses a simplified calculation method.

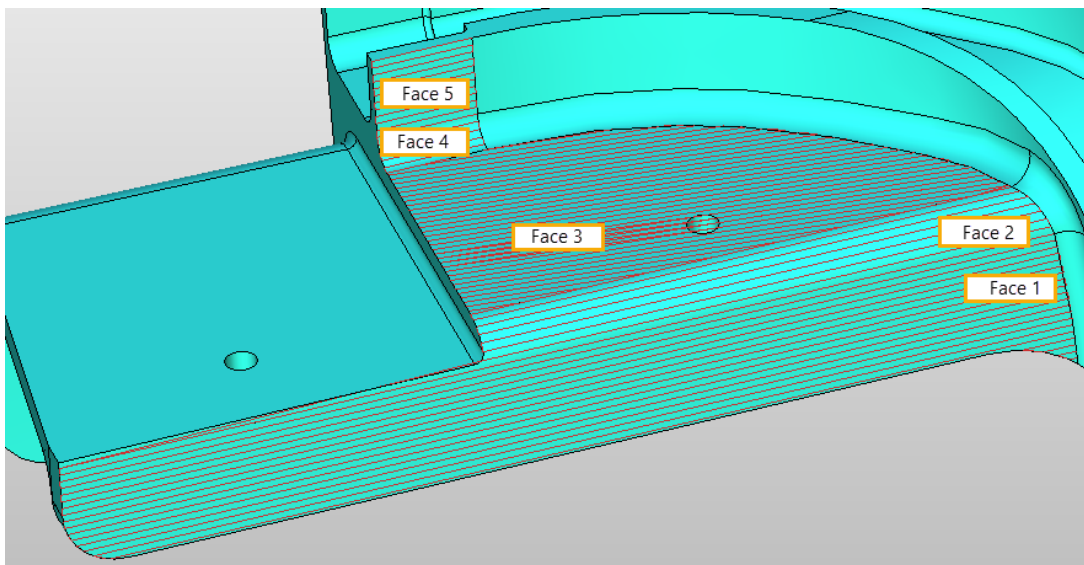
Added recognition of incomplete holes when selecting a face.

Selecting Multiple Faces in 5D Zone Milling

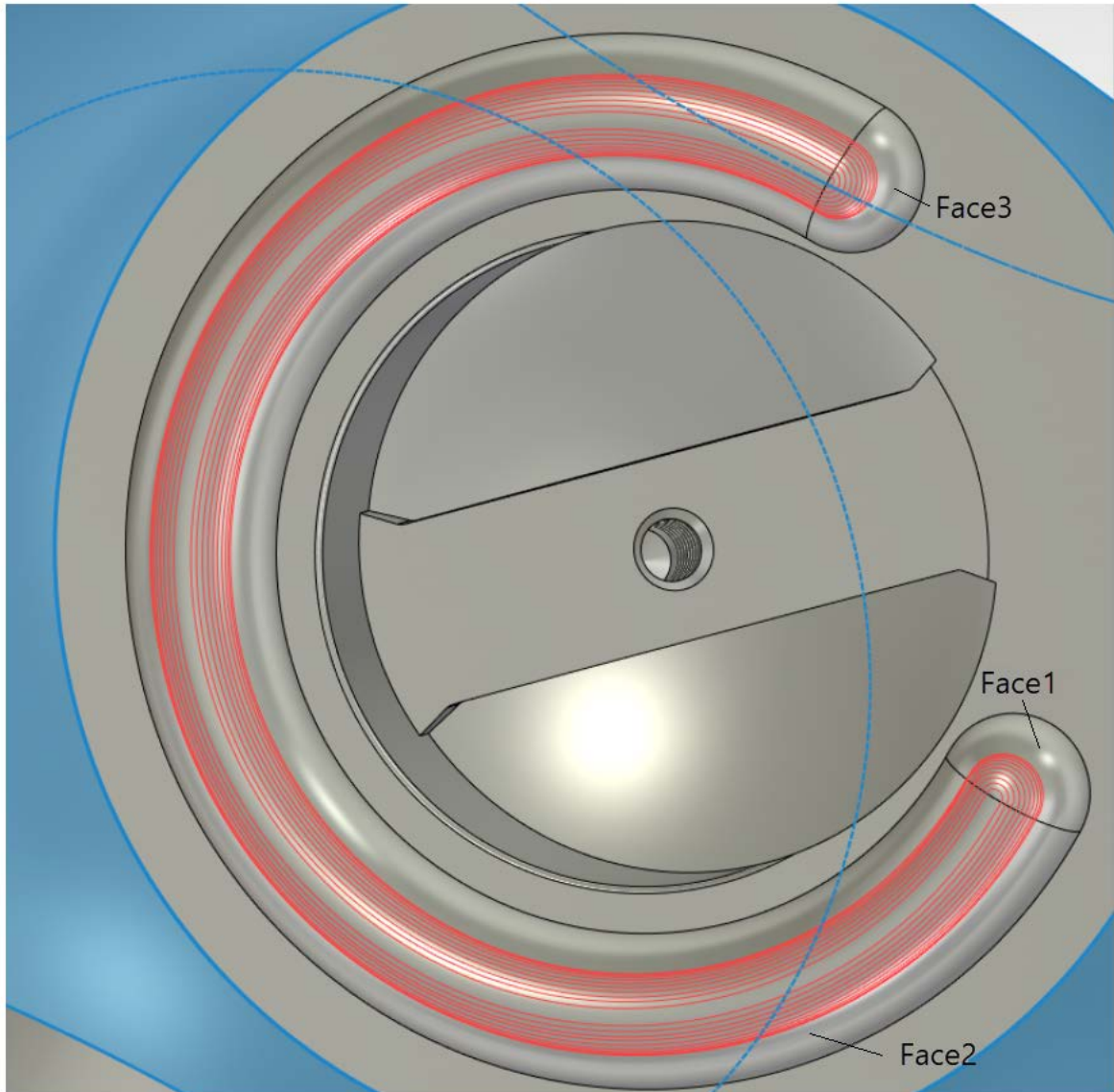
The ability to select multiple faces in 5D zone milling with the ability to quickly transition from one face to another has been added.



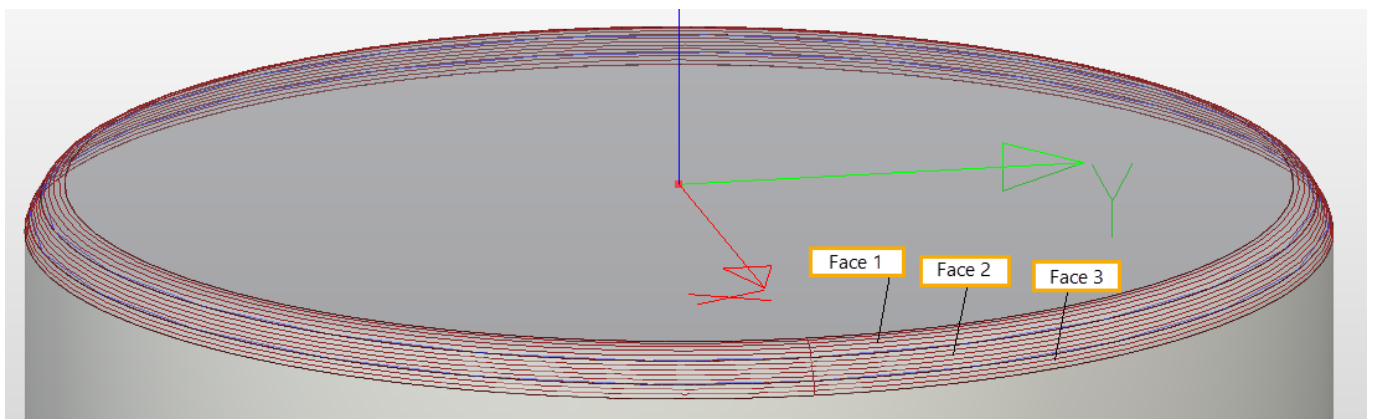
The merging of the passes of individual faces has been implemented, where possible.



The case of closing passages is taken into account (for example, for spherical sections at the ends of a toroidal groove, as shown in the figure).



The case of closed passes of separate faces is taken into account.



Variant of processing ruled surfaces (without the flag "2nd curvature") is taken into account.
Added the ability to invert the order of passes.

Accelerated Recalculation

The first calculation of the toolpath occurs in the usual way, but with the preservation of the intermediate calculation results (ICR) of the most time-critical section.

If, when editing any parameter, it does not affect the ICR - this is determined automatically - then the mode of accelerated toolpath recalculation (ATR) is activated, in which the critical section is simply bypassed and replaced with the previously saved ICR data. Experience shows that the gain in time due to the use of ART can reach an order or more.

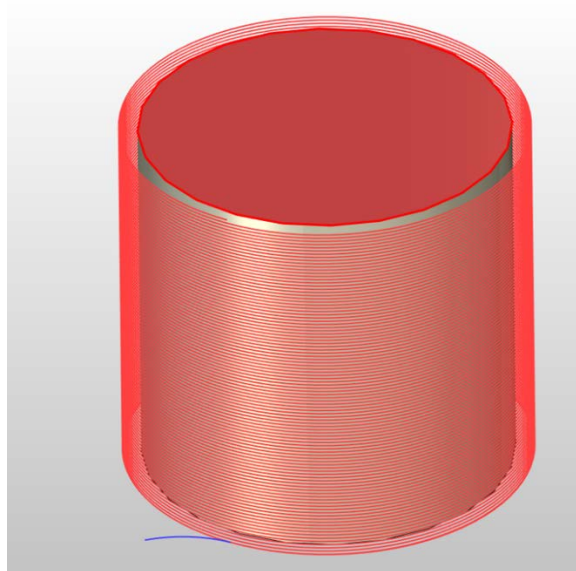
The parameters, the change of which starts the ATR, include all technological parameters - feed rate, spindle frequency, etc., **Allowance, Thickness, Step of a deepening**, parameters of **High Speed Machining, Trimming, Plunger Machining, Radius Correction** (for a pocket), **Offset Correction, Circular Interpolation** flag, **Moving On/Off** parameters, etc.

Examples of parameters for which the ATR mode does not work are any geometry change, step and type of passes, the **Open Pocket** mode, **Contour Filter, Side Offset, Fillet Radius, Radius Correction** (for pencil milling) and etc.

The ATR mode is effective only when editing parameters, namely in the preview mode or exit by clicking the **Finish Input** button. In a normal recalculation of the toolpath by RMB, the normal recalculation is in effect, but the ICR is preserved.

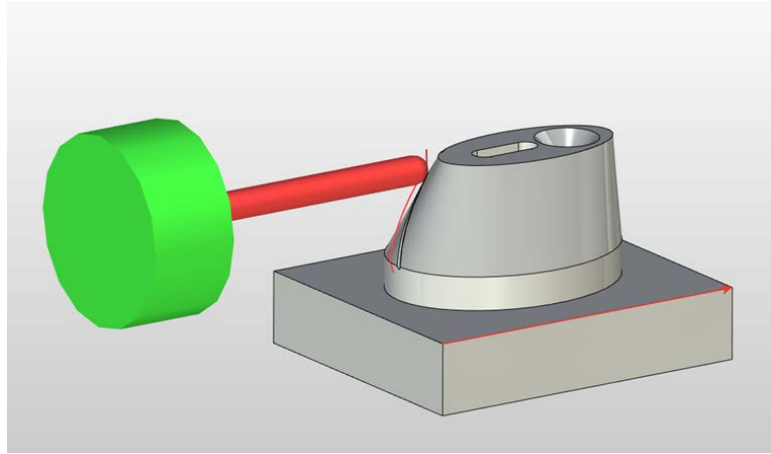
Cutting-in, Start/Return point, Moving on/off Updating

Previously, each creation or changing of start/return point, as well as cutting-in and moving on/off required recalculation of the main toolpath, which in some cases led to a significant (at least twice) increase in the total time for creating the toolpaths. Now this is not required.



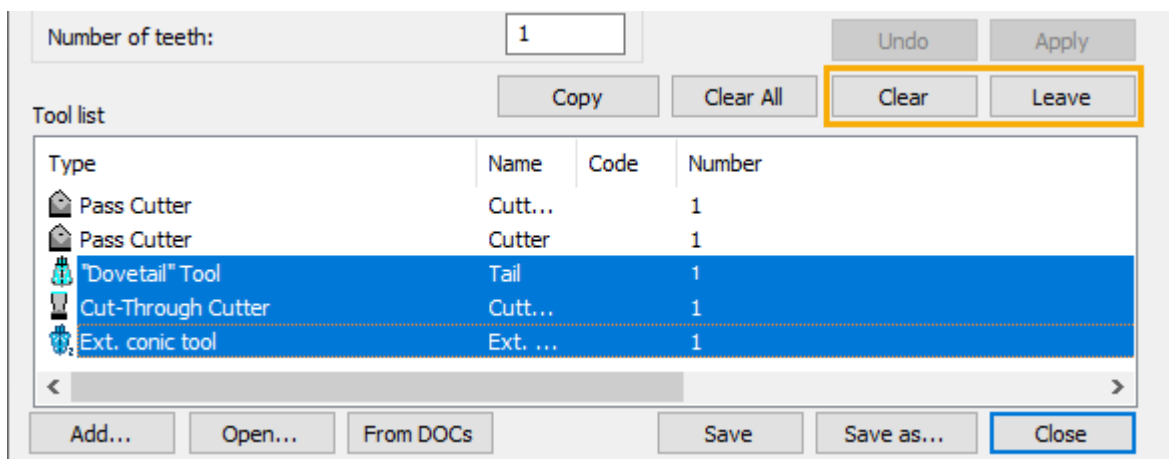
Tool Vector

In 3D zone milling, the ability to generate a tool vector by selecting edges and workplanes has been added.



Tool Editor

In the tool editor, the ability to delete or leave the selected toolbox has been added. This is achieved by introducing a multiple selection of items in the list of tools and **Leave** button, when pressed, all except the selected items in the list are deleted. The **Clear One** button has been replaced by **Clear** and now allows you to delete all selected list items.



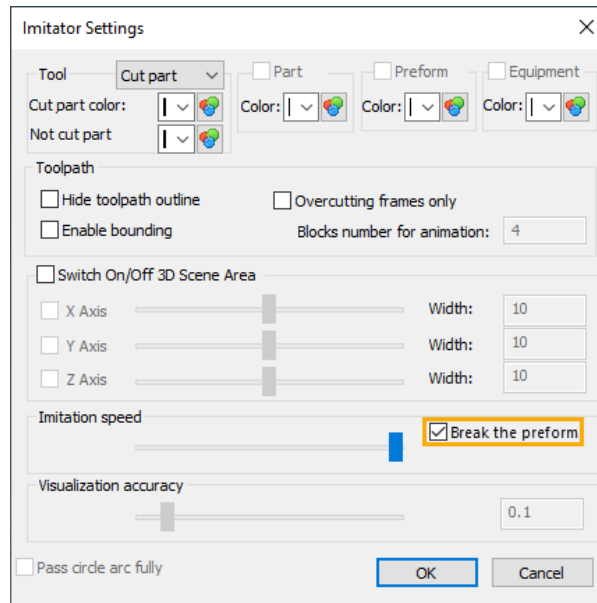
Also, now you don't need to redefine the tool after correcting its name in the tool editor. The system remembers the position (index) of the selected instrument in the combo box list. But there is a condition that it is impossible to change the position of the tool in the list simultaneously with the name.

CNC Imitator

«Break the preform» option

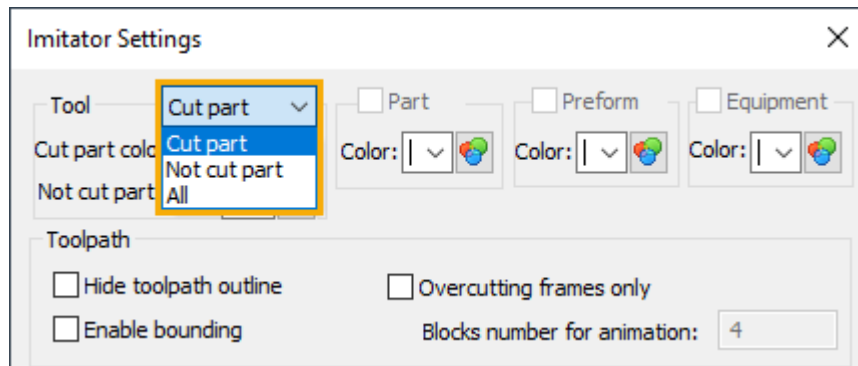
To speed up the process of simulation with removal, the **Break the preform** option has been added, which splits the workpiece into several bodies.

This function allows you to practically exclude the non-linear effect of slowing down the simulation speed as the number of workpiece faces increases in the process of interaction with the tool. As the experiment has shown, this allows in some cases to speed up the simulation process and the generation time of the resulting model by 2-3 times.



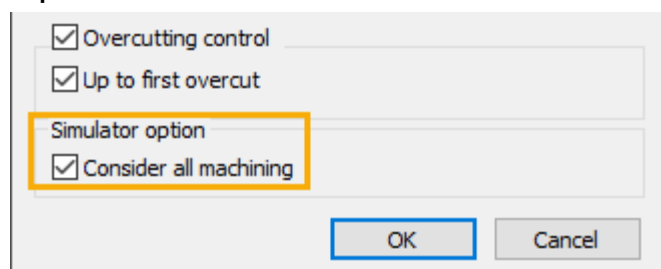
Simulation taking into account the non-cutting part of the tool

The ability to set simulation taking into account the non-cutting part of the tool has been added. For this, a drop-down list with a set of options and lists appeared in which you can set the colors of the cutting and non-cutting parts of the tool.



«Consider all machining» option

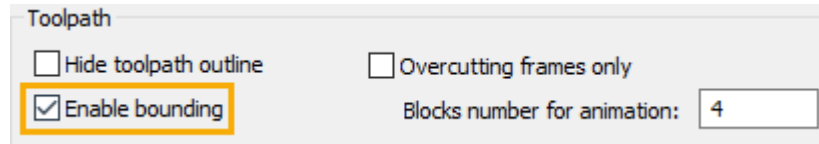
The ability to take into account all machining during simulation with automatic change in the scene of the bodies of the rig has been added. For this, the flag **Consider all machining** have been added in the **CAM System Options, Simulator option** section.



By default, it is cleared - work is carried out taking into account only active machining. The bodies of the part and the workpiece, as well as all the allowances, are the same for all machining and are taken from the toolpath of the active machining parameters.

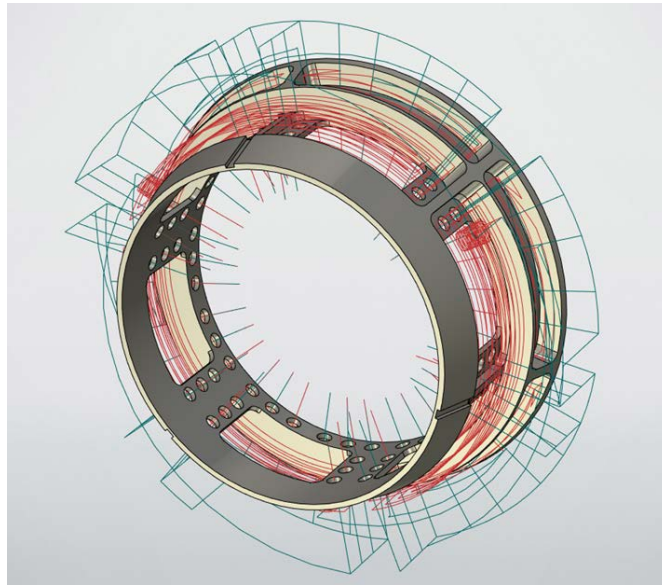
Now, when entering the machining simulator, a warning is displayed indicating the name of the toolpath, if a tool is not specified for any of them.

Also in the simulator, the operation of the **Enable bounding** of 3D scene option has been restored and optimized.



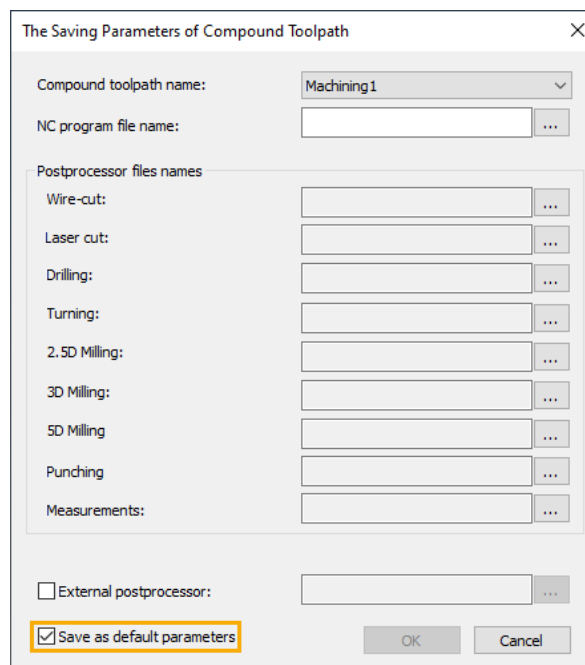
Going on G0 with Changing the Tool Vector

Now, when displaying in a 3D window and simulator, this transition is made not along a straight line, as before, but along an arc of a circle, which makes it possible to exclude false cuttings.



Saving Parameters of Compound Toolpath

Added the **Save as default parameters** flag to the dialog. If you set this flag and exit with **OK**, then the name of the file with the G-program, the name of the external postprocessor, and the names of all internal postprocessors are saved in the registry and when creating a new NC these values will be used by default for all machining. The user only needs to install and save the required settings once.

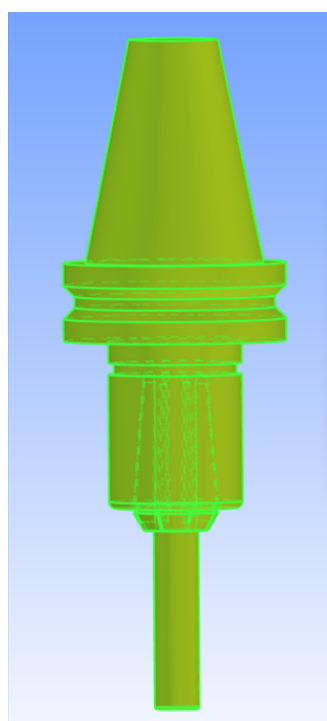
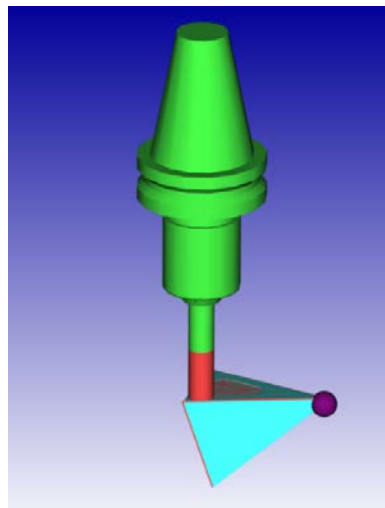


Using the Fragment of an Instrumental Assembly

A mechanism for using a parameterized 3D fragment of an instrumental assembly (FIA) as a tool body template for several toolpaths with the same type (for example, conical), but different tool parameters has been added.

Using the FIA as the body of the instrument allows you to solve the following tasks:

- use to simulate the real shape of the cutting and non-cutting part of the tool;
- select a more suitable tool by analyzing the simulation result and changing one or another FIA parameter (for example, the length of the cutter);
- use a shaped (non-standard) tool that is not available in the tool editor (TE);
- use one FIA of this type (for example, a cone cutter) as a template for several toolpaths at once with different tool parameters;
- control the parameters of the tool of a given toolpath in the file of the instrument (TE), by changing the corresponding variables of the FIA and vice versa from the TE change the parameters of the FIA;
- use additional FIA variables to control the shape of the instrument and, accordingly, the result of simulation, which are lacking in the RI;



Name	Comment	Expression
m_H	Cutting length	40
m_CR	Cutting edge radius	1
m_D	Cutter diameter	20
m_L	Tool length	150
m_OFF	Disable	0

☒ Variables List
☐ Preview

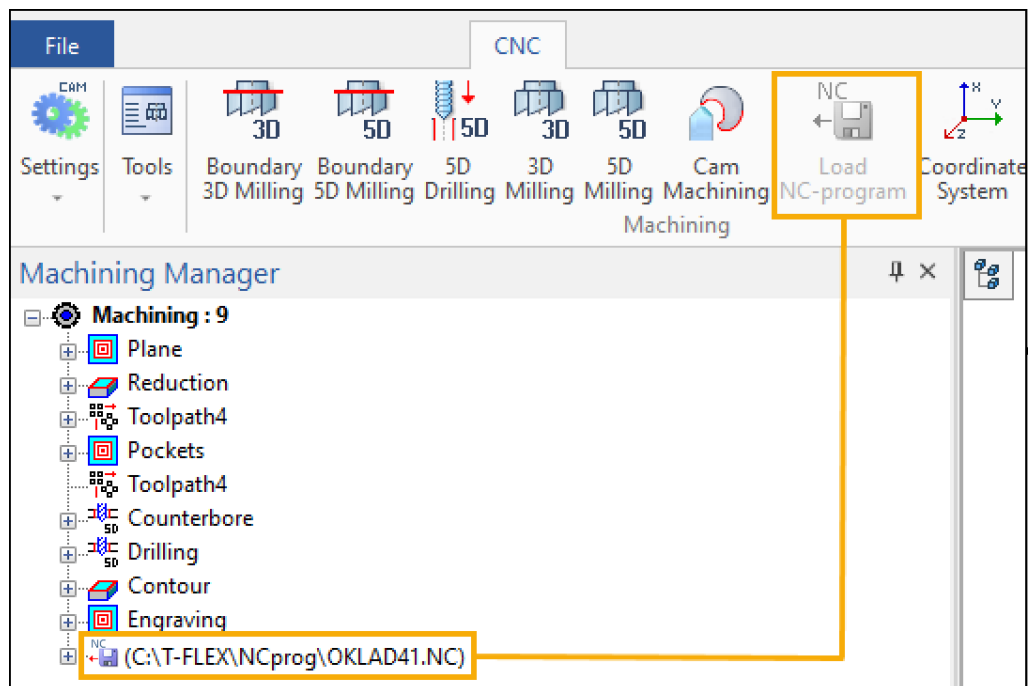
OK Cancel

Toolpath creation from NC

The ability to create a toolpath using NC has been added. This possibility is realized by means of repost-processing of the NC, that is, the transformation of a specific NC into a set of standard G-commands perceived by T-FLEX CAM. Repost-processing is carried out with the help of special .dll repost-processor (RPP), developed, as well as the software, taking into account a specific machine and CAM rack and loaded dynamically when the NC is loaded. The user can use either the default RPP .dll (BM12_500_E200CNC_5D1.dll), or order the development of a specialized RPP for his specific machine.

To correctly take into account the angular axes, the kinematic RPP (KRPP) is used, which is a text file in which the command matrix of the kinematic diagram of the machine is written. In accordance with this matrix, the tool is rotated and/or moved along the axes. In other words, this file determines how and in what sequence the movements are processed by the machine. The path of the KRPP file must match the corresponding RPP with the .dll extension replaced with .txt.

To load NC in the automenu of all basic types of paths (3D and 5D zone, 5D drilling, 3D and 5D milling), the **Load NC-program** button has been added, which allows loading the required NC. Simultaneously there is a dynamic loading of the .dll RPP, if it has not yet been loaded, and the corresponding KRPP file. If the RPP file is absent by default, the NC load button is deactivated, and the elements of the RPP file task in the machining parameters dialog become invisible.



Other Innovations

- The name for any reason not calculated toolpath is displayed with a special message.
- In pencil milling, you can specify a tool of the "Drill" type.
- In pencil milling, when using R-correction, the offset direction is tied to the correction type (left/right).
- In pencil milling, added a flag for rounding inner corners.
- In pencil multilayer milling added the ability to specify the offset angle of the pass relative to the tool vector.
- The parameters of the tool setting map have been expanded.
- Added Q parameter in cycle G84 - **Tapping**.
- Added the ability to use circular and spiral interpolation for "3D spiral".
- Added the ability to use the **Moving On/Off** for the "3D spiral".

- For the technological toolpath and all 2D milling operations, including engraving, added the ability to edit parameters from the 3D window.
- When cutting-in into a finished hole, angle A is measured in increments relative to the tangent to the toolpath.
- For 5D zone milling in the **Positional** mode and the type of pass **Spiral** with a constant vector of the tool, the options for **Circular Interpolation** and **Moving On/Off** are added.
- When selecting multiple faces in 5D zone milling, merging of passes of individual faces has been added.
- In 5D zone positional milling, added flags for the first and last pass in spiral milling.
- In 5D zone positional milling, the ability to change the movement of the cutter in the direction across the passes has been added.
- In 5D zone milling with the "Spiral" pass, the ability to tilt the cutter by a specified angle has been added. A constant tilt of the cutter is useful, for example, for chamfering with a conventional or disc cutter.

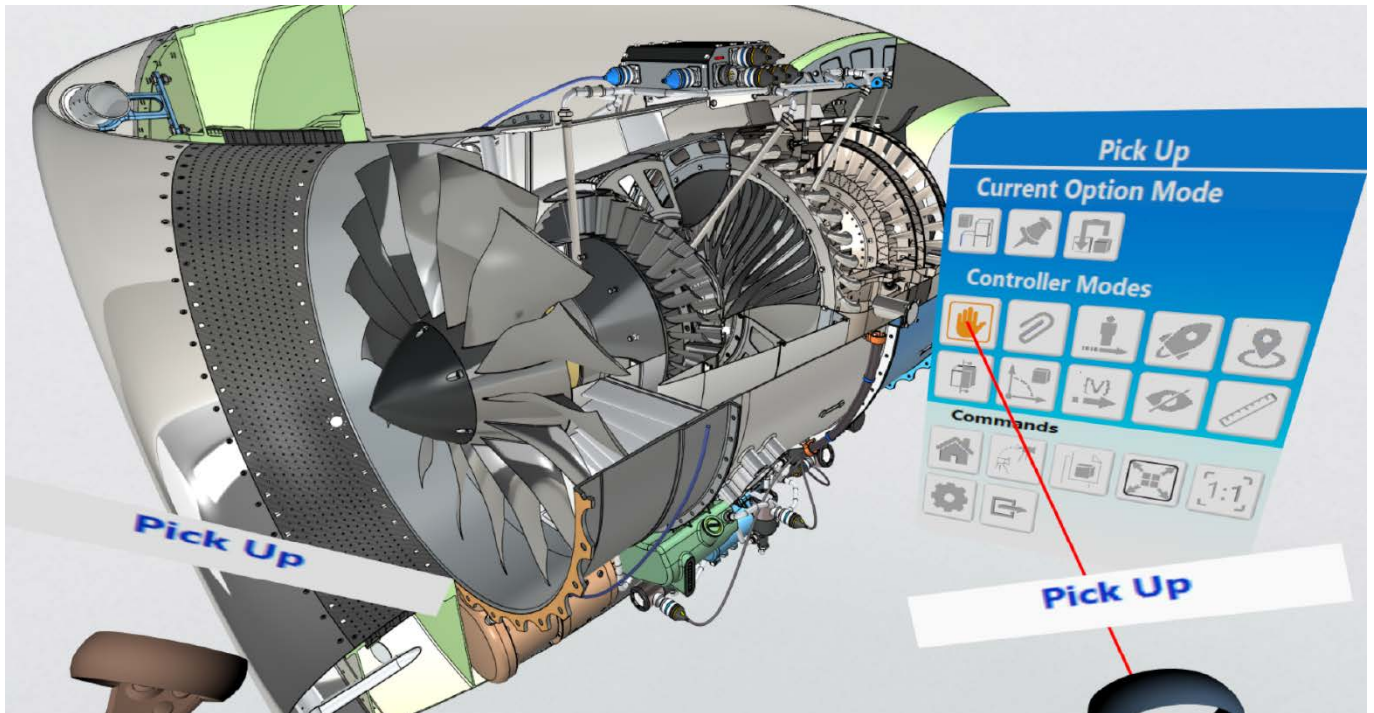
T·FLEX VR 17

What's New



T-FLEX VR

The new T-FLEX VR module continues to evolve. The dialogue with the model in VR space has become more logical and convenient. New commands for measuring the model were added.




VR menu is organized in the following manner:

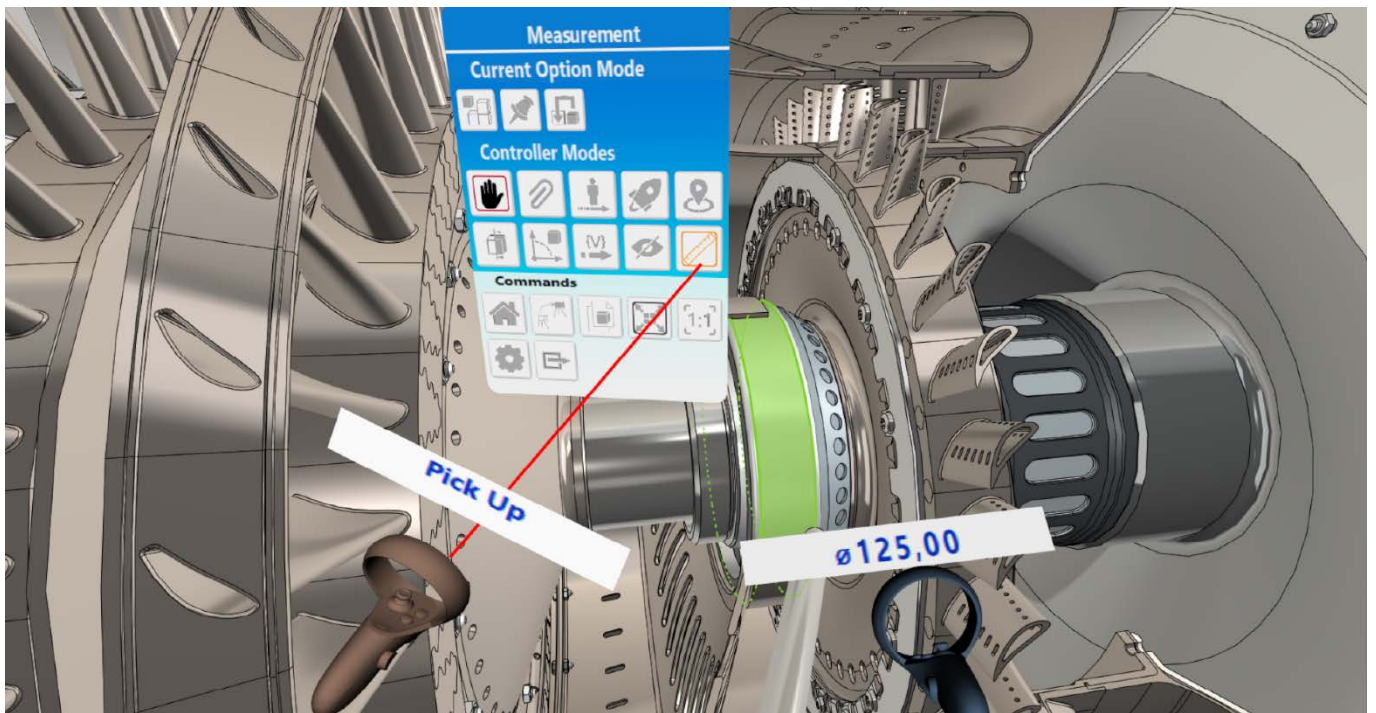
- Upper part of the menu displays current mode of VR controller.
- A list of additional options for current mode or command of VR controller is shown below.
- Next the list of modes available for VR controller is disposed.
- Bottom part displays the commands available for selection in the VR menu.

Both modes and commands allow to change the virtual space or set the representation options. There is always one active mode at any moment of work, **Take** mode is active by default. User defines the necessity of command running.



Now in VR space you can take measurements. A new mode has been developed for this.

 **Measurement.** Four measurement variants and dynamic highlighting option (active by default) are available.



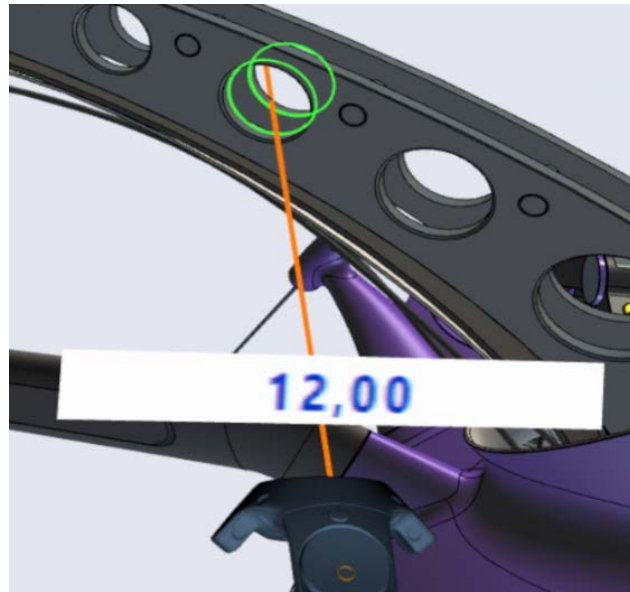
Distance. The distance from the ray origin point to the scene point of ray incidence.

Elements. If a straight edge is selected, the length of the edge is measured; if the arc of a circle is selected, the diameter of the circle is measured.



Between Vertices. The distance between two vertices selected on the objects.

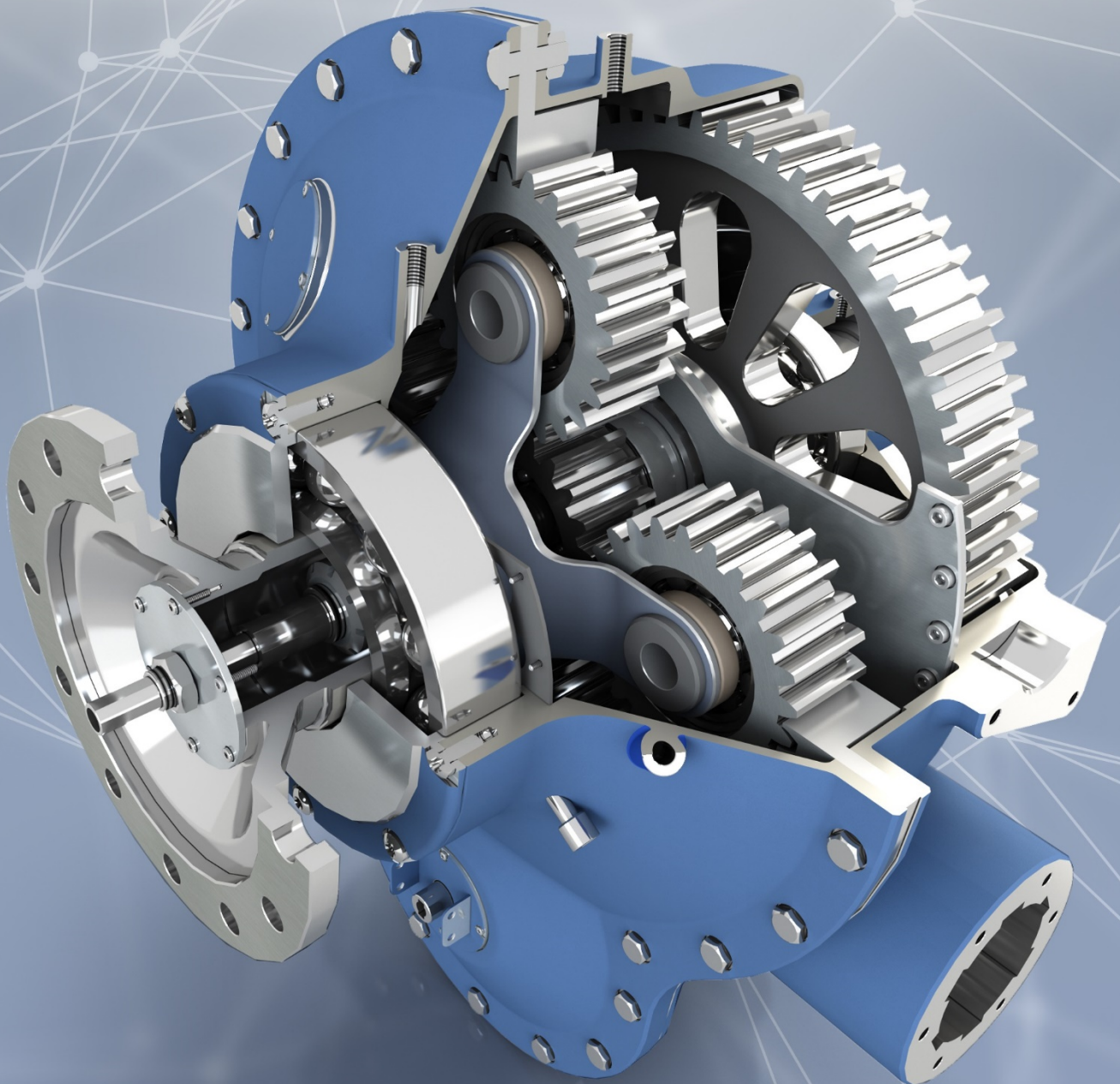
Between Elements. The option measures the minimal distance between two selected elements. The edges and vertices can be selected.



Dynamic Highlighting. The option highlights the elements pointed by the ray.

T·FLEX Gears

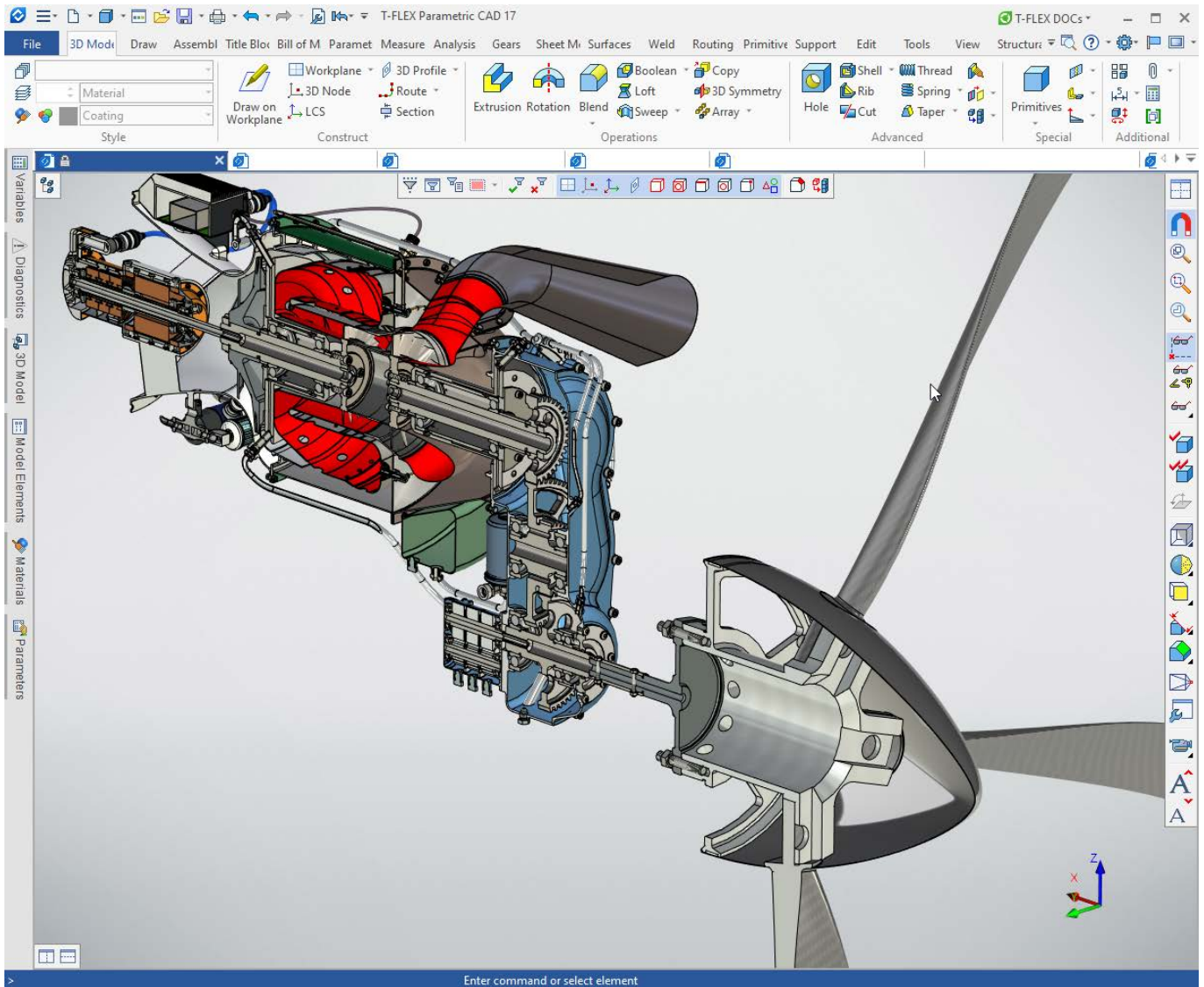
NEW APPLICATION



T-FLEX Gears 17

T-FLEX Gears application is a part of T-FLEX PLM complex, which serves for design, analysis and calculation of gears, gearings or complete mechanisms. Apart from creating 3D models and model-based drawings (in accordance with various standards) it also allows calculations.

Resulting models fulfil all requirements to critical parts, so the application can be used not only in general mechanical engineering, but also in hi-tech industries such as aerospace, engine building, shipbuilding, instrumentation engineering etc.



Calculation and Analysis

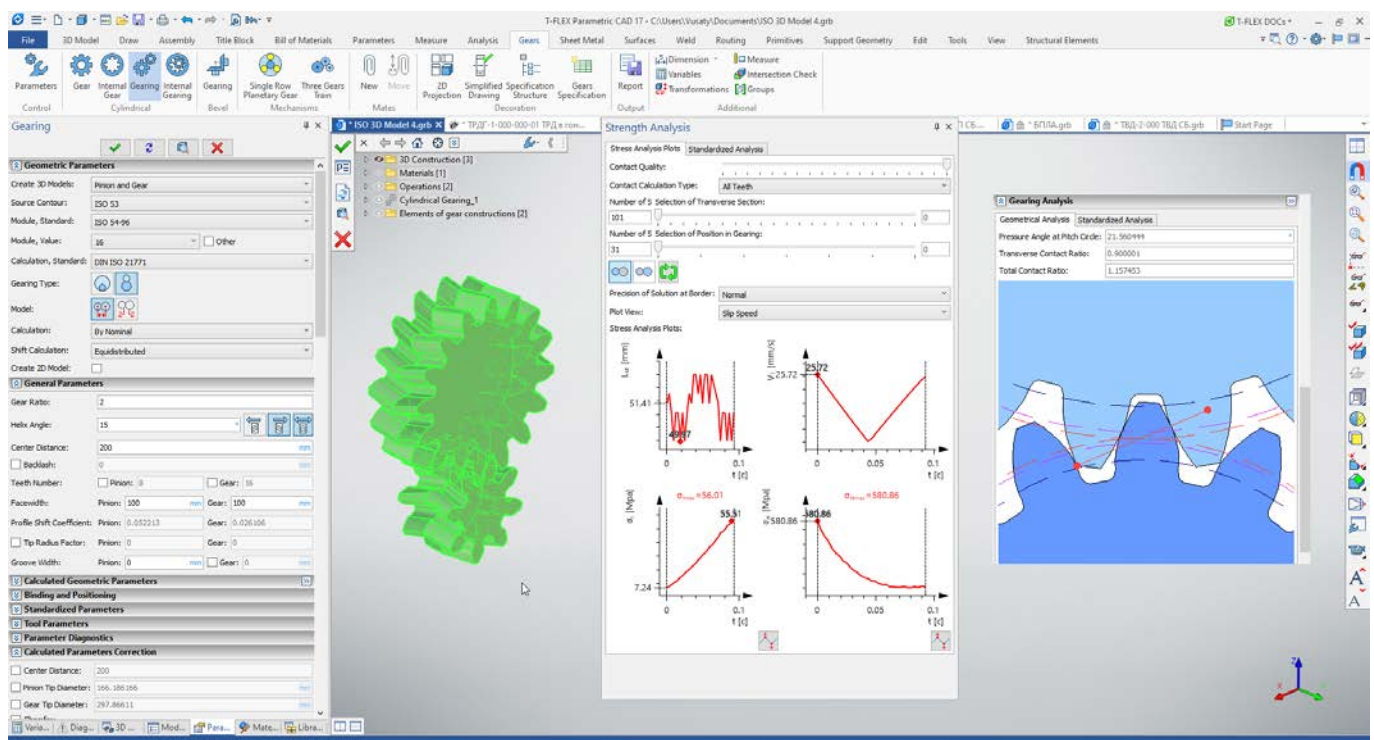
GOST, ISO or DIN standards can be used in the application. Following calculations are available:

- Geometrical analysis;
- Accuracy Parameters calculation;
- Strength analysis.

You can also use non-standard user-defined geometrical and accuracy parameters. The effect of custom accuracy parameters can be evaluated using calculation by middle of tolerance range.

User interface of T-FLEX Gears application provides easy control of calculation results.

All necessary data (geometrical, strength, animation) is provided in separate tabs, so you don't need to create reports for viewing results. Quick recalculation allows displaying updated results in real time.



It's worth to notice, that in the strength analysis, an estimate of the stress distribution according to a given load is available based only on geometrical parameters. If a recalculation was made to the middle of the tolerance field, it will also be taken into account. Empirical dependencies from standard calculation methods are not required here. Stresses in each end section and along the path of action are presented in the form of graphs. In addition, graphs of the path of action length change and slip rate depending on gears rotation are available.

Gearing animation is an additional tool for mechanism quality control. Animation represents the current position of gears along the path of action and selected end section. It also represents the shape of contact and transitional surfaces resulting from selected tool parameters and considering the recalculation to the middle of the tolerance field. Animation allows to easily detect possible intersections that could lead to a jam.

Gears and couplings classification

T-FLEX Gears provides the ability to design cylindrical, bevel, worm gears and gearings as well as combining them in various mechanisms. Input data can be taken from a standard (GOST, ISO, DIN) or typed in manually. So the maximum productivity is achieved without unnecessary restrictions.

Cylindrical Gears and Gearings

Cylindrical gearing can be calculated for spur, helical or double helical gears on parallel shafts. Gears with different width might be centered or positioned with a user-defined shift. Gears position affects strength calculation and contact ratio. Gears are always meshing upon generation of 3D model.

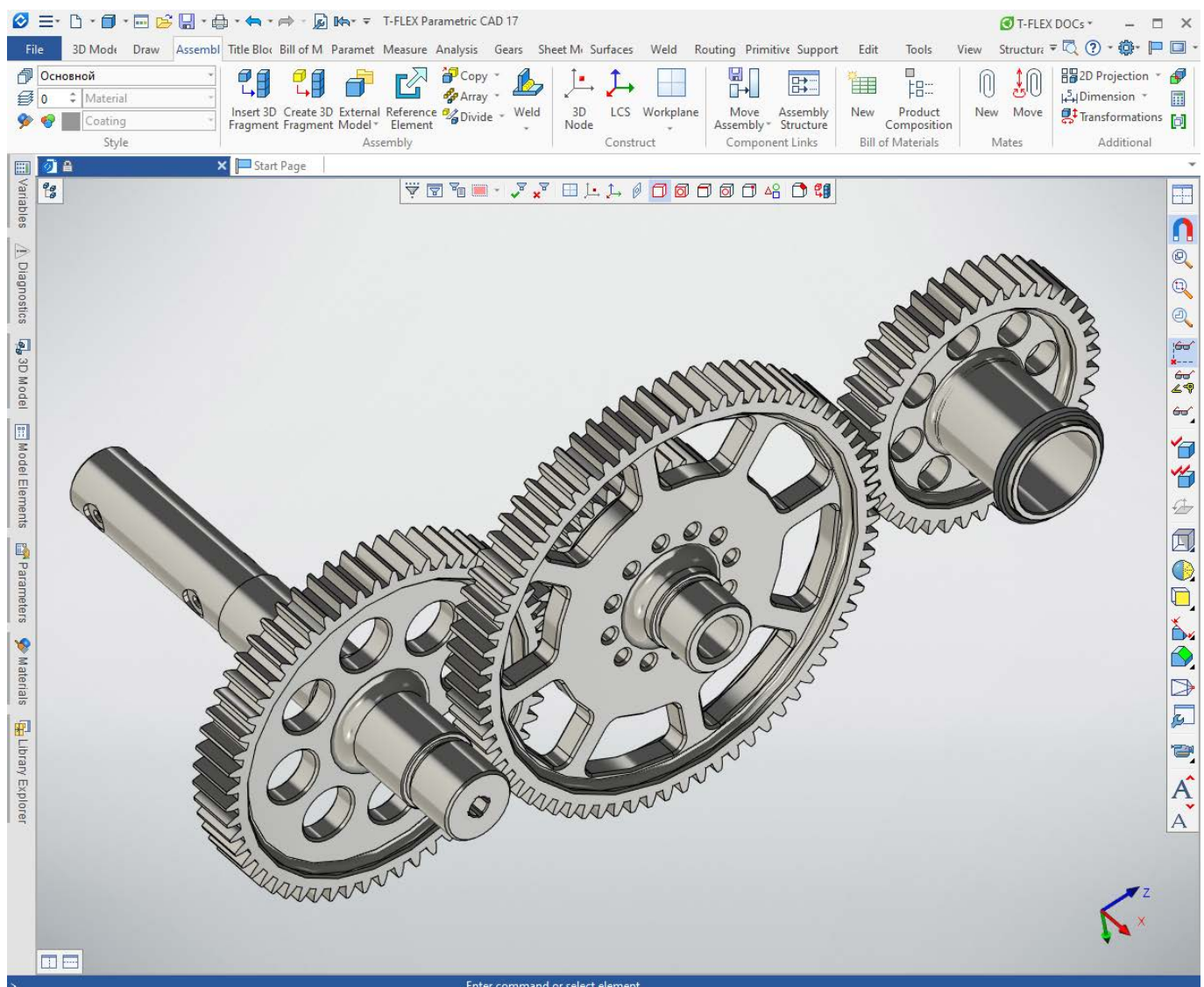
Gearing can be designed without reference to the whole mechanism or integrated into an existing design. For instance, it's enough to select two shafts and set module and ratio.

Following types of cylindrical gears and gearings are available in T-FLEX Gears:

Spur gears and gearings

This is the most popular gear type. Apart from module the gearing can be defined by teeth number, by center distance and ration or by center distance and teeth number. Shift can be defined manually or using one of the available options, e.g.: optimize by bending stress.

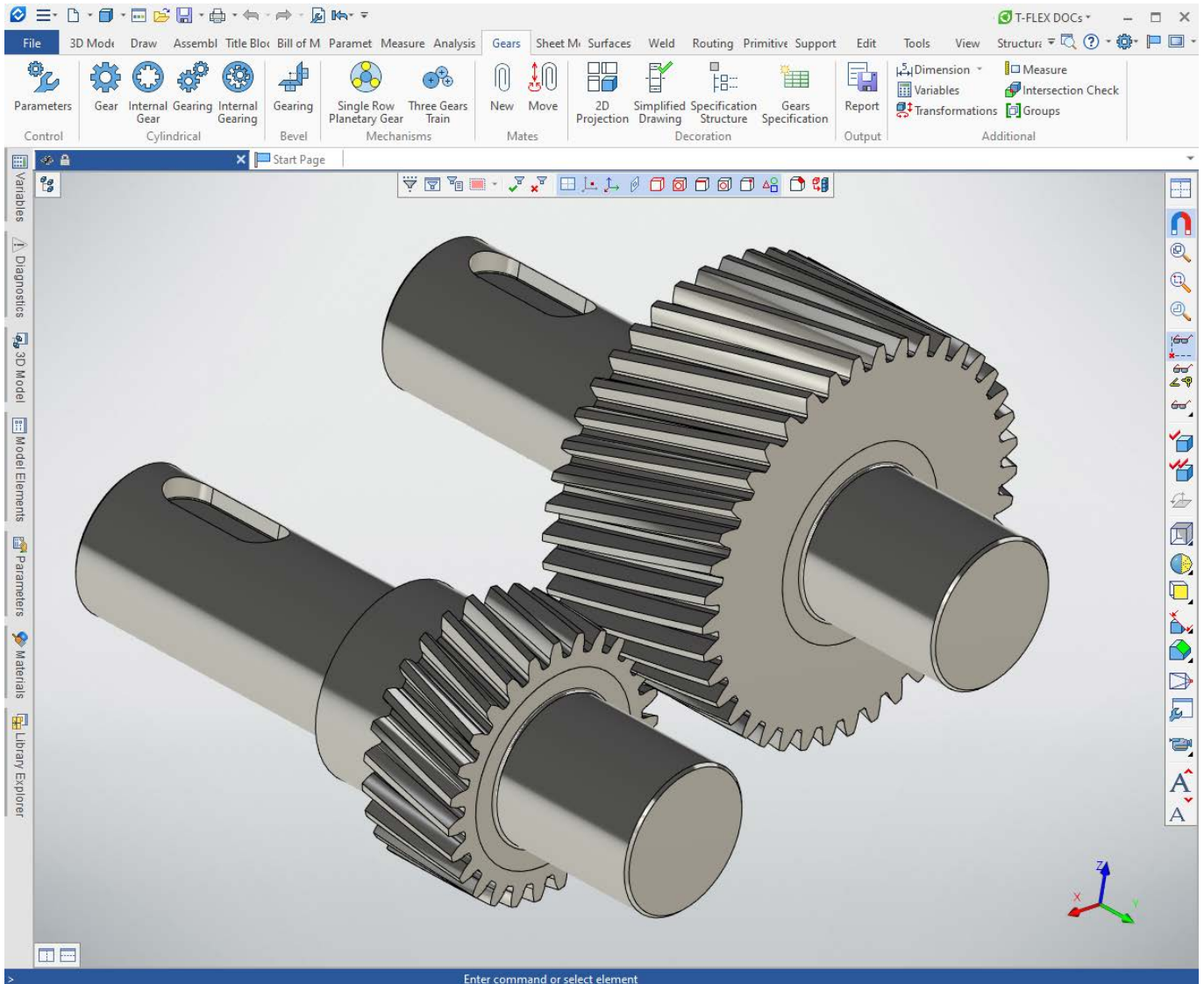
Spur gearings are used in low-speed transmissions.



Helical gears and gearings

Helical gear is formed from rotating each subsequent end section by a same angle. Apart from that, the design is similar to spur gears. The ability to see each section's animation and stress graphs provided by the T-FLEX Gears is very useful in this case.

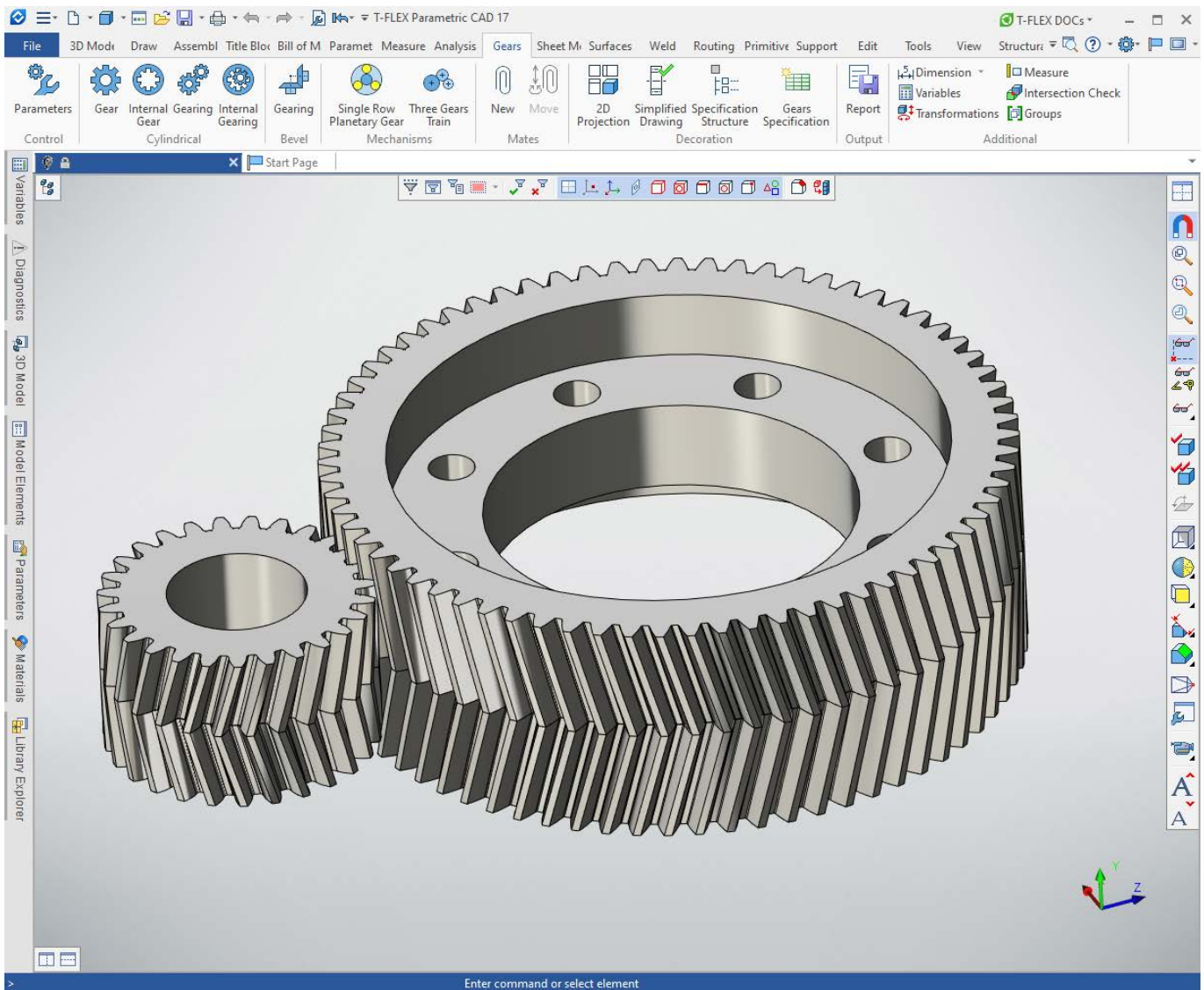
This type of gears ensures more reliable gears meshing. It leads to more smooth functioning and prolonged service life of a mechanism. Such gears are used in medium and high speed mechanisms of high importance.



Double helical gears and gearings

Select Double Helical gear type and specify width of a groove between mirrored teeth of a gear. Gearing parameters and strength analysis take double helical gears specifics into account.

Compared to other types double helical gears can transfer higher power with less noise. They are used in heavy-loaded transmissions.



Bevel gears and gearings

Bevel gears calculation utilizes the same advantages as cylindrical: variety of available standards, detailed results visualization, various analysis types, reports generation and others.

The application allows to design straight helical and spiral gearings. Special attention is paid to bevel gears 3D modelling quality. Gearing is calculated from spherical sections with the ability to adjust the number of calculated sections and the number of points on them, as well as indicating the spline interpolation degree. All the latest features of T-FLEX CAD 17 are applied upon generating a 3D model.

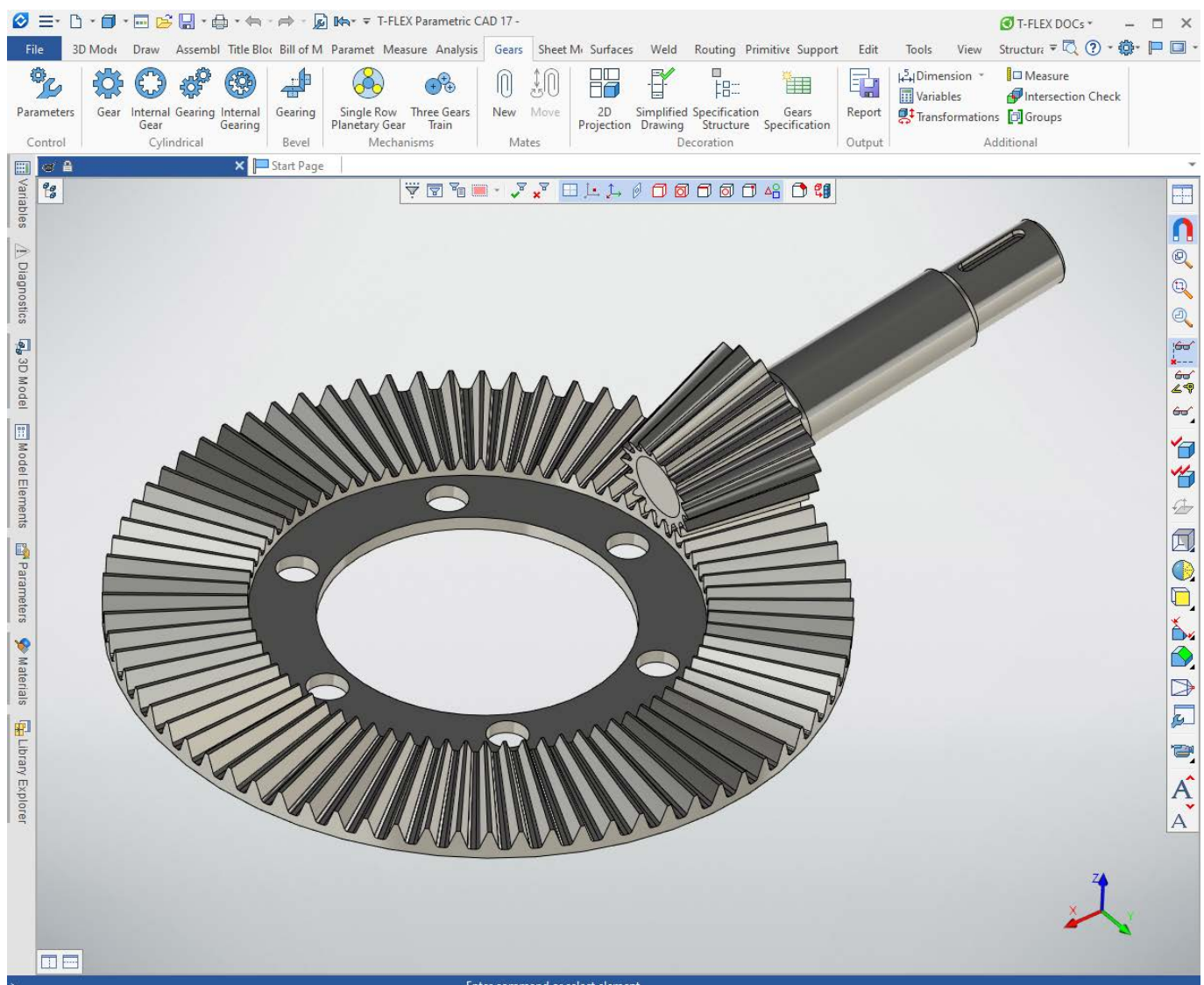
Following bevel gear and gearing types are available in the T-FLEX Gears:

Straight bevel gears and gearings

This is the simplest bevel gear type in terms of calculating geometrical and meshing parameters. However, even here, simplified geometry creation methods will give a significant inaccuracy. The gearing geometry is calculated on spherical sections with support for all precision settings.

Straight bevel gear is a particular case of helical bevel gear with zero tooth angle. You can also create spiral bevel gears with zero angle.

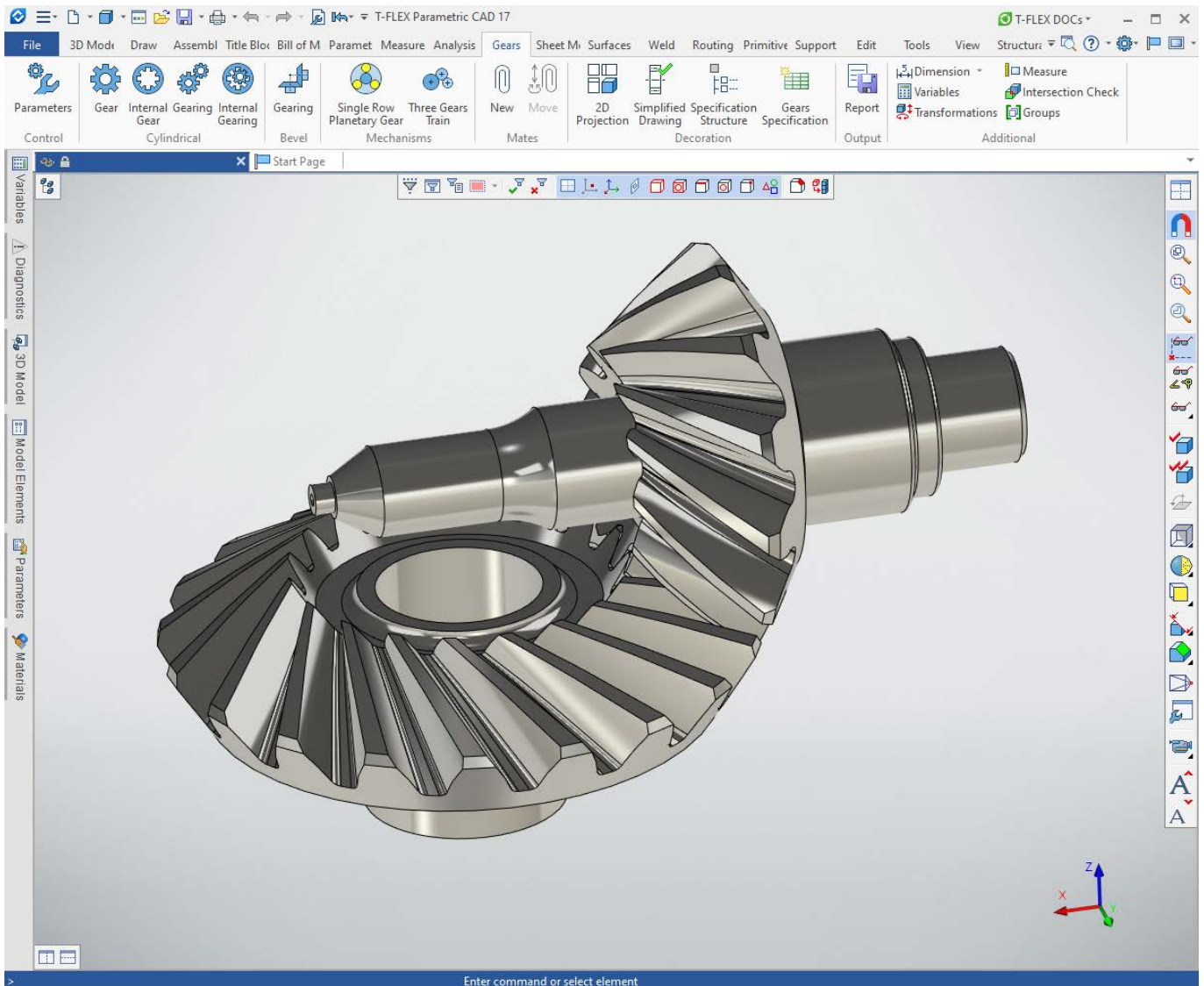
Straight bevel gearings are the most popular bevel gearing type due to simple manufacturing process. They are used in mechanisms with circumferential speed up to 3 m/s.



Helical bevel gears and gearings

It has straight angled tooth. Tooth line remains straight upon cone unwrapping. Same line of each tooth is tangent to the same circle.

Despite being simple in theory, this tooth type is rarely used due to complicated manufacturing process. Such gears are used in mechanisms with circumferential speed up to 12 m/s.

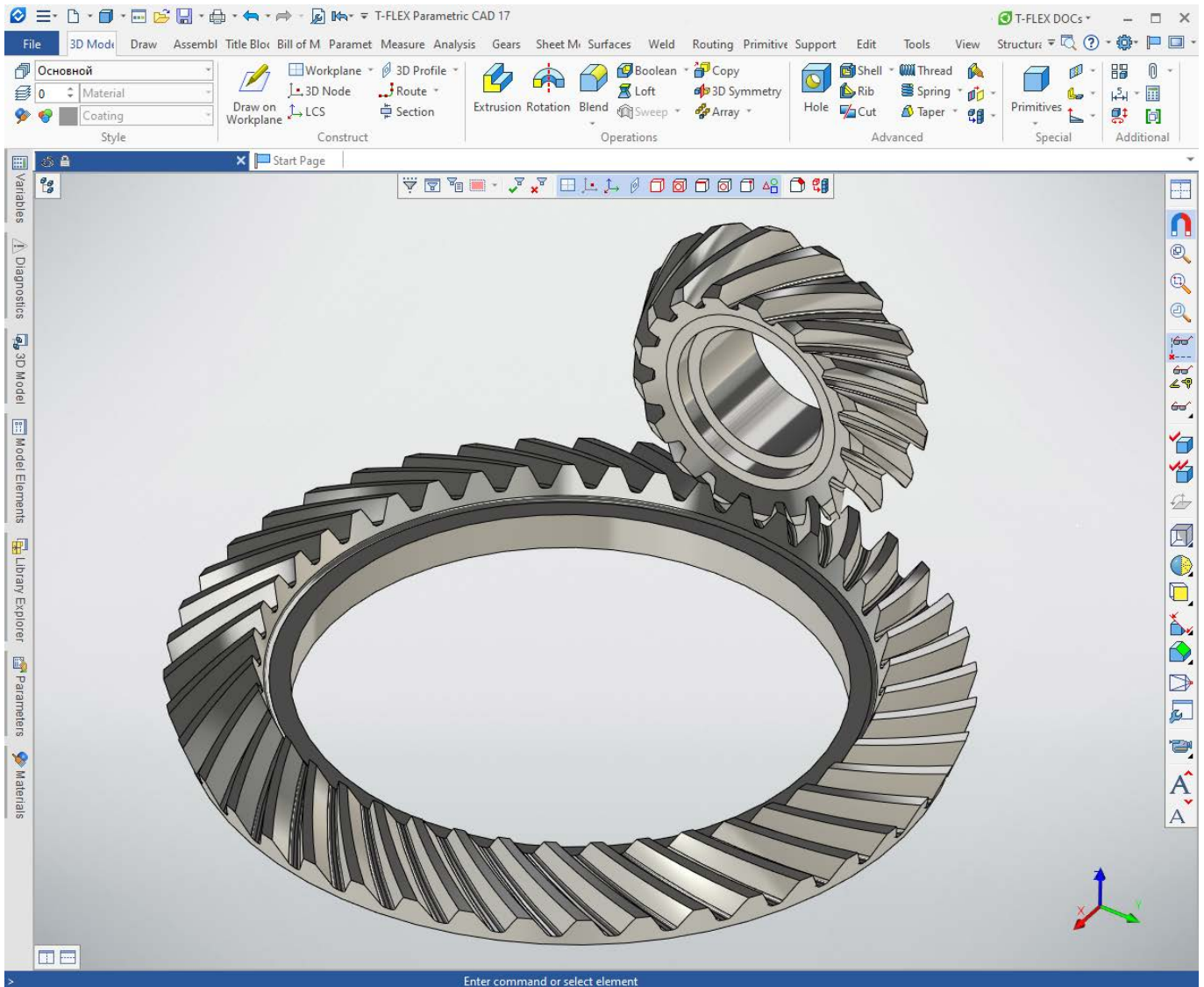


Spiral bevel gears and gearings

There are two types of spiral bevel gears available in the T-FLEX Gears:

- **Spiral (Constant)**
A line tangent to cone have the same angle to cone generatrix at each point of a tooth spiral. This type of spiral tooth is used when manufacturing process of gear cutting doesn't matter (e.g. for 3D printing)
- **Spiral**
Tooth spiral parameters are defined by manufacturing standards.

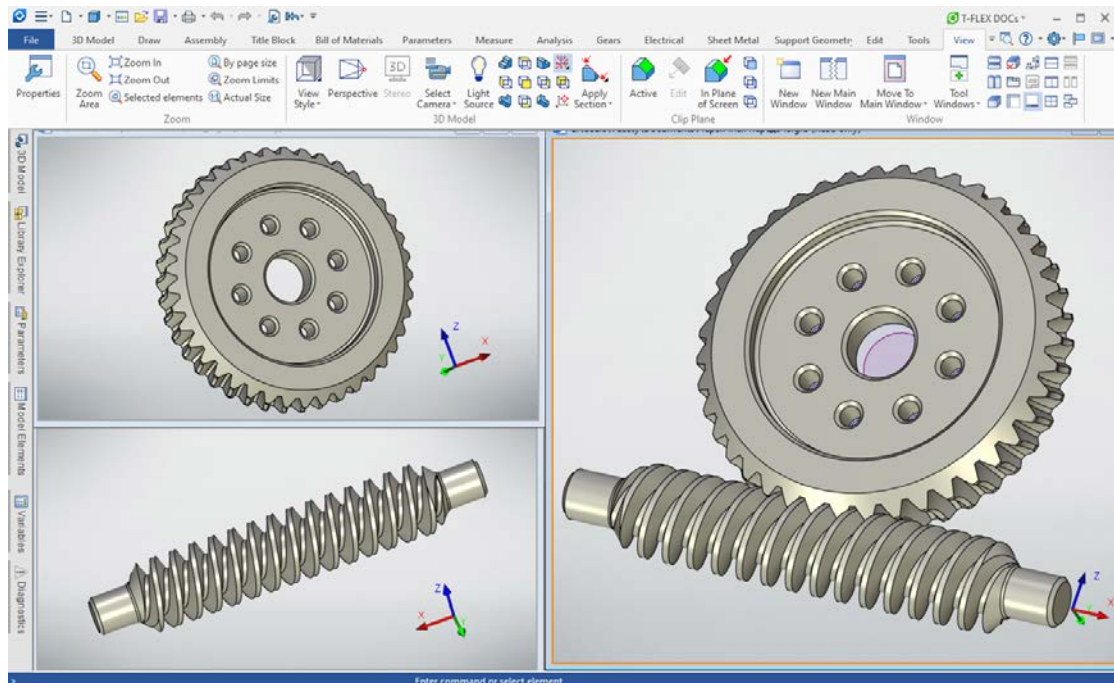
Spiral bevel gears are used in mechanisms where shafts are orthogonal to each other.



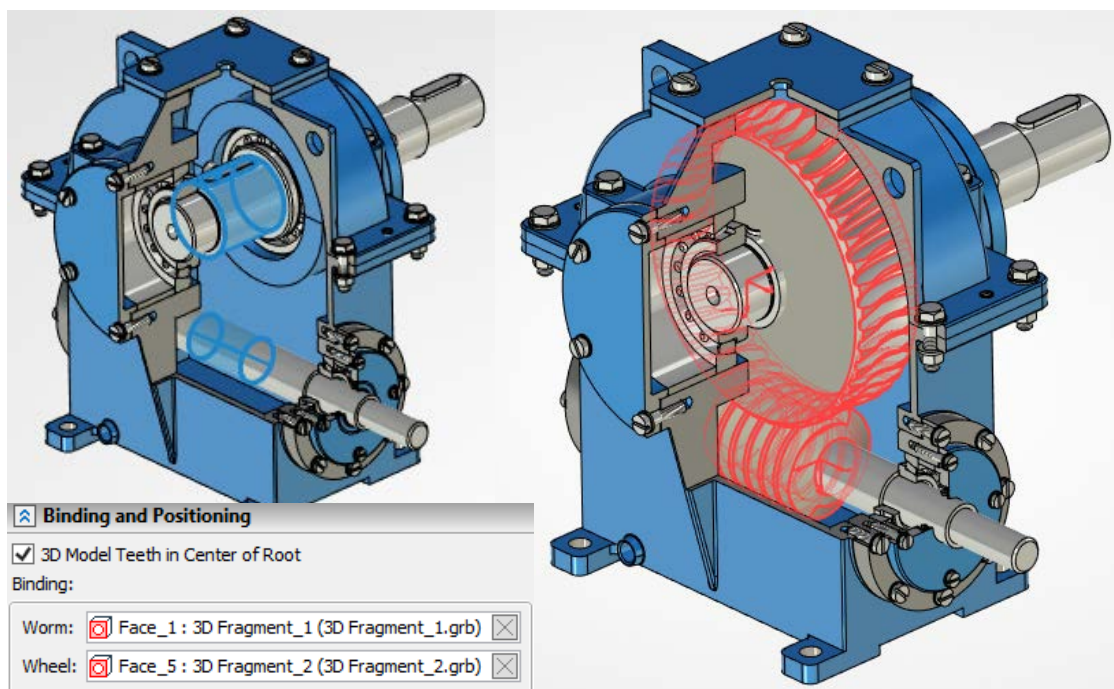
Worms, worm gears and gearings

Worm gearings transfer motion at normal angle with high ratios and torques. T-FLEX Gears provides tools for creating a worm drives. Generated models are automatically set into contact with high precision and without gaps/clashes of contacting surfaces. This is combined with the great performance – generating a model takes between 3 and 30 seconds. Worm drive can be defined:

- By center distance and shift;
- By lead and profile shift;
- By pitch diameter and shift;
- By center distance and pitch diameter.



Moreover, you can create worm drives automatically by defining only two shaft surfaces, ratio and pitch or module.



Mechanisms

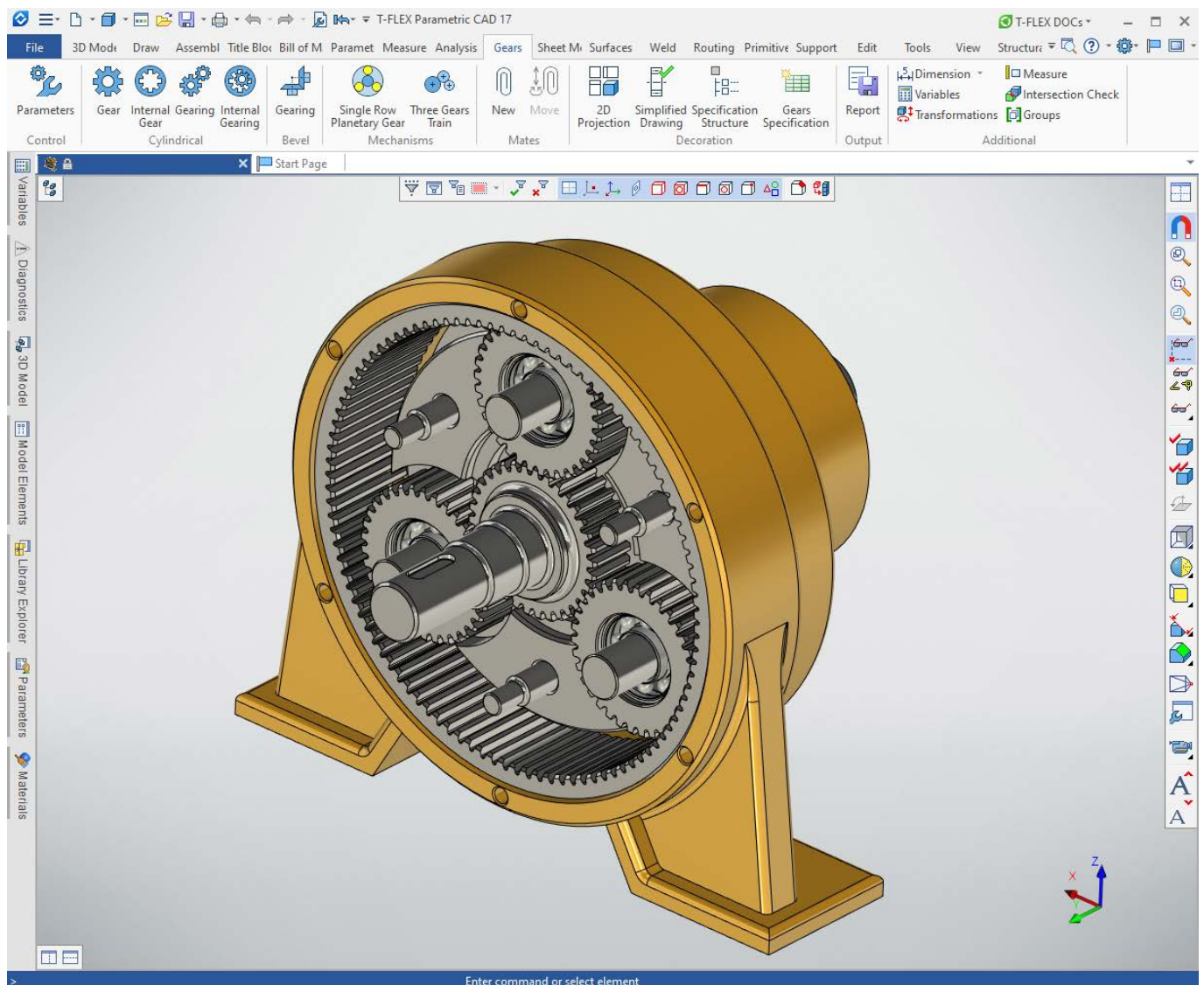
Mechanisms commands generate the whole transmission mechanism from initial parameters. It doesn't require the complicated process of positioning all meshing components.

Each gearing in a mechanism is represented as a separate gearing object similar to described above with all of the analysis and calculation possibilities.

Following mechanism types are available in T-FLEX Gears:

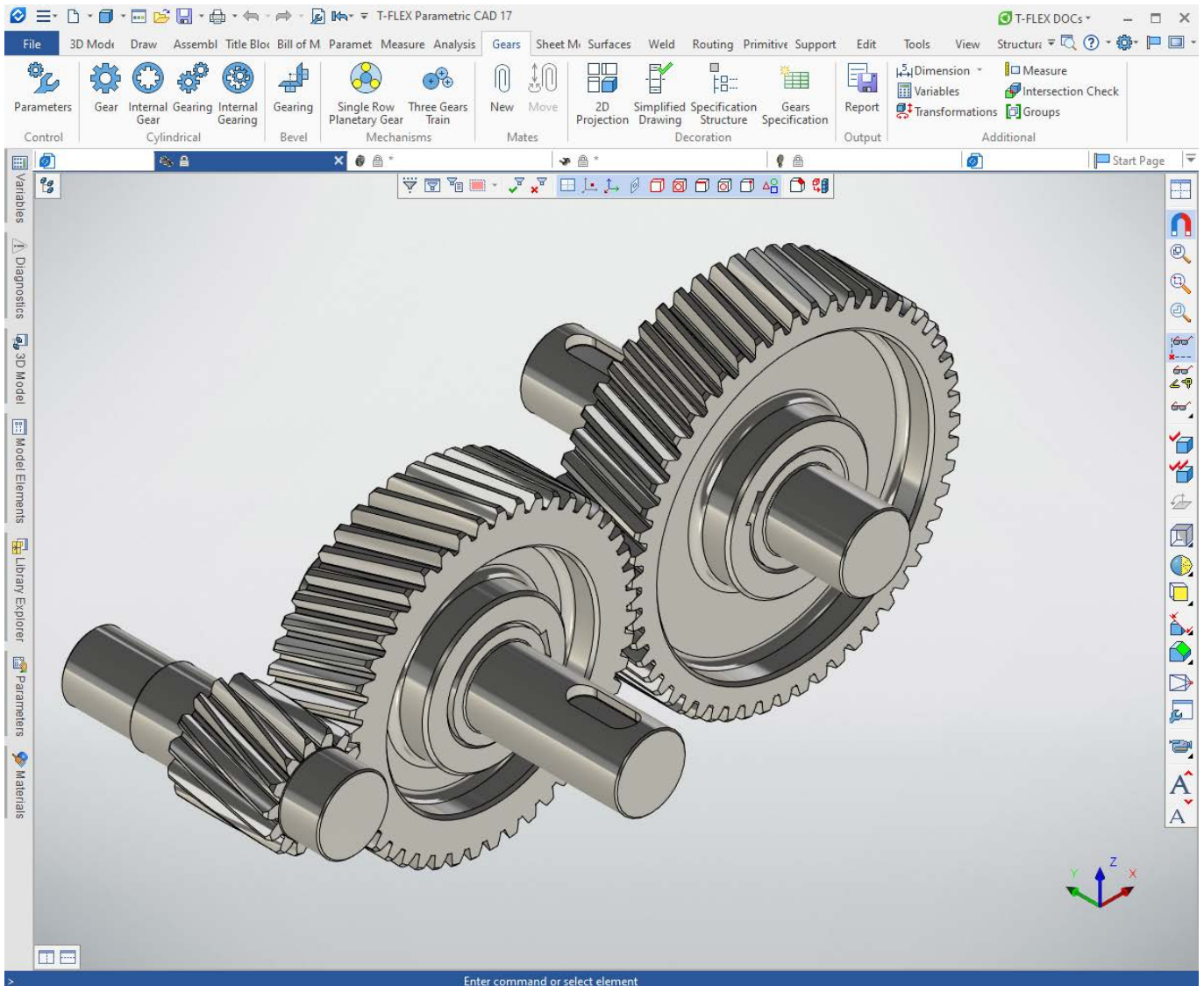
Single Row Planetary Gear

Minimum data required are the input and output element types and a ration. It's also possible to set additional parameters, such as ration search accuracy, facewidth, gears number etc. A list of possible solutions ordered from max to min module (which means from lesser to higher teeth quantity) is provided in result, so the user can choose an optimal solution.



Three Gears Train

First you need to select an input data model for each couple of gears and set the required parameters (ratios, teeth number, center distance, shift). Additional parameters are set in a similar way to the planetary gear. User can choose an optimal solution from the list of possible solutions. After selecting a solution, a train of three gears forming two gearings is generated.



All resulting gears, gearings and mechanisms can be saved to a custom library. Additionally, you can create custom templates using the T-FLEX CAD parametrization tools. All of the above helps to simplify the design process and reduce time consumption.

Decoration and Output

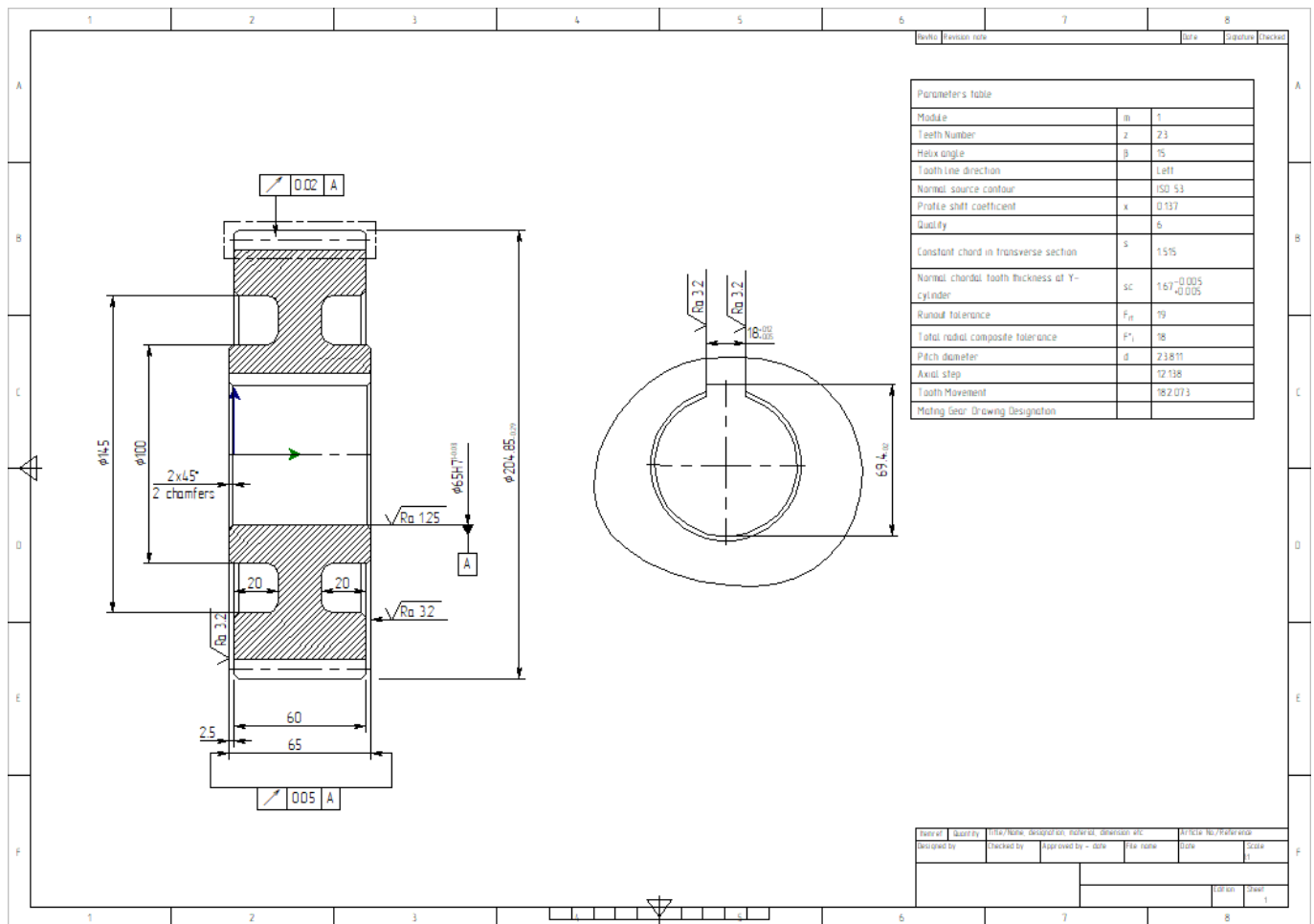
T-FLEX Gears provides tools for creating reports and generating drawings in accordance to selected standard.

The report can be generated for all calculation results at once or you can have several reports for each calculation. Reports support PDF and HTML output formats.

A gear drawing should contain a gears specification table and a special simplified teeth representation.

The Gears Specification command allows to create uniform customizable tables for all gears in assembly or for each gear separately. Size, position and content of the table can be compliant to the standards or user-defined. Default table parameters can be customized.

Simplified drawing command adds a simplified gear or gearing representation to existing 2D projections.



Mates

Automatic mates creation can be activated when creating a gearing. It allows to visualize the gears rotation and simplifies use of the resulting model in the T-FLEX Dynamics application.

