

What's New

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WELCOME TO T-FLEX CAD 17

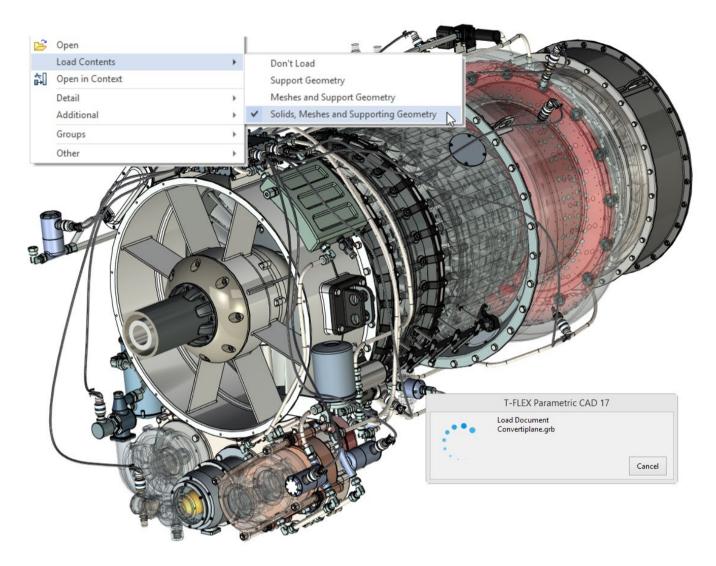
T-FLEX CAD 17 offers a large number of new functionality and improvements, many of which are implemented according to users' requests, including: a significant increase in performance when working with assemblies, teamwork using T-FLEX DOCs, a new mechanism for 3D design, new operations for surface modeling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



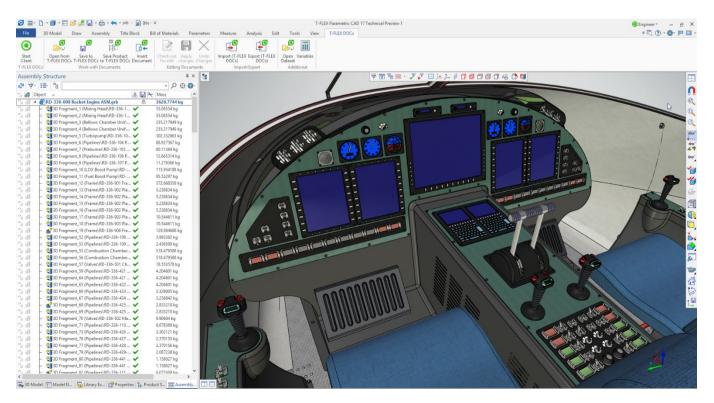
Teamwork on Large Projects

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.

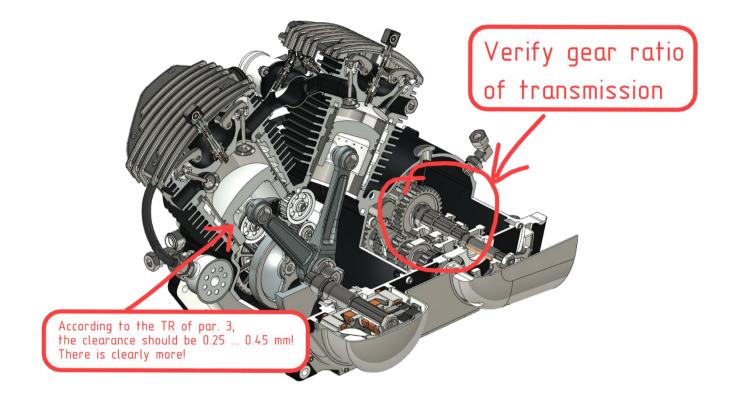
Working with large assemblies was improved: loading, editing, and recalculating assemblies is up to ten times faster thanks to new mechanisms for loading assemblies and reducing memory consumption. Now you can work with assemblies with unloaded geometry and mesh, including the insertion of such fragments. 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.



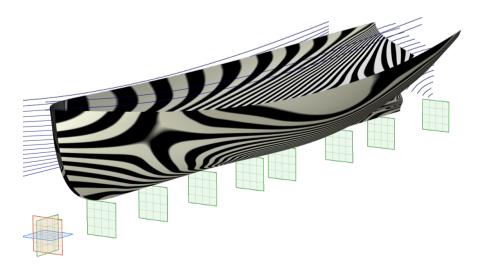
A new remarks mechanism was added for teamwork. The new **Remarks** window enables you to control 2D and 3D remarks - new object types. Remarks are stored in separate files or in the new dataset **Remarks** in T-FLEX DOCs. You can edit remarks in the same way that you edit fragments in the context of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

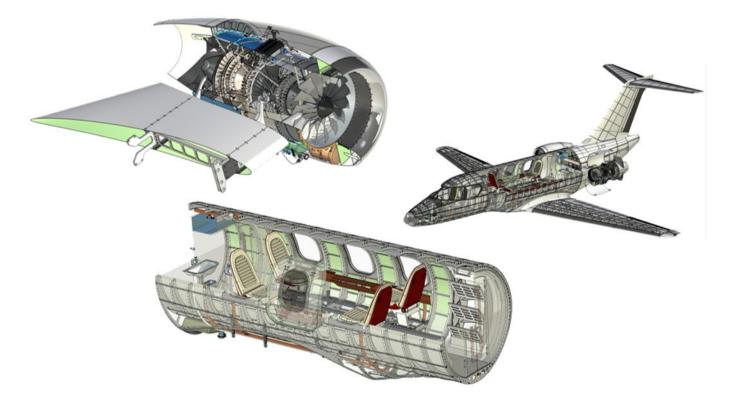
New approach to the loading of assembly fragments significantly improves performance in comparison with the previous versions of T-FLEX CAD and other CAD systems.

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.

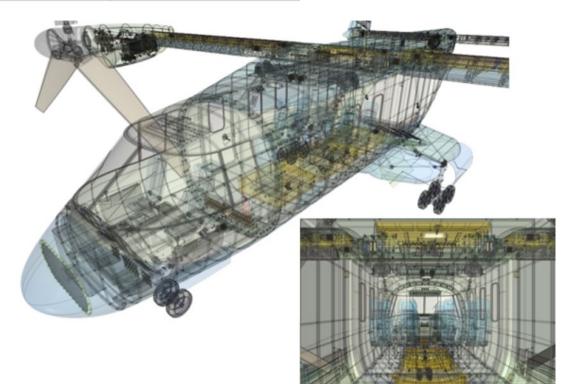
Aircraft model, 48372 bodies.

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.



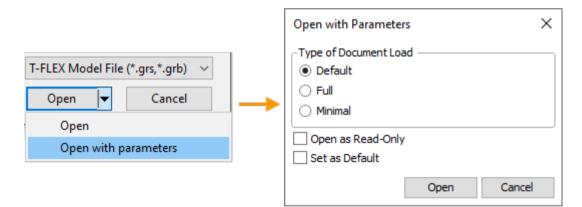
PC specifications on which the assemblies were opened: OS Windows 8.1, RAM 20 GB, SSD (!), Intel Core i5-2500 CPU 3.30 GHz, Video: NVIDIA GeForce GTX 1080.

• 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.

	🇊 3D Fragment_1 (00.00.004-Gearcase.gi	5.	Cancel Section applying			
⊳	🖌 3D Fragment_2 (00.00.003-Worm.grb)	î.	Fix component			
⊳	3D Fragment 4 (00 00 006-Sleeve arb)	× .				
⊳	3D Fragment_6 (00.00.008-Gasket.grb)	a)	Rollback Model			
⊳	3D Fragment_7 (00.00.008-Gasket.grb)		Variables			
⊳	O 3D Fragment_8 (00.00.007-Bush.grb)	Z	Open			
	🗢 3D Fragment_9 (00.00.002-Gearcase co		Load Contents	•		Don't Load
⊳	/ 3D Fragment_10 (00.00.005-Shaft .grb)	<u>۽</u>	Open in Context			Support Geometry
⊳	③ 3D Fragment_16 (00.01-01.000-Worm		Detail			Meshes and Support Geometry
⊳	Ø 3D Fragment_21 (00.00.007-Bush.grb)		Detall		_	
	3D Fragment_22 (00.00.008-Gasket.grl		Additional	•	×	Solids, Meshes and Supporting Geometry
⊳	3D Fragment_25 (00.00.009-Hatch gas	×	Suppress			
Þ	3D Fragment_26 (00.00.010-Hatch.grb		Move to Folder	•		
Þ	Bearing cap_5 (Standard Parts\Bearing		Go to elements			
⊳	Bearing cap_11 (00.00.011-Bearing cap					
К	Rearing can 20 (Standard Parts) Rearin		Open in new window			

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.

		Assembly S	tructure		4 ×
			· • • • • • • • • • • • • • • • • • • •	P ⊕	
		Object 🔺	🔐 🛃 🕼 🔚 🎘 Mass		^
		4 🌋 00.00.00)-Worm ge 🐾 🗂 🛛 🗎 🔭 143.928217 kg	3	
		 ⇒ \$\$\$ 3D Fr → \$\$\$ 3D Fr → \$\$\$\$ 3D Fr → \$\$\$\$\$\$\$\$\$ 3D Fr 	Open file Open with	0000	
ø	Don't Load		Load Contents	•	
Ø	Support Geometry		3D Fragment_1 (00.00.004-Gearcase.grb)		
ø	Meshes and Support Geometry	/	3D Fragment_2 (00.00.003-Worm.grb)		
	Solids, Meshes and Supporting	Geometry	3D Fragment_4 (00.00.006-Sleeve.grb)		
		⇒ Õ 3D Fi	3D Fragment_6 (00.00.008-Gasket.grb)		

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.

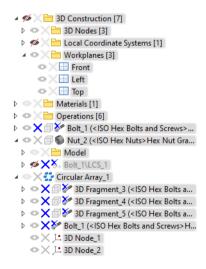
Object	Geometry Source	Get Document Paramet
Reference Face_1	Reference Elements.grb	Reference Elements.grb
Reference Face_2	Reference Elements.grb	Reference Elements.grb
Reference Face_3	Reference Elements.grb	Reference Elements.grb
Copen Geom	2	
Show in Refe	rence Source Document	

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.



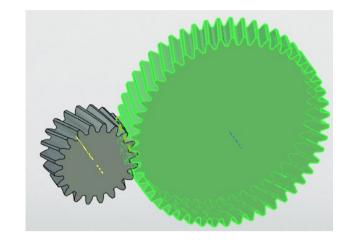
When suppressing fragments or array/copy/symmetry of fragments in the 3D model tree, the suppression immediately applies to all reference elements, "raised" objects and created fragments.



MATES

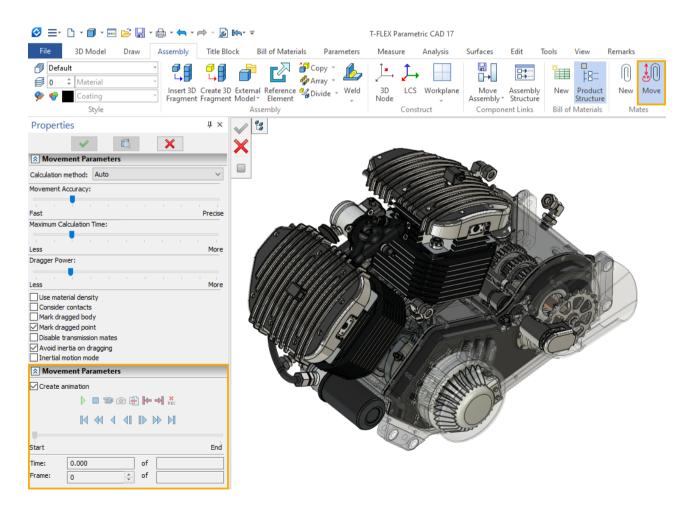
New Types of Mate

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.



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.

WORKPLANE

The command for creating workplanes has a new interface.

Workplane		₽×
× .		
S General Parameters		
	- 4 🛹 🔤	
	l Front and Top	
	l Front and Left	
	<u> </u> Three Standard Views	
Additional Parameters	;	
External		
Show in 3D View		
Show 2D Annotations in 3	D View	
Show all 2D Elements in 3	D View	
Show Lines of Intersection	ns with Workplanes	
Show Coordinate System		
Do not delete in "Purge" of	ommand	
Transparency:	· · · ·	0.9

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.

Workplane	μ ×
✓	
S General Parameters	
Plane: Front	\times
Tangent: O Face_1: Cylinder_1] iii
Offset: 10	- -
Select Direction of Workplane Axis	
Additional Parameters	
External	
Show in 3D View	
Show 2D Annotations in 3D View	
Show all 2D Elements in 3D View	
Show Lines of Intersections with Workplanes	
Show Coordinate System	
Do not delete in "Purge" command	
Transparency:	*

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

Workplane	4 ×	
•	 X 	
A General Parar	neters	
Path:	S 3D Path_1	
Point:	1. 3D Node_1	and the second
Coordinates Type:	🦙 By Offset 👻	and the second sec
Base Point:	🏑 Start 🗸	
Offset:	45	
Select Direct	ion of Workplane Axis	and the second sec
Additional Par	rameters	
External		
Show in 3D View	I Contraction of the second	and the second se
Show 2D Annota	ations in 3D View	
Show all 2D Elen	nents in 3D View	
Show Lines of Ir	ntersections with Workplanes	
Show Coordinat	e System	
Do not delete in	"Purge" command	
Transparency:	0.9	

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 note that all changes from the beginning of drawing will be canceled, and not only those that were on the workplane.

T-FLEX Parametric CAD 17	\times
? Document has been changed. Cancel changes?	
Yes No	
Don't show this message again	

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:

Shell	Ąх	
✓✓		PΞ
General Parameters		X
		đ
Operation: arr Casing WSP465	\times	0
Common Thickness: 3	*	
Faces:		ß
Removed Different Additional		B
Face_2 : Blend_1	^	
	■×	
	\times	I
		L
Pierce Smooth Surfaces		
—		L
Keep Source Body		L
Blend Convex Edges		

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.

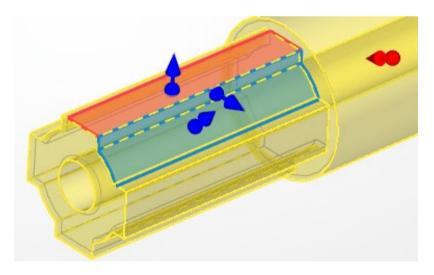
🖻 SI	hell 👻
đ	Shell
	Offset Body

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.

Offset Body	₽×	
🖌 🔍 💐 🗙		
Seneral Parameters		
Operation: 🕜 Extrusion_1	\times	
Offset: 1	*	
Faces:		
Element Offset	~	
Face_5 : Blend_1 29	•×	
Face_6 : Blend_1 21	· 🛛 🔀	
Face_7:Blend_1 25	;	
—		
Face Offset: 29	* *	
Face Offset Direction: Inside	~	
Keep Source Body		

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.

Seneral Parameters	
Operands Selections Faces Matches	
Target Bodies:	
🔗 Blend_1	\times
Tool Bodies:	
Element	
Cylinder_3	
🕜 Extrusion_1	\checkmark \times
🕜 Extrusion_2	

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.

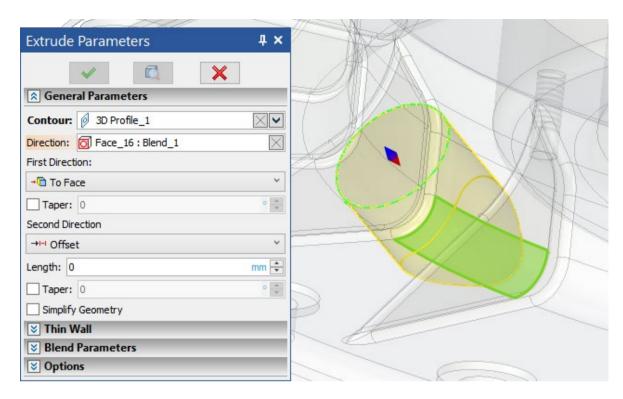
S General Parameters						
Contour:	Ø 3D Profile_1					
	3D Profile_2	■×				
	□ ···	\times				
Axis:	📥 3D Axis_1	\times				
Angle: 🕄	64	*				
Start Angle:	32	*				

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

Rotation Parameters	Ψ×	\checkmark
✓ <a> 		PΞ
Seneral Parameters		~
Contour: Ø 3D Profile_1	imes	$\mathbf{\nabla}$
Axis: d 3D Axis_1	\times	
Angle: 🕄 255	*	Ø,
Rotation Range		0/
First point: Select element		
Start Angle: 0	*	+ Q
Simplify Geometry	_	¥'≬
Blend Parameters	-	4
Source Blend		1 1 1
5	- A	
Target Blend		$\left(\begin{array}{c} 1 \\ 1 \\ 1 \end{array} \right)^{2}$
5	*	4
Fillet		
Radius: 5	- A- - V	
Options		Ě
Use Manipulators		Ø
 ✓ Show Labels ✓ Dynamic preview 		5
Create Driving Dimensions		

EXTRUSION

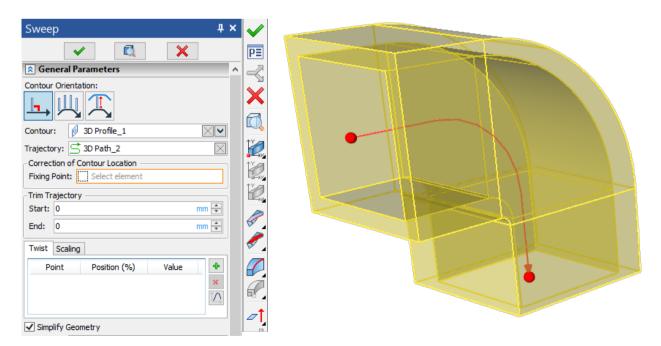
The operation interface has been updated, the ability to control the contours and direction in the operation parameters dialog has been added.



Added ability to select units and Simplify Geometry option.

SWEEP

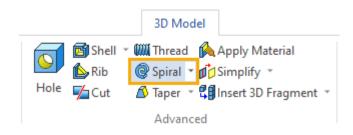
The command interface was improved: selected elements can be managed, the types of created operation are switched using icons.



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.

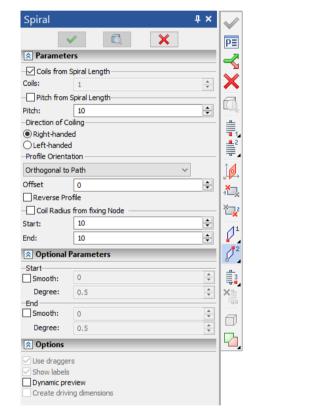
SPIRAL

The operation of creating a spiral has been updated.



The operation works on the basis of a spiral 3D path (see <u>Helix 3D Path</u>), 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

Spiral		4 ×			
	~				
Seneral	Param	eters			
Axis Type:	Ти	vo Points Y			
First Point:	j,	3D Node_1			
Second Poir	nt: 庄	3D Node_2			
Start Point:		Select element			
Start Angle:	0	• *			
Accuracy:	0.0	0001 m 束			
Profile					
Perpendicul	ar to Pa	ath ~			
Contour:		🖉 3D Profile_1			
First Fixing) Point	: 🗇 Vertex_1 : 3D Profile_1			
Second Fixing Point: Vertex_2 : 3D Profile_1					
Scale:		1			
Parameters					
Length and	Coils	×			
Length:	61	A. 			
Pitch:	6.1	A			
Coils:	10	revolution 🖶 🕄			
Distance —					
Diameter		~			
Start dia	meter:	50 🗘 🛃			
End diam	neter:	50			
Options					
Dynamic p	review				

T-FLEX CAD 17

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

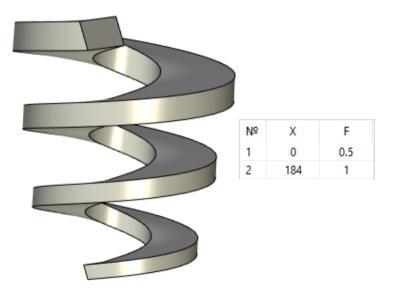
Multiple edge selection is available.

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

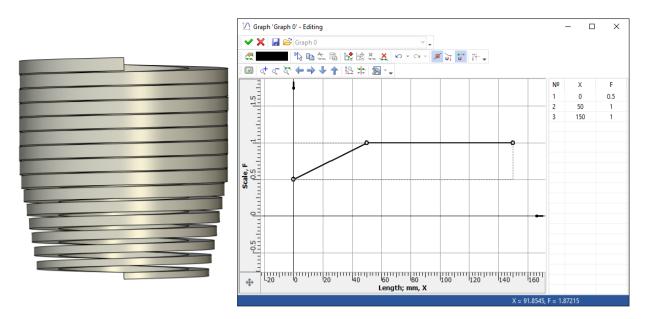
The **Scale** option is an analogue of smooth from version 16. The scale can be set as a constant. A value other than 1 resizes the spiral along its entire length.

	Profile			
	Perpendic	ular to Path		~
	Contour:		3D Profile_1	
	First Fixin	ıg Point:	Vertex_1 : 3D Profile_1	
	Second F	ixing Point:	Vertex_2 : 3D Profile_1	\times
	Scale:		1	÷ 🛃
E C				
	Profile			
	Perpendicular to Path			~
	Contour:	🥖 3D Pro	file_1 🛛 🖂 🗸	÷
	First Fixing Point:	Vertex	_1:3D Profile_1	\square
	Second Fixing Point:			
		_		
	Scale:	0.2	•	*

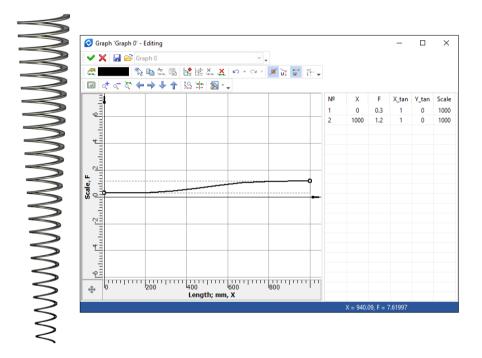
You can select graphs as a scale. A Polyline graph with two points scales linearly from the beginning to the end of the entire length:



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				μ×
	×		5	
🔉 Ger	neral Par	ameters		^
Type:				

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



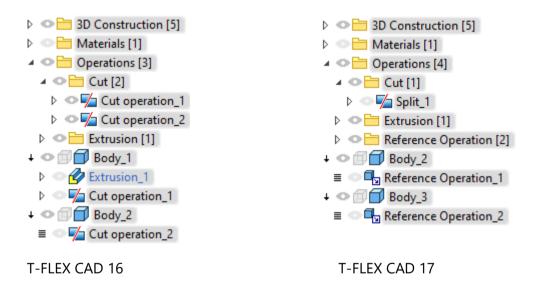
Split - a new type of cutting in which the object is split by the selected type of secant elements, while remaining in the form of single resulting body.



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.



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

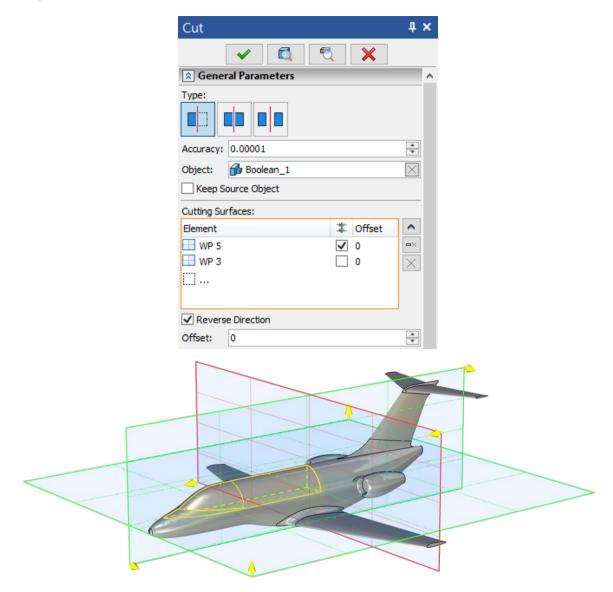
You can now quickly select options for cutting elements in the filter toolbar: **3D Sections**, **Workplanes**, **3D Profiles/3D Paths**. By default, all variants of cutting elements are active.



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.

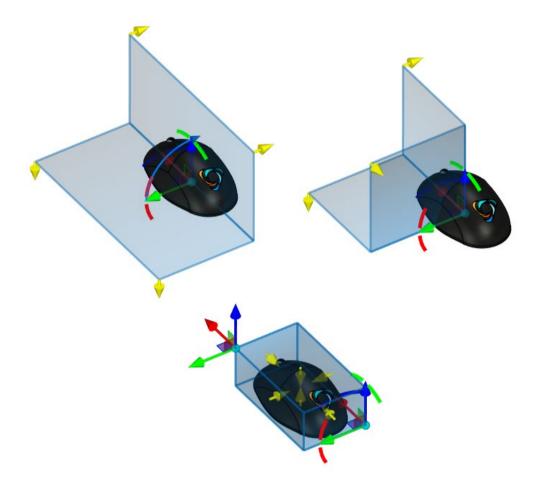
Section 4 ×	Image: A start of the start	Section 4 ×	\checkmark
✓✓	PE	✓	PΞ
Section Parameters		Seneral Parameters ∧	X
Apply to All Elements	~		8
		Apply to:	
	→	· · · · · · · · · · · · · · · · · · ·	_₩←
			-€⇒
			X Y
		Apply Section	
Geometric section		Geometric Section	
Apply section		Color of Sectioned Surfaces: Body Material	

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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.

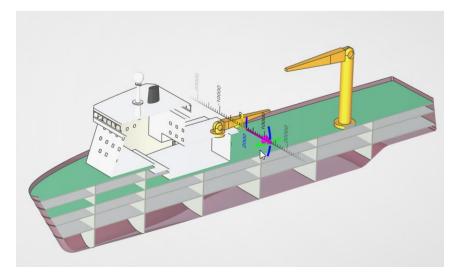
To create sections by plane angle, octant and parallelepiped, the ability to change the direction of the section using the manipulators or a special button in the automenu has been added.

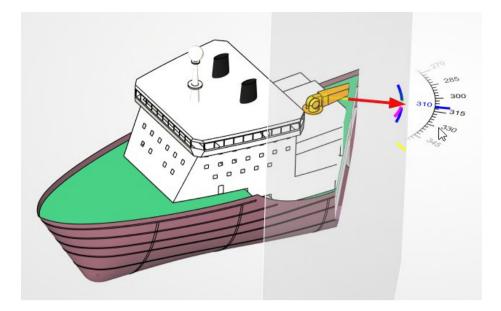


CLIP PLANE

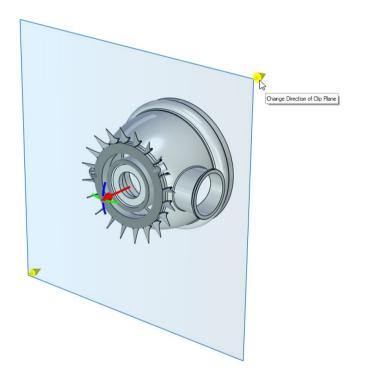
Clip Plane command, which is used to quickly create simple sections by a plane on a 3D model, was improved.

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

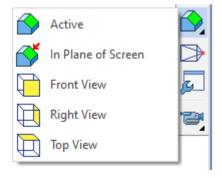




- Added manipulator to change the direction of the clip plane;



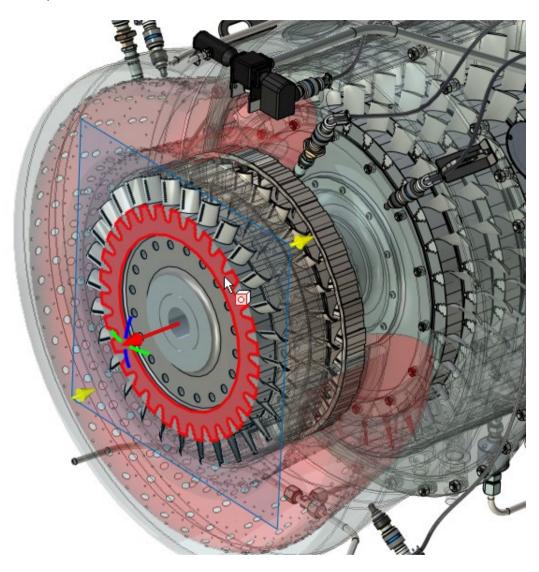
- Added standard views for quick dissection of the model - Front View, Right View, Top View. The section for the Back, Left, Bottom views can be obtained by quickly switching the direction change manipulator;



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.

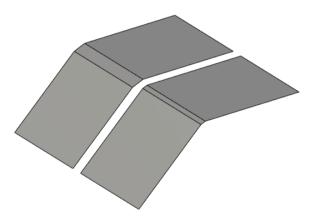
When saving the clip plane - exit the mode with confirmation of creating the clip plane - to move the clip plane, you can use **ctrl** + **shift** + **LMB** - as in previous versions of the system.

BLEND

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

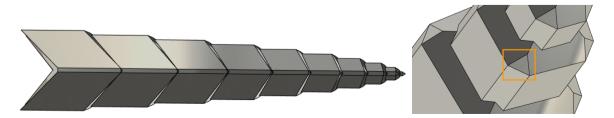
Edge Blend	₽×
 <	
A General Parameters	
Edges Vertices Boundary Trimming	
Element Radius	^
Edge_1:Box_1 [5]	■×
	\times
	1 1
Common Properties	
Offset 1: 5	*
Offset 2: 5	*
Reverse	
✓ Propagate	
Apply Convex Blend First	
✓ Delete Elements	
Curvature-continuous	
S Overflow Processing	
Options	

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



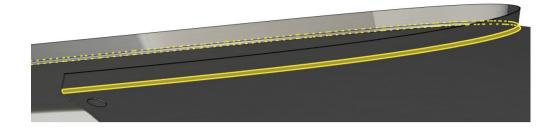
As you can see from the figure, in the new type of chamfer **Offsets along Face** are offset by 5 mm along the edges, while in the old type of **Offset** 5 mm values are converted to other values due to the angle between the edges other than 90 degrees.

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.

end 🛛 🕹 🕇 🗙	Face Blend	
🔍 🔍 🗙	✓	
neters	Seneral Param	eters
nt Width 🗸 🗸 🗸	Mode:	Radius
	Left Faces:	Face_2 : Box
	Right Faces:	Face_3 : Box
	Spine:	S 3D Path_1
all v	Radius:	5
Not selected	Type:	Rolling Ball
Bounds	ShapeConic	
Reverse	Rho: 0=Cirde	
Delete	Bounds Edges: Select	lement
Not selected	Type: Tangent	
Not selected	Limit Planes	
ious Curvature 🗸		element
	Second: Select	
to Wall 🗸 🗸	Notch:	
New Body 🗸 🗸	Tangent:	
~	Accuracy:	0.01
Not selected	Solutions:	All
	Trimming:	Faces to Wall
	Result:	In Left Walls
	Inside Tight Corners	No
	Options ■	
	✓ Dynamic preview	
's: ~		

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

Face Ble	end		4 3	×
[×			
Sener	al Parame	eters		^
Mode:		Radius	~	
Left Faces:		Radius		
		Offset		
Right Faces	s:	Width		
Spine:		Select element		

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.

Spine:	🔿 3D Path_1	\times	
Offset 1:	10	÷ 🛃	
Offset 2:	30	÷ 🛃	
Type:	Rolling Ball	~	
-Shape			
Conic		~	

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:

Spine:	😅 3D Path_1		
Width:	30	* *	
Ratio:	1	* *	and the second sec
Type:	Rolling Ball	~	- Andrew - A
Shape]	

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

Spine:	G 3D Path_1		
Width:	30	* *	
Ratio:	0.5	* *	and the second s
Type:	Rolling Ball	~	and the second se
Shape			

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**.

Shape	
Conic	Ý
Conic	
Chamfer	
Continuous Curvature	
Edges: Select element	~

There are two possibilities for constructing a conical section. The first is with the **Rho** option turned off:

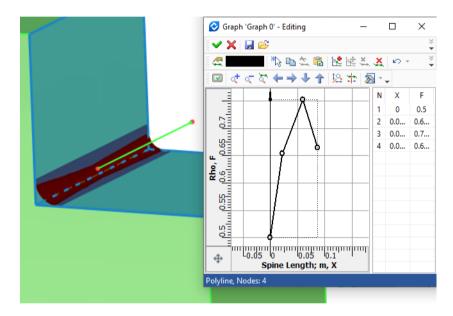
-Shape		
Conic		~
Rho:	0.5=Ellipse	÷ 🛃

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.

Shape Conic	~	
✓ Rho: 0.001	÷ 🛃 🖌	
Bounds		the
Edges: Select element		
Type: Tangent	~	
Shape Conic		
	÷ 🗠	
Bounds		A

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.

Shape -			
Continu	ious Curvature	¥	1
Type:	Depth	~	100
Offset:	0.9	÷ 🛃	1
Depth:	0.7	÷ 🛃	1111

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.

Primitives		ұ ×	
	 ✓ ✓ ✓ 		
General Pa	arameters		
		•	
Diameter:	0	*	
✓ Height:	100	*	
✓ Angle:	45	*	
Diameter 2:	200	*	
Symmetry			
▲ Thin Wall			▲
- Thin Wall			•
Thickness:	No	\sim	
Value:	2	*	
Bottom:	2	*	
Top:	2	*	
Transform	ations	>>	
		4	
		1	
		•	
L			
Options			
Create Drivi	ing Dimensions		

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.

Circular Array	×
✓	
Seneral Parameters	
Nut_4 (<iso hex="" nuts="">Hex Nut G Array of Fragments</iso>	^
Nut_13 (<iso hex="" nuts="">Hex Nut Grade C ISO 4034.g</iso>	■×
Bolt_3 (<iso and="" bolts="" hex="" screws="">Hex Bolt ISO 401</iso>	\times
Alternative Face Selection	

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,

array/copy/symmetry are fully functional fragments. This allows you to apply, for example, transformation exploded view scenario, and other operations to selected elements of arrays/ copies/symmetries.

🕨 👁 🗊 🎒 3D Fragment_1 (Body.grb)
Image: State of the state of
Image: Second
🖻 👁 🗊 👕 3D Fragment_4 (Bush.grb)
🕨 👁 🗊 🌑 3D Fragment_5 (Ball.grb)
🕨 👁 🗊 🚱 3D Fragment_6 (Seat.grb)
🕨 👁 🗊 😳 3D Fragment_7 (Seat.grb)
👂 👁 🗊 🧆 3D Fragment_8 (End Cap.grb)
👂 👁 🗐 🧇 3D Fragment_9 (End Cap.grb)
🕨 👁 🗊 🖍 3D Fragment_10 (Handle Subassembly
Image: Solt_3 (<iso and="" bolts="" hex="" screws="">Hex</iso>
▷ 👁 🗊 懮 Nut_4 (<iso hex="" nuts="">Hex Nut Grade</iso>
Image: Second
🔺 🗢 🛟 Circular Array_1
Image: Stranger of the stra
Image: Stranger St
Image: Stranger of the stra
Image: Solid State Solid State Solid State St
.1. 3D Node_1
I 3D Node 2

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.

					(
👖 Paramet	ters	:	<				
Variables:							
Name	Comment	Expression					
\$Finish		"Was 🗸 🖞					
d		12 🗸 🖞					
L		45 🗸 🖻		∄≿∏⊴≬	🄊 🗇 🗇 🖉	2 🗐 🔫 🛛	PE 🛛 »
res		0 6					
Variables I	link						
Preview	List						
	OK	Cancel					
			~				
🛷 Linear /							
	ragment_3 (<iso bolt<br="" hex="">ragment_4 (<iso bolt<="" hex="" th=""><th></th><th></th><th></th><th></th><th></th><th></th></iso></iso>						
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	ragment_6 (<iso bolt<="" hex="" th=""><th></th><th></th><th></th><th></th><th></th><th></th></iso>						
	ragment_7 (<iso boli<="" hex="" th=""><th></th><th></th><th></th><th></th><th></th><th></th></iso>						
	ragment_8 (<iso bolt<="" hex="" th=""><th></th><th></th><th></th><th></th><th>H</th><th></th></iso>					H	
👂 🏈 3D Fr	ragment_9 (<iso bolt<="" hex="" th=""><th>ts and Screws</th><th></th><th></th><th></th><th></th><th></th></iso>	ts and Screws					
Þ 🚺 3D Fr	ragment_10 (<iso plain="" th="" w<=""><th>/ashers>Wash</th><th></th><th></th><th></th><th></th><th></th></iso>	/ashers>Wash					
D 3D Fr	ragment_11 (<iso plain="" th="" w<=""><th>/ashers>Wash</th><th></th><th></th><th></th><th></th><th></th></iso>	/ashers>Wash					
D 3D Fr	ragment_12 (<iso plain="" th="" w<=""><th>/ashers>Wash</th><th></th><th></th><th></th><th></th><th></th></iso>	/ashers>Wash					
Þ 🔇 3D Fr	ragment_13 (<iso plain="" th="" w<=""><th>/ashers>Wash</th><th></th><th></th><th></th><th></th><th></th></iso>	/ashers>Wash					
	ragment_14 (<iso plain="" th="" w<=""><th></th><th></th><th></th><th></th><th>V U</th><th></th></iso>					V U	
Þ 🔾 3D Fr	ragment_15 (<iso plain="" th="" w<=""><th>/ashers>Wash</th><th></th><th></th><th></th><th></th><th></th></iso>	/ashers>Wash					
Þ 🔾 3D Fr	ragment_16 (<iso plain="" th="" w<=""><th>/ashers>Wash</th><th></th><th></th><th></th><th></th><th></th></iso>	/ashers>Wash					

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.

Circular Array 4 ×	
✓	
Seneral Parameters	
Rotation (Rows)	
Parameters: Number of Copies and Total Angle	
Copies: 3	
Step 120	
Angle 360	
Parallel Offset	
Half-Step Shift	
Reverse Direction	
Symmetric	
Radial Translation (Columns)	
Parameters: Number of Copies and Step v	
Copies: 2	
Step 187 👻	
Length 187	€•• €•
Half-Step Shift	
Reverse Direction	
Symmetric 🗸	
Axial Translation (Height)	09
Parameters: Number of Copies and Step	
Copies: 3	
Step 400 👻	
Length 800 🗘 🐼	
Half-Step Shift	
Reverse Direction	
Imitations	
Exclusions	
🔢 Var 🕂 Dia 🜉 3D 🎫 Mo 🙀 Lib 🚰 Pro	

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.

Linear Array	4×
✓✓✓	
Seneral Parameters	
Extrusion_1	^
	=×
	\times
↓ ↓	
Layout: Row Shift	~
A First Direction (Rows) ←	
Direction/First point: Direction/First point: Direction/First point:	\times
Parameters: Number of Copies and Step	×
Copies: 4	÷
Step: 85.5	÷ 🕅
Length: 256.5	
Half-Step Shift	
Reverse Direction	
Symmetric	~
Second Direction (Columns)	
S Third Direction (Height)	
Eimitations	
 ✓ Exclusions ✓ Options 	

ARRAY BY PATTERN

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

	3D	Mod	el			
🏟 🄗	🕝 Boolean 👻 🔏 Loft	_	opy D Symi	metry		🔂 Shell 🖒 Rib
Rotation Blend	🕅 Sweep 🔹	% A	rray *		Hole	Cut
Ō	perations	*	Linea	r Array		1
		88	Circu	lar Arra	iy	
		6 6	Array	by No	des	
		:2	Array	by Pat	h	
		82	Paran	netric A	Array	
		**	Array	by Pat	tern	
		₽ ₽	Array	by Tab	le	

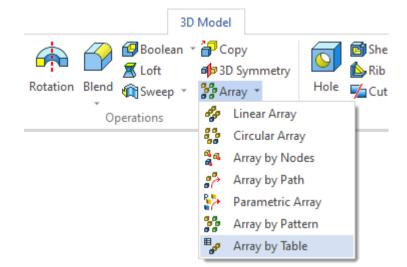
You can select any array as a pattern array.

Array by Pattern	ł ×	\checkmark	$\times \Leftrightarrow \Rightarrow \textcircled{1} \ominus \boxdot$	% - ≪ ∃
 ✓ ✓ ✓ 		₽≣	▲ [●] 3D Construction [3]	
General Parameters	-		Workplanes [3]	
	_	$ \mathbf{X} $	Geometric entities [5]	
			Materials [1]	
			Operations [5]	
G Hole_2	\mathbf{h}		+ • 🗊 🗐 Body_1	
	■×		Cylinder_1	
	\times	đ	▷ 🗃 Shell_1 ▷ 🖸 Hole_1	
		-	I and the second sec	
			▷ O Hole_2	
Alternative Face Selection:	·			
Pattern: 🌮 Linear Pattern_1 🔶	- 🖂			
Options				
Copy to one Face			•	
Extend Faces				•
Check Intersection with other Faces			•	
✓ Don't check body boundaries			•	
			•	
			•	
			•	
				<i>F</i>
			0	
			•	
		$ \rangle$		
		`		

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.

Array by Table	9		Д ×	:		
×		×				
🕿 Array by Table	2		_			
Carew_1 (<isc Mut_2 (<iso h<="" th=""><th></th><th></th><th></th><th>-</th><th></th><th></th></iso></isc 				-		
🖁 Washer_3 (<is< td=""><td></td><td></td><td></td><td>il 🧲</td><td></td><td></td></is<>				il 🧲		
Uasher_4 (<is< td=""><td>O Plain Washers:</td><td>>Washer N</td><td>orm</td><td></td><td></td><td></td></is<>	O Plain Washers:	>Washer N	orm			
Database:	a		~			
Number of Copie	es: 1		*			5
Parametric Arra						
Coordinate System	1 ———					
Cartesian			Ý			
Source CS:	Auto					
Target LCS:	Global CS					
coordinate:	X_coord		Ý	1		
/ coordinate:	Y_coord		Ý			
Z coordinate:	Z_coord		Ý		67	
Angle around X Axi	s: [not set]		~			
Angle around Y Axi	: Angle_arour	nd_Y	Ý			
Angle around Z Axi	s: [not set]		Ý			
a		1	1	1	1	1
Ng	X_coord			Angle_around_X	-	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.

Array by Table 4	×						
Array by Table							
Washer_4 (<iso plain="" washers="">Washer Normal Grade A ISO</iso>	^						
	■×						
Nut_2 (<iso hex="" nuts="">Hex Nut Style 1 ISO 4032.grb)</iso>	\times						
Bolt_1 (<iso and="" bolts="" hex="" screws="">Hex Bolt ISO 4016.grb)</iso>							
Database: a ~							
Number of Copies: 1	*						
✓ Parametric Array							
Variable: d (Washer_3 (<iso plain="" washers="">Wa Y Column: Y_coord Y</iso>							
Variable: res (Nut_2 (<iso hex="" nuts="">Hex Nut S 🗸 Column: Z_coord 🗸 🗸</iso>							
Variable: [not set] · Column: [not set] · ×							
Add Variable							

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.

Corner		₽×
	× 🔍 🗙	
S Opera		
🔉 Gener	ral Parameters	
- Bend Re	elief	
Corner:	Round	~
Isolating:	[No]	
Extend	Relief	
Depth:	[3] 🗘 Width: [3]	×
Gap type:	Without overlap	~
Gap:	[0.1]	*
	ns	
Dynami	c Preview	

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You can now select boundary faces in an operation.

Operands	
	×
Face_2 : Flange_2	

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.

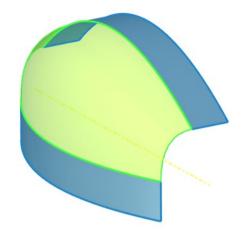
Jog 🕈 🕇 🕹	
 ✓ ✓ ✓ 	
Operands ^	×
3D Profile_1	
×	
	120
S General Parameters	P 22
Radius [2] mm 🔹	
Height 🕇 70 mm 🛨	
Angle 🥑 90 • 🛨	
Offset mm +	
Position Vene vene vene vene vene vene vene ven	
Height Reference External Dimension Y	
Fix Projection	
Boundary	
✓ Jog face completely	
Bend Relief: No ~	
Depth: [3] mm 👘	
and the second sec	

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** group.

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.

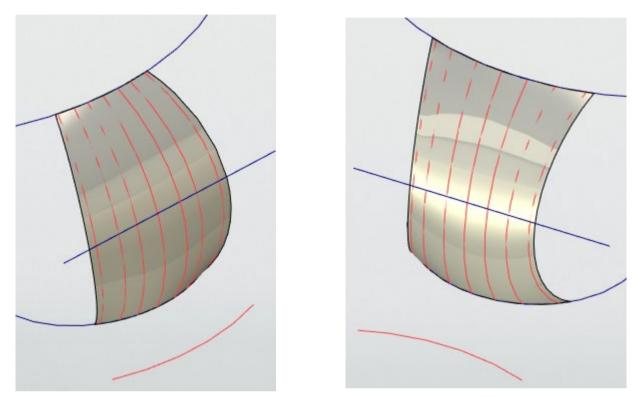


The command contains ten options for constructing transition surfaces and will be developed. Depending on the selected option, the edges of the surface can be set as curves or be free. In this case, the shape of the surface edge can be set by a graph.

$\overline{\mathbf{A}}$	Conic – Discriminant
	Conic – Path
	Conic – Tangent
\supset	Conic – 4 Points
$\overline{\bullet}$	Circle – Tangent to Surface
	Circle – Radius
R R	Circle – Radius and Angle
	Circle – Three Points
-•~•	Line – Angle
\mathcal{I}	Line – Tangent

Surfaces

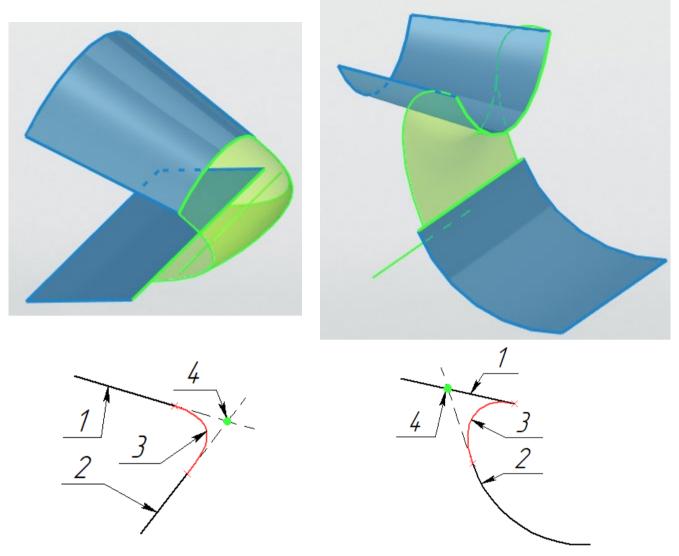
The basic element of any construction option is the reference curve. The reference curve can be defined by multiple smoothly conjugated edges, profiles, a trace, or a 3D path. The reference curve determines the direction of the tangents. The planes are located along the normals to these tangents. The 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.



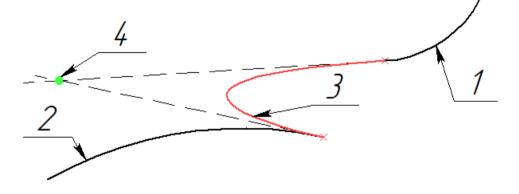
Influence of the reference curve on the location of invisible forming profiles

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.



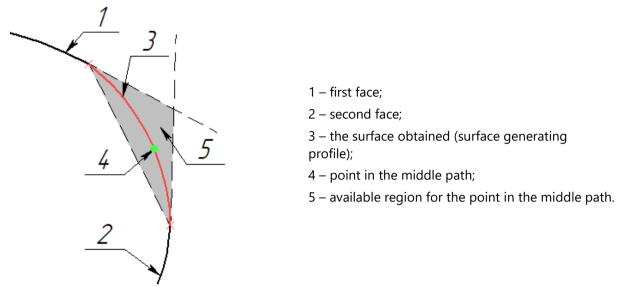
1 - first face; 2 - second face; 3 - the surface obtained (surface profile); 4 - tangent lines intersection point.



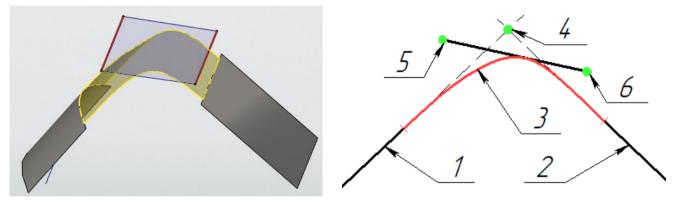
Conic – **Discriminant**. It creates a conical surface that has the form of a smooth rounding between two curves that lie in the specified surfaces. The degree of rounding of the resulting surface is determined by the **Discriminant** parameter. The parameter varies can be set from 0 to 1 (not including 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.

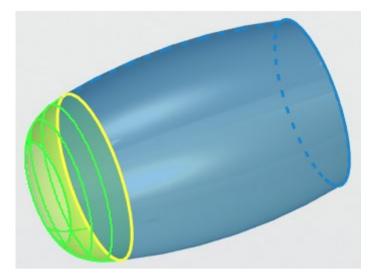


Conic – Tangent. It creates a conical surface that has the form of a smooth rounding between two curves that lie in the specified surfaces. The shape of the surface rounding is determined by another surface (forming surface) defined by two curves. The surface to be created must be tangent to this surface. 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.

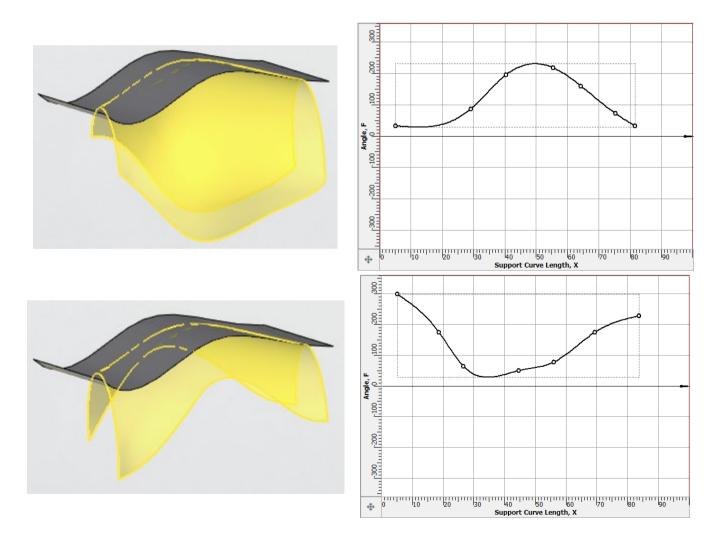


1 – first face; 2 – second face; 3 – the surface obtained; 4 – tangent lines intersection point; 5 – first reference curve point; 6 – second reference curve point.

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.



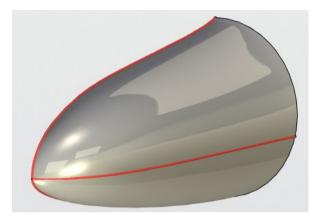
Circle – Tangent to Surface. Creates a surface that is tangent to the specified surface. The cross-section of the surface has the shape of a circular arc with a specified angle relative to the point of contact, the center of the arc lies on the specified curve. The arc angle can be constant along the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



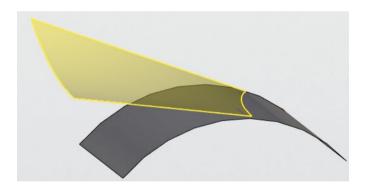
Circle – Radius. Creates a closed surface, which forming profiles are circles of a given radius. The radius can be constant or variable. The position of the forming profiles is determined by the reference curve. The center of the circles is determined by the specified curve.

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.

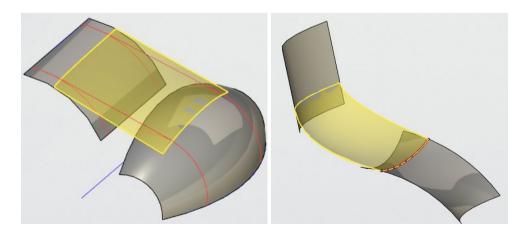
Circle – Three Points. Creates a surface that passes through three specified curves. The cross-section of the surface has the shape of a circular arc. The arc is created based on three points of intersection of the invisible profile plane with three specified curves. Curves can meet at a single point.



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.

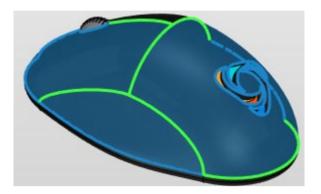


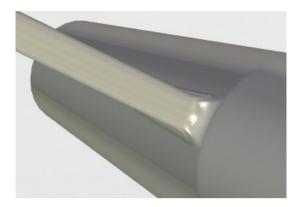
Line – Tangent. Creates a surface whose forming profiles are segments. Each segment will lie in a plane perpendicular to the reference curve. A segment will be tangent to two curves formed at the intersection of the plane, in which the forming segment lies, and the two surfaces specified by the user.



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.

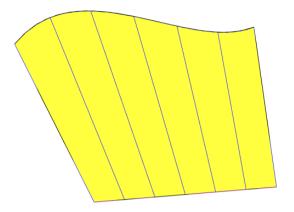
	Ruled Surfa	се	₽×	
	× .		×	
	A General Par	ameters		
	First Section:	j≞ 3D Node_1	\boxtimes	
	Second Section:	🖉 3D Profile_1	\times	
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There is an opportunity to change the direction of the entire section. When you select the second section, the first one will automatically reverse.

Sections can be smooth or with jogs.

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

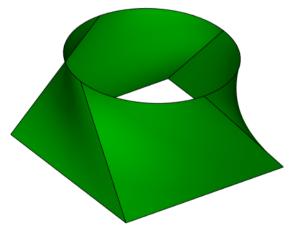
The essence of the **By Arc Length** type is the distribution of isoparametric curves, i.e. the intervals between adjacent isoparametric curves are equally related to the total length.



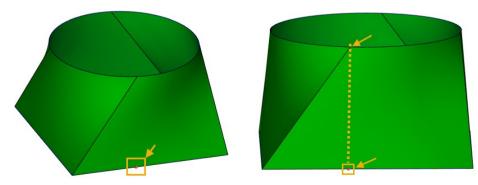
There is possibility to create surfaces between closed contours, provided that both contours are closed. In this case, fields will appear that allow you to set the start of the first and second sections.

Alignment	
By Arc Length	~
Start Point of First Section:	Select element
Start Point of Second Section:	Select element

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.

	4 ×	_	
✓ 🔍	×		
S General Parameters			
First Section: 😥 3D Profile_2			
Second Section: 😥 3D Profile_3			
Alignment			
By Support Curve	~		
3D Profile_1	\times		
Accuracy: 0.01	mm 🖨		
Simplify Geometry: 🗸			
Options	/		
✓ Dynamic preview	/		

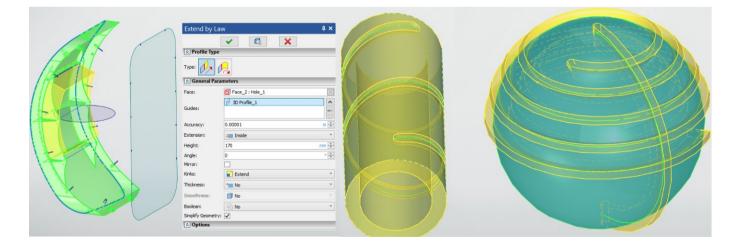
There is also possiblity to specify closed sections with jogs.

EXTEND BY LAW

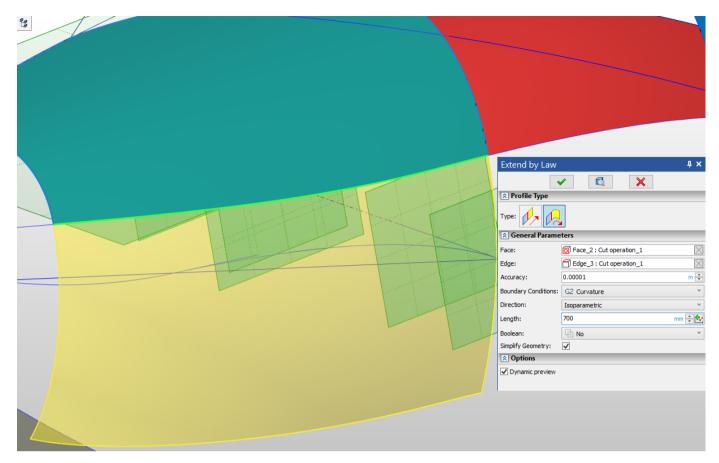
The command is designed primarily for surface modeling, but it also enables you to work with solid geometry. This command enables you to build a surface with one edge defined by a guide curve that lies on a face.

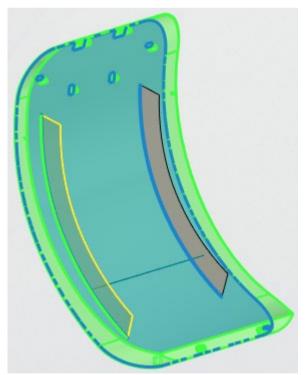
Two types of profile are available in the command: Line and Circle Arc.

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.



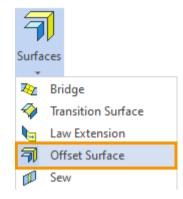


This command enables you to create surfaces, along a guide curve of any complexity. This is useful when working with body parts that have complex spatial geometry. You do not need to perform a complex sequence of operations. It is only required to project the desired contour on the face or create a guide curve using the wire geometry commands and create the surface in one operation.

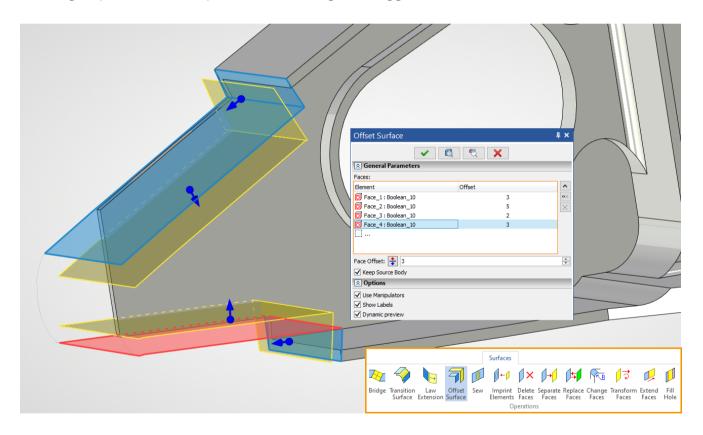
In solid modeling, the command is useful for creating grooves obtained by milling. The shape of the groove is set by the guide curve, and the width is set by the thickness parameter. If you specify the **Thickness** parameter, a solid body will be created instead of a surface. To create a cavity, specify 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.

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.



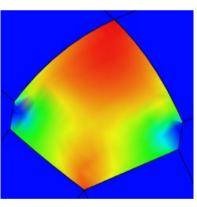
A new dialog allows you to manage the set of selected items: remove them from the list, add them, and separately edit parameters for each item or group of items.

AREA FILLING

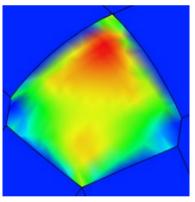
The command has a new dialog style. All selected edges are displayed in the editable list. For each edge, you can define the type of the conjunction of surfaces (boundary condition): G0, G1, G2. By default, the G2 conjunction is created. The **To All** button enables you to set the selected type of boundary condition for all other edges.

Bound Edges:					
Element Ty			pe		^
🗇 Edge_13 : Face R	Edge_13 : Face Removal_1		G0		■×
🗇 Edge_14 : Face R	emoval_1		G0		\times
🗇 Edge_15 : Face R	emoval_1		G1		
🗇 Edge_16 : Face R	emoval_1		G1		
Edge_17 : Face Removal_1			G2		
Edge_18 : Face Removal_1			G2		
Edge_19 : Face Removal_1			G2		
Boundary Condition:	G2		~	΄ Τ	o All
Minimize Topology	G0				
Reverse G1					
✓ Attach to Source BG2					

You can analyze the smoothness of the transition between surfaces by using the **Surface Curvature** command, which displays the change in surface curvature.



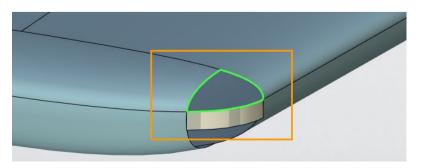
G1 is set for all edges



G2 is set for all edges

The pictures show that the curvature jumps under the boundary condition G1. And under the boundary condition G2, the curvature function has no discontinuities (except for places that were obtained using other operations: smoothing, contour extraction, etc.).

Now you can create a filling surface between edges of different bodies using the command. This simplifies the creation of connecting surfaces between bodies and extends the capabilities of surface modeling.



WIRE GEOMETRY

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

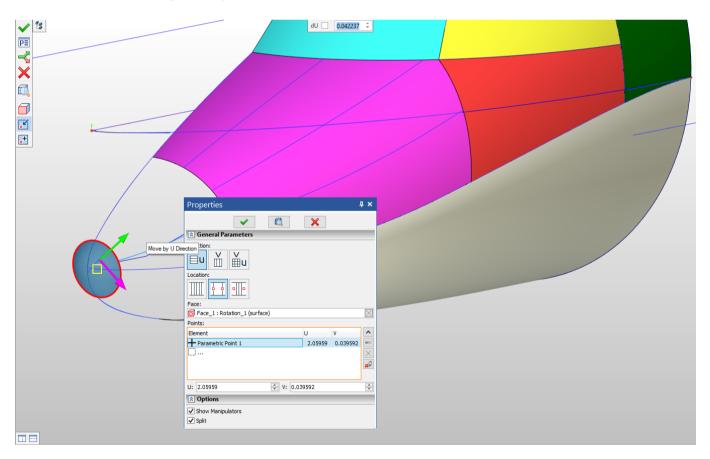
3D PATH

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.

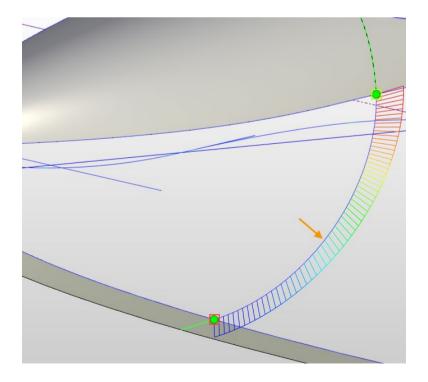
The new **Conic Curve** 3D path option was added. It has six options for creating a conic curve. You can select an option by clicking the corresponding dialog icon.

\rightarrow	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
$\mathbf{\overline{\mathbf{x}}}$	By 4 Points and Guide

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

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	~

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.

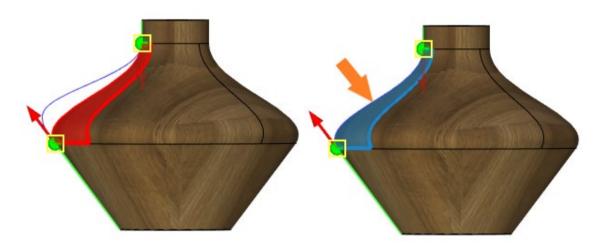


Transition Path

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.

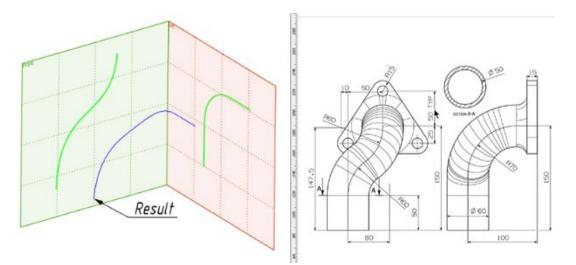
Continui	ty:			
G0	G1	G2	G3	
Tangent	magnitu	de 🕶	[1]	

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



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
181	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.

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Compound 3D Path	
Elements:	
Edge_10 : 3D Fragment_16 (00.01-01.0	
Edge_27 : 3D Fragment_2 (00.00.003	MAR AND A VIME
🗇 Edge_34 : 3D Fragment_9 (00.00.002-G	
🗇 Edge_36 : 3D Fragment_1 (00.00.004-G	
🗇 Edge_37 : 3D Fragment_1 (00.00.004-G 📇	
□ ➡	Ch Last The last
Reverse Direction	SA TRAFFIC P
Find Transition Sequence:	
<i>#</i> 4	TO NIT OF DUE

Modification

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

\$3	D Path 👻	
J	Route	
Ŝ	3D Path	
W	Helix 3D Path	
÷	3D Axis	
X	Change 3D Path Length	
æ	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.

🙎 Length (hange		
Mode:		_	
Extension Le	ength		
Symmetric:			
Extension at Start:		100	mm
Extension at	End:	150	mm
Start Shape:	∫ ^t \ N	atural	¥
End Shape:	∫ ^{I−} Lir	near	¥

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



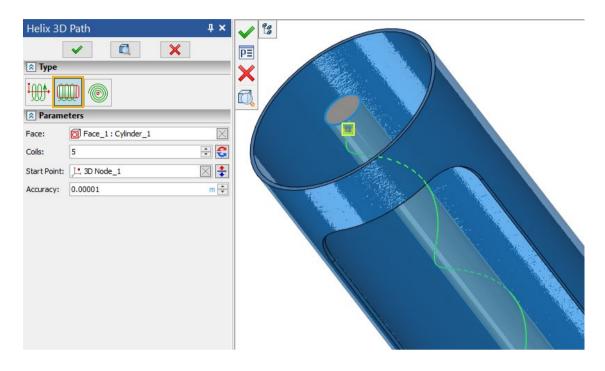
When you call the modification command again, the parameters of the previous modification will already be set for the path in the dialog. You can change the modification parameters, or you can delete one of the two modification options - or both modification options at once.

Modification	ų×	
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Seneral Parameters		X
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ModifyPath3dAuto.Modifications	»	
Rounding	+	
	-	
	+	
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		ATE 21 ON
1		
		AND FILE A
Rounding	_	
Rounding Radius: 100	mm	

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.

Helix 3D Path 4 ×	
 ✓ ズ 	
Я Туре	
S General Parameters	
Axis Type: Two Points ~	
First Point: 1 3D Node_1	
Second Point: 1 3D Node_2	
Start Point: Select element	(((💷))))))))]]
Start Angle: 0 •	
Accuracy: 0.00001 m 🛓	
Parameters	
Pitch: 1.75	
Coils: 10 revolution 🗧 🚭	
Distance	
Diameter	
✓ Start diameter: 15	
✓ End diameter: 50 +	

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:

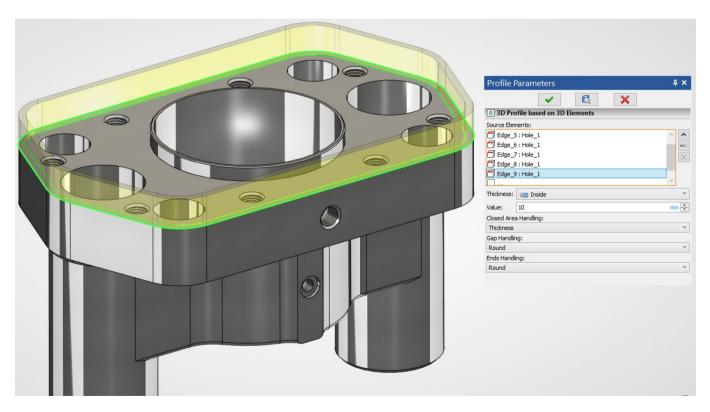
- 3D Profile by 2D Elements
- 3D Profile by 3D Elements
- 3D Profile Projection
- 3D Profile by Unfolding

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.

Ø 3	D Profile 💌
Ø	3D Profile
02D	3D Profile by 2D Elements
BaD	3D Profile by 3D Elements
品	3D Profile Projection
3	3D Profile by Unfolding

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.

Planar Profile:	
Ø 3D Profile_1	\times
Reverse Direction	
Direction or Point:	
Face_2 : Cylinder_1	\times
Faces or Body:	
Cylinder_1	^
	■×
	$\overline{\mathbf{X}}$

3D Node

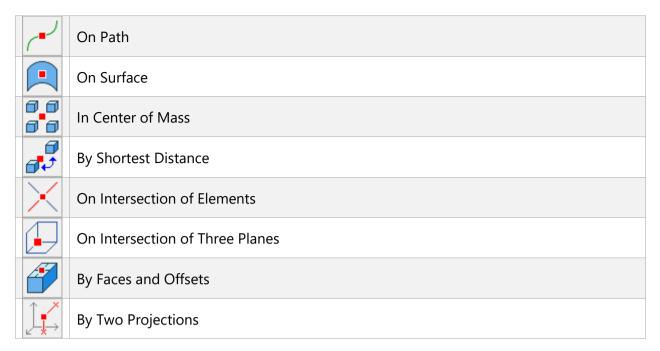
The command dialog was updated. Options for creating a 3D node are divided into 9 types. Each type has a corresponding icon in the dialog that is duplicated in the command automenu.

Properties	Ψ×	
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Туре		PΞ
	$\begin{array}{c} \uparrow & \swarrow \\ \downarrow & \checkmark \\ \downarrow & \checkmark \end{array}$	X
Coordinates		Ľ,
X: 0	mm 🜲	~
Y: 0	mm 🚔	
Z: 0	mm 🜲	
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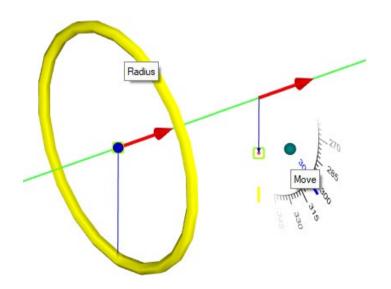
As in all renewed dialogs, creation modes can be selected by clicking the icon at the top of the dialog.



In Coordinates



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.

Properties 4 ×
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🗒 🏲 🖪 📽 🎜 🔀 🗗 🍠 🎼
S On Intersection of Elements
First Element:
📑 3D Path_1 🛛 📈
Second Element:
🗇 Edge_2 : Box_1
Edge_4: Box_1
Edge_5:Box_1
Intersection Point: 0

Bindings are now available for 3D node manipulators.

2D DESIGN

Working in a 2D scene has become even more convenient. Added a command for creating a 2D array from a table. 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

Work has begun on updating the **Projection** command. The command has been substantially redesigned and transferred to a new interface. Added new types of isometric projections. 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.

Projection	4 ×						
~							
Seneral Parameters							
	* Scale: [1]						
Perspective Projection Consider Hidden Lines Options							
Update:	Manually and On Full Regeneration 💙						
Calculation of Bodies Penetration:	No						
Exploded View Scenario:	· · · ·						
Configuration:	No						
Precision:	[Default]						
Options							
Options Image: Convert Polylines to Splines Convert Polylines to Splines Simplify Geometry Save Information for 3D Dimensions Create Outline Area Last Version of Projection Algorithm Geometry Search							
Welds: Consid	ler and Fill 🗸 🗸						
Minimum Size of Elements: 0	* *						
Opposite Side View							
Very Projection Elements							
➡ Lines							
Sections							
🚹 Diagno 🛛 🚛 3D Mo 🛛 📰	Model 📴 Library 😭 Proper						

A more detailed description of the command changes is expected later.

SPLINE DRAWING

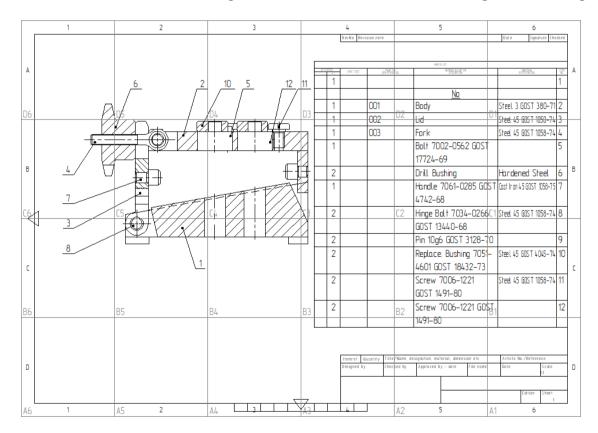
Spline command now supports a continuous curve input holding the left mouse button pressed, i.e. implemented is the feature of spline painting.



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.



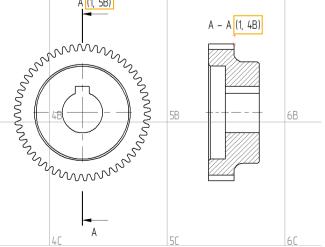
In the document parameters, you can now select numbering within one page or consecutive horizontal numbering as a detection of zones for bill of materials.

Page: Active (Page 1)	 Ÿ Filter	
Document	/ Zones	
▶ 3D	Detect zones for bills of materials	No
2D Fragment	Visibility	No
3D Fragment	Show first symbol	Numbering within One Page
Large Assembly Management	Draw Last	Consecutive Horizontal Numberin
Performance	Color	🔲 Dark Gray 😽
External Variables	▲ Number	
Naming Rules	X	6
Visibility of Pages	Y	4
Save	▲ Direction	
All	Horizontally	From Right to Left 🔹
Page (Page 1)	Vertically	From Bottom to Top 🔹
▲ Paper	▲ First Char	
Zones	X	1
E		

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:

Style Font Common Dimensions Font	
Text Text Parameters Text over Line: B - B Offset: 4 Auto Auto Additional Text:	
Zone and Sheet: Don't show Zone and Sheet: Don't show Don't show	~
Text under Line: Part Number Zone Name Zone and Sheet Position: Don't show Sheet Number of Part Number Sheet Number and Part Number Zone Name Offset From Arrow: Sheet Number of Image Coordinates Sheet Number and Part Number Zone Name Offset From Arrow: Sheet Number and Image Zone	Zone Name
X: 0 V: 0 Offset Along Arrow: Auto Underline Text Line Width: [0.1] Standard: Default V	
Level: 0 Image: Default Image: Default Image: Default Priority: 0 Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default Image: Default	~
Set as default OK Отмена Set as default OK C	Отмена



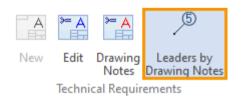
Also, synchronization between the division of the page into zones and zones of title blocks is implemented.

		T-	FLEX Parametri	c CAD 17		×				
	Zones visibility has been changed through "Document Parameters". Change "Show Zones" parameter of title block?									
			Don't show	this message	again	Yes	No			
V	ariables								Ą	×
	Basic paramete	rs								
	Itemref Qua	antity	Title/Name, d	esignation, m	material, dimension etc Article No./R			Reference		
	Designed by		Checked by	Approved by	y - date	File name	Date	Scale		
							~		1:1	
							Edition	Sheet		
									1	
	Additional parameters									
	Color: 0 V Show zones Show revision note									
	List of Variable	ec								

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

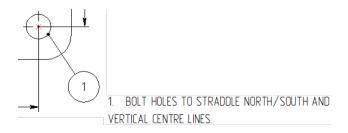
NOTES ON TECHNICAL REQUIREMENTS

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



Drawing Notes							
ß	🗅 Document Technical Requirements 🛛 🕹						
#	Text						
1	1. BOLT HOLES TO STRADDLE NORTH/SOUTH AND VERTICAL CENTRE LINES.						
2	2. DIMENSIONAL TOLERANCE 1mm						
3	3. ANGULAR TOLERANCE 1mm						
4	4. PIPE WELDING TO ANSI B31.3						

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.

Properties		ą×			C	+		
Objects selection	1							
+ _≡ Add		~]					
마] Fragment 0xD0000	14 (<iso and="" bolts="" hex="" screws="">Hex Bolt I</iso>]					
		■×						
		\times			ſ			
Array by Table								
Database:	a	~						
Number of Copies:	1	*				+		
Coordinate System —								
Cartesian		~						
Source CS: Page	Coordinate System							
Target LCS: Page	Coordinate System							
X coordinate:	X_coord	~	j I					
Y coordinate:	Y_coord	~						
Angle:	[not set]	~		а				
Scale:	Scale	~]	Nº	X_coord	Y_coord	Angle	Scale
	Add Variable]	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.

Add Variable								
Variable:	P *	Column:	[not set]	Ý				
	[not set]		[not set]					
	Р		X_coord					
			Y_coord					
			Angle Scale					
			Scale					

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).

•	 Applicable standards/speci Dimension and Tolerances Cast fillets to be <u>R0.9</u> and noted * Dimensions according to specific terms 	Cor	ner Radii <mark>R</mark>	_			
L		А	<u>F</u> ont	Ctrl+F2	1		
		≣¶	<u>P</u> aragraph	Ctrl+F3			
		Ж	Cut	Ctrl+X			
		Ēþ	<u>C</u> opy	Ctrl+C			
		Ē	<u>P</u> aste	Ctrl+V			
			<u>D</u> elete	Del			
			Select <u>A</u> ll	Ctrl+A			
			Insert	*		<u>T</u> ext	F6
			<u>S</u> how Symbol	> >	S	Hyperlink	Alt+F6
		4	-	,	VAR	<u>V</u> ariable Product Structure Position Number	F8 Alt+F7
		8	Check Spelling	Ctrl+F11	5	Position Number (T-FLEX DOCs)	Alt+F8
					5	BOM Position Number	Alt+F10
)=д	Technical Requirements	Alt+F11
					A_2^1	Index	Alt+F12
					A1	Index	
					<u>123</u> 456	Fraction	Ctrl+F12
					123 430	Fraction	
					~	Radical	
					5	Radical	
					₹.5∕	<u>R</u> oughness	Ctrl+F8
					₽	<u>G</u> D&T	Ctrl+F9
					머	<u>F</u> ragment	Ctrl+F10

T-FLEX CAD 17 and Add-on Modules. Release Notes

Hyperlink Parameters 4 ×	Hyperlink Parameters 4 ×
Type: → C:\ → ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓	Type: Image: C:\Program Files\T-FLEX Parametric CAD 17\Libr File: =C:\Program Files\T-FLEX Parametric CAD 17\Libr Text: <lesson -="" 3="" 3d="" assembly="" model="">_Ball Valve.grb</lesson>
Hyperlink Parameters 4 ×	Hyperlink Parameters 4 ×
Type: $ \bigcirc \ \square \$	Type: Type: Type: Dimension: $\downarrow 5$ \Box \Box \Box Dimension: $\downarrow 5$ Linear Dimension 16 (0x4000001) Nominal Tolerance Deviations
Hyperlink Parameters 4 ×	Hyperlink Parameters 4 ×
Type:	Type: ← C:\ → ↓ ↓ ↓ ↓
	Macro: Macrosamples.Macrosamplesciass.createspil
First Corner: X: 165 Y: 90	Text: MacroSamples.MacroSamplesClass.CreateSpline
Second Corner: X: 163 Y: 65	
Text: Page 1	

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.

Insert Table X				
Number of Columns:	5			* *
Number of Rows:	2			*
Column Width:	50.8			
Prototype:	proto6	proto6 ×		
test	bott Dm fowe	top reverse	70 width	variable A
20	<u>red underline</u>	green bold	blue tnr	regular
			OK	Cancel

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.

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

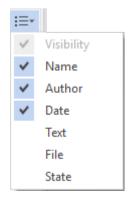
To work with remarks: create, delete, and edit notes, you need to use a special **Remarks** window.

Remarks		1 ×
🎁 🍾 🍳 🛅 🛩 🧋	24	-≣י Q
Name	Author	Date
Remark 2	Admin	1/22/2020 5:56:01 PM
Remark 3	Admin	1/22/2020 5:56:20 PM
Remark 1	Admin	1/22/2020 5:55:41 PM

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

	New Remark
≁	Edit
a	Show
	Delete
# *	Create Remark Plane
2	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.

Autor. 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.

Remark Plane		Ψ×
✓ 🔍	×	
Seneral Parameters		
Select element		~
Activate		

The Remarks tab automatically becomes active in Ribbon.



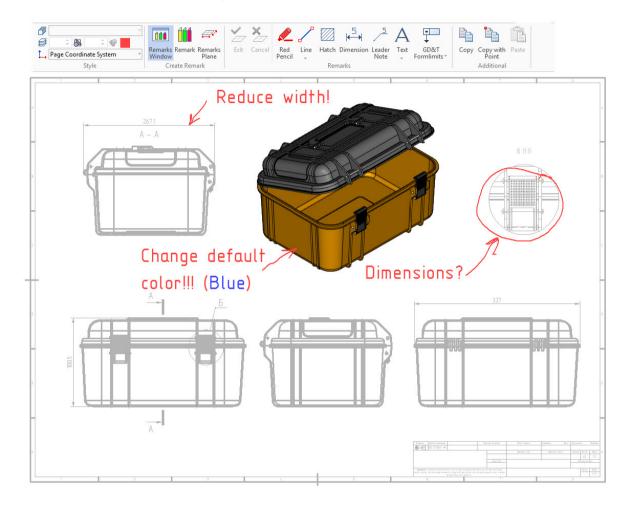
Remark and sketch tools are available on the tab. The combination of remark elements and sketch elements enables you to create a comment on a model or drawing in a free, hand-drawn style. For example, in the spline drawing mode (**Red Pencil** command), you can create a remark that indicates a defect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

<mark>₩</mark> ₩ ₽	D 🖓 📢 -
--------------------	---------

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 STRUCTURE

The option **Concatenate Unique Values with Quantity** in the column of summing values when merging has been added.

Column Proper	ties	×
Properties		
Name:	Format	Show column in Product Structure window
Synonym Name:		☐ Group by column ✓ Sum values when merging
Category:		Concatenate Unique Valu 🔻
Туре:	String •	Summation
Unit:	None 🗶	Concatenate strings Concatenate unique strings
Column Width:	Auto	Concatenate Unique Values with Quantity
Value		Number format
Default Value:		Enabled
List of values:		Image: Second sec
- Data Assignment -		
Format		•
		OK Cancel

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

Product Structure			
🔢 ∄•• 🚰 🛃 ∋↑ ∋↓ 🥔 🔊 ﷺ ** 💱 🕅 📴 Eneral BOM	▼ 📄 Bill of Ma	terials 👻 🗘 Update 💌	PE 🛱 💷 🔺 🖽 📷
Description 👻 Part No.	Forma	t Purchased Items	1
▲ No group			
°abs □	1 📑 A3		8
+ ¹	5 A2; 2A	3; A4; A5	A
¹	1 📑 A4		e
¹	1 🗳 A3		e
¹	1 🗳 A5		e
¹ 0	1 🖆 A3		e
<u></u>	1 🗳 A2		e

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.

Quality Control	4 ×
2 🖻 🔌 🖭 🕨 🖡	- א י≡י
Name	Information
▲ §≣Script	C:\Program Files\T-FLEX Parametric CAD 17\
-> 🛅 Parameters	
-> 🛅 Actions 🔚	
-> 🦰 Results	
4 Em Fixes	

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.

Qu	uality Control		μ×
2	🖻 🔌 🖻 🕨 🖡	ر ب	0 ;≡,
Nan	me	Information	•
4	Script	C:\Program Files\T-FLEX Parametric CAD 17\Program\Qualit	
	-> 🔚 Parameters		
	Actions		✓
	→ 🗗 2D		✓
	→ 🔁 3D		✓
	-⊳ <mark>⊡</mark> 2D/3D		✓
	– > Layer Names	Action with this UID is not registered	✓
	-> > Page Format	Action with this UID is not registered	✓
	↓ > MD5 CheckSum	Action with this UID is not registered	✓
	🛥 🛅 Results		
	→ →> Result of Check	Total: 37, Errors: 10	
	↓ →> Result of Check	Total: 37, Errors: 6	
	– 🚫 Layer Names	Action with this UID is not registered State: Error, Entries: 0	
	– 🚫 Page Format	Action with this UID is not registered State: Error, Entries: 0	
	– 🚫 MD5 CheckSum	Action with this UID is not registered State: Error, Entries: 0	
	 Unused Objects 	No unused objects were found State: Succeessfully, Entries: 0	
	 Unused 2D Constructions 	No unused objects were found State: Succeessfully, Entries: 0	
	 Unused 2D Nodes 	No unused objects were found State: Succeessfully, Entries: 0	

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.

R ²⁴	Update
ß	Open Script
2	Settings
PE	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

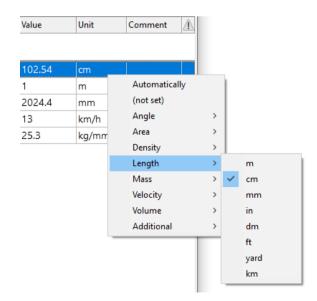
The variable editor has become even more convenient. Now, when the wrong separator of the integer and fractional part is entered, it is corrected automatically. For each variable, you can specify a measurement unit, and the system automatically checks the correctness of measurement units in the calculated 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.

<u>۱</u>	/ariable Ec	altor				×
Fil	e Edit	Variable View 🕐				
(v)	a		- 2	à 62 🛅 🛱 "≩ ∋	L ⊒† 📰 🖌	ß
2	Name	Expression	Value	Unit	Comment	l
	roup:	1m+1in	102.54	cm		Ĩ
	b	1	1	m		
?	с	a+b	2025.4	mm		1
2	S	100mm ²	0.155	in²		1
	R	(1m+1dm)/1cm	11			
?	W	1mm*R	0.011	m		
?	1	1in+1ft	13	in		
	Angle	1rad+10°	1.174533	auto(rad)		
?	Sin	sin(Angle)	0.92251			
2 ?	V	1 (km/h)	1	auto(km/h)		
	Sq	1m*1cm	0.01	auto(m ²)		T

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.

Line P	arameters			×]
		11 dn	n		
Distan	ce:	1 dm+10	6	Undo	
Value:		-66.11	X	C <u>u</u> t	Ctrl+X
Line length:			Ē	<u>C</u> opy	Ctrl+C
		Default	a	<u>P</u> aste	Ctrl+V
			\mathbf{X}	<u>D</u> elete	Del
Level:		0		Select <u>A</u> ll	Ctrl+A
Layer:		Default		Insert <u>S</u> ymbol	Alt+F9
Color:				<u>R</u> epeat Symbol	F9
Line St	yle:	Default		Insert <u>V</u> ariable	F8
Derdare			Insert Unit	Ctrl+F9	
🗌 Set	as default	0		Enter Value	Ctrl+F8
				Dictionary	F6
				Insert <u>F</u> raction	Ctrl+F
	An	gle	>		
	Are	a	>		
	De	nsity	>		
	Ler	ngth	>	m	
	Ma	ISS	>	cm	
		ocity	>	mm	
		ume	>	in	
	Ad	ditional	>	dm	
				ft	
				yard	
				km	

The selected measurement unit is added to the expression.

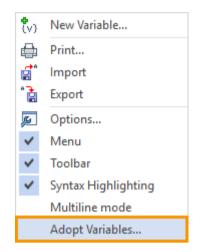
Line Parameters X				
Distance:	1 dm+10 in			
Value:	3.54			

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.

Adopt Fragmen	nt Elements		×
Element	Value 150 100 20	Comment	
✓ Variables Suffix:	☑ Databases		* +/-
Variables Group N	lame Fragme	nt 0xD000001 (
		ОК	Cancel

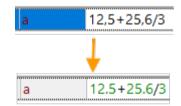
The values of the borrowed variables will depend on the values of the variables in the fragment from which they were borrowed. The file from which the variables were borrowed does not have to be inserted as a fragment in the assembly. Borrowed variables and databases will be marked as borrowed with the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

📅 Va	riable Edito			_		×
File	Edit V	riable View ?				
{v}		PE 🚧 🚽 🖓	¢ i i i	e e e	↓ <u>⊒</u> † [≍] ⊟ _{VAR} \$	VAR ÷
₽?	Name	Expression			Value	Â
🖃 Gro	oup:					
 - ?	Height	Adopted (Fragment 0xD000001 (IS	O 3D Mo	del 3.grb))	150	
2	Length	Adopted (Fragment 0xD000001 (IS	O 3D Mo	del 3.grb))	100	
2	Width	Adopted (Fragment 0xD000001 (IS	O 3D Mo	del 3.grb))	20	
				OK	Cancel	

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.

Variable Edito	or —		X
File Edit V	/ariable View 🕐		
) 🖏 🖨 🖨 P	⇒ <mark>۲ ایش ا</mark> ا	↓ <u></u> † [≍] ⊟ VAR	VAR ÷
P ? Name	Expression	Value	Â
Group:			
🟳 ? Height	Adopted (Fragment 0xD000001 (ISO 3D Model 3.grb))	150	

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:

Read Param	eter File X
Input File:	.par
	Replace Comments Create New Variables Only Marked Variables Load Connectors Group Variables
	OK Cancel

When importing variable values to a file, the ability to display group names is also added.

Save Parame	ter File X
Output File:	.par
	Marked Only
	With Expression
	With Comments
	Export Hidden Variables
	Export Connector Values
	Export Only Selected Variables
	Export Group Names
	OK Cancel

MEASUREMENT

The model measurement tools were significantly were 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.

MEASURE COMMAND

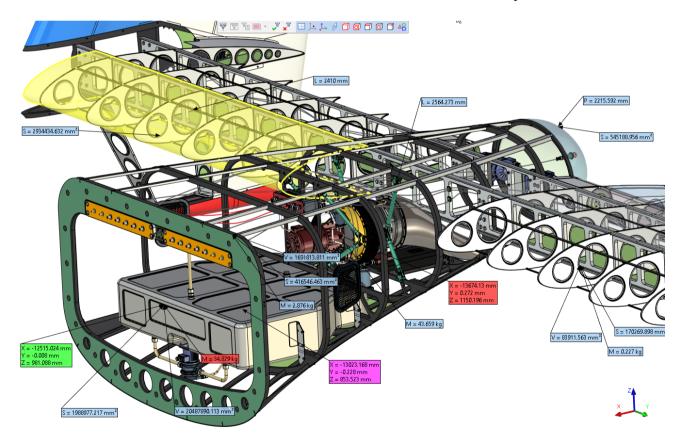
The Measure command has new measurement properties and new visualization tools.

Decorations

Now decoration of measurements can be displayed.

Value:	50	mm
Expression:	get("Edge_2", "EndY")	
Show De	corations	

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.

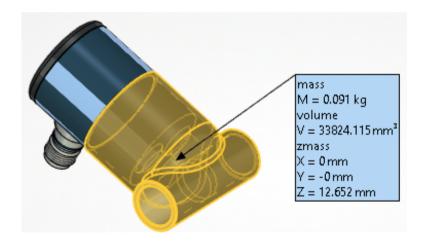
Measure		џ	×	
	✓ 2 🗙			
A Measure				×
				7
Element:	Edge_1: 3D Fragment_2 (00.00.003-Worm.grb)) [\times	t⊑
Filter:	All		~	Save Measure Result <s></s>
System of units:	From document		~	Save measure Result < 5>

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.

A Measurements for Report	
Measurements for Report	₽×
Representation:	
Text Table	
======= Edge_1 : 3D Fragment_2 (00.00.003-Worm.grb) ========= Diameter: 32 mm Diameter	
======= Edge_2 : 3D Fragment_10 (00.00.005-Shaft .grb) ========= Diameter: 60 mm Diameter	
	×

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**.

Add to Report							
[0]	Description	Value	Units	^			
✓	Surface area	30000	mm²		Ī		
~	Perimeter	1200	mm		Ì		
	Maximum X coordinate	50	mm		l		
	Minimum X coordinate	-50	mm				
	Maximum Y coordinate	50	mm				
	Minimum Y coordinate	-50	mm				
	Maximum Z coordinate	100	mm	\sim			
<			>				

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 **I**. 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.

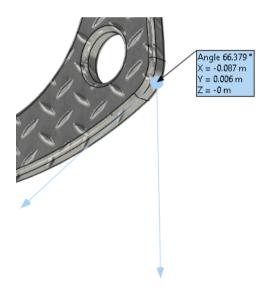
- for proprties: StartX, StartY, StartZ, EndX, EndY, EndZ-a 3D node is created on the path (at the beginning or end, respectively);
- for proprties: LocationX, LocationY, LocationZ-a 3D node is created on the surface at the starting point of the surface;
- for the Xmass, Ymass, and Zmass properties, 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

Measurement

Thread Measurement

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

Measure					₽×
	~	2	×		
A Measure					
	þ				
Element:	Face_2 : Thread_3				\times
Filter:				~	
System of units	From document				~
Add to Repo	ort				
Merge Label	s				
	Description	Value	Units	Property	^
Diameter of th		24	mm	ThreadDiameter	
Pitch of thread		2	mm	ThreadStep	
Height of threa Number of threa		1.082532 1	mm	ThreadHeight ThreadNumberOfVisit	
Thread hand	eads	I Right-handed		ThreadDirection	
Notation of thr	read	M24x2		ThreadNotation	
Offset from th		0	mm	ThreadStartOffset	~
Value: Me	etric thread GOST 8724-2002				
Expression: tg	et("Face_2", "ThreadStandard	7			
Show Decor	ations				
Measuremer	nt Coordinate System				
Variables					

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.

A Measure	Measure						
	Intersection Result_1	<u>^</u>					
F lower by	O Torus_1	■×					
Elements:	🗇 Box_2	\times					

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.

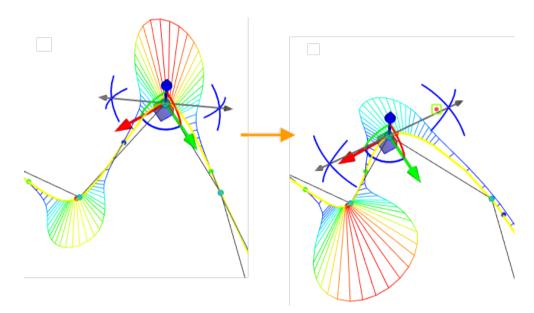
Value: 3.937008	in				
Expression: get_u("Edge_1","Length", in")		Unit {0}			
		Meter	Convert: 0.1 m		
		Centimeter	Convert: 10 cm		
		Millimeter	Convert: 100 mm		
	~	Inch	Convert: 3.937008 in		
		Decimeter	Convert: 1 dm		
		Foot	Convert: 0.328084 ft		
		Yard	Convert: 0.109361 yard		
		Kilometer	Convert: 0.0001 km		

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.

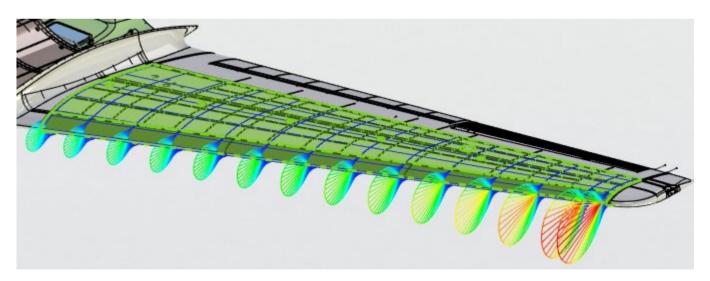
CURVATURE ANALYSIS

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

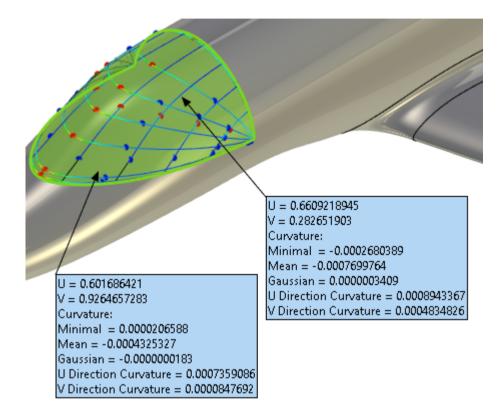
A new tool for analyzing surface curvature is available: Surface Curvature by Sections command.



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.

🗸 U Step:												
50 🔺	-					_(ļ					
V Step:												
50 🗘						-	ļ					1

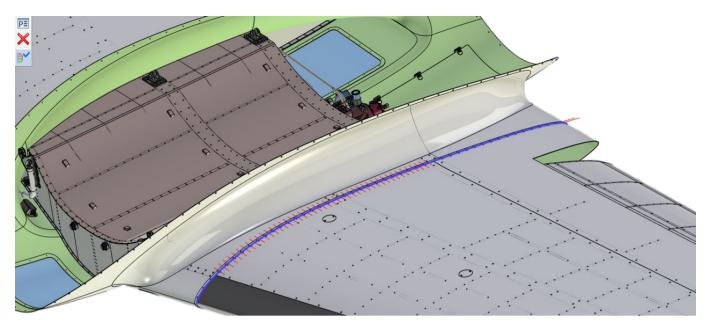
Visual display settings are available: color scale, image scale, image line density, output range. In addition, the display of measurements in points is available.



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.

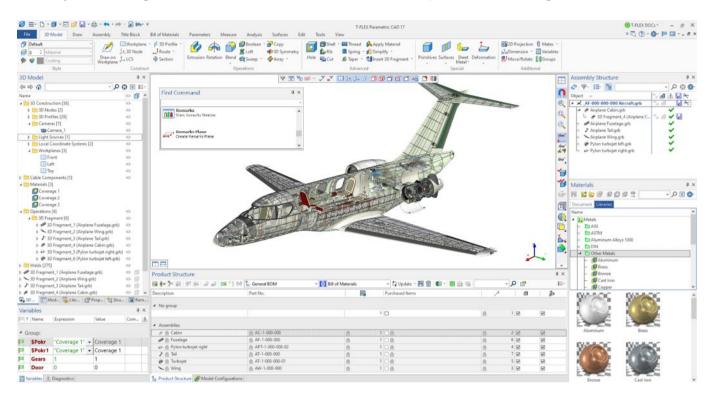
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<u>^</u> N	Aeasure Res	ults						
N₽	Result	х	Y	Z	Distance	Angle	^	₽ ₽
10	~	-5692.555517	906.934368	-792.82477	1.002273	0.16057		
11	~	-5708.337283	906.730307	-788.95385	1.0022	0.160215		
12	~	-5724.169018	906.537259	-785.2922	1.002151	0.156238		
13	~	-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		
24		F0C0 040400	005 000004	701 10010	1 000000	0.110011	~	

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.



HIGHLIGHTING THE PARAMETERS OF OPERATIONS/COMMANDS

Highlights for various operation parameters have been implemented.

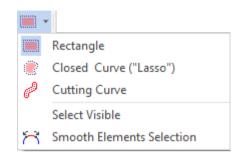
Extrude Parameters	4 ×	
✓	×	
☆ General Parameters		
Contour: Select element		
Direction:		
First Direction:	Rotation Parameters	4 ×
✓ Automatically		
Length: 10	✓	×
Taper: 0	Seneral Parameters	
	Contour: Ø 3D Profile_1	\bowtie
	Axis: Select element	Boolean
	Angle: 🕄 360	
	Start Angle: 0	
	Simplify Geometry	S General Parameters
		Operands Selections Faces Matches
		Target Bodies:
		Select element
		Tool Bodies:
		Box_1

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

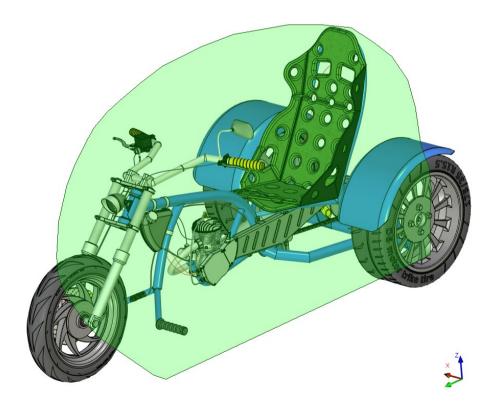
The feature of selecting elements by a random outline was added. An element selection mode control button was added to the filter panel. Selection modes are available for all the operations and in the command stand-by mode.



The following variants are available.

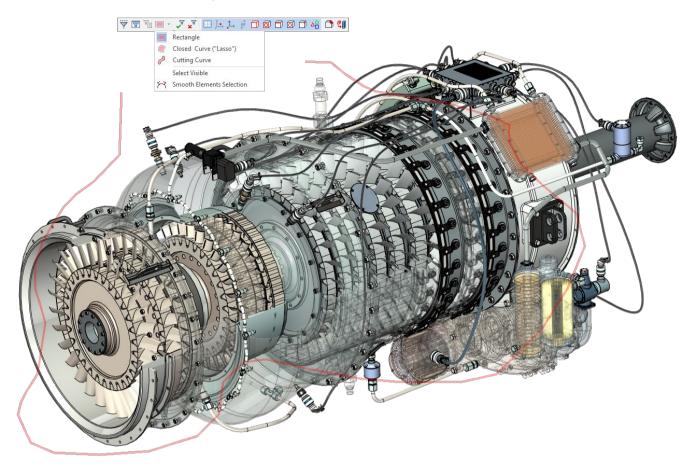
Rectangle. A common selection mode actual for the previous versions.

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.

Interface and interaction with the system



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.

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.

User	Pind Find	
Startup		
2D	 Workplanes 	
3D	Standard	ISO (First Angle)
Windows	Font	ኳ Tahoma
Folders	Number of grid lines	3
Bill of Materials	Select by all grid lines	
Files	Brightness of 3D elements when drawing on a workpl	50
Fragments	Create 3D Axes by 2D Axes	\checkmark
Snap	 Size of coordinate system 	
Constraints and Dimensions	Scene	128
Dynamic Toolbar	Dragger	160
Colors	Scene Rotation	
Additional Options	Using arrows	15
Performance	Using Shift+Arrows	90
Save	Smooth Rotation, sec	✓ 0.5
T-FLEX DOCs	Turn workplane on activation	\checkmark
Quality Control	Rotate with mouse button	Left
All	Rotation Speed, %	Left
	4 3D	Right
	When object is created	Middle
	Color for clip plane	Body Color
	Projections	
	Don't show warning messages while creating projecti	

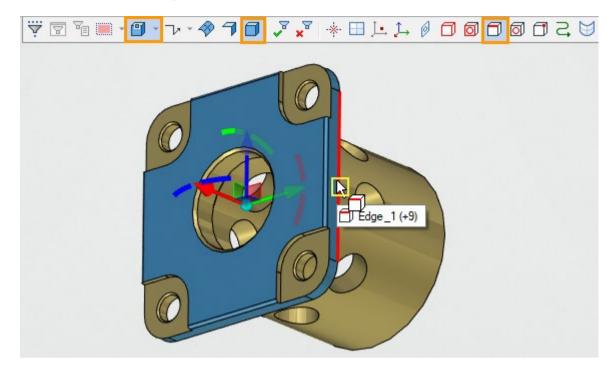
SET CENTER OF ROTATION

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

Ce	nter of Rotation	₽×
	✓	
C		
x:	0.0499999449	*
Υ:	0.0007581376	*
z:	0.0805744976	*

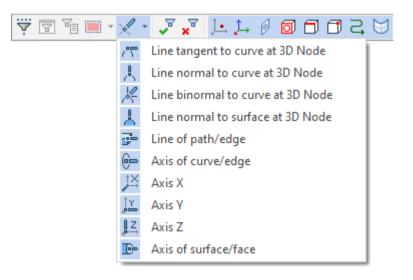
FILTER OF SELECTION BY GEOMETRIC TYPE

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.



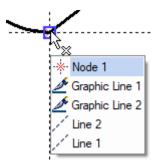
Another set of available geometric types is in the **3D Node** operation: **Point**, **Wire**, **Surface**, **Sheet**, **Solid**. There is a list of objects for **Point** and **Wire** geometric types.



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

CUSTOMIZABLE ELEMENT SET UNDER CURSOR

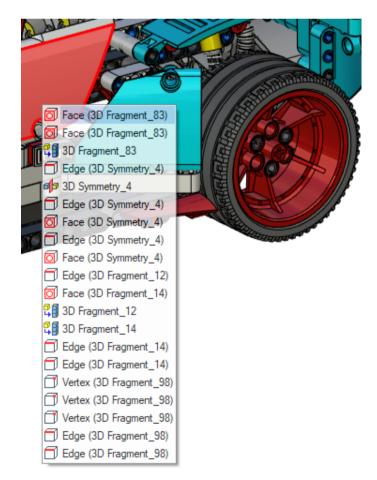
Element selection from the list under the cursor was improved.



You can set the appearance time of the pop-up tip for the element under the cursor in the **Option** dialog in the **Additional Options** tab. In case there are several elements, set the element list. The time is set in milliseconds.

Additional Options		
Right mouse button action	Cancel in command and menu without command	
Double-click action	Edit element parameters	
Invert mouse wheel direction on zoom		
Tooltip delay for element under cursor (ms)	500	
Delay of "Elements under cursor" list show (ms)	2000	
Transparency of "Elements under cursor" list (%)	50	

List transparency can also be set in the **Additional Options** tab: it is convenient when you need to see the selected element under the list. Restart T-FLEX CAD for the set 3D graphics display parameters to come into effect.



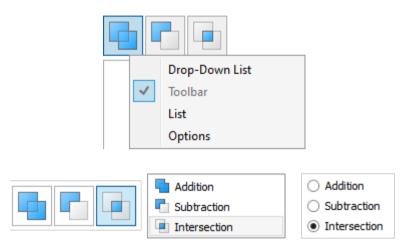
When necessary, you can call the element list immediately by using <Tab> key or by one click on the center mouse button.

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.

Contour Orientation:		
Contour: OFace_1:Box_1		
Trajectory: T Edge_2: Box_1	3	
Correction of Contour Location		
Fixing Point: Vertex_3 : Box_1		
Additional Point: Select element]	
Point that defines Rotation: Select element		
Trim Trajectory		
Start: 5		
End: 25 mr	Unit {0}	
	Meter	Convert: 0.005 m
Twist Scaling	Centimeter	Convert: 0.5 cm
Point Position (%) Value	Millimeter	Convert: 5 mm
Start 0 0	Inch	Convert: 0.1968503937 in
End 100 15		Convert: 0.05 dm
Position (%): 100	Foot	Convert: 0.0164041995 ft
Angle: 15	Yard	Convert: 0.0054680665 yard
	Kilometer	Convert: 0.000005 km

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

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Element	Radius	^	
Face_1 : Extrusion_1	[5]	■×	
Edge_2 : Extrusion_1	[5]	\times	
Edge_3 : Extrusion_1	[5]		
[]			
	~	Vertic	al Lines
Common Properties	~	Horiz	ontal Lines

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

Description	Value	Units	Property		
Perimeter	605.4	mm	perimeter	~	Description
Surface area	22897	mm²	area	~	Value
X coordinate of plane origin	0	mm	LocationX	3	Units
Y coordinate of plane origin	10	mm	LocationY	*	Units
Z coordinate of plane origin	0	mm	LocationZ	~	Property
X component of plane normal	-0		NormalX	· · · · ·	D .
Y component of plane normal	1		NormalY		Reset
Z component of plane normal	-0		NormalZ		~

NEW WINDOW «LAYERS»

An auxiliary window Layers was added.

Layers										4 ×
🗖 AI [A] 🖪	2 💼 📃									- ∕ ≔-
Name	🖗 Hidden	8	X	8	M	onochrome		Same lin	?	Group
Default	v≩≏ 0	8		8		0		0	?	
Layer1	vi}~ 0	8	闼	•		10	≣	0	?	
Layer2	18 · 0	8	Ø	8		4	≣	0	?	
Layer3	🏫 1	8	\square	8		3	Ξ	0	?	

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.

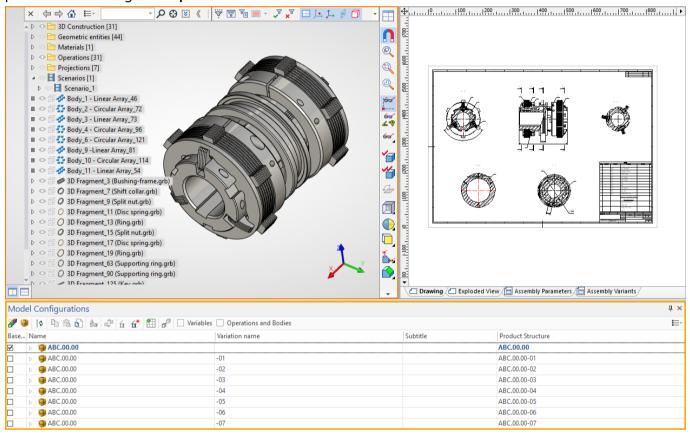
«Model Configurations» window

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.

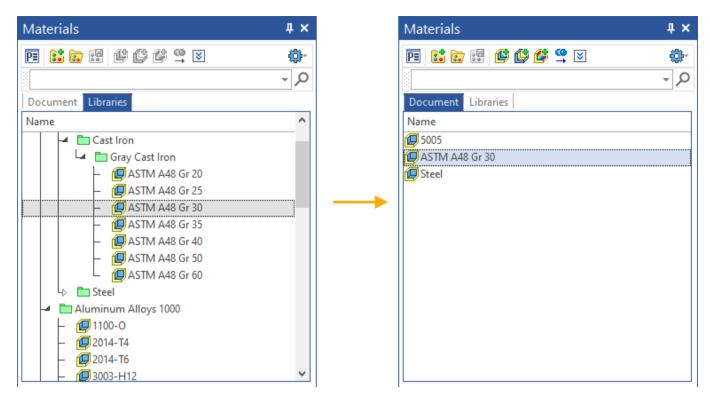
Work in the **Model Configurations** window is available without saving the file. In addition, import of performances using the **Import E** command is available.



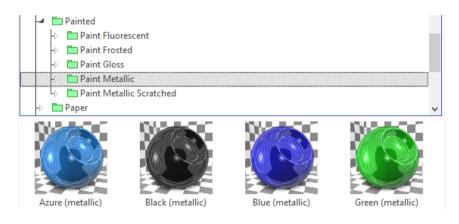
«MATERIALS» WINDOW IMPROVEMENT

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.

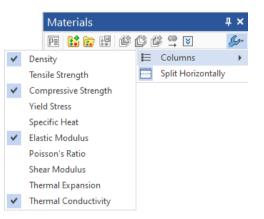
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A preview icon was added to the left from the name of the material. Besides, material preview is available in the bottom panel of the **Materials** window. In case a particular material is chosen, it is the only one to be displayed at the preview panel.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

Ma	terial	s						д,
P≡	8	.	8	eee:				ß
3								- 5
Do	cumen	t L	ibra	aries				
Nar	me				Density	Compressive Strength	Elastic Modulus	1
4	; Met	tals						
ŀ	- 📁	AISI						
	La		Ste					
		H.		Stainless Steel				
			-	但 AISI 201	8000 kg/m³	640 N/mm²	200000 N/mm ²	
			F	🔲 AISI 202	8000 kg/m ³	700 N/mm²	200000 N/mm ²	
			F	🗐 AISI 303	8000 kg/m³	585 N/mm²	200000 N/mm ²	
			F	🗐 AISI 304	8000 kg/m ³	215 N/mm²	200000 N/mm ²	
			L	AISI 309S	8000 kg/m ³		200000 N/mm ²	
			L	AISI 316Ti	8000 kg/m ³		200000 N/mm ²	
			L	AISI 321	8000 kg/m ³		193000 N/mm ²	
			L	🗐 AISI 409	7760 kg/m ³		217000 N/mm ²	- 1
			L	AISI 420	-	1165 N/mm ²	218000 N/mm ²	
			L	🗐 AISI 430	7800 kg/m ³		200000 N/mm ²	
			L	AISI 439	7700 kg/m ³		206000 N/mm ²	
			L	AISI 441	7700 kg/m ³		220000 N/mm ²	
			L		7900 kg/m ³		200000 N/mm ²	

«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.

A Delete Unused Construction	
✓ 3D Nodes	B
✓ 3D Paths	
Workplanes	
Visible Unused Variables	
Element	(j)
Visible Unused Variables (1)	
var ✓ a = 10	
 3D Nodes (3) 	
3D Paths (1)	
S ☑ 3D Path_1	
Only on Active View	

In the commands dialog, you can cancel the deletion of an item, or a group of items. Inverting the list \square , clearing the list \square , selecting all items \square , and information about the selected item 1 are available.

TOOL WINDOWS UPDATED VIEW

All tool windows have been moved to the new interface.

3D Model	Ą×
	B -
	- P
Library Explorer	4 ×
🏪 🖻 🗉 🗵	=-
	ب ک
Model Elements	₽×
Model Elements	4× ≣∹
	E.
Current View	•⊒ م_

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.

3D Model	д×			
	Sp-			
	, 15	Columns	•	ID
Name	0 1	Search Panel		Description
D Construction [3]	•	Parameters		Part No.
Materials [1]		Use active View Filter		Recalculation Time
		Appearance		File Name
	×	Hide Fragment Connectors		File Path
		In Order of Creation		Color
			_	Material and Color
				Coating
			×	Visibility
				Suppress
				Wireframe
			×	Semitransparent View
				Fixed Position
				Load Contents
				Creation Order

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.

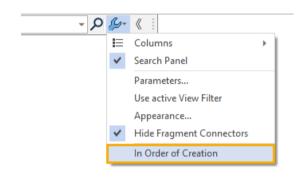
Interface and interaction with the system

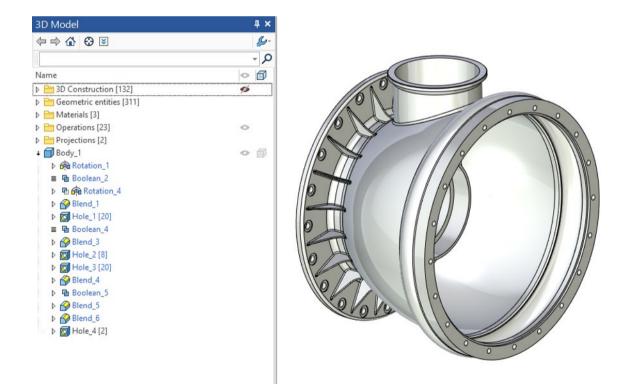
3D Model	4 × 🔮
	Ry-
	🗧 Columns 🔹 🕨
Name	💿 🗸 Search Panel

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

× 🗇 🔿 🚯 😨 💽 🗸	B -	« : 🐺 😨 🖷	× ×
	IE	Columns	→
	~	Search Panel	
		Parameters	

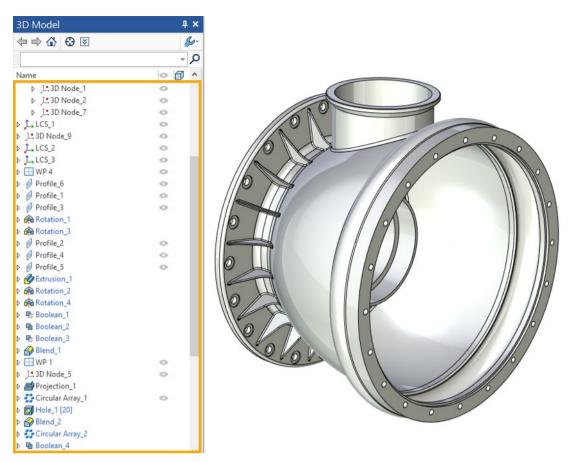
In the case of the model tree, the search bar is now disabled by default. Also added the ability to sort created items in the order they were created.





Standard View

T-FLEX CAD 17 and Add-on Modules. Release Notes





Added Fixed Position column.

3D Model	д ×
	\$ -
	م -
Name	🗢 🗊 🕹

The column ensures quick assigning and control of fixed bodies. Added **Load Contents** column.

3D Model	4 × 🔮
	B -
	Q -
Name	o 🗗 🗇 🔶
 4 3D Fragment_1 (Body.grb) 	
J 3D Fragment_2 (Shaft Subassembly.grb)	Don't Load
SD Fragment_3 (O-Ring.grb)	• 🗊 Support Geometry
Image: Pragment_4 (Bush.grb)	o 🗊 🦷 🦾
3D Fragment_5 (Ball.grb)	Meshes and Support Geometry
SD Fragment_6 (Seat.grb)	 Solids, Meshes and Supporting Geometry
S SD Fragment_7 (Seat.grb)	○ ∅ ∅
N # 3D Fragment & (End Can orb)	

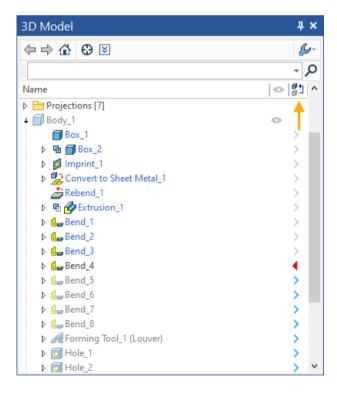
The column allows you to quickly select content download options for each fragment for performance and memory management.

Added Creation Order column.

3D Model			‡×
$\Leftrightarrow \Rightarrow $			\$ -
			ب -
Name	0		Creation Order
3D Fragment_1 (Body.grb)	0	Ø	1
👂 🚦 3D Fragment_2 (Shaft Subas	0	Ø	6
Image: Solar Stragger (Contemporation of the second sec	0	ø	20
3D Fragment_4 (Bush.grb)	0	ø	27
3D Fragment_5 (Ball.grb)	•	ø	10
SD Fragment_6 (Seat.grb)	•	ø	21
Solution Stragger Stragger (Seat.grb)	•	A	22

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

Added Rollback column.

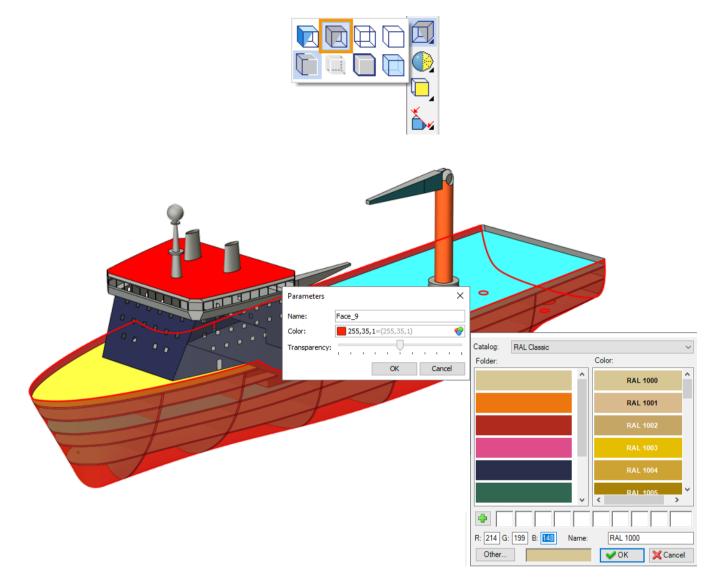


The column allows you to display the operations to which the rollback was performed.

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.

Interface and interaction with the system



HOT KEYS TO CONTROL AUXILIARY WINDOWS VISIBILITY

Hot keys were added for auxiliary windows output commands.

				Customize	
			2	Options	
			₽	Applications	
			2	User Commands	
			Ē	Interface Elements	►
ß	Open Folder			Tool Windows	•
Ē.	Library Explorer	Alt+1	1	Theme	►
Â	Diagnostics	Alt+2	2	Environment	►
a ,	3D Model Tree	Alt+3	×	Reset Settings	
P	Properties	Alt+4			
Þ	Macros	Alt+5			
	Variables	Alt+6			
□→	Assembly Structure	Alt+7			
9	Materials	Alt+8			
:	Product Structure	Alt+9			
	Bird's Eye View	Alt+0			
8	Model Elements	Shift+F1			
^t\$	Structural Elements	Shift+F2			
***	Remarks	Shift+F3			
3	Model Configurations	Shift+F4			
ð	Layers	Shift+F5			
[]	Studies	Shift+F6			
1	Weld Project	Shift+F8			
8:2	Quality Control	Shift+F9			
	Find Command				

START PAGE

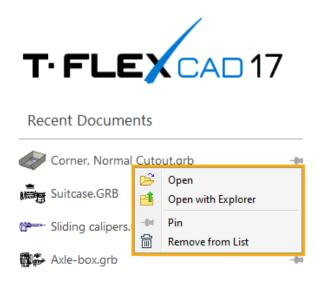
Start page information is divided into thematic groups. Now all the necessary information, contacts, files and learning materials are easier to find.

(i) Information	Learning Materials		Contacts
 T-FLEX CAD 17 Release Notes T-FLEX Products Overview 	• Tutorial • Manuals (PDF)	T-FLEX Resource Center Design Examples Free T-FLEX Viewer	 Official Web Site Official Forum Contact Us Technical Support Image Content Image C

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.

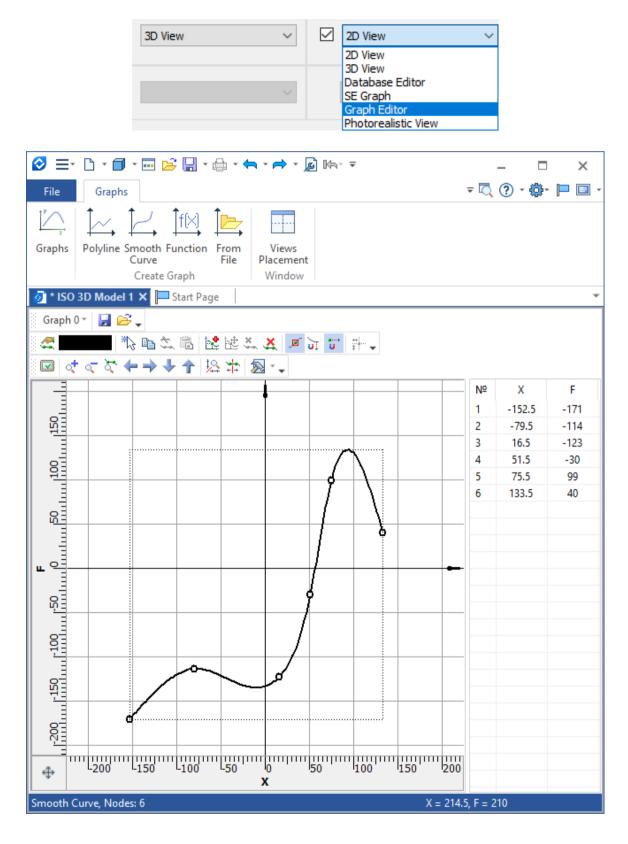
		✓ Startup	
		Load previous session documents on start	
		Disable macros automatic execution	
Open Create	•	Number of "Recent Files" in menu	10
		Close welcome page after document is loaded	
		Show welcome page at startup	\checkmark
		Run as Server	
	Piston.grb	*	
	Isoparametric Curve.	grb 7	
	Base member with h	oles.GRB -	

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.



VIEWS PLACEMENT COMMAND

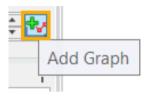
Using the **Views Placement** command, the Graph Editor can now be placed in a separate document window. When working with a graph, a special ribbon tab **Graphs** will be automatically activated.



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**.

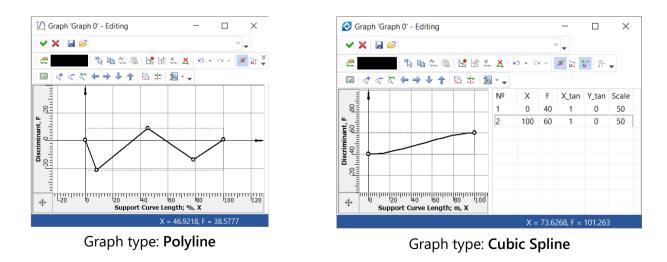
You can call the graph editor from the command dialog in which the creation of a graph is available.



Special modes were added:

Select Graph X		Select Graph		×	
Create New Graph Use Existing Graph		Create New Graph Use Existing Graph			
Name:	Graph 0		Name:	Graph 0	
Туре:	Cubic Spline	~	Туре:	Cubic Spline	~
Axis X Type:	Percent	~	Axis X Type:	Polyline	
	(not set)			Cubic Spline	
	Length			By 3D Curve	
	Percent	-			_
	Unit Length				

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.



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

😤 By 3D Curve		
Support Curve:	Select element	
Law Curve:	Select element	
Approximation Accuracy:	0.0001	m 🛓

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**.

	Transition Surface
	✓ Q × Q × Q × Q × Q × Q × Q × Q × Q × Q
The	Section Type
	(a) By Discriminant
	Support Curve: Edge_3 : Edrusion_1 (surface)
	First Guide: 🔲 Edge_4 : Extrusion_1 (surface)
	First Face: Sec. 1: Extrusion_2
	Second Guide: Tedrusion_1 (surface)
	Second Face: 5 Extrusion_2
	Discriminant: 0.5=By Low K
	Approximation Accuracy: 0.1 mm
	Calculation Accuracy: 0.01 mm E
	Simplify Geometry:
	Calculation Accuracy: 0.01 mm smpdf/ Generaty: V ute Guides: V
	S Options
	Properties # ×
	✓ © X
	a By 3D Curve
	Support Curve: 0/ 30 Profile_1 2
	Approximation Accuracy: 0.0001 m 2
	1 W 1 1
···· 10	

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.

Axis X Type:	Length ~	
Length Units:	(not set)]
	Length	
Percent		
	Unit Length	

New types of graps 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.

Recalculation Parameters						Ą	١x
✓	٢	×	•	3			
Seneral Parameters							
Name	Value	Tolerance		†	Recalculation Type	^	■×
 Not Recalculated (13) 							\times
[45] Linear Dimension 1.5 (0x4000002)	1.5		-0.49	0.49	+		¢.
[슈] Circle Dimension R 12.45 (0x4000008)	12.45		-0.1	0.2	∓ ^{∞∞}		- -
4 ⁵ → Linear Dimension 29.5 (0x4000009)	29.5		-0.35	0.45	↑ ⁰⁰⁰		
Linear Dimension 5zb8 (0x4000005)	5	zb8	0.05	0.068	₹ ⁰⁰⁰		∲ {∀}
Linear Dimension 4 (0x400000B)	4		-0.32	0.54	★ ⁰⁰⁰		(j)
Linear Dimension 130E11 (0x400000C) التحي	130	E11	-0.4	0.7	★ ⁰⁰⁰	\sim	₿
Delete Symmetrical Tolerances							
Preview:							
Correctly Calculated Dimensions: 9=Li	ght Blue						•
Incorrectly Calculated Dimensions:	Light Red	ł					•

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 (for versions 2015-2020), FBX.

Added support for formats of new versions of systems: Inventor 2019/2020/2021, SolidWorks 2019/ 2020, Solid Edge 2019/2020, Parasolid V31/32, JT 10.2/10.3, Revit 2020, NX 1847/1872/1899 Series, ACIS 2020.

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, Siemens NX, Creo, CATIA, Solid Edge, Rhino, Revit, I-DEAS, VDA-FS, JT, PRC, CGR, U3D, FBX formats - added import option layers.

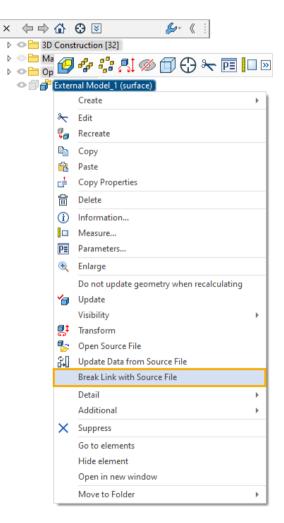
Added the ability to import drawings from KOMPAS v.18/19.

Improved automatic generation of the product structure when importing assembly models.

In the **T-FLEX CAD Extended Import** module, the ability to directly read the CATIA v.5 (R2020) format has been added, as well as support for new versions of Creo 5/6 systems.

BREAK LINK WITH SOURCE FILE

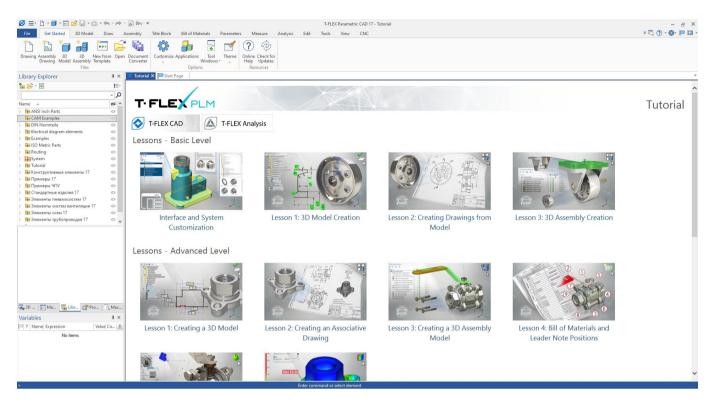
Added the ability to break the link with the source file of the external model. When this option is activated, all parameters associated with the source file will be removed from the operation context menu.





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.



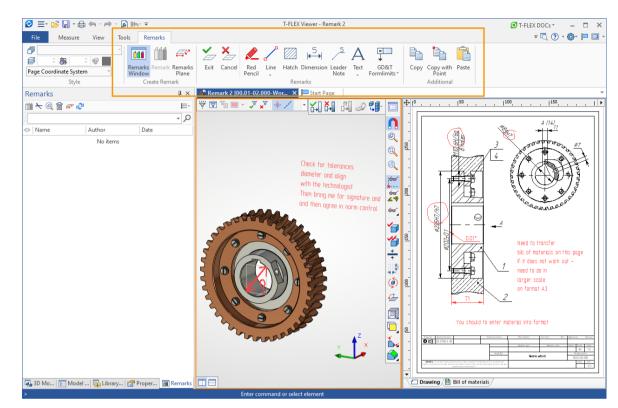
Now the washer **Washer ISO 10673** is used as the model under consideration. This lesson is the final one in the parameterization section and includes all the information learned by the user in the previous lessons. The lesson explores the classic approach of creating a model - the 2D to 3D method.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

T-FLEX VIEWER 17

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

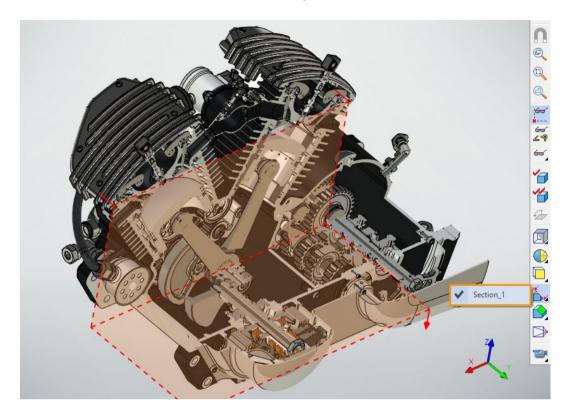


• A new remarks mechanism is available, including work in the **Remarks** window.

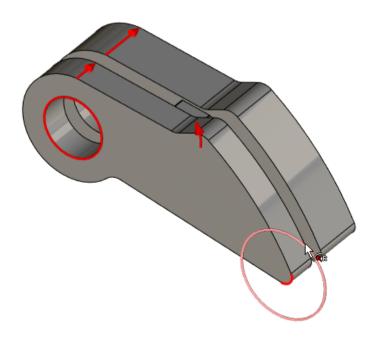
• You can measure models using the Measure command and calculate mass-inertial characteristics.

Measure				Ąх	🍰 🔒 Decorations.grb 🗙 🔚 Start Page
×	2	×			✓ 🐮 🔍 🖓 🖓 🐨 🐨 🗴 💭
A Measure					×
Element: O Face	_1 : Blend_2			\times	L = 101.244 mm
Filter: All				~	
System of units: From doe	ument			~	
Add to Report					
Merge Labels					
Description	Value	Units	Property	^	Angle 66.379*
Perimeter	1078.7800	mm	perimeter		X = -86.88 mm
Area	10181.591	mm²	area		Y = 6 mm Z = -0.16 mm
X coordinate of plane origi	0	mm	LocationX		X = -5 mm
Y coordinate of plane origi	8	mm	LocationY		Y = 6 mm
Z coordinate of plane origi	0	mm	LocationZ		Z = 111.34 mm
X component of plane nor	0		NormalX		
Y component of plane nor	1		NormalY	\sim	P = 1078.78 mm
Value: 1078.780017				mm	X = -55.622 mm
Expression: get("Face_1", "perimeter")			Y = 6 mm		
Show Decorations					Z = 23.66 mm
Measurement Coordinat	te System				
Variables					R = 12 mm

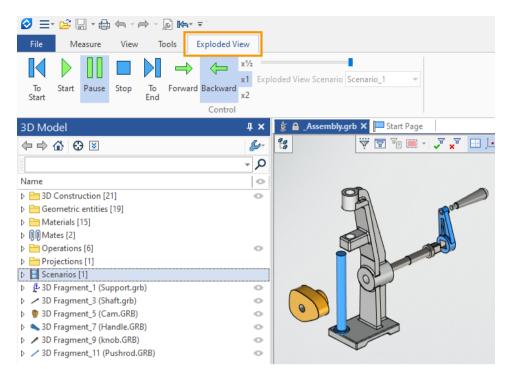
• For a detailed view of the models, commands for using 3D sections and clipping planes are available, in addition, visual display options are available, as well as options for models searching in the 3D scene and the **3D Model** and **Assembly Structure** windows.



• 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.

	EX Viewer - Start Page	🕑 T-FLEX DOCs 👻 💷 🗮 🗙			
File Get Started Measure View Tools Remarks					
Open Customize Applications Tool There Files Options There Check for					
Remarks # × Fart Page ×		¥			
	_				
Name Author Date No items T- FLEXVIEWER 17 Recent Documents Boss.grb Ball from copies.GRB 0.0.01-02.000-Worm wheel (assembly).grb Assembly.grb Decorations.grb					
م ع م ال Mod الم المات ال Pro الا Rem	T-FLEX CAD 17 Release Notes T-FLEX Products Overview • Manuals (PDF)	Resources Contacts • T-FLEX Resource Center • Official Web Site • Design Examples • Official Forum • Free T-FLEX Viewer • Contact Us © 🕐 🔥 🛃			
> Enter command or selec	t element				



00

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.

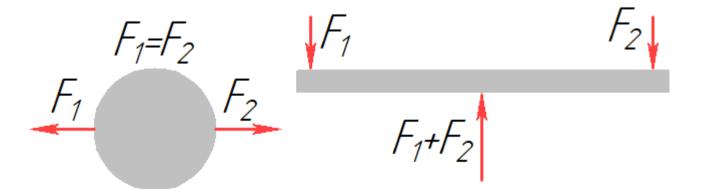
New Algorithms for Solving

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**.

General Sc	olve Nonlinear	Thermoeffects	Results		
Solving Method					
Automatic	assignment				
ODirect		5	Settings		
OIterative		5	Settings		
Relative accur	acy:	0.0	001 ≑		
Maximum num	ber of iterations:	500			
Stabilize System Settings					
Element Type:	Linear Tetrahedron		~		
Recommended for calculations without numerical values for stress limit estimation					
□ Alpha Method 0.6 🗘					

Inertial Relief

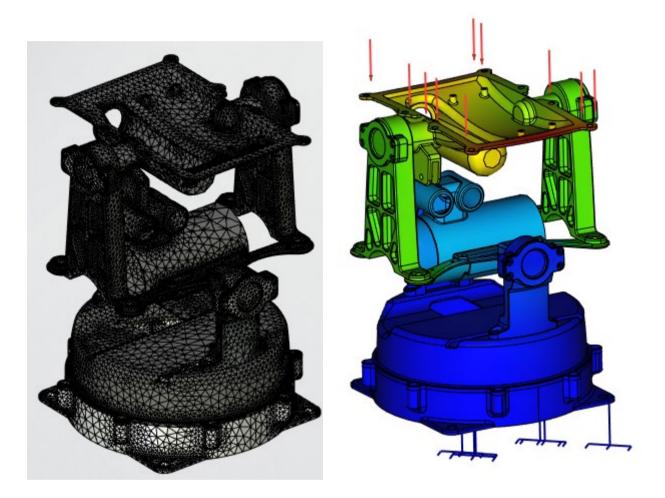
In contrast to stabilization, which is aimed at calculating studies with insufficient restraints, **Inertial Relief** enables you to solve studies where the equilibrium of the system is a consequence of the balance of forces.



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.

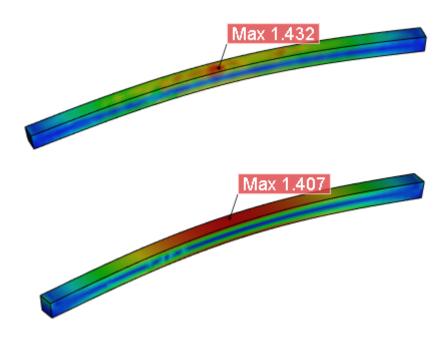
Smoothed FEM

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.



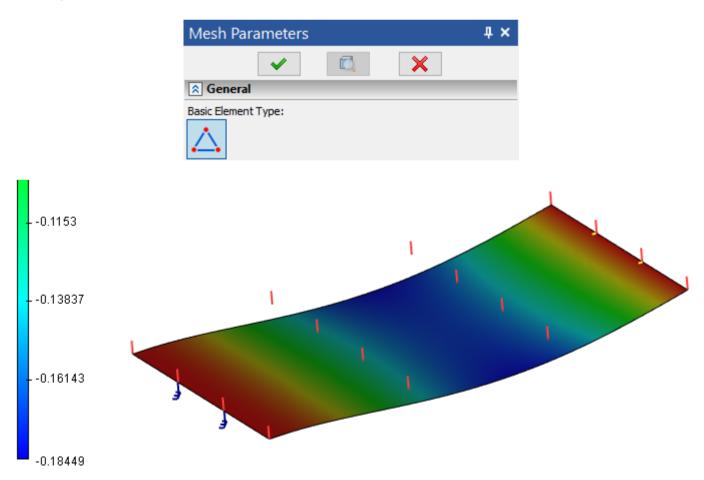
Improved Stress Calculation

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.

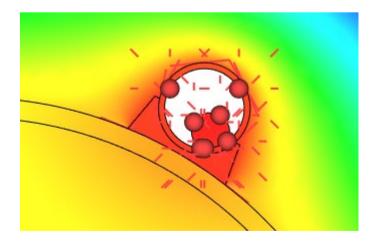
Radiation	Paramete	rs			
Face_1:Box_ Face_2:Box_ Face_3:Box_ Face_4:Box_ 	2 1		-		
Type:					
Radiation between Faces 🗸					
Radiation to Sp Radiation betw					
Temperature:	20		℃≑ 🛃		

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

Type:		
Radiation betw	een Faces	~
Radiation:	0.5	÷ 🛃
Temperature:	20	∞≑ 🔩

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 Parameters	
Contact Type:	
Faces of First Body:	
Face_1: Box_2	\mathbf{h}
	■×
	\times
Faces of Second Body:	
O Face_2:Box_1	$\mathbf{\wedge}$
	■×
	\times
Visual Scale: 1	*

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

 \blacksquare and \boxtimes to delete extra faces. You can use the icons $\boxed{\bigcirc_1}$ and $\boxed{\bigcirc_2}$ 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.

Symmetry Type:	Mirror	~
First Mirror Plane:	Mirror	
	🕀 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.

T-FLEX CAD 17 and Add-on Modules. Release Notes

First Mirror Plane:	
Face_3 : 3D Fragment_2 (C:\Users\	^
	■×
	\times
Second Mirror Plane:	
Face_5 : 3D Fragment_1 (C:\Users\	^
	×
	\times
Third Mirror Plane:	
Face_8 : 3D Fragment_2 (C:\Users\	^
	■×
	X

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 2007.

PRODUCT STRUCTURE IN ANALYSIS REPORT

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

Report	×
General	
	List of Product Structure Schemes X
Report Sections: Study Information Model Materials Mesh Properties Restraints Probes	General BOM
Image: Comments for Selected Section:	Parameters Compound Title Section Part No. Compound Part No. Material Weight Price Price Price Remarks Variation Name
Product Structure Schemes OK Cancel	OK Cancel
Display report Create Cancel	



What's New

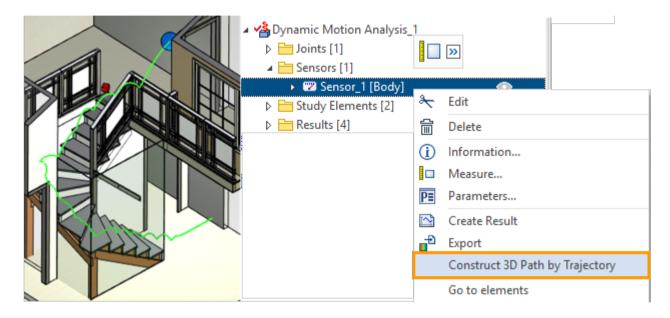
5

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.

Measure				4	ι×
	×	2	×		
A Measure					
	a				
Element:	Referen	ce to 3D Path_1			\times
Filter:	All				¥
System of units:	From docum	ient			¥
Add to Report	t				
Merge Labels					
Descrip	otion	Value	Units	Property	^
Length		744.172420682		Length	
Perimeter		744.172420682		perimeter	
X coordinate of s			mm	StartX	
Y coordinate of s			mm	StartY StartZ	
Z coordinate of s X coordinate of e			mm	EndX	
Y coordinate of e			mm	EndY	
Z coordinate of e		100	mm	EndZ	
Minimum X coord		-50	mm	XMIN	
Minimum Y coord		-50	mm	YMIN	

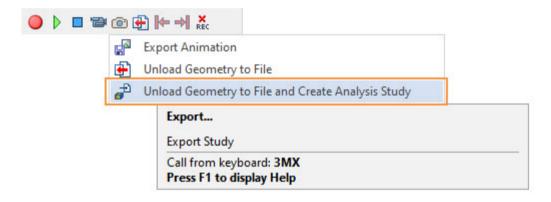
UNLOADING MODEL CALCULATED STATES

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.

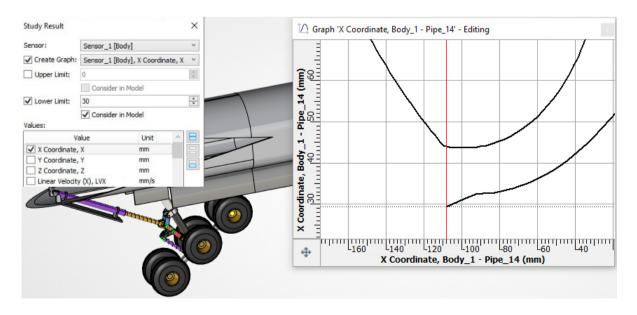
	Export Animation	
e	Unload Geometry to File	
₽	Unload Geometer to File and Create An Export Export Drawing Or Mod	
	Call from keyboard: EX. Press F1 to display Hel	

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.

🕂 Warning	×
Dynamic Study contains some calculated Results Continue Calculations Start Calculations from Beginning	
Don't show this Dialog in future	
OK Cancel	

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

The study has results calculated with other parameters	
The study has results calculated with other parameters	
 Continue Calculations Start Calculations from Beginning Don't show this Dialog in future 	
OK Cancel	-

GRAPHS AS CYCLOGRAM

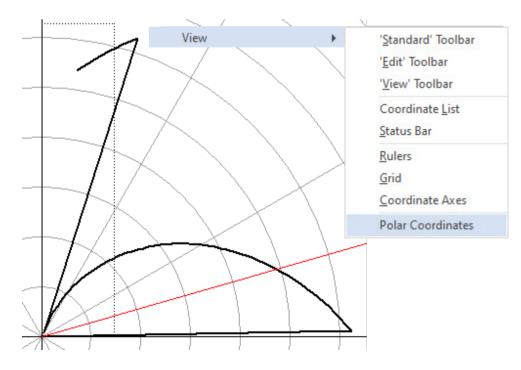
Calculation of mechanism cyclogram is an important and convenient functionality for evaluating the joint operation of the product units under study. As is known, most mechanisms have a rotary drive, and therefore, the cyclograms of the machine components are usually built in the coordinates given by 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.

Study Result	×	
Sensor:	Sensor_1 [Body]	
✓ Create Graph:	Sensor_1 [Body], X Coordinate, X 💙	
Upper Limit:	Sensor_1 [Body], X Coordinate, X	~
	Sensor_1 [Body], Y Coordinate, Y	
	Sensor_1 [Body], Z Coordinate, Z	
Lower Limit:	Sensor_1 [Body], Rotation Angle around X Axis, RX	
	Sensor_1 [Body], Rotation Angle around Y Axis, RY	
Values:	Sensor_1 [Body], Rotation Angle around Z Axis, RZ	
Va	Sensor_1 [Body], Linear Velocity (X), LVX	
X Coordinate,	Consor 1 [Radul Linear Valacity (V) LVV	
Y Coordinate,	Sensor 1 [Reduil Linear Velecity (7) 11/7	
Z Coordinate,	Concer 1 [Reduil Linear Velecity (magnitude) 1V	
Rotation Angle	Sensor 1 [Body] Angular Velocity (V) AVV	

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

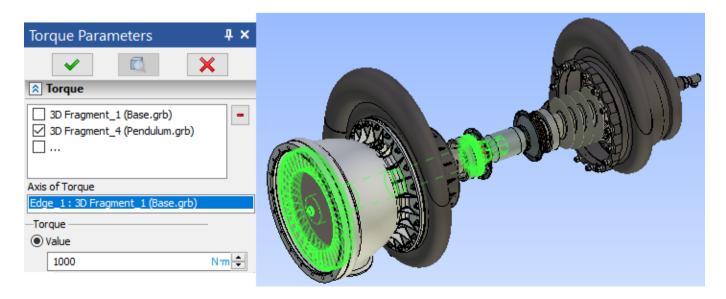
If the driving link of the mechanism has a constant speed and moves rectilinearly in a given direction, you can set the law of its movement using the new **Translation** command.

Starting Velo	city Parameters	Ψ×
~		×
Seneral Para	ameters	
🕜 Blend_13		^
··· ··		■×
Element:	Select element	
	on is bound to body:	
Select element	t	
LCS:	Select element	
Value:	0	m/s 🌲
X:	0	m/s ≑
Y:	0	m/s 🚔
Z:	0	m/s 🚔
Limited Force:	0	N

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.

Force Paran	neters		₽×
×		×	
Refere			
-	t_5 (Little p	um.grb) endulum.grb) endulum.grb)	-
Force			
Value			
100		1	N 🜩

INTERFACE AND VISUALIZATION

Unit Setting

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

Spring For	rce
First Point:	Vertex_1 : 3D Fragment_7 (Little pendulum.grb)
Second Point:	Edge_2 : 3D Fragment_4 (Pendulum.grb)
Force Units:	Newton 🗸
Initial Length:	0 mm ≑

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.

Dragger Force: More More Stop by Dragger Mark Dragger Point Show Sensors
Auto Repeat

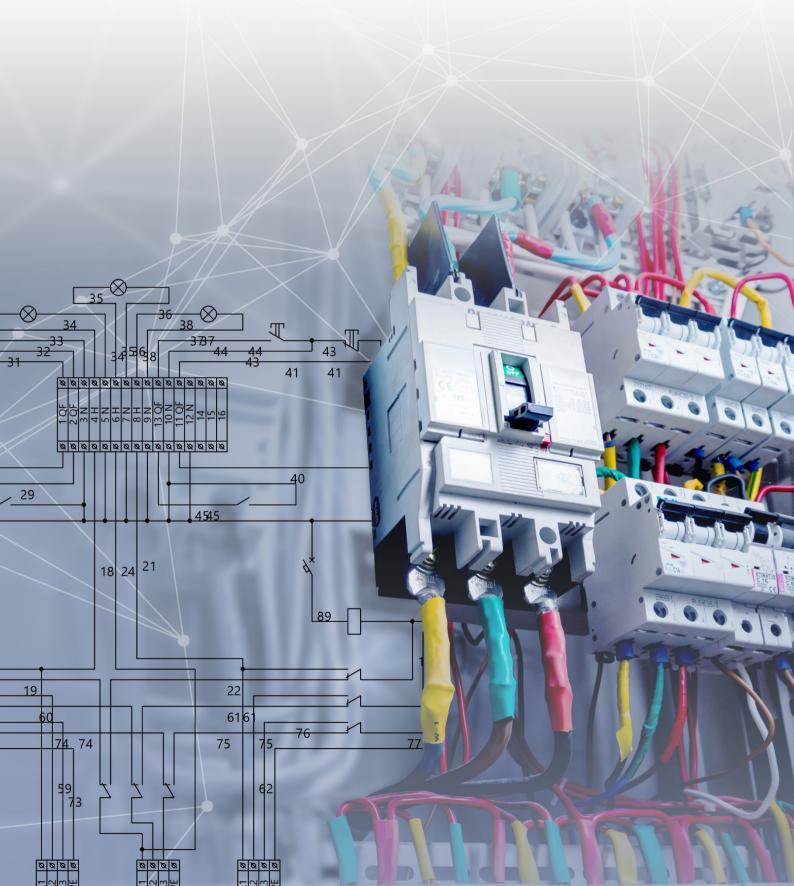
TIME COUNTER FOR CALCULATING STUDIES

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.

Properties				₽×		80							
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Simulation	Parameters												
) 					·					Z		
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Animation Ste	:p: 1			-						20	a	1	
◯ Time Scale:	1	_		*				d	- A		902	3	
Time:	233.888	of	3600.000				4	1 3				Š	
Calculation Time:	3:53.889	of	3:53.889					A	X II				
Frame:	11695 🐥	of	180001								1		X
Dragger Force: More Stop by Dragg Mark Dragger Show Sensors Auto Repeat	Point		1 I	More		<u>H</u>							



What's New



T-FLEX ELECTRICAL 17

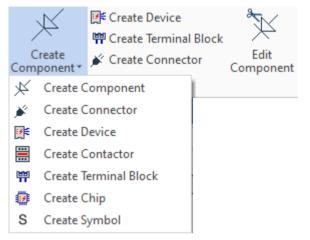
The tools of diagram design and cable components design are expanded in the new version of the T-FLEX Electrical. This made the module even more flexible for solving tasks in the field of electrical engineering.

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 <u>Full/Address Tag Name and Local Tag Name</u>). 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.

Create device				џ
[~			
Electrical Item T	ype			
🔉 Component Stru	icture			
Name	TN	Des	2D Representation Full TN	P.
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> CP 3	3		Contact (number 🖓	£
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I -E USBA_F	X2		USBA_F table 🔐 🗌	65
Common				
-• Power+	1		Fragment Connector	- 6 :1
-• Data-	2		Fragment Connector	-
-• Data+	3		Fragment Connector	
				-
-• Body	4		Fragment Connector	1
	4		Fragment Connector	~
-• Body	4		Fragment Connector	~
	4		Fragment Connector	V
Rarameters	4			
Parameters Parameter	4			~
Parameters Parameter For Reports	4			~
Parameters Parameter For Reports Data for PS from 3D	4		Value	
Parameters Parameter For Reports Data for PS from 3D Description	4		Value USBA_F	~
Parameters Parameter For Reports Data for PS from 3D Description Group Name	4		Value USBA_F Connectors	~
Parameters Parameter For Reports Data for PS from 3D Description Group Name Include in Reports	4		Value USBA_F Connectors	
Parameters Parameter For Reports Data for PS from 3D Description Group Name Include in Reports Manufacturer	4		Value USBA_F Connectors	
Parameters Parameter For Reports Data for PS from 3D Description Group Name Include in Reports Manufacturer Part No.	4		Value USBA_F Connectors	

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.

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Component Structu	re						<u>.</u>	• • •		· · ·	Ĕ		
Name	TN	Des	2D Representation	Full TN	P,	<mark>2</mark> 40	X	• • •	• • •				
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Relay with two	1		Relay with two wind		<u>+</u>	-	_	• • •			Γ		
🖻 🔂 Relay coil	2		Relay coil		69	23	R	• • •	•••		1		
Changeover co	3		Changeover contact				Auto T-0	• • •					
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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

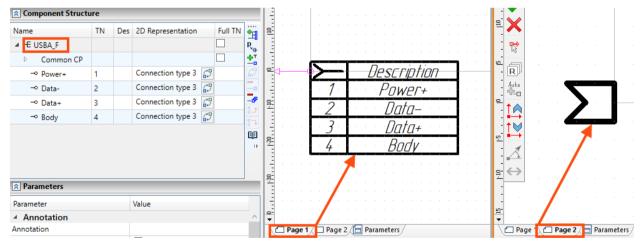
Parameter categories have been added for all diagram elements. This makes it easier to find the desired element information for the user. The following categories of parameters act as the main groupings: For **Reports** (parameters that directly affect the element information included in a text report), **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.

T-FLEX CAD 17 and Add-on Modules. Release Notes

	St	ructural element properties		×
	Pa	arameters Contacts		
	6			^
		Data for PS from 3D	0	A
		Description	Single-phase AC motor	Single-pha 🗎
		Group Name	Motors	8
		Include in Reports	1	8
		Manufacturer		8
		Part No.		8
		Report Name	{Name} {Standard}	Single-pha 🗎
		Short Name	Single-phase motor	Single-pha 🗎
		Standard		8
		Graphic Representation		
		2D Source	<ac motors="">Single-pha</ac>	sse AC m 🔁 🖱
		3D Source		<u> 2</u> 8
M1	· · · •	Tag Name		
I I I M A		Family Name	M	8
\cdots $\sqrt{1}$		Full Tag Name	M1	6
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			OK Cancel	Apply Help

Revised Create Connector Command

Connectors have two 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.

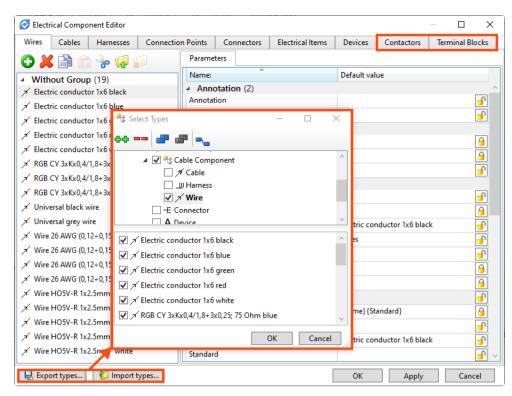


COMPONENT EDITOR

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.

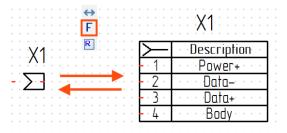
Options		×
Colors	Common File of Components	
Additional Options	Path	😂
Performance	Synchronization Mode	Disabled 💌
Save		Disabled
T-FLEX DOCs		One Direction - From
Electrical		One Direction - To
All		Both Directions

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).

DEVELOPMENT OF ELECTRICAL DIAGRAMS

Converting Connector Graphics on the Diagram

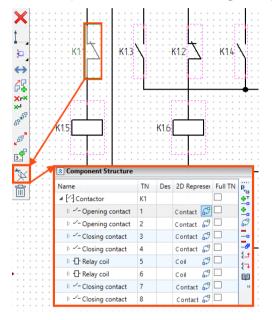
After the work carried out on the connector data model and expanding its capabilities for storing additional graphics, the user can switch the connector symbol from simplified to full (tabular) directly in the diagram context at any time (**F** button in the automenu of connector).



Viewing the Component Structure in the Context of Diagram

Due to the **Component Structure** block has been redesigned (see <u>Diagram Elements</u>) 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.

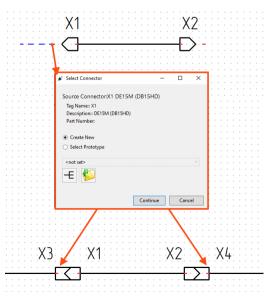
Viewing the complete composition is especially relevant for elements that have a <u>splitted representation</u>, because sometimes it is difficult to find all parts of them on the diagram quickly.



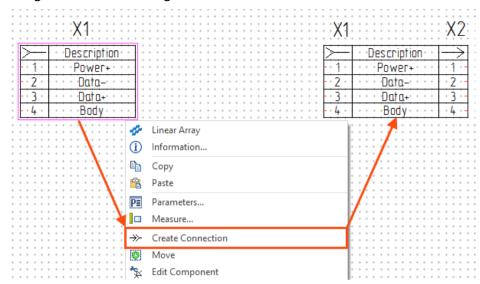
Selection of Mating Compatible Connectors

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.

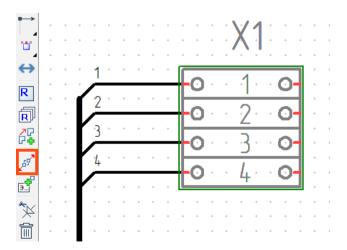
Splitted Representation for All Types of Compound Elements

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 functional groups (see <u>Functional</u> <u>Blocks</u>).

1 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.

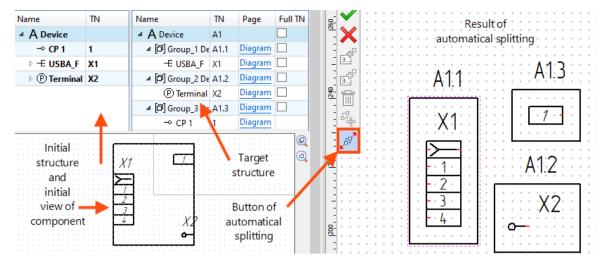
A separate **Splitted Representation** command has been created for splitting elements. It runs from the automenu of the selected item.



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**.

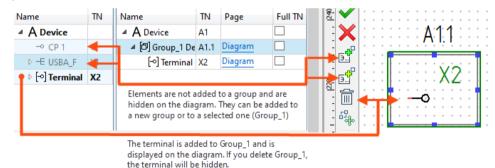
There are two splitting modes available in the command - automatic and manual.

T-FLEX CAD 17 and Add-on Modules. Release Notes



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.

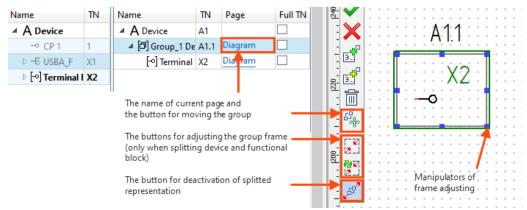
When exploding manually, you can **Add Element to New Group** (the group will be created automatically) or **to Selected Group** (the element will be added to one of the existing groups; the group must be preselected). The element included in the group is marked in bold. For such an element, the commands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

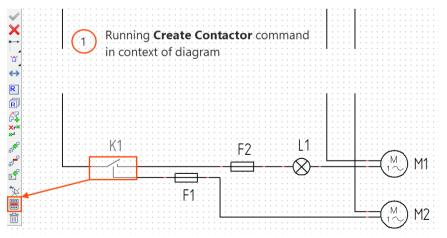
By default, the tag name of group consists of tag name of the initial component, the **Separator** and the **Group Suffix**. In the **Diagram Parameters** window, you can hide the group suffixes in all splitted components in the diagram.

🧭 Diagram Parameters		-	- 🗆	×
✓	Parameter	Value		
Font of Tag Names Font of CP Designations	Tag Names of Element	^		
Font of Designations on Connection Lines	Default Position of Compour	Top Right		•
	Basic Position	Top Center		-
	Visibility of Tag Names	✓		_
	Visibility of Group Suffix			
	Group Separator			_
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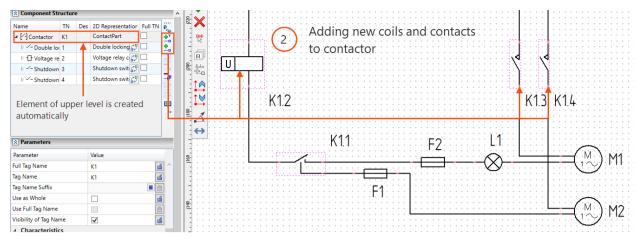
You can also change the separator character, for example, by replacing point with a colon.

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.



T-FLEX CAD 17 and Add-on Modules. Release Notes



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

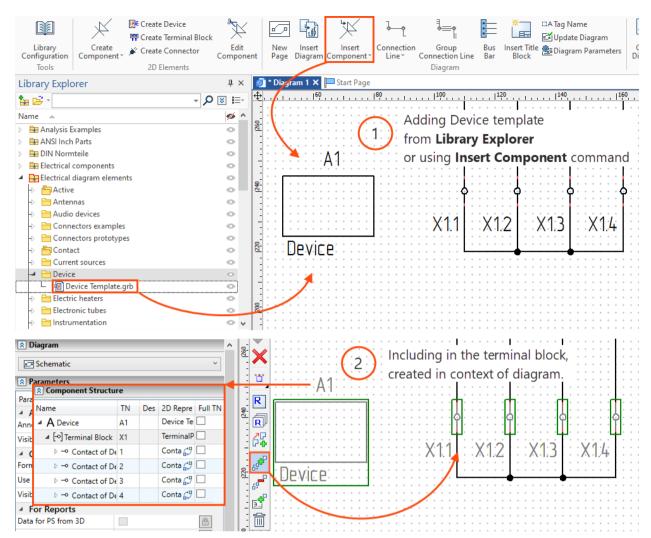
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In addition to the previous two methods of contextual creation of elements, the system also provides a mode of combining several elements into a compound one. According to this principle, it is possible to create both switching elements (**Contactors**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

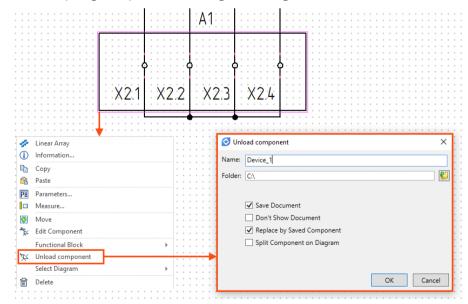
To use this mode, the user needs to add all "parts" of compound element to the diagram (device can be inserted without filling the structure). When you select device, terminal, relay coil, contact or chip, special commands Add Link with Element and Delete Link with Element will appear in the automenu.

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

T-FLEX Electrical 17



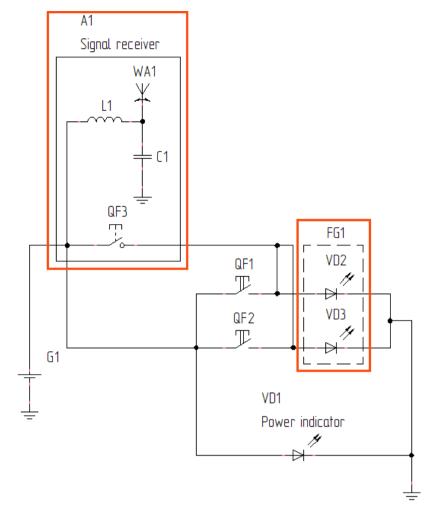
Each element created in any of the above ways can be saved in the library using **Unload Component** command. This means that it has become even easier to create elements and now these actions can be performed without interrupting the process of diagram design.



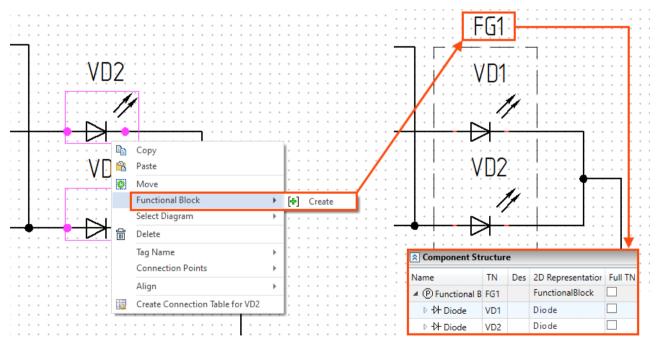
Functional Blocks

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.

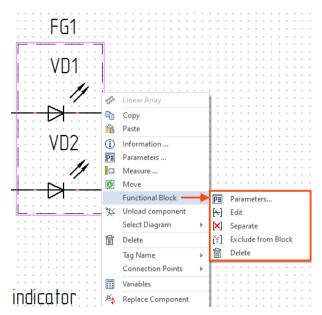
T-FLEX CAD 17 and Add-on Modules. Release Notes



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.

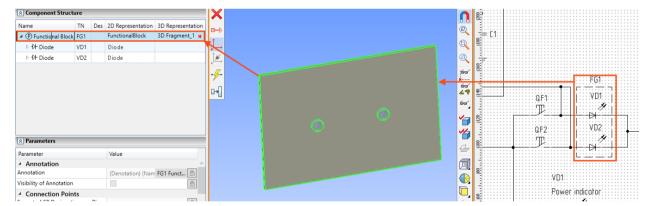


A functional block can be displayed with or without an outline (solid or dashed). In the latter case, the system will automatically turn on the display of the Full Tag Name (see <u>Full/Address Tag Name and Local Tag Name</u>) for elements included in the FG.

Splitted Representation command is available for functional blocks (see <u>Splitted Representation for All</u> <u>Types of Compound Elements</u>).

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

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.

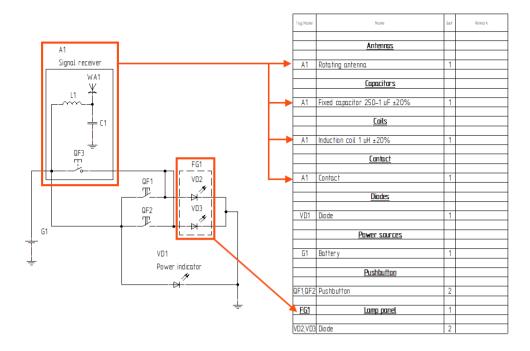


Diagram Parameters

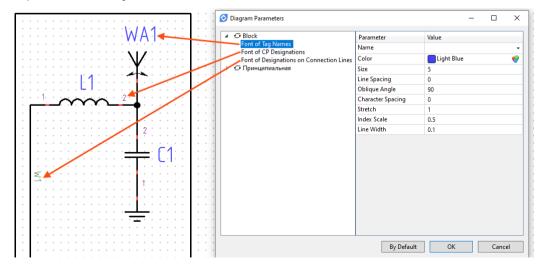
Diagram Parameters command has been significantly improved.

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.

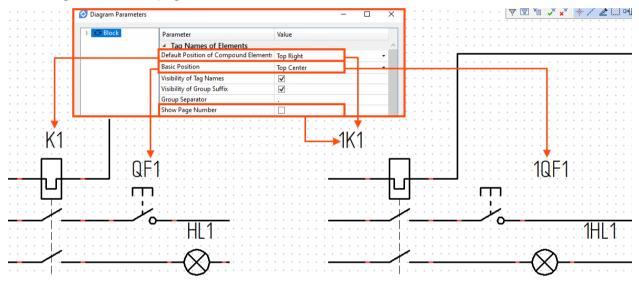
Insert Diagram			ų×
✓		×	
🔉 Diagram File			
C:\			6
▲ Diagrams			
Description:	Impo	rt Diagram P	arameters
✓ Schematic	✓		

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.



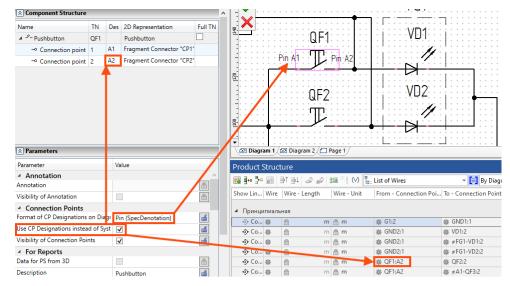
Visibility of Group Suffix and **Group Separator** options are designed to work in splitted representation and are described above (see <u>Splitted Representation for All Types of Compound Elements</u>).

Connection Point Denotations

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

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.

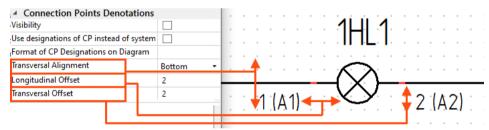
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Use CP Designations instead of System Values checkbox replaces the system designations of CPs with custom ones (in the connection table). The custom value will be taken from **Designation** parameter of connection point.

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.

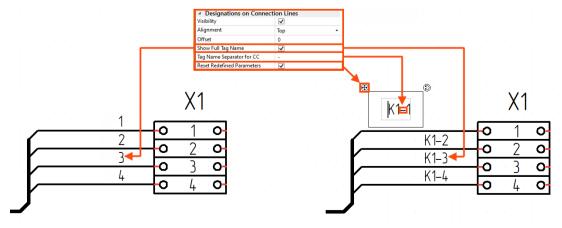
Longitudinal Alignment and **Longitudinal Offset** parameters control the position of the CP signatures relative to the connection lines. The **Transversal Offset** parameter controls the position of CP signatures relative to 2D connectors.



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.

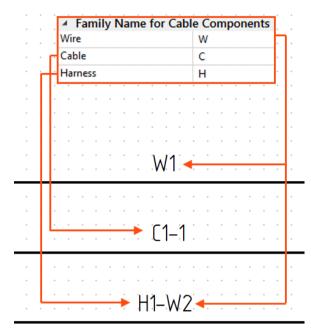
The **Show Full Tag Name** checkbox allows displaying the complete tag name of cable components on the CL. The **Tag Name Separator for CC** parameter sets the character to be inserted between the parts of the full tag name of CC.



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.



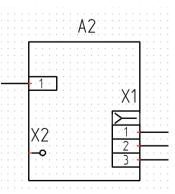
Thus, all cable components will be assigned the tag names using a unified "mask" specified in the diagram parameters.

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.

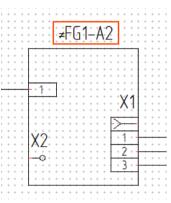
For example, we see X1 connector in A2 device on the diagram.



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.

Structural element properties	×
Parameters Groups Contacts	
2D Source	<connectors examples="">Socket 🔁 🖆 🔺</connectors>
3D Source	2 🔒
Tag Name	
Family Name	X
Full Tag Name	≠FG1-A1-X1
Tag Name	X1 🖆
Tag Name Suffix	✓ A
Use Full Tag Name	0
Visibility of Tag Name	✓ 1

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.



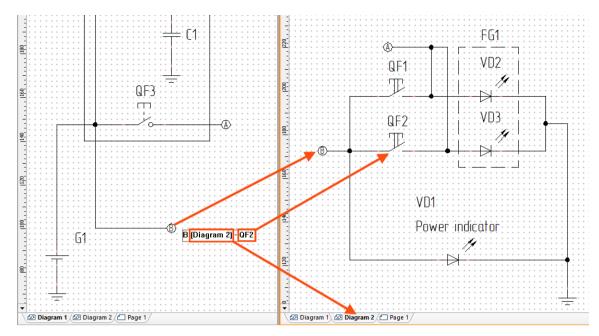
Quick access to **Use Full Tag Name** checkbox is provided in all component editing dialogs at all nesting levels (**Full TN** column).

Component Str	ucture					^	250				Į	+	+	*	*	*	
Name	TN	Des	2D Representation	Full TN	 ₽,₩			X	• •	• •		•	•	•	•	•	•
▲ A Device	A1		Device Template		-1# 			₽\$				+	+	+	+	+	+
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▷ -E USBA_F	X1		USBA_F table 🛛 🖧 🖞	✓		Γu	se F	ull TN	for	Ne	este	d E	len	ner	nts		
▷ [-•] Terminal	X2		Terminal			D	on't	Use F	ull 1	τN	for	Ne	ste	d E	ler	ne	nts
						In	ivert	for N	este	ed	Eler	ner	nts				
						E	(pan	d Ne	sted	Ele	eme	ents	;				
						С	ollap	ose Ne	este	d E	lem	ien	ts				

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.



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.

Edit electrical item		ų×
~		
Scomponent Structur	e	
Rarameters		
Parameter	Value	
 For Reports 		^
Data for PS from 3D		e
Description	Fixed capacitor	A
Group Name	Capacitors	e
Include in Reports	\checkmark	A
Manufacturer		B
Part No.		B
Report Name	{Name} {U}-{Nominal} {Nc Fixed cap	B
Short Name	Capacitor	B
Standard		≜ ~

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.

Edit electrical item		4 ×	🔊 Fixed capacitor.grb 🗙			
Electrical Item Type			 ⊕1⁻²⁰ = ⊕1⁻²⁰ ⊕1⁻¹⁰ ⊕1⁻²⁰ 	<u>10111011201</u>		
S Component Structure						
Parameters						
Parameter	Value		₽₩	•		
Annotation		^				
Annotation	{	Fixed cap				
Visibility of Annotation	Name		Value	Comment		
Connection Points	{Annotation}	{Name} {U}-{N	lominal} { ±{Tolerance:perce	Annotation		
Format of CP Designations on Diagran	{DataForPsFrom3D}	0		Data for PS from 3D		
Use CP Designations instead of Systen	{Denotation}			Tag Name		
Visibility of Connection Points	{FileLink2D}	<capacitors></capacitors>	Fixed capacitor.grb	2D Source		
For Reports	{FileLink3D}			3D Source		
Data for PS from 3D	{FullDenotation}			Full Tag Name		
Description	{GroupName}	Capacitors		Group Name		
Group Name	{ }	0		Rated Current		
	{l:unit}	А		Unit - Rated Current		
	{IncludeReport}	1		Include in Reports		
	{Manufacturer}			Manufacturer		
	{Name}	Fixed capacito	or	Description		
	{Nominal}	1		Rated Capacitance		
	{Nominal:unit}	μF		Unit - Rated Capacitance		
	{PartNumber}	-		Part No.		
	{PinFormat}			Format of CP Designations on Dia		
	{Power}	0.5		Rated Power		
	{Power:unit}	w		Unit - Rated Power		
	{PreDenotation}	С		Family Name		

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 **Part is located in the dialog of Create <Component>** and **Edit Component** commands.

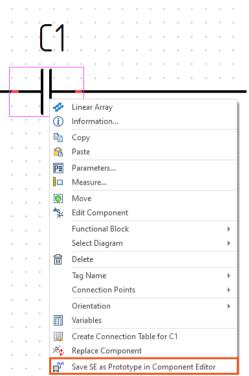
The command is available when creating the compound components in a separate file and in context of diagram.

Edit connector							ų.		👰 * Connector 1					_	agram 1		Start Pa	-			
[~			×					⊕	-10		⁰	1	. 10	n da s		tra	30		140	
Component Struct	ure							^													
Name	TN	Des	2D Repre	sentation		Full TN	•		×												
✓ -E USBA_F							P		원 🙀 🗤 🗤												
Common CP								÷	DE15F (DB15HD)												
-• Power+	1		Connect	ion type 3	69		69	-E	DE15M (DB15HD)			4								-	
-• Data-	2		Connect	ion type 3	69			Æ	Socket				• • •	T		2	•	1.		- X	
-• Data+	3		Connect	ion type 3	69	<	< .	÷	USBAF		┣	•>			L	IPS	ΕΓΙμ	ЭТIC	- חו		
-• Body	4		Connect	ion type 3	69		5	÷	USBAM			<u> </u>	1	1		D.	21.10			-1	
							60	¢۵	Select				1			PL	JWE	// +			
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Parameters		_			_]			-		e	.3			Ľ	Tate	<u>[</u> +]	
Parameter			Value						<u>~</u> →≻- <u>8</u> -	4		.	1				Ree	4/			
Annotation							^		E	1.			7			<i>L</i>	100	/		┛╹	
Annotation									-	1		1									

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.

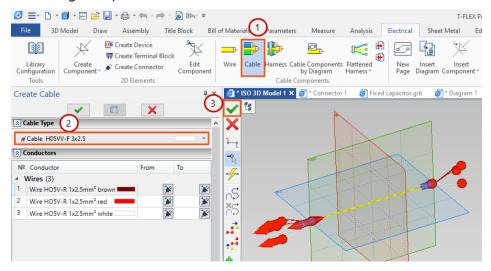


This function is also useful if parameters of a library element have been changed in the diagram, and a prototype with such parameters may be useful in the future.

3D CABLES AND HARNESSES

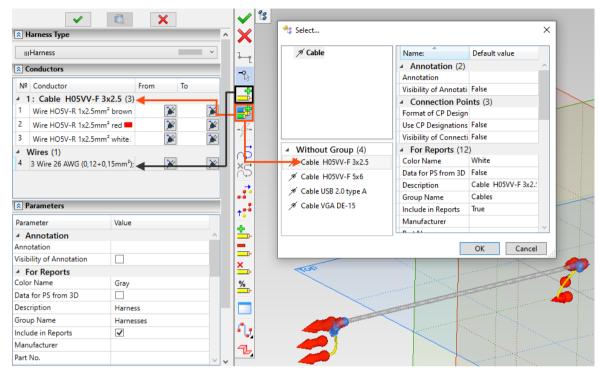
Creating Cable Components without Diagram

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.

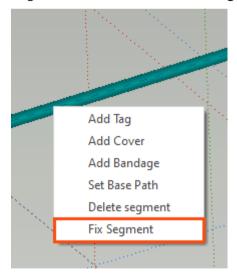


New Functions for Managing Topology and Composition of Cable Components

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

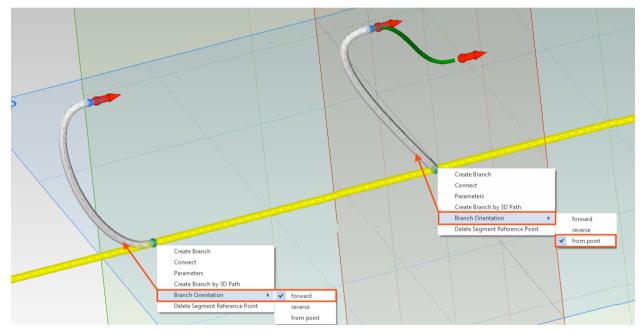
The entry and exit points for wires into and out of the harness can now be set manually. To do this, use the **Connect** command. The command is run from the context menu of endpoints and branch points of the harness.

		Result
Add Stripping of Shield Create Branch Connect Segment Reference Point Branch Orientation	Select - X Not specified Not connected	
	4 ОК Отмена	

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**.

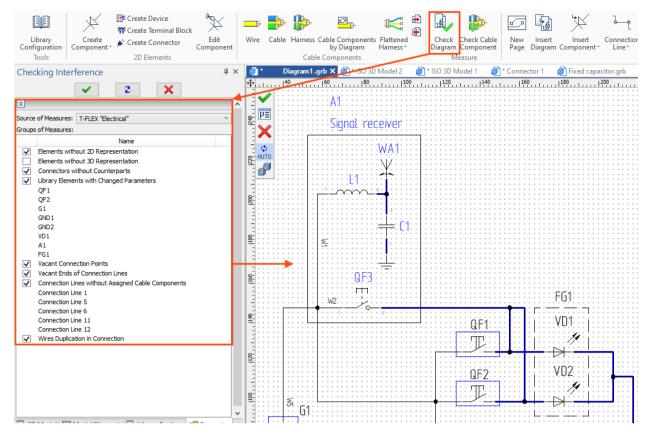


FUNCTIONS OF CONTROL OVER DESIGNER'S ACTIONS

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 diaplayed in the corresponding service window when the checking parameter is selected. When errors are detected, the system highlights the results in the 2D scene.

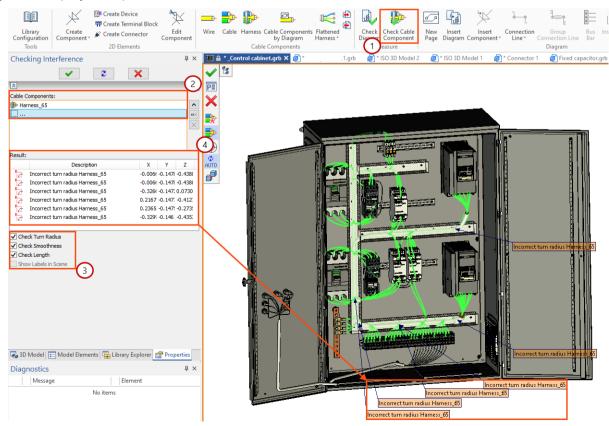
T-FLEX CAD 17 and Add-on Modules. Release Notes



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.



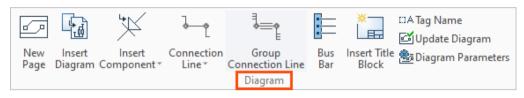
The result of the check can be saved as a permanent set of labels in the 3D scene, which is provided by a special option in the automenu of **Check Cable Component** command.

INTERFACE

Redesigned Ribbon

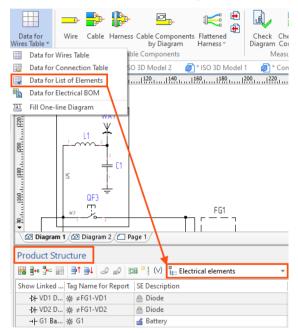
The Electrical tab of the Ribbon panel has been redesigned.

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.



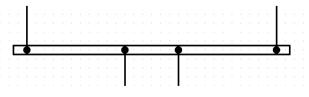
New Commands

New commands were added to PS (product structure) 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 Structure** 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.



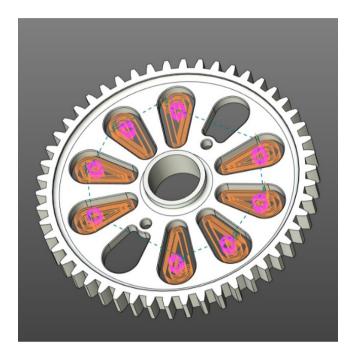
What's New

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).

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4,9	

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.

T-FLEX CAD 17 and Add-on Modules. Release Notes

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Toolpath:	Copy Arra	y1		
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Symmetry		~		
Step:		10	ð	
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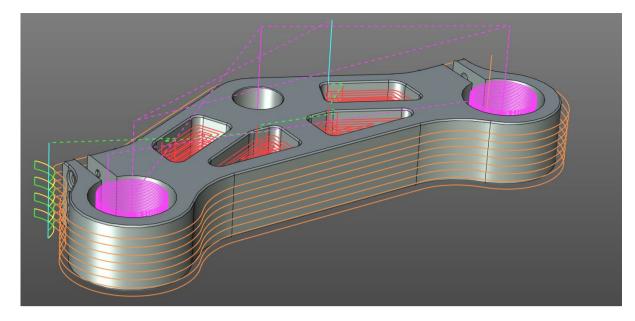
Start/Return Point has been added to override the start of the first and return of the last toolpath of the array.

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		Paste		
		Delete		
		Movin On/Off		
		Start/Return Point		
		Cutting-in		

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.

Toolpaths Properties		×			
Relative Coordinate System 3D Mill Wire-cut Laser Turning Drilling	ng 5D Milling Milling Punching	5D Drilling Measurements	Section type:	Main 🗸	
Compound toolpath status Active Level: Default V	Not active Level: 0 🜩 Layer: Default	~		Idling Approach	
Color: Co	Color:	§ ↓ Absolute		Withdrawal Leadin Leadout Cutting-in	
Section type: Main V Save colors as default settings	ОК	Отмена		outing in	

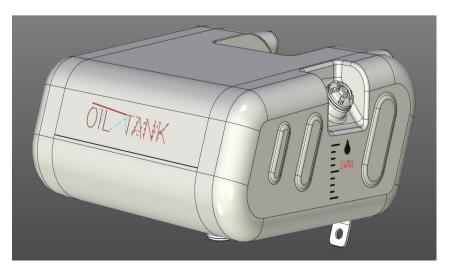


ENGRAVING

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.

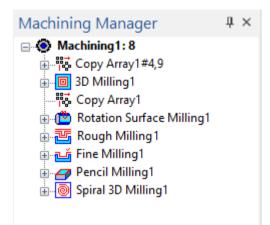


USER INTERFACE

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, cuttingin - 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.

achining1 : 6 Pocket Milling1	Toolpaths Common Parame	ters	×
Pencil Milling1 5D Zone Milling1 5D Zone Milling2 5D Drilling1 5D Drilling2	Toolpaths parameters Toolpaths name: Tool file: Tool name:		
	Technological parameters Working feed rate: Rapid feed rate: Spindle RPM: Cooler:	50 100 1000 On/Off	
	Approximation Precision: Circular interpolation: Cutting control:	0.1 On/Off On/Off	

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.

T-FLEX CAD 17 and Add-on Modules. Release Notes

CAM Options	
Working folders:	
C:\Program Files (x86)\T	T-FLEX CAM 17\Program
CAM Libraries Postprocessors NC Programs Tools	C:\Program Files (x86)\T-FLE C:\Program Files (x86)\T-FLE C:\Program Files (x86)\T-FLE
Toolpath calculation paran Recalculate toolpaths Disable all trajects whe Show calculate time Active display of equip	on complete recalculation en creating the new one
Tooltips settings	
✓ Overcutting control ✓ Up to first overcut	
Simulator option	
ОК	Cancel

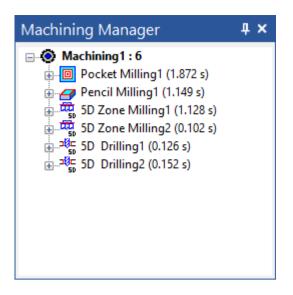
If the flag is cleared, then the scene displays the equipment bodies of all machining, and if set, only the active machining. To activate the option after changing the state of the flag, double-click on any of the machining in the machining manager tree or on the **Update Model** button. In this case, 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**.

CAM Options	×
Working folders:	
C:\Program Files (x86)\	T-FLEX CAM 17\Program
CAM Libraries Postprocessors	C:\Program Files (x86)\T-FLE C:\Program Files (x86)\T-FLE
NC Programs Tools	C:\Program Files (x86)\T-FLE
Toolpath calculation para	meters on complete recalculation
	en creating the new one
Tooltips settings	
Show tooltip window	
Overcutting control	
Up to first overcut	
Simulator option	
Consider all machining	I
	OK Cancel

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.

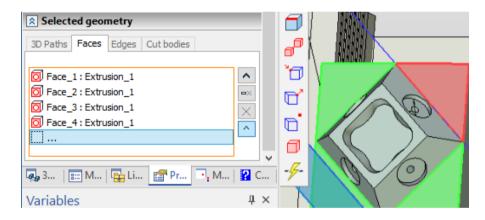


The calculation time is not displayed in inactive toolpaths. To remove the calculation time, you need to clear the flag and recalculate any trajectory by RMB.

"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 ¹) 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.

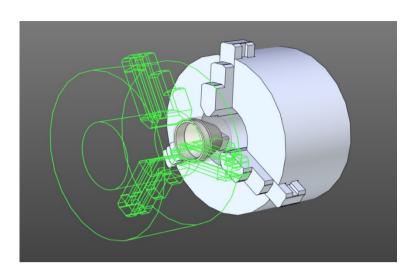
In addition, it became possible to use the button \frown to change the position of elements in the list by moving the selected element up in the circle. Thus, it is possible to change the machining sequence of individual elements when calculating the toolpath. This button is only available for faces 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.

T-FLEX CAD 17 and Add-on Modules. Release Notes

Machining Parameters	₽×
✓ <a> 	
Parameters	
Trajectname: Machining Parameters1	
Preform Allowance: 0.1	
Equipment Allowance: 1	
Cutting Allowance: 0.1	
Tool Changing Point	
X: 0 Y: 0 Z: 0	
Tool file:	
Workpiece by part dimensions:	
Parallelepiped Cylinder Cylinder axis: X Y Z	
Selected geometry	_
	52
Preform: 2 3D Fragment_2	
Part: Cylinder_1	\times
Equipment	
3D Fragment_1	^
SD Fragment_1	
[L	



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.

Machining recount X		×
Tr 🔽	ng control: ansitions betwe polpaths	en trajectories
OK Cancel		Cancel

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, recountion 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 O Ser		
—Previous mashining para	m ifinished meters ——	
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

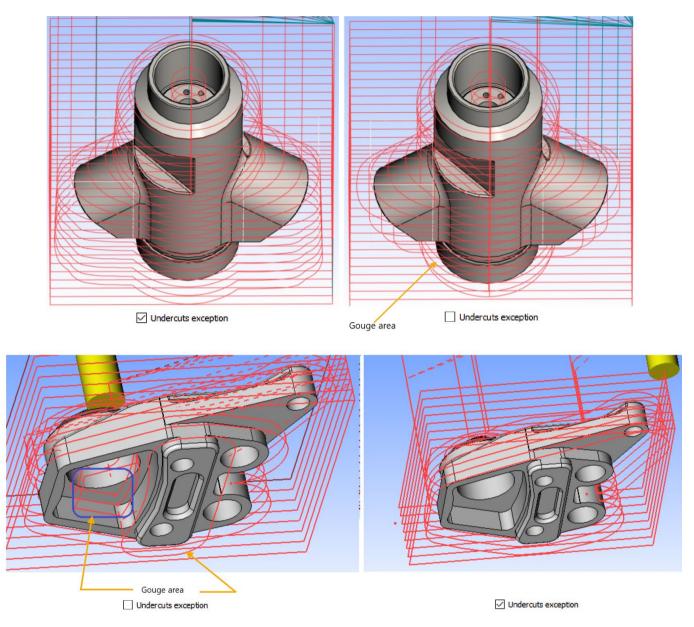
Previously, machining for some types of passes was performed not in layers, but in local zones ("wells"), where possible, in order to reduce the number of transitions at idle speed. In some cases, this can lead to an incorrect toolpath, in which the tool cuts immediately along the bottom layer bypassing 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.

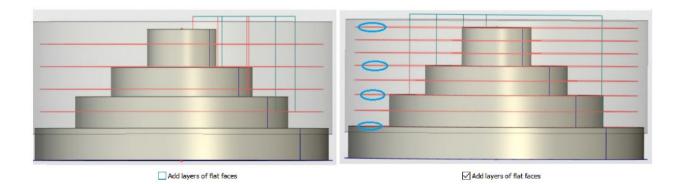
Now, to take into account undercuts, it is enough to set the corresponding flag in the **Additional Parameters** tab. In this case, each subsequent section is created taking into account the machined parts of all upper sections, which excludes the cuttings associated with undercuts.

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.

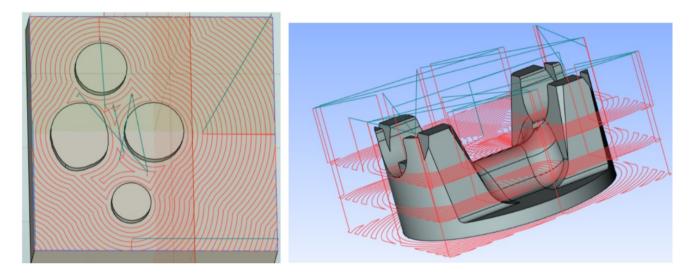
Expansion of functionality for certain types of passages

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.

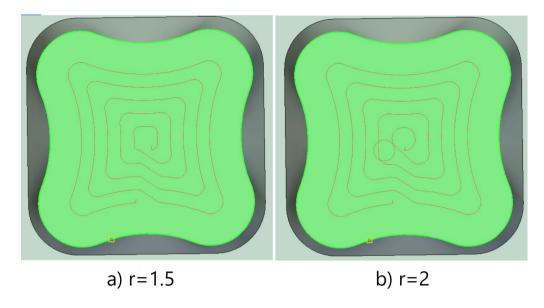
This option is necessary to reduce the inertial load on the machine's actuators. This allows you to increase the machining modes and surface roughness, and hence reduce the manufacturing time.

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.

Additional Parameters	
Boundary Allowance:	0
Fillet Radius:	0
✓ Smooth transitions , R:	0
Outside offset	0
Use part dimensions testing	
Cut by layers	

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, 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.



1 Note: This option has also been added to pocket milling.

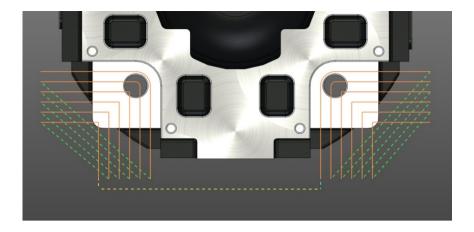
POCKET MILLING

Open Pocket

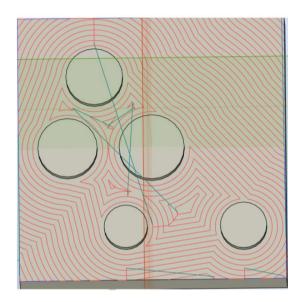
In the boundary 3D milling type **Pocket Milling**, the **Open Pocket** option has been modified. In particular, for the **Equidistant Zigzag** pass, an **Outside Offset** was added, and for a pocket with a closed outer contour, the case of parts with several inner protrusions or shaping contours was taken into account. Previously, only one was possible.

Pocket Milling		ąх
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Pass type		
Equidistant Zigzag	\sim	
──loollift: ── ○Abs.	5	
-		
O Step of passes: 10		
Crest height:		
Processing overlap: 0.5	_	
		- 1
Stock Removal Para	meters	
Thickness: 0	0	
Step of a deepening: mm	10	
Side offset:	0	
Pass direction (grad.):	0	
Contours Filter		
All	\sim	
2.5D Plane Milling		
Passing milling Cour	nter milling	
🗹 Optimised Offset		
🗹 Open Pocket		
Outside offset:	1	
😵 Path trimming		
Plunge Machining		
Vigh Speed Machini	ng	
Additional Paramete	rs	~

T-FLEX CAM 17

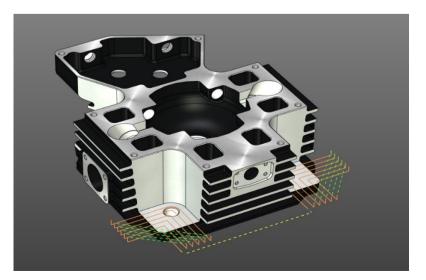


This option allows you to process open pockets using different strategies. Pockets can have a variety of configurations of two main types: with one or two open sides without internal protrusions (see figure above) and fully open, bounded by a closed loop with one or more internal protrusions.



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.



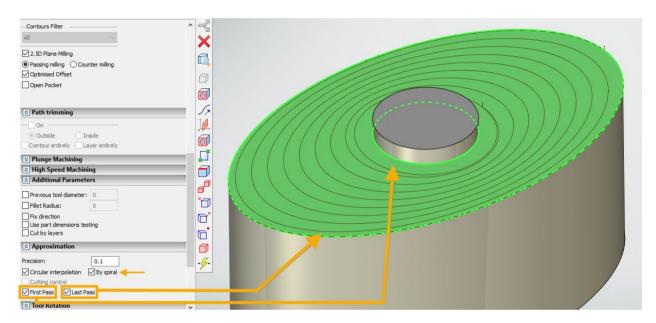
Setting with one face for a fully "open pocket" with internal ridges

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.

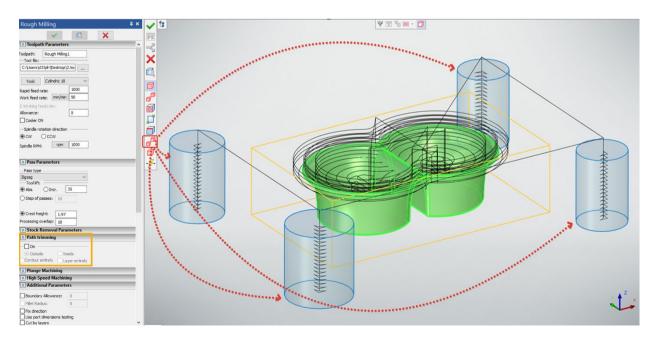
"By spiral" Option

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.



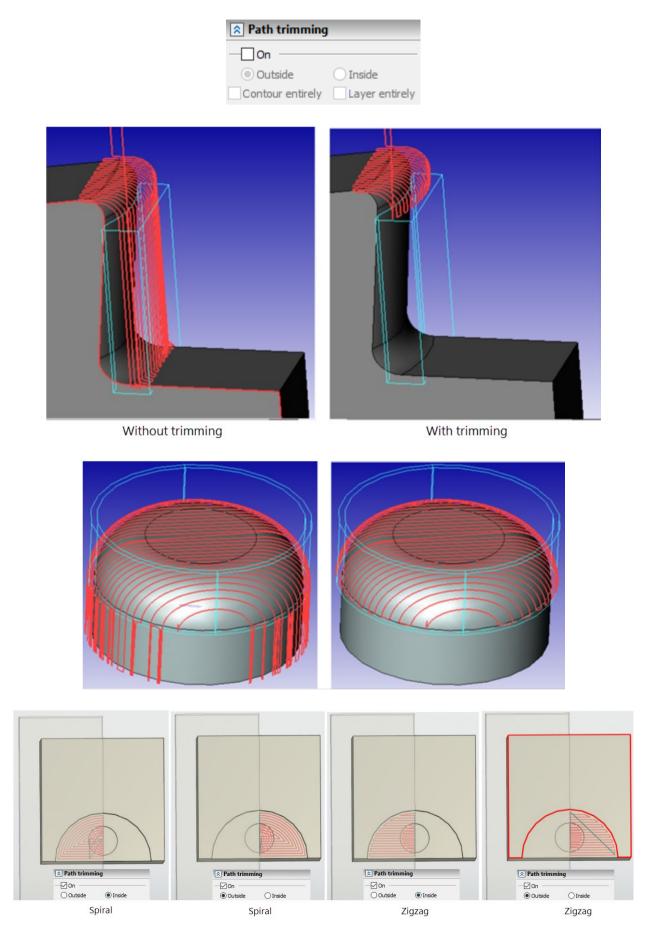
PATH TRIMMING FOR BOUNDARY 3D MILLING

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.

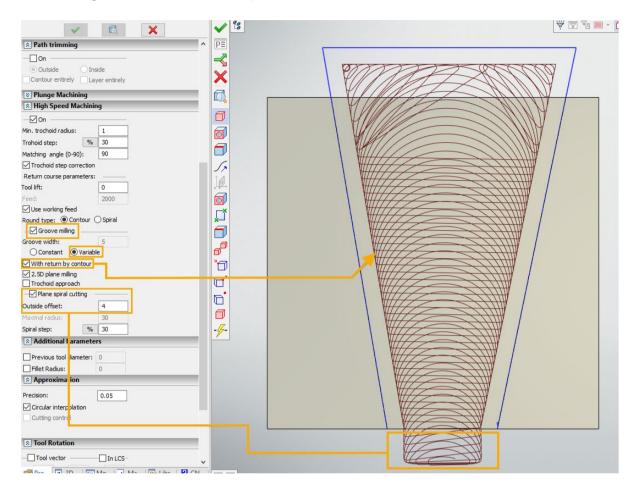
The path and layer options allow you to remove both the path inside the trim body and the entire layer. The set of options may differ for different types of machining.



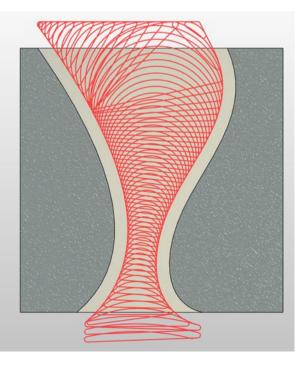
PENCIL MILLING

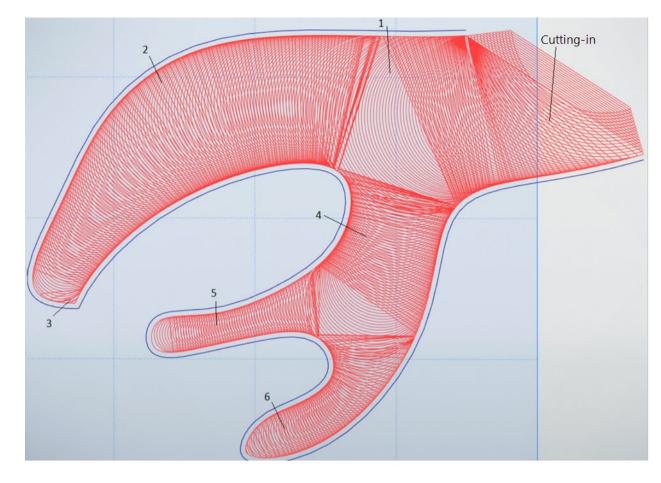
High Speed Machining

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.



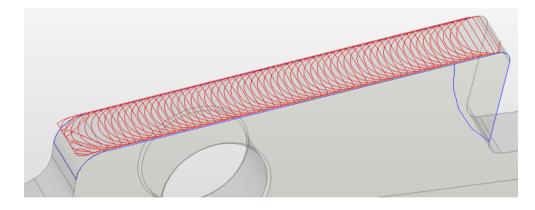
Now it is possible to machine grooves of variable width, the shape of the grooves is practically unlimited.





Provides the ability to work with a complex guide path that has multiple branches, as shown in the figure.

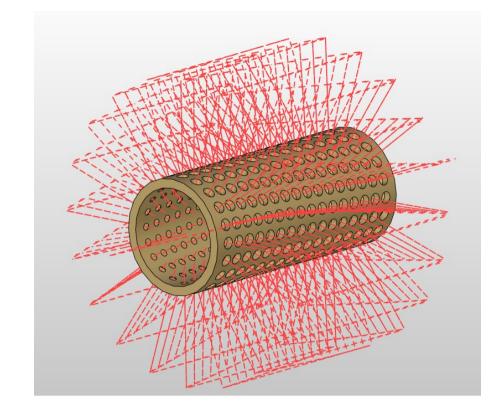
Resetting the **2.5D plane** flag allows you to project with collision control (by analogy with Pocket Milling) the high speed milling toolpath onto an inclined plane or curved surface.



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.

▲ Tool Rotation	
Tool vector	In LCS-
X: 0 Y: 0	Z: 0
Kotation angles	
Z-axis;	0
🔿 X-axis: 🔘 Y-axis:	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.

A Hole Filter		
— Diameter Filter		
From: 0	To: 1000	
— Depth Filter —		
From: 0	To: 1000	
_Depth		
🔵 Equal 🖉	Auto	

The **Auto** mode determination of the depth of through holes when selecting their cylindrical faces has been improved.

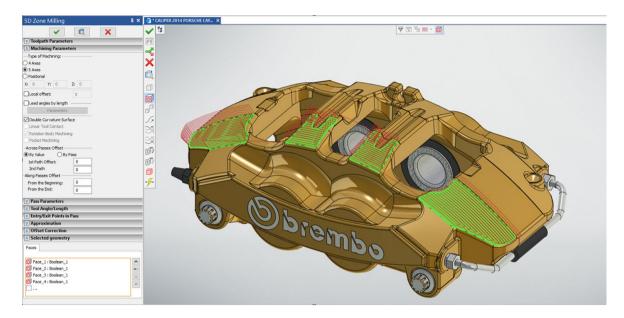
When forming the NC, the case of drilling holes of different orientations in one toolpath was taken into account.

Selecting one flat face uses a simplified calculation method.

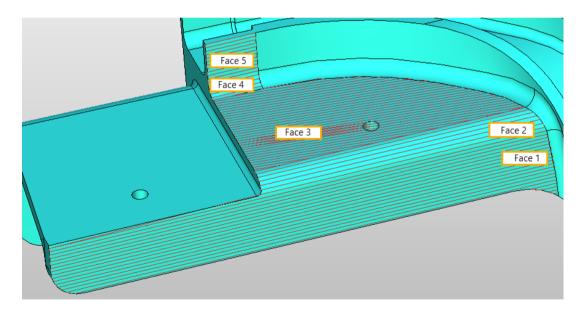
Added recognition of incomplete holes when selecting a face.

SELECTING MULTIPLE FACES IN 5D ZONE MILLING

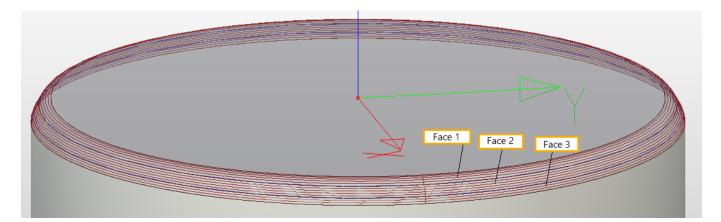
The ability to select multiple faces in 5D zone milling with the ability to quickly transition from one face to another has been added.



The merging of the passes of individual faces has been implemented, where possible.



The case of 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.

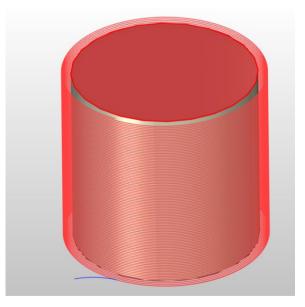
The parameters, the change of which starts the ATR, include all technological parameters - feed rate, spindle frequency, etc., Allowance, Thickness, Step of a deepening, parameters of High Speed Machining, Trimming, Plunger Machining, Radius Correction (for a pocket), Offset Correction, Circular Interpolation 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.

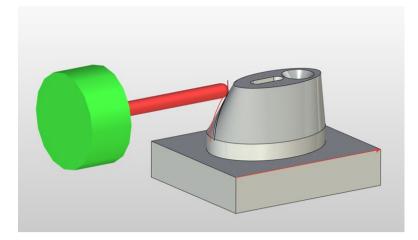
CUTTING-IN, START/RETURN POINT, MOVING ON/OFF UPDATING

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.

Number of teeth:	1		Undo	Apply
Tool list	Сору	Clear All	Clear	Leave
Туре	Name Code	Number		
Pass Cutter	Cutt	1		
Pass Cutter	Cutter	1		
🎄 "Dovetail" Tool	Tail	1		
📱 Cut-Through Cutter	Cutt	1		
🏶 Ext. conic tool	Ext	1		
<				>
Add Open From DO	OCs	Save	Save as	Close

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.

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Imitator Settings		×
Tool Cut part Cut part color: Not cut part Toolpath	Part Preform	Equipment
Hide toolpath outline	Overcutting frames only	
Enable bounding	Blocks number for animation:	4
Switch On/Off 3D Scene Area		
X Axis	Width:	10
Y Axis	Width:	10
Z Axis	Width:	10
Imitation speed	Break	the preform
Visualization accuracy		0.1
Pass circle arc fully	OK	Cancel

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.

Imitator Settings	×
Tool Cut part Cut part colc Cut part Not cut part All	Part Preform Equipment Color:
Toolpath	
Hide toolpath outline	Overcutting frames only Blocks number for animation: 4

«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.

Overcutting control		
Simulator option		
	ОК	Cancel

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.

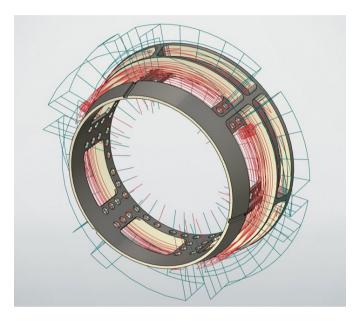
Now, when entering the machining simulator, a warning is displayed indicating the name of the toolpath, if a tool is not specified for any of them.

Also in the simulator, the operation of the **Enable bounding** of 3D scene option has been restored and optimized.

Toolpath		
Hide toolpath outline	Overcutting frames only	
Enable bounding	Blocks number for animation:	4

GOING ON GO 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.

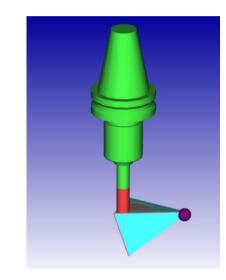
The Saving Parameters of Compound Toolpath				
Compound toolpath name:	Machining1	~		
NC program file name:				
Postprocessor files names				
Wire-cut:				
Laser cut:				
Drilling:				
Turning:				
2.5D Milling:				
3D Milling:				
5D Milling				
Punching				
Measurements:				
External postprocessor:				
Save as default parameters	OK	Cancel		

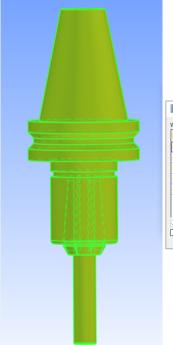
USING THE FRAGMENT OF AN INSTRUMENTAL ASSEMBLY

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

TOOLPATH CREATION FROM NC

The ability to create a toolpath using NC has been added. This possibility is realized by means of repostprocessing 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.

File					CNC						
	<u> </u>		111 50	≬ ↓ [5D	177 30	50 50	\mathbf{D}	NC ←		1	*
Settings *	Tools *	Boundary 3D Milling	Boundary 5D Milling	5D Drilling	3D Milling	-	Cam Machining	Lo NC-pre			dinate tem
						Ma	chining				
Machir	ning M	lanager							д	×	8
	I Plane Reduc Toolpa Pocke Toolpa Count Count Conto I Engrav	tion ath4 ts ath4 erbore 9 vur	g\OKLAD41	.NC)							

OTHER **I**NNOVATIONS

- 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".

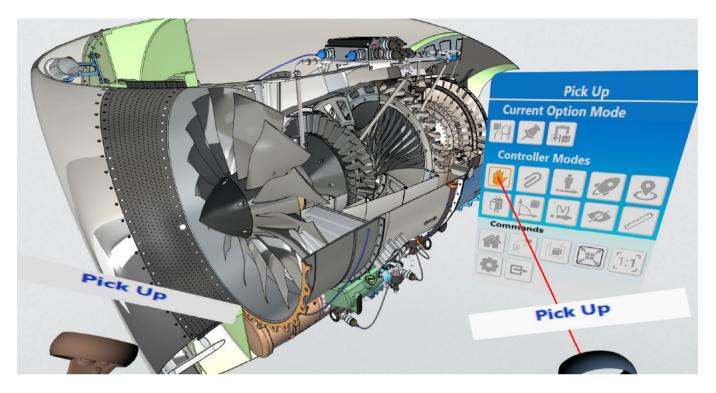
- For the technological toolpath and all 2D milling operations, including engraving, added the ability to edit parameters from the 3D window.
- When cutting-in into a finished hole, angle A is measured in increments relative to the tangent to the toolpath.
- For 5D zone milling in the **Positional** 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.



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:

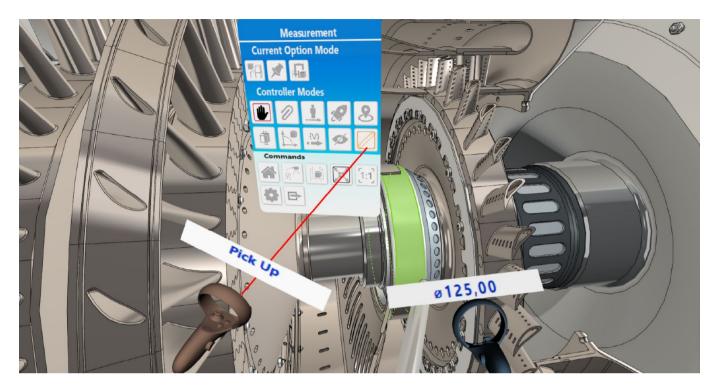
- Upper part of the menu displays current mode of VR controller.
- A list of additional options for current mode or command of VR controller is shown below.
- Next the list of modes available for VR controller is disposed.
- Bottom part displays the commands available for selection in the VR menu.

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.

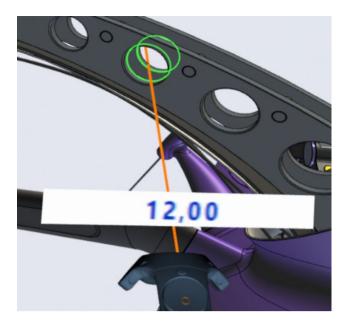
Elements. If a straight edge is selected, the length of the edge is measured; if the arc of a circle is selected, the diameter of the circle is measured.



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.

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Dynamic Highlighting. The option highlights the elements pointed by the ray.

Contact us to request information about T-FLEX software, our Academic Program, or if you have ideas on cooperating with Top Systems

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