

T-FLEX CAD 16

WHAT'S NEW



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WELCOME TO T-FLEX CAD 16

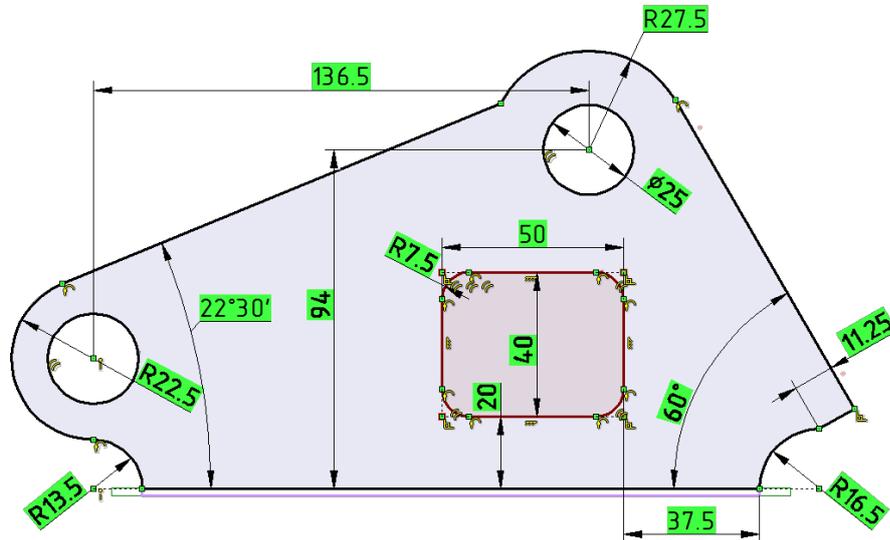
T-FLEX CAD 16



T-FLEX CAD 16 offers various new features and improvements, many of which are implemented in accordance with the requests of users. Among them - new possibilities for parametric drawing, effective tools for working with complex assembly products, new technological solutions for electrical engineering and sheet metal, design in virtual reality and much more.

New Tools of Parametric Drafting

New mechanism for creating parametric drawings based on variational parametrization is added. It allows you to create parametric drawings and create profiles of future 3D models without the use of construction lines. For the sketch lines, you can now apply various geometric and dimensional constraints that determine the geometry of the project. The new mechanism is another way of defining the relationships of the drawing elements along with the existing mechanism based on the construction lines.

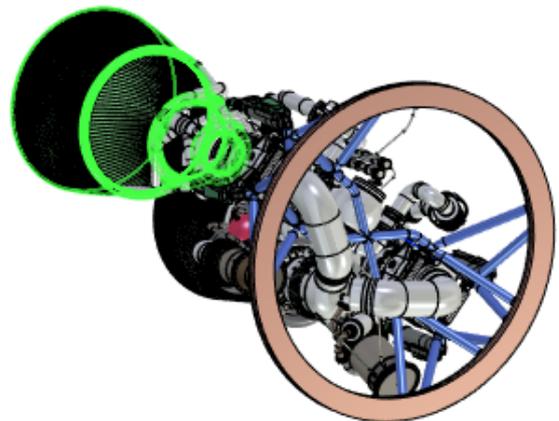
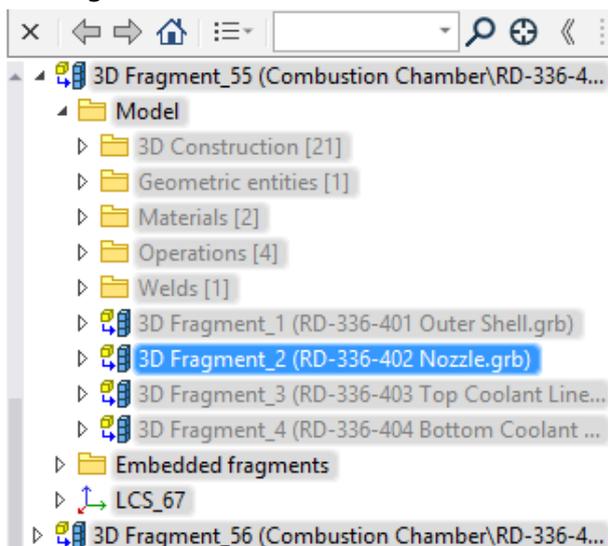


With this way of defining relationships users do not need to think in advance the order of lines creation, since their position and relations to each other can be changed at any time.

Both mechanisms can be used both individually and in combination, providing users with a wide range of options for solving problems of different nature.

Flexibility and Efficiency when Working with Large Assemblies

Improvements in the new version for working with assembly models will be of great use in the design of complex products, solving common problems of such industries as automotive, aerospace, shipbuilding, engineering, etc.

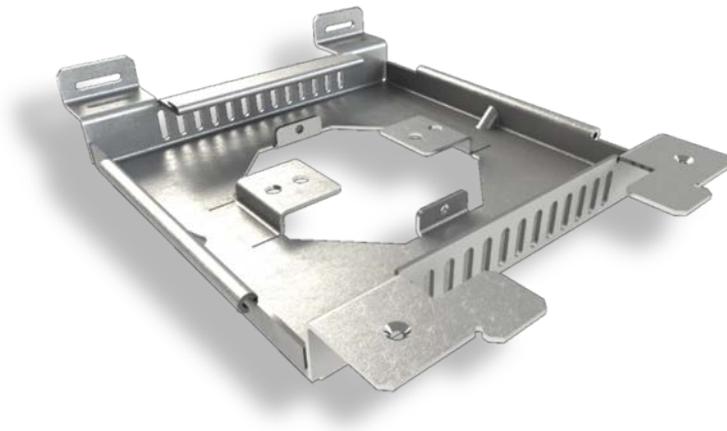


New tools and improvements to existing assembly modeling mechanisms provide efficient ways to work with any objects throughout the entire hierarchy of a complex product, edit parts directly inside the assembly itself, use geometry of any objects as the basis for new parts.

Sheet Metal Design

Tools for designing sheet metal parts were significantly improved in the new version.

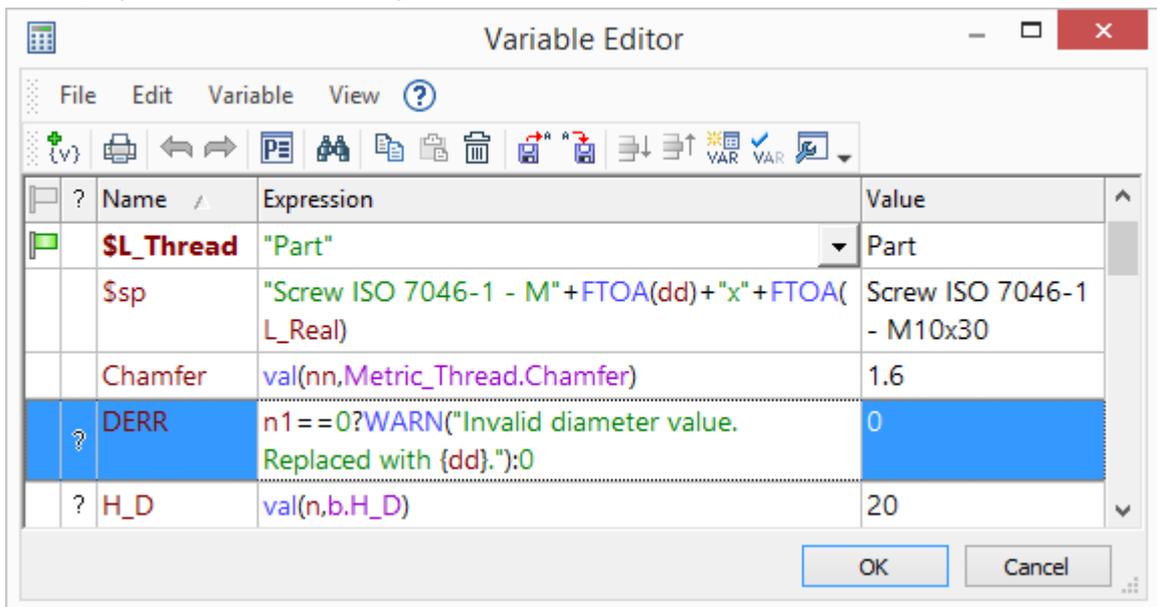
New operations - "Corner", "Normal Cutout", "Gusset" and «Hem" - were added.



It is now possible to create flanges on a custom 3D Profile. User interface of operations has been updated, new features have been added to the existing commands, a semi-transparent dynamic view has been implemented, and functionality of manipulators has been improved.

Updated Variable Editor

Variable editor now displays expressions on the screen as a whole, highlights the syntax in different colors and helps you find the necessary functions and variables from the list.



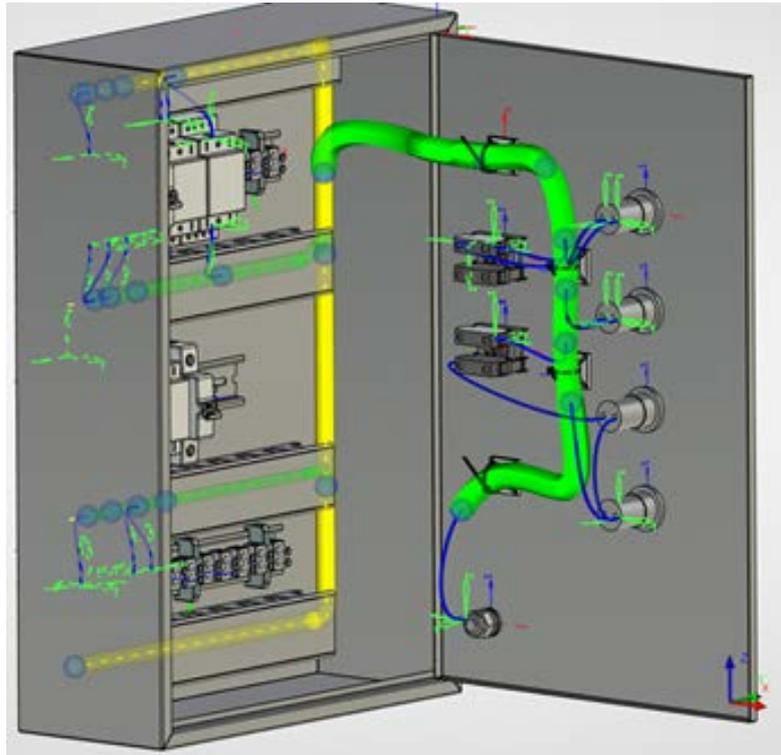
Hints and new functions were added, and much more.

If several documents use the same variables, they can now be stored and updated in one single file.

Detailed information on these and other innovations in T-FLEX CAD 16 will be provided later.

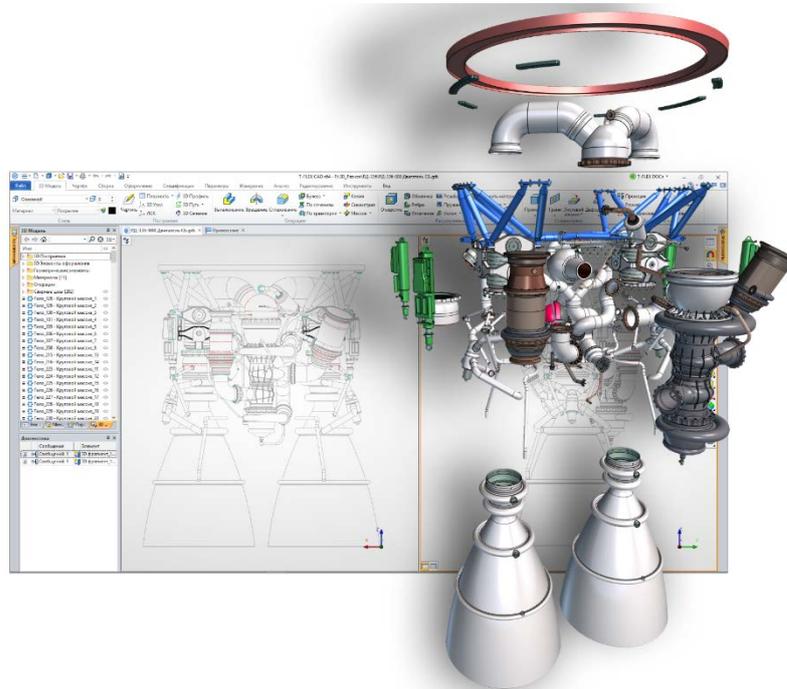
New Add-on Application for Electrical Design

T-FLEX Electrical - a new fully integrated add-on application for T-FLEX CAD. It provides a wide range of tools for development of electrical systems: from construction of two-dimensional circuits to the placement of 3D models of electrical components, creation of wires, cables and connections, harnessing and generation of fully integrated specifications, which include both mechanical and electrical parts.



Design Using Virtual Reality

T-FLEX VR is a new integrated application, which allows you to work in T FLEX CAD 16 using virtual reality technologies.



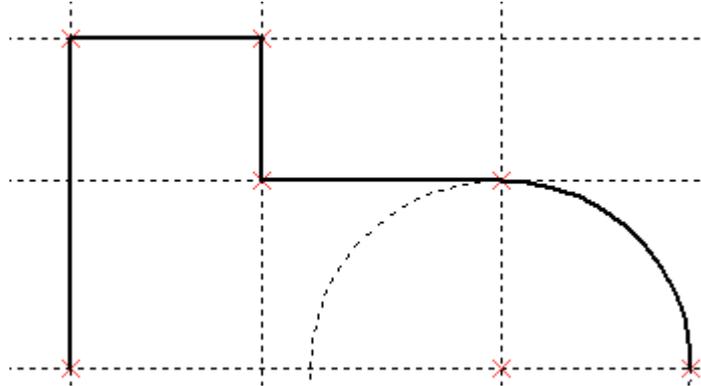
Users can view models using the virtual reality helmet, as well as move objects in the scene and invoke commands with manipulators.

The results of actions are displayed in real time thanks to synchronous updating.

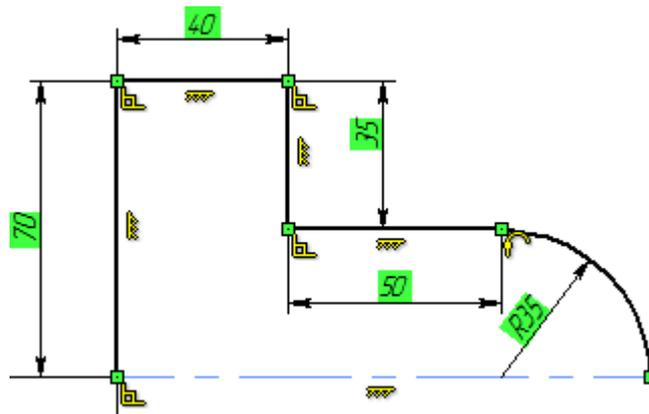
NEW TOOLS FOR PARAMETRIC DRAFTING

In the new version, mechanisms of parametric design, which are the basis of the functionality of the system, were improved.

For a long time, the only parameterization mechanism in T-FLEX CAD was the *mechanism based on the history of the constructions*. To form the geometry of the drawing, the construction lines and parameters of these lines were used.



Now, along with this mechanism, the mechanism of *variational parametrization* is available to users. The new mechanism allows you to set geometric and dimensional dependencies directly on the elements of the drawing.



First, the mechanism is convenient for quickly creating profiles of future 3D- models.

A large number of drafting commands, which were previously intended for nonparametric drawing, were updated to support the new parametric drafting mechanism.

VARIATIONAL DESIGN. NEW POSSIBILITIES OF PARAMETERIZATION

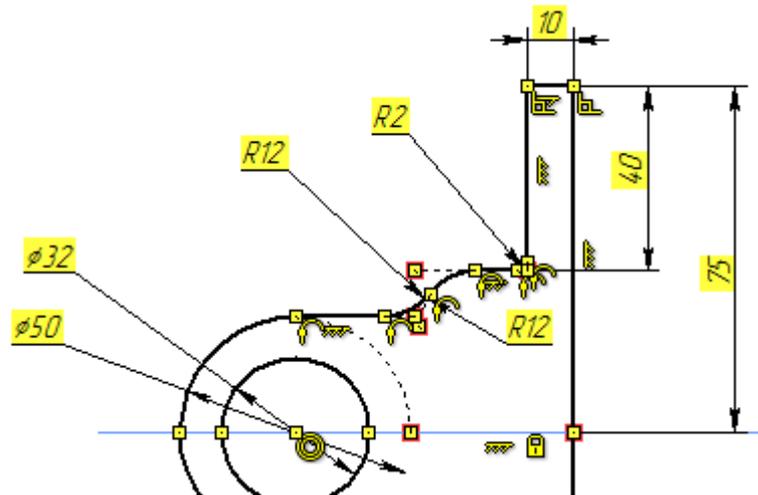
New variational design features are built into existing parameterization mechanisms and tools, providing users with even more flexible parametric capabilities.

Thanks to the tools of variational design, it became possible to define the relationships between *geometric elements* (lines, circles, curves, etc.) by means of *constraints* -geometric and dimensional relations.

The use of constraints makes it possible to eliminate dependence on the order of the constructions, giving the designer greater freedom when working on projects. To set or remove constraints between elements, you can change the values of the model parameters at any time while designing.

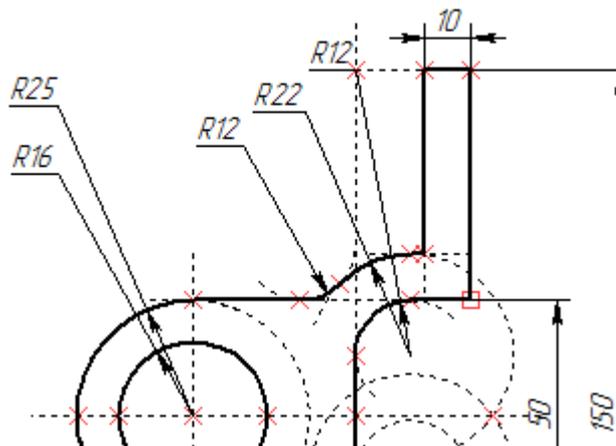
Freedom in setting constraints also allows you to deal with cyclic dependencies. Geometric elements do not have rigid hierarchical dependencies and therefore restrictions can be freely assigned between elements in any order.

The user is given the opportunity to work with not completely defined parametric drawings.



In addition, variational design allows you to create parametric drawings and set the initial data for 3D modeling without the use of construction elements.

It should be noted that the "classical" mechanism for T-FLEX CAD parameterization based on the construction lines, and the new one, based on the constraints, have their distinctive advantages and features that should be taken into account when using.



Both technologies can be combined in one model and even on one page, using the same additional parametrization tools: variables and expressions, databases, user dialogs, parametric and adaptive fragments, etc.

Constraints (geometric and dimensional) can also be created between construction lines and image lines.

Design based on the history of constructions	Variational Design
The position of the geometric element relative to the previously constructed elements is determined when it is created	The position of the geometric element can be set or changed at any time by means of constraints
Cyclic dependencies are not possible (previously constructed elements can not depend on the elements constructed later)	Circular dependencies between elements are permissible
The parametric model is always completely and unambiguously defined	The model may not be completely defined, which on the one hand makes it convenient to modify it, but on the other hand it can lead to unpredictable results upon parametric changes

The recalculation of the model requires only a single passage, which makes it possible to quickly calculate models with a large number of elements	The recalculation of the model requires a specialized calculation of the constraint system, which limits the complexity of the model and the computational speed
Uses the construction elements on the basis of which the image lines are created	Does not require the use of construction lines. Dependencies are added using constraints and control dimensions.
Variables are used when specifying parameters of the construction elements	Variables are used as values of control dimensions, as well as for suppression of constraints and control dimensions for the purpose of qualitative changes in the parameterization scheme

The new mechanism of variational parametrization does not compete with the long-term successful practice of parameterization mechanism based on the history of constructions. It is a complement to the existing mechanism, which allows:

- to solve more conveniently a certain class of parametric problems, including situations where the problem is not completely defined at the initial stage;
- to work in the familiar style for users who have moved from other systems;
- to avoid usage of construction lines for those who, for whatever reason, do not want to use them.

The existing parametrization mechanism based on the construction lines continues to be the key functionality of T-FLEX CAD and will continue to evolve in the future. In particular, a number of significant novelties on it were added in the 16th version. Read more about them below.

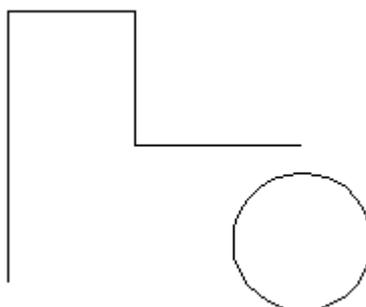
It can also be noted that T-FLEX CAD is by far the only system on the market in which two parametrization mechanisms are realized at once, and they coexist harmoniously with each other and can even be used simultaneously within the framework of one project. This approach provides users with the maximum level of flexibility in terms of the use of certain parametric tools, taking into account the nuances of implementing specific projects and personal preferences.

GEOMETRIC CONSTRAINTS

Let us analyze the main points of work with the new mechanism on a simple example.

First, you need to create the basic geometric elements of the future model using the **Sketch** command. The mechanism does not depend on the build sequence, so you do not have to immediately specify the relationships between the drawing elements.

Constraints and dimensions for created items can be created automatically. Automatically created dimensions and constraints can be replaced by others. The way to create them will be described later in the text.

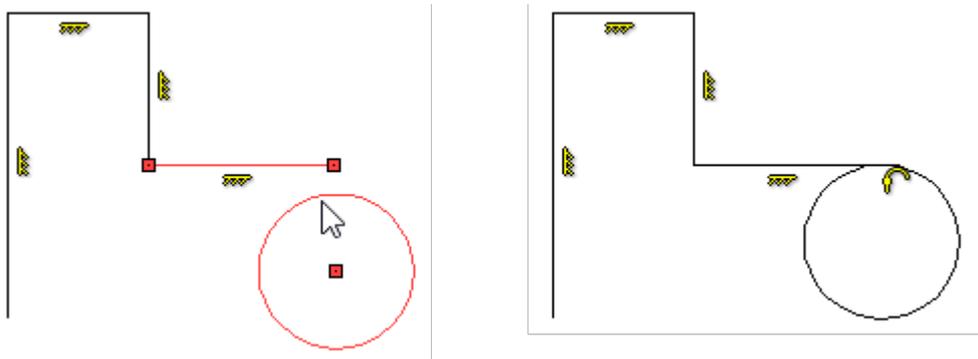


Constraints are used to create relationships. They determine which elements are parallel, which have an equal radius, etc. Thus, the constraints set the geometry of the model and allow it to be controlled.

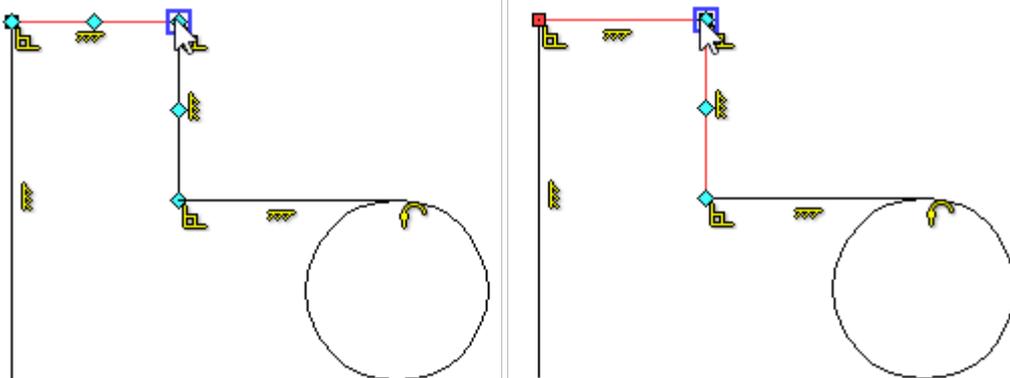
To create a constraint, select it in the Ribbon panel or call the command **SC: Constraint** and specify the elements for which a constraint will be created. Constraints of the same type are created continuously, i.e. you only need to select new elements to create constraints.



For example, to create a tangent, you need to specify a segment and a circle.



To create a coincidence for the two lines, you must point the cursor to the line and then select a point that will coincide the point of the other element. This is important for creation of constraints, because if you select just two lines, the system cannot accurately determine the future geometry.

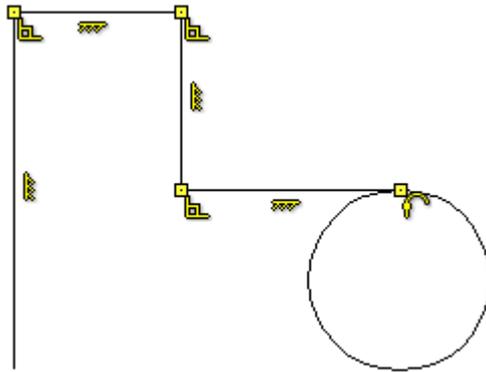


The mechanism of snapping to characteristic points of elements is realized. Characteristic points can be used to create constraints. For example, you can fix a circle by selecting a characteristic point of its center.



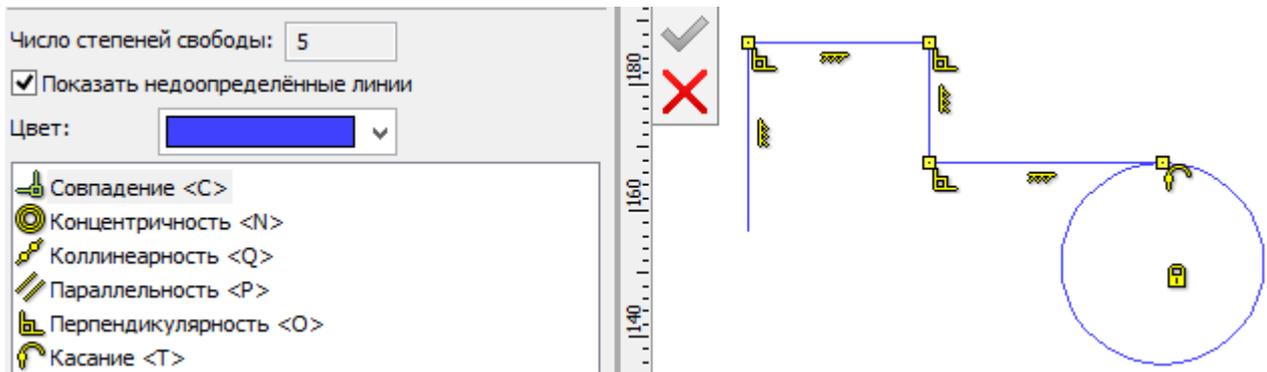
Description can be found in the section "[Snaps](#)".

In the example, horizontal , vertical , perpendicular , coincidence  and tangency  constraints were created. They are all displayed on the drawing in the form of icons next to the elements.

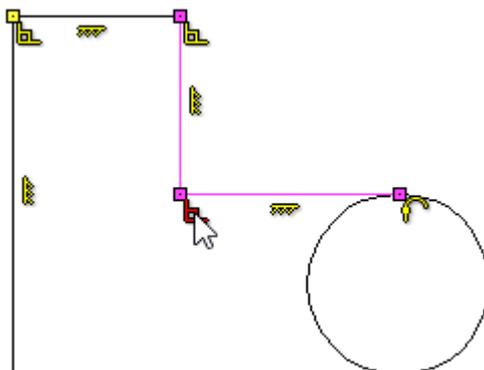


Command **SC: Constraint** allows not only to create constraints, but also provides information about the drawing

The **Degrees of freedom** field shows how many degrees of freedom remains for the drawing elements. You can select all elements that are not fully defined using the **Show Under Defined lines** option.

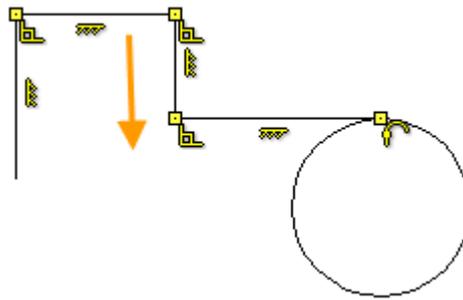


When you hover over the icon, the elements, for which constraints were created, are highlighted. You can delete the constraints by selecting the icon and pressing the <Delete> key.



To simplify the design process, the **Auto Parametrization**  option is available on the filter toolbar. If it is active, the main constraints, such as parallelism, perpendicularity and tangency, will be created automatically by the system.

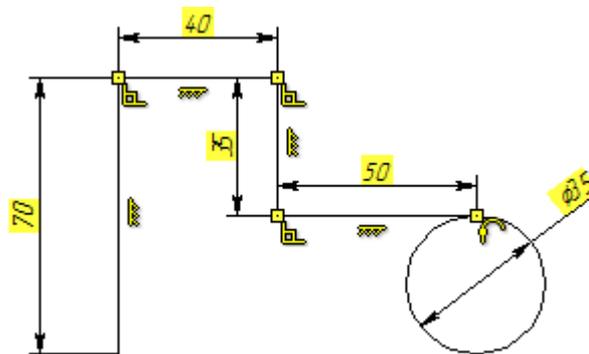
Constraints allow you to determine the position of the elements only partially. If you try to move the elements, they will not change the position relative to each other, however their size will change.



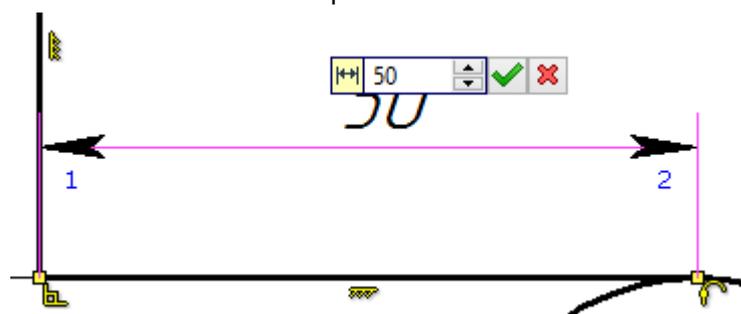
PARAMETRIC DIMENSIONS

In order to fully define the drawing dimensional constraints should be created in addition to geometric constraints.

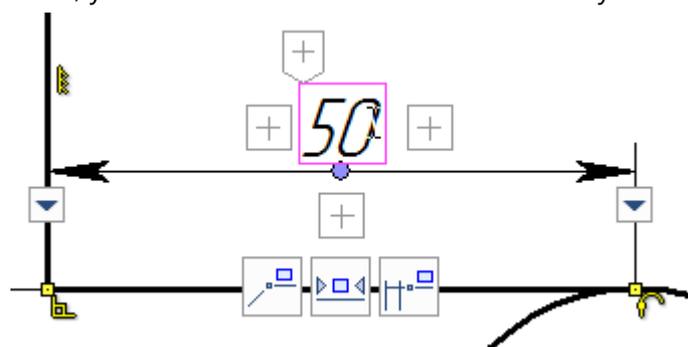
Dimensional constraints are created using the command **D: Dimension**.



To increase the convenience of creating driving dimensions, the ability to immediately edit the dimension nominal without leaving the command is added. To do this, the **Edit driving dimension value on creation** option must be active in the dimension parameters.



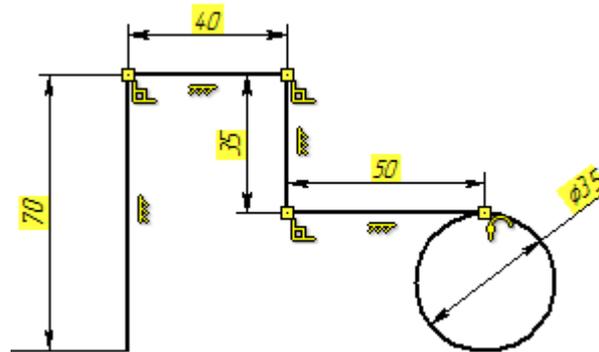
When the dimension is created, you can enter the nominal edit mode by clicking on the nominal.



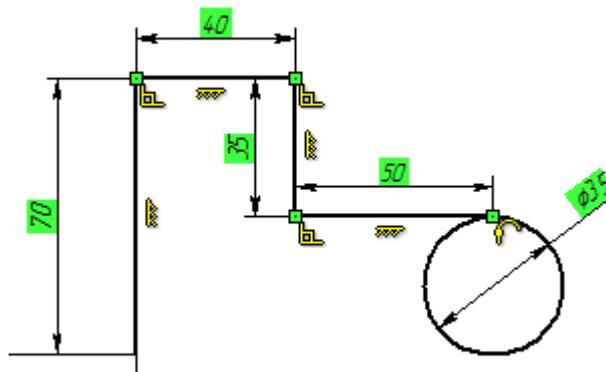
To work with the mechanism of variational design, the "Driving Dimension" was introduced. It is responsible for the geometry shaping, so when you change it, the element changes.

The values of the control dimensions have background. The background color of the dimensional value shows different states of dimension and geometry in general:

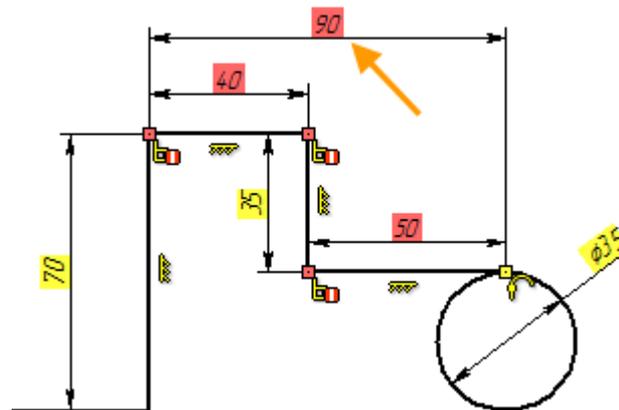
The system is under defined - the background is yellow. If the system is under defined, then some elements still have degrees of freedom, so they can be changed and thus affect the overall geometry of the model.



The system is defined - the background is green. If the system is defined, then the elements do not have degrees of freedom, and the resulting geometry can not be changed. When the drawing is finished, it is recommended that the system is defined.

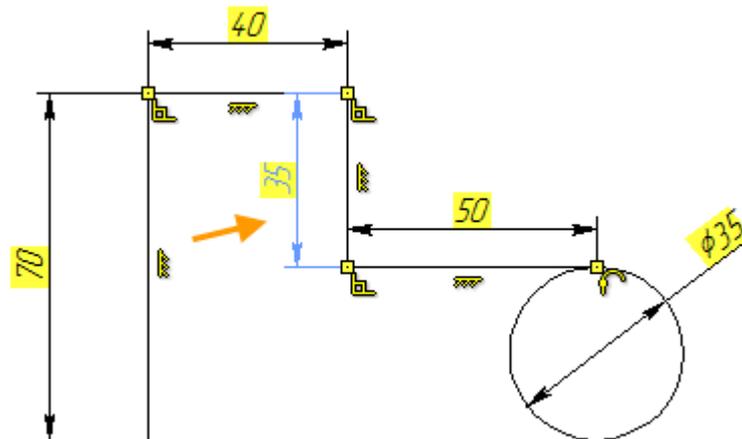


The system is over defined, or there is an error setting the value of the driving dimension- the background is red. This is the case when user has created unnecessary driving dimensions for the element, or has set dimensions, the values of which cannot be provided.

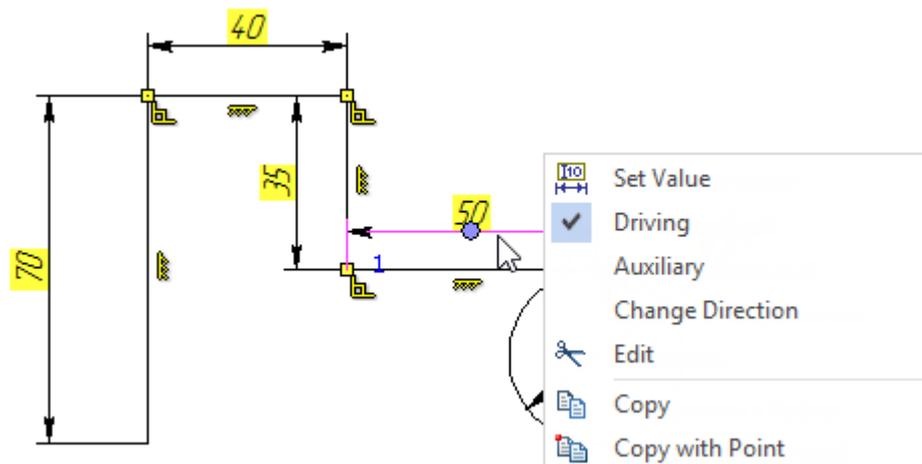


The option  allows you to always create control dimensions in the command D: Dimensions.

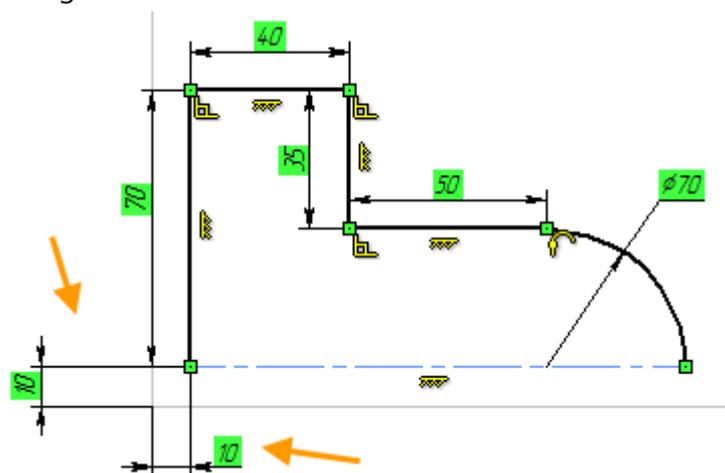
Also the "Auxiliary dimension" type is available for lines. Auxiliary dimensions can simultaneously be driving, but they are not printed and can be hidden in the scene.



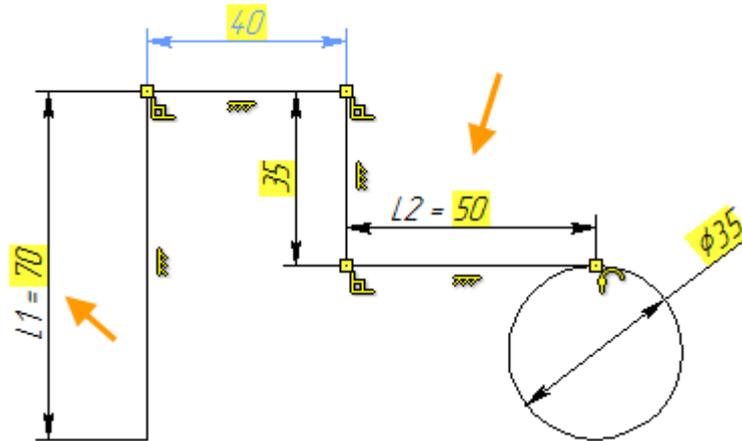
You can change the dimension type in its context menu. When you manually set the dimension value, it automatically becomes driving.



The user can also create a dimension relative to the origin of the drawing. Sometimes it is necessary to completely define the drawing.



With the help of driving dimensions, the user can control the parameterization of the drawing. To do this, you must specify a variable or an expression for the dimension value. The name of the entered variable is displayed before the value of the dimension nominal.



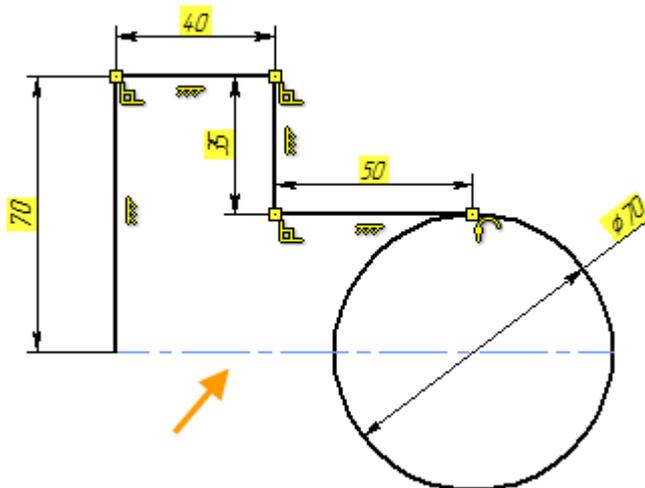
Dimensions for the elements can be created automatically. To do this, enable the **Create Dimensions Automatically**  option on the filter toolbar.

New dimension types that allow you to control parameterization using the new mechanism were added. More details about these dimensions can be found in the section "[Dimensions](#)".

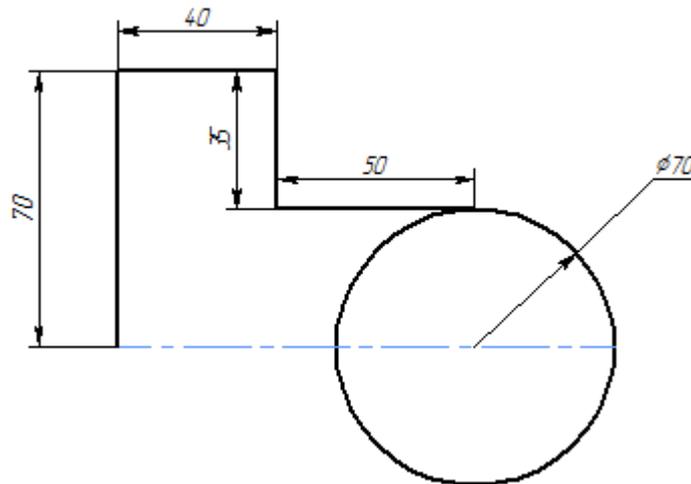
UPDATED SET OF COMMANDS AND DRAWING TOOLS

Not only dimensions can be auxiliary, but also other elements (lines, circles). They are not printed and can be hidden using the **Hide constructions**  option.

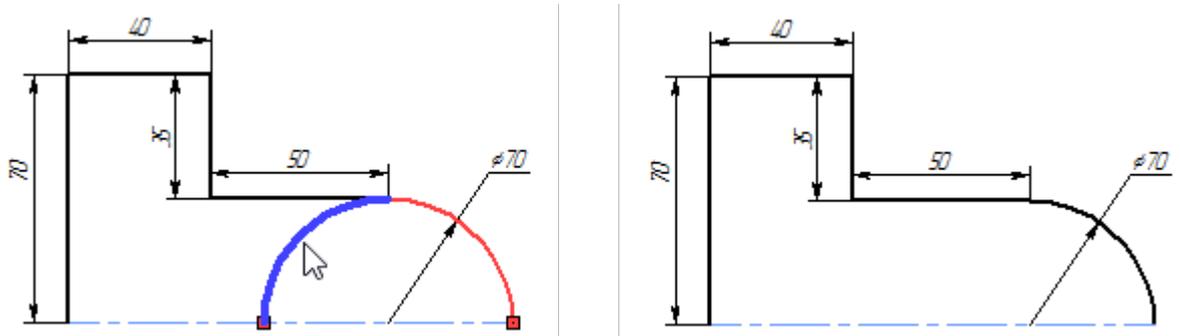
It is important to remember that the auxiliary lines are full-fledged elements of the drawing, so constraints and dimensions are created for them too.



If necessary, all constraints and background for the driving dimensions can be hidden by the option **Show/Hide Constraints** .



To complete the drawing, you can use the **Trim**  command to delete all unnecessary constructions.

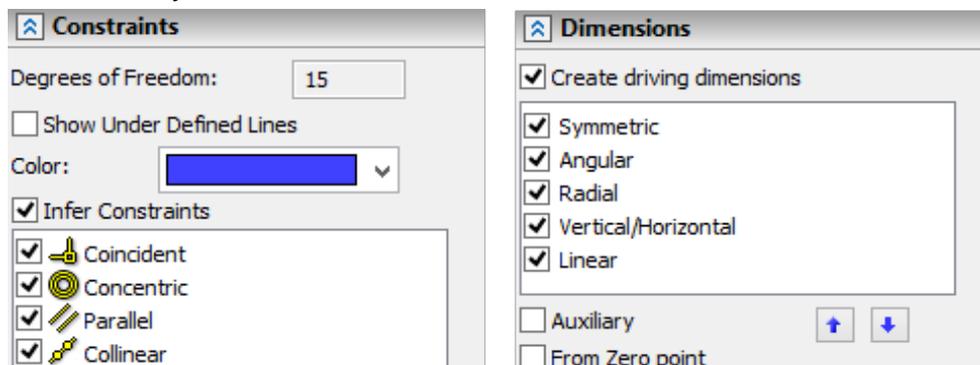


A filter of automatic creation of constraints and dimensions is also available for the trim command.

Constraints for the finished drawing can be created automatically. To do this, use the **Infer Constraints**  command. This is a convenient tool for quickly creating a large number of constraints.

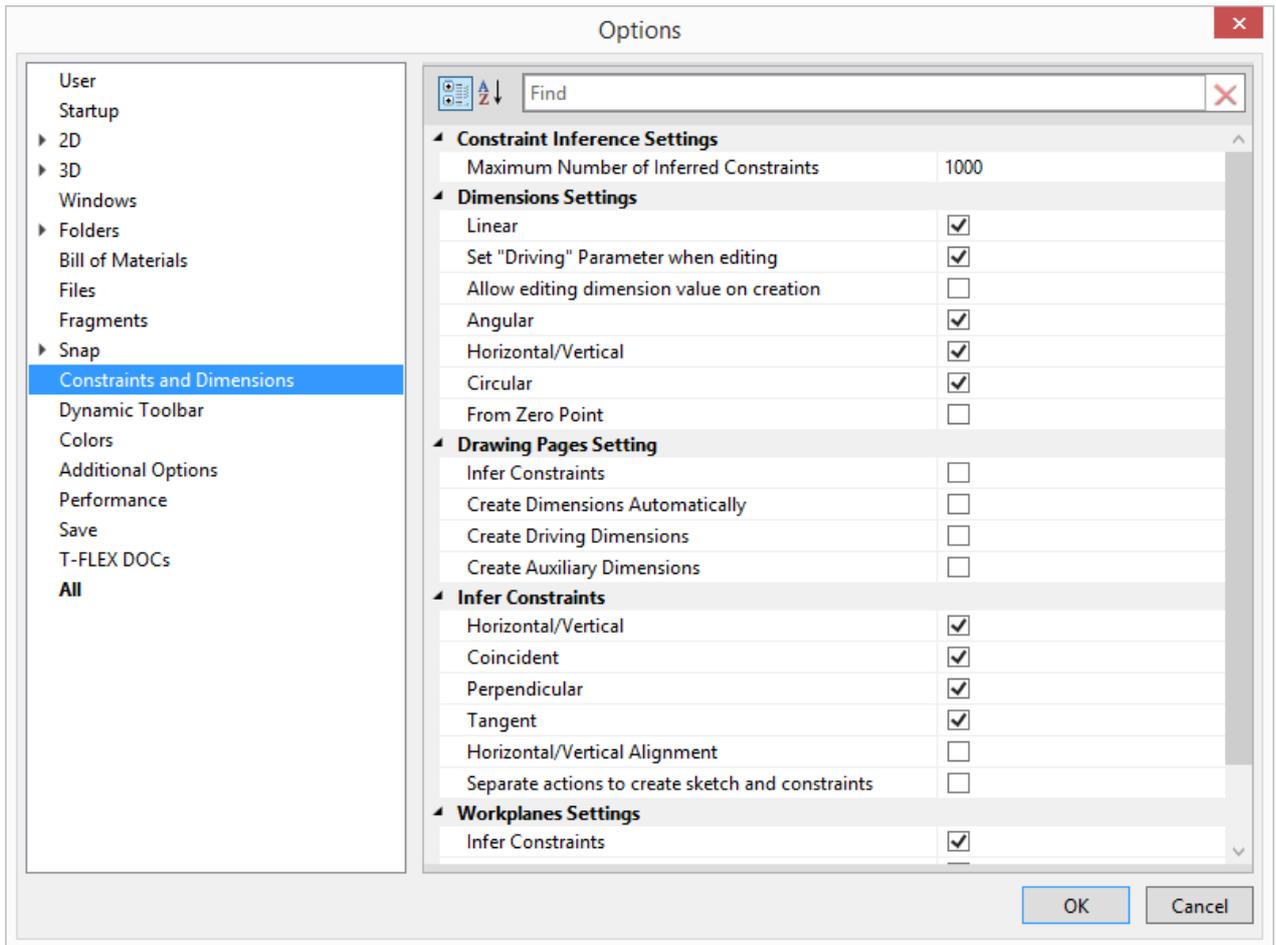
The command displays the number of degrees of freedom and you can select under defined elements with color.

In the **Constraints** and **Dimensions** sections, you can manually specify, which constraints and dimensions will be created automatically.



You can assign a creation priority for dimensions. The **Auxiliary** option allows you to make all created dimensions auxiliary, and the option **From Zero point** allows the system to use the origin to create dimensions.

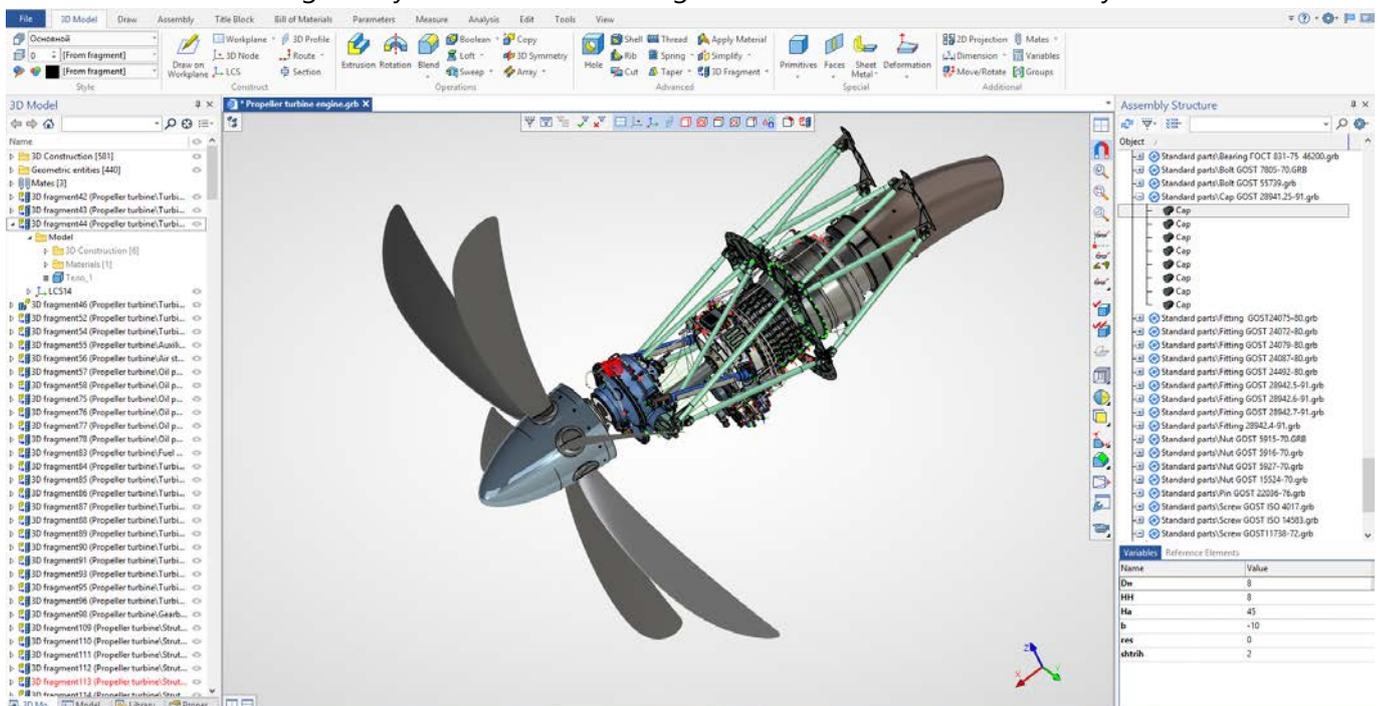
In the **Options** dialog the **Constraints and Dimensions** tab is available. User can specify parameters for the automatic constraints creation on it.



WORKING WITH ASSEMBLIES

When developing a new version of T-FLEX CAD, great attention was paid to the management of the objects geometry to improve the convenience of working with complex assemblies and details. Increased efficiency of the old mechanisms and added a number of new features.

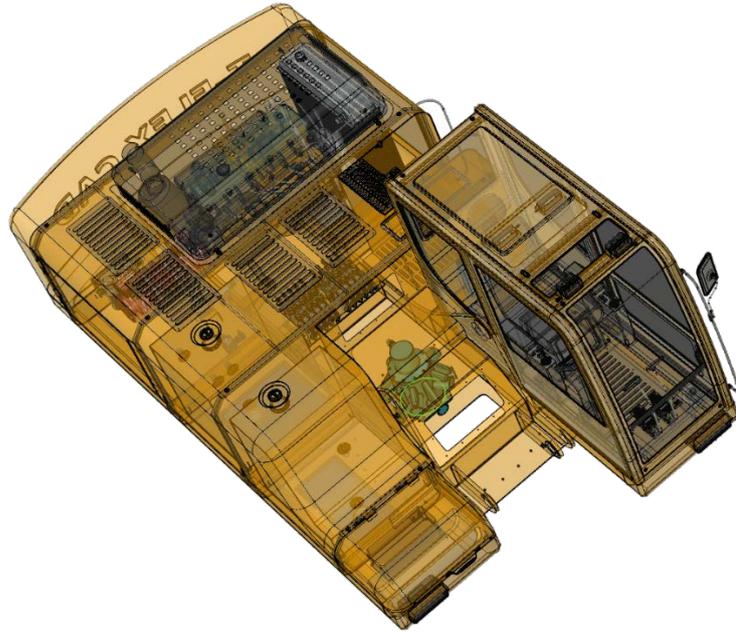
- ✓ The mechanism of borrowing geometry for designing elements in the assembly has been improved. Now user can not only create parts in the context of the assembly based on the existing geometry, but even upload this geometry to a separate file and track its change. This opens up wide opportunities for teamwork on complex projects;
- ✓ Improved assemblies navigation. The model tree and the **Assembly Structure** window now displays information about all the elements of the subassembly at all levels. User can immediately switch to editing of any of these elements right in the context of the assembly.



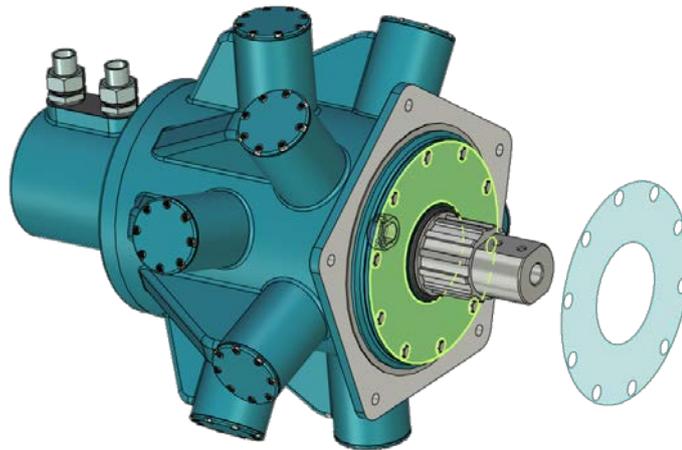
REFERENCE ELEMENTS

The system implements a new mechanism of reference elements, which allows you to borrow a three-dimensional geometry of objects for subsequent operations. The basic elements of the model (3D nodes, coordinate systems, workplanes, 3D profiles, 3D paths, bodies of various types), and topologic elements (edges, faces, vertexes) can be borrowed.

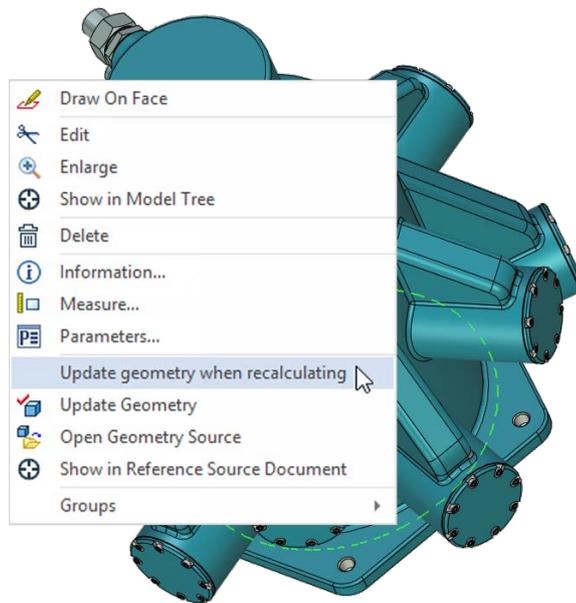
In the example below, the face of the excavator body was borrowed to snap the hydraulic motor model.



If this face changes, or if you need to move the motor/excavator body in the scene, users will not have problems with object snapping. The body and the hydraulic motor will move and change independently of each other.

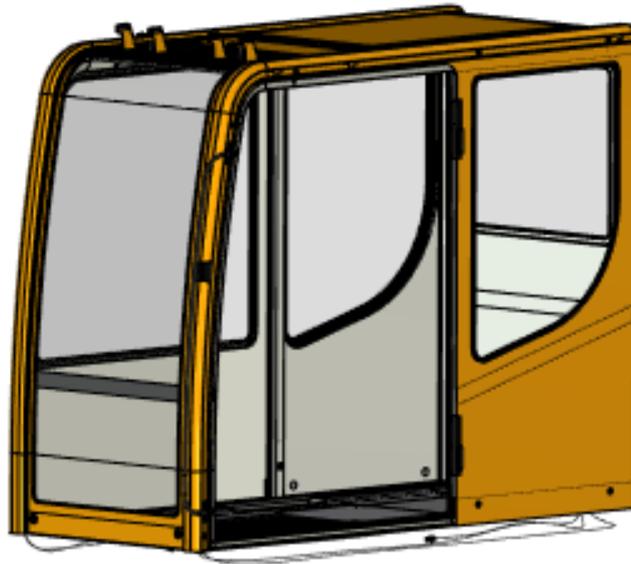


To update the face of the body to which the motor is attached, you can use the **Update geometry when recalculating**.

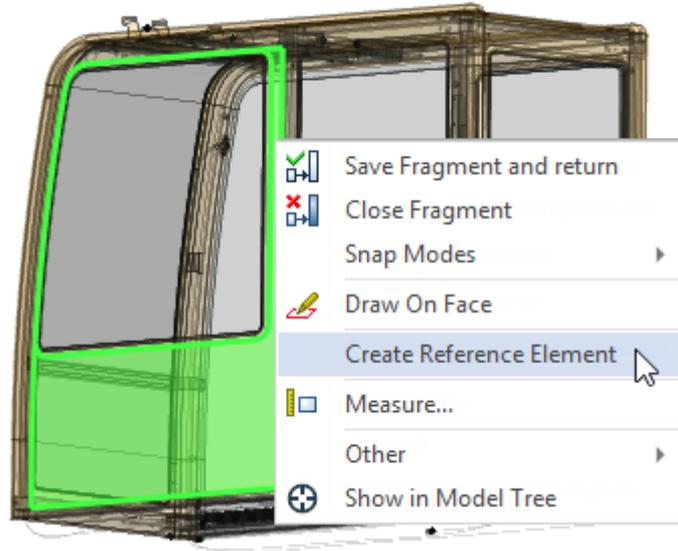


The **Create Reference Element** command appeared in the system. It creates an object of the appropriate type, which stores a copy of the geometry of the source element. This copy "remembers" the information about the source, and its geometry can be updated either automatically or manually, depending on the corresponding parameter of the reference element.

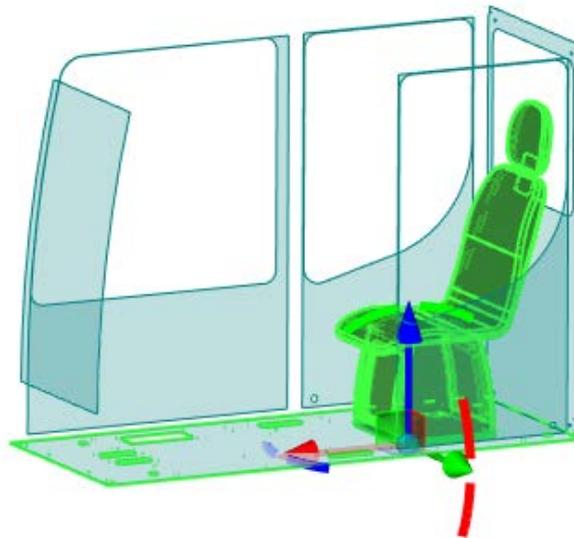
For example, you need to unload the cabin to fill it with equipment. One user develops the cabin, and other users do not have access to this file.



The geometry of the cabin can be uploaded to a separate file so that several users can work in parallel. To do this, a fragment is created in the context of the assembly, and reference elements are unloaded into it.



On the basis of these unloaded elements another user can create equipment in the cabin, having full data on the current dimensions of the cabin. In this case, it is possible to snap to the unloaded geometry and update the unloaded geometry in accordance with the changes in the source file of the cabin.

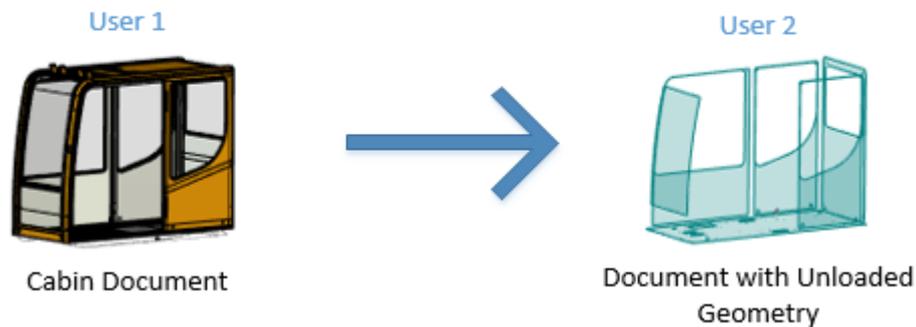


The mechanism of reference elements allows to solve the problems with the development of complex assemblies. For example, the user can borrow the geometry of elements from the elements of higher-level fragments in the assembly structure, or from fragments located in "parallel" subassemblies.

Example of the cooperative work scheme:

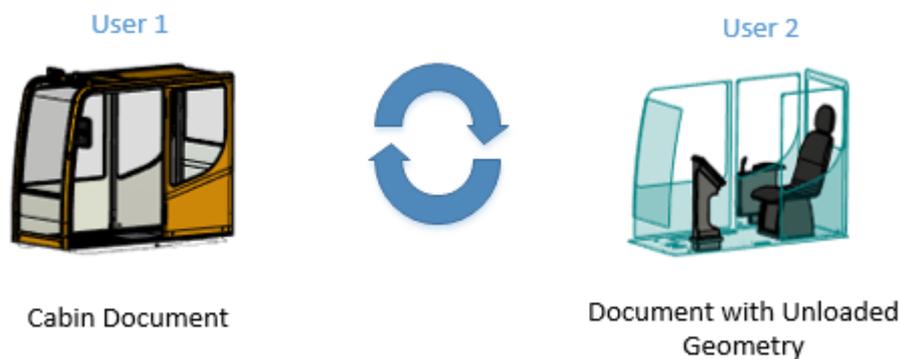
A geometry is unloaded into a separate file in the context of the assembly.

Upload Geometry to file



The user can receive changes of the uploaded geometry from the source document or update the source document in accordance with the changes in the uploaded file at any time.

Parallel Work on Project



Result of Parallel Work

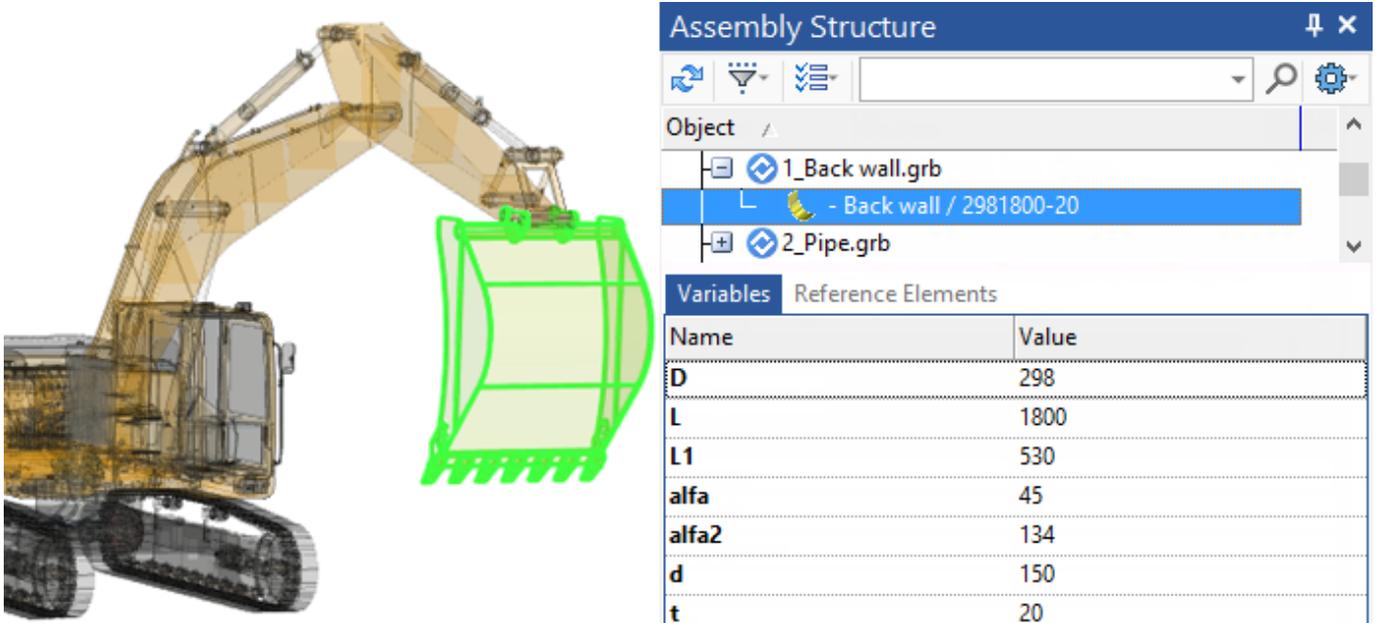


The mechanism of reference elements ensures that there is no loss of borrowed geometry. It also ensures integrity of the model in case of absence of the source object of the original geometry. For example, when you rename or delete a fragment file that is the source of referential geometry, when body recalculation errors occur, etc.

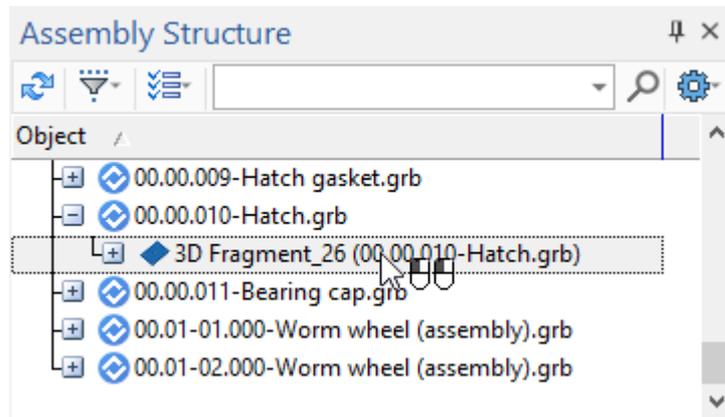
The introduction of the reference element mechanism allows T-FLEX CAD to be taken to a new level and greatly simplifies the design of complex products for users.

"ASSEMBLY STRUCTURE" WINDOW

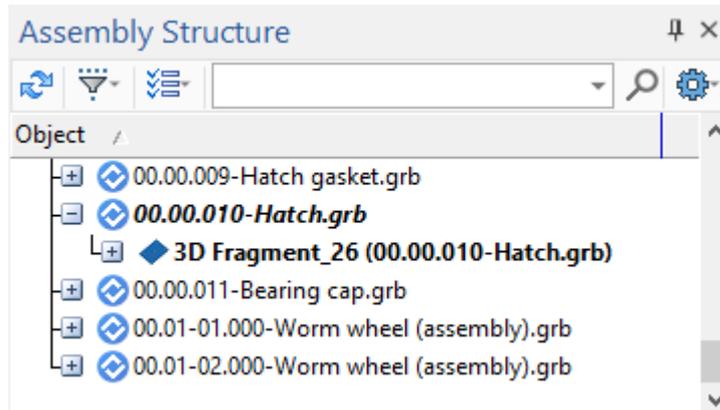
The **External Links** window has been renamed to the **Assembly Structure** window and new features have been added to it. The window provides functionality for managing the multi-level structure of the assembly model. Information is presented in the form of a tree structure. When selecting fragments, a list of variables for these fragments and referential geometry are displayed.



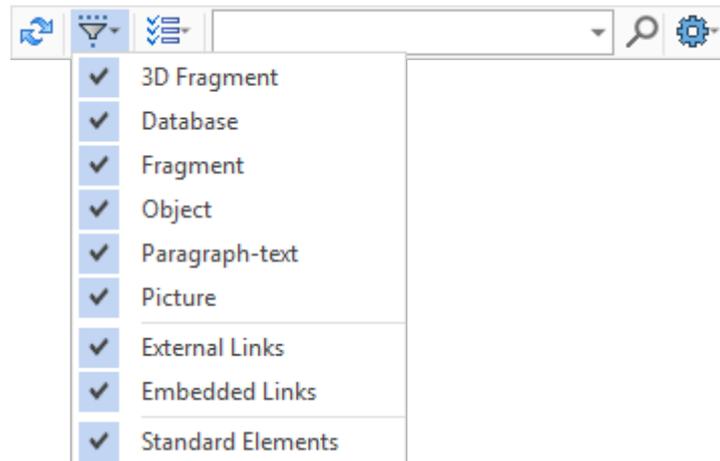
For fragments of any level, the command **Open in Context** is available. You can also go to editing in the context of the assembly by double-clicking on the fragment.



All fragments of the upper level, the assemblies of which includes the fragment edited in the context of the assembly will be marked in italics. This will simplify the search for the edited fragment.

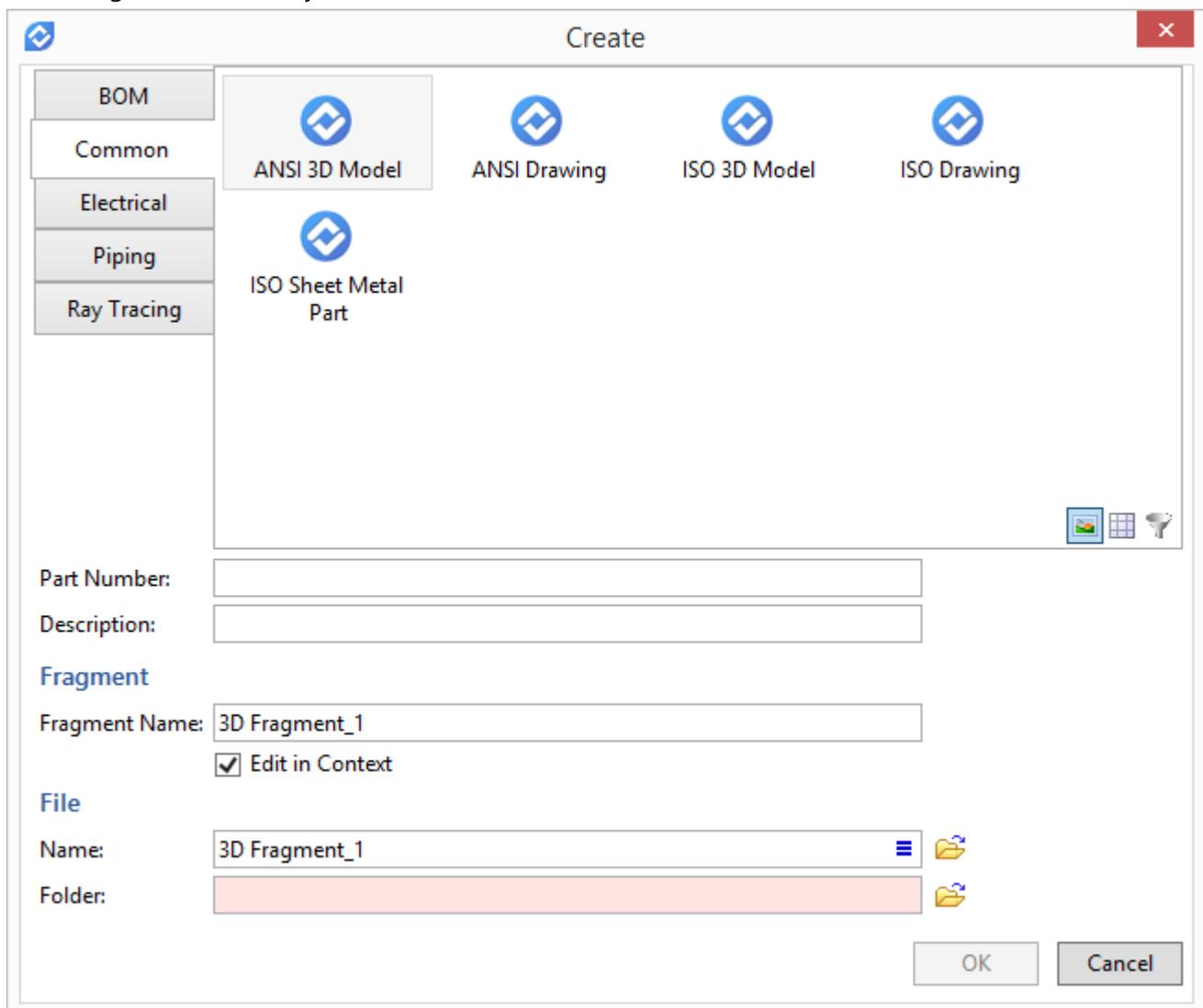


The possibility to hide standard elements in the list was added to simplify designing of large assemblies. There is also a possibility of filtering assembly structure elements by types and other characteristics.



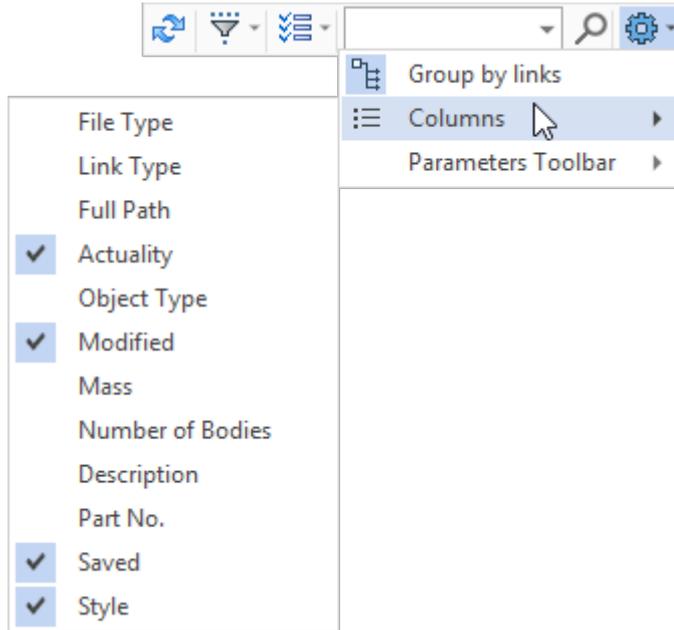
All geometry borrowed from other files is displayed on the **Reference Element** tab.

The **Create 3D Fragment** command is now available on each level of the assembly structure. The command allows specifying general parameters of a new assembly component and, if necessary, begin its editing in the assembly context.



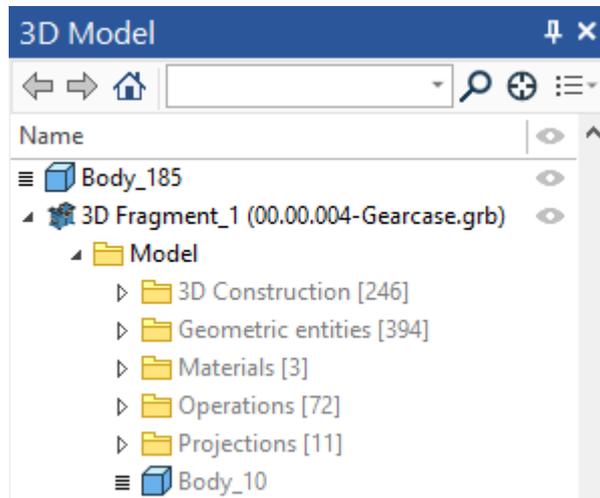
The updated dialog of 3D fragment creation allows selecting of a prototype based on which a new fragment document will be created. You can also specify its part number and description, specify a fragment name in the assembly.

User can select columns that will be displayed in the window.



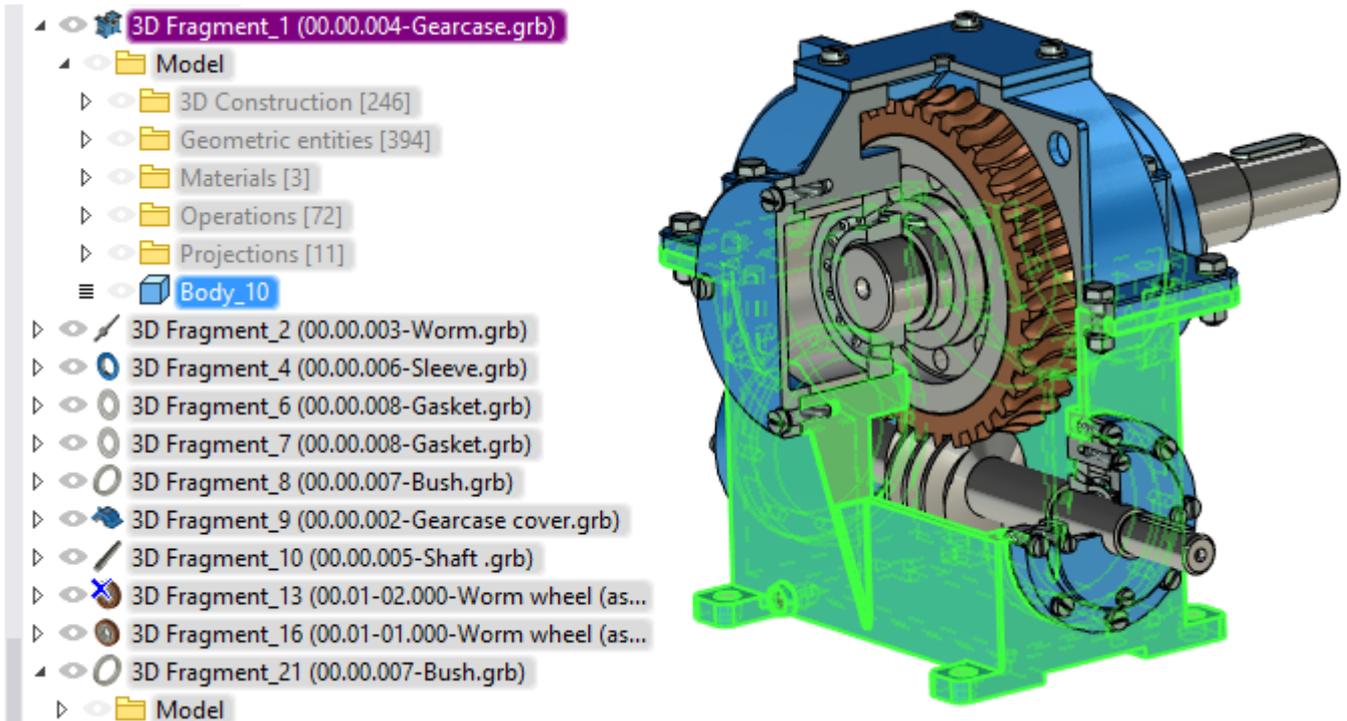
DISPLAYING THE ASSEMBLY STRUCTURE IN THE "3D MODEL" WINDOW

The **3D Model** window has been refined to provide control of the assembly structure. Now the window displays the structure of all fragments and subassemblies that are included in the assembly. Open the "Model" folder for the subassembly fragment to view this structure.



The structure view is available for all levels of nesting.

Marking of bodies and operations also occurs at all levels of nesting.



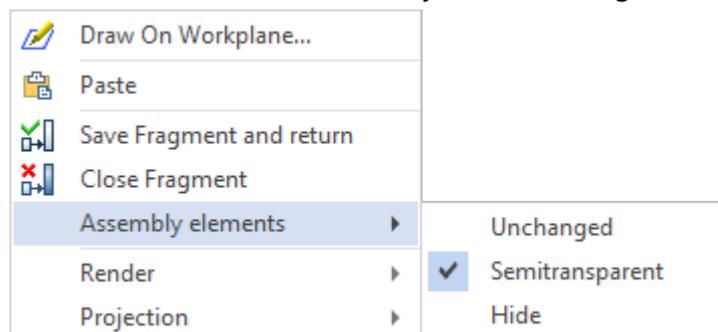
For items in the "Model" folder, the **Open in Context** command is available. If you call the command for a fragment or an operation, the system will go into editing it in the context of the assembly.

Now all the fragments and operations can be edited directly in the assembly.

The number of objects in a folder is displayed in the model tree next to the folder name.

VISUALIZATION UPON DESIGNING IN CONTEXT OF ASSEMBLY

When a fragment model is edited in the context of an assembly a geometry of the assembly model becomes semitransparent by default. The system allows managing the visualization. An assembly geometry can be displayed in the context of the assembly without changes or be hidden.

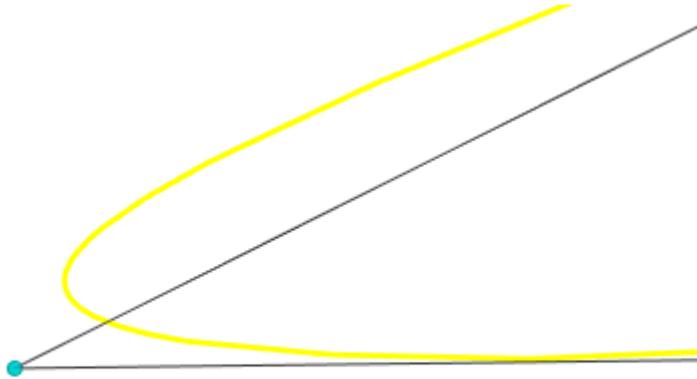


The **Assembly structure** window allows managing visibility and visualization mode of all fragments beyond the edited model. It regards to the assembly elements on the levels higher or lower than the model being edited as well as the fragments on the same level with it. The special commands and columns in the assembly structure are used for it.

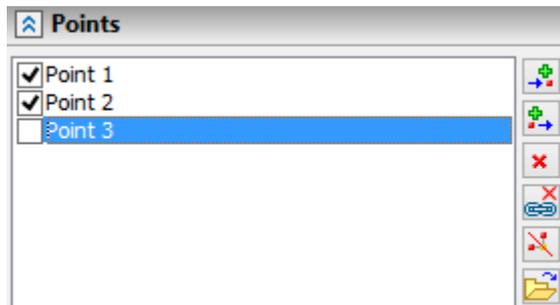
3D GEOMETRY

ROUTE

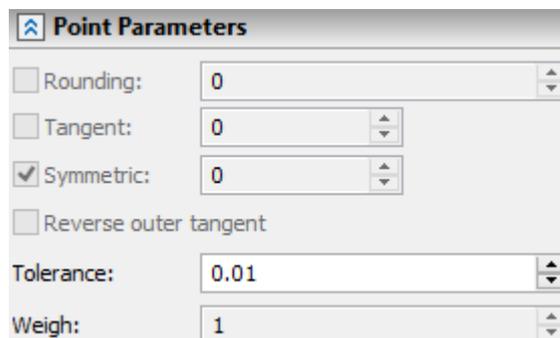
The ability to build a curve that does not require an accurate passage through all specified points is added. The new mechanism allows smoothing a large number of points of a smooth curve with a given accuracy.



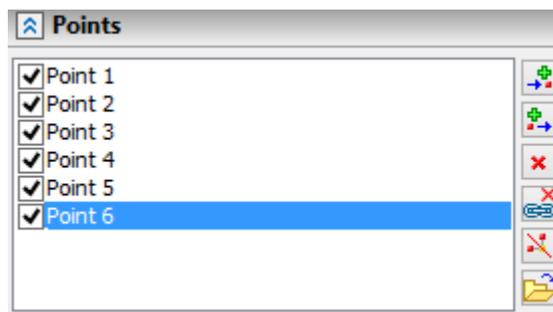
The construction of such a curve occurs in the **Spline through points** mode. An **Interpolation point** flag is available next to each of the points. All the default points are interpolation points. If you remove a flag from a point, then the deviation will be taken into account for the point.



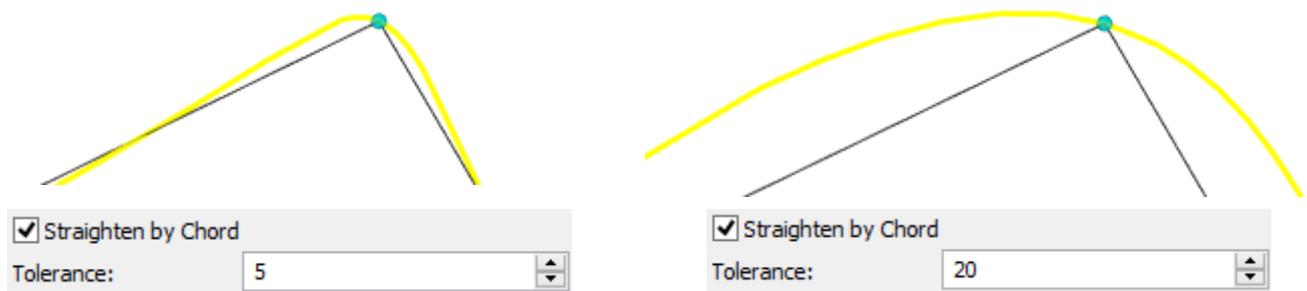
The value of the curve tolerance from the points is given immediately for the entire spline in the parameters dialog.



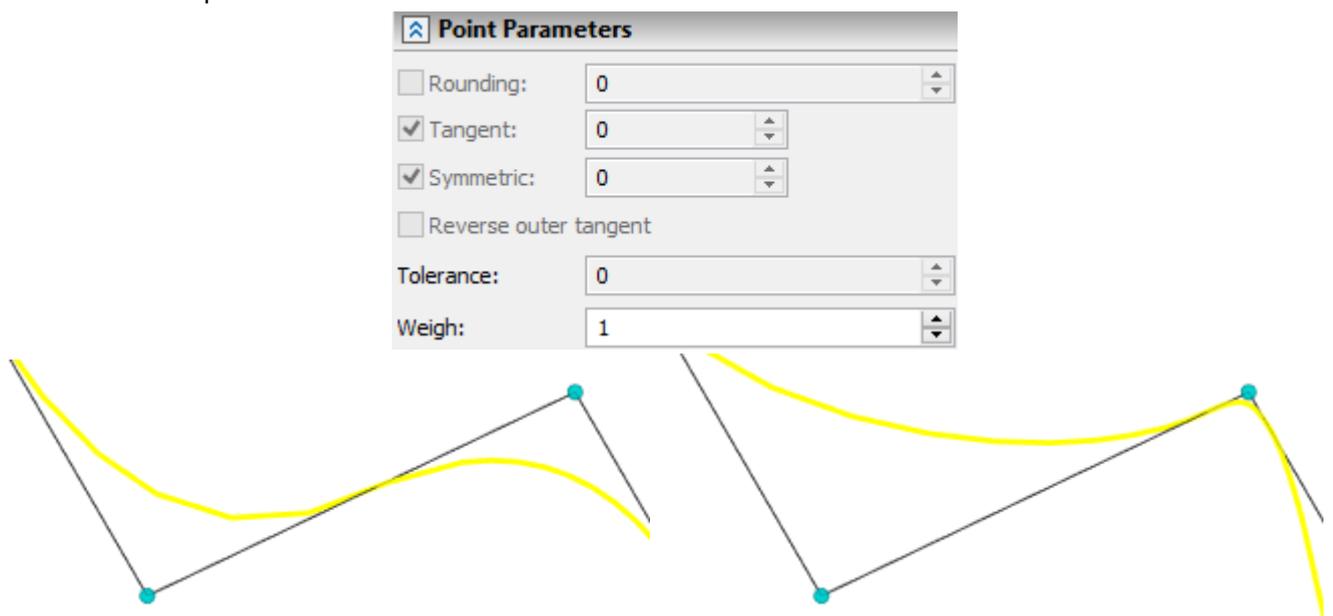
The list of points can be downloaded from a TXT or CSV file using the **Load from file**. To specify each point you need three coordinates, which you can enter from a new line.



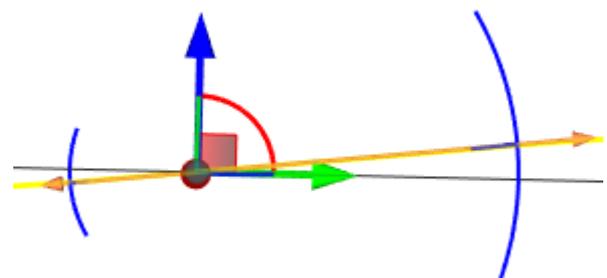
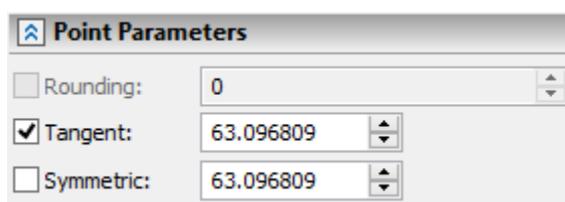
In the **Spline through points** mode, the **Straighten by chord** option was added.



For the **Spline by polygon** the **Weight** parameter is added. The greater the weight of the point, the closer the curve passes to it.



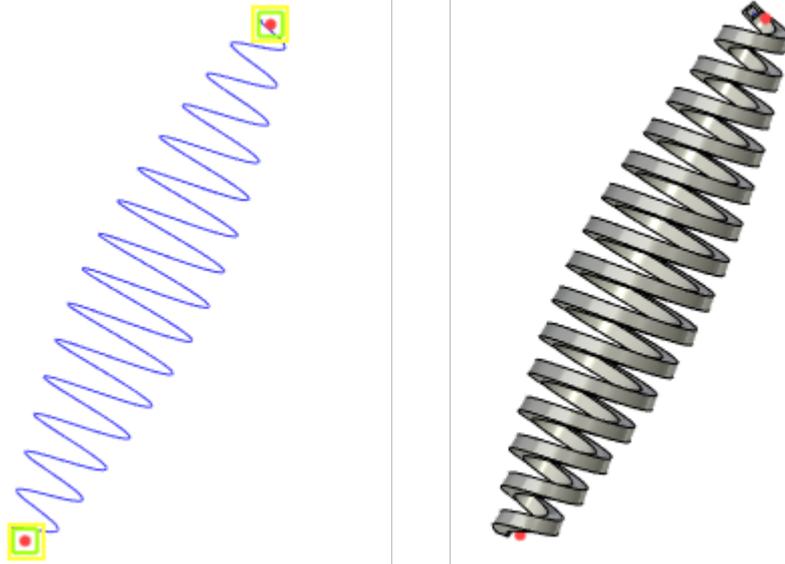
For **Spline through Points**, the option **Symmetric** is added. The option allows you to separately control the tangency at a point.



Now when you connect two routes, it is possible to specify a touch at the end to control the boundary conditions.

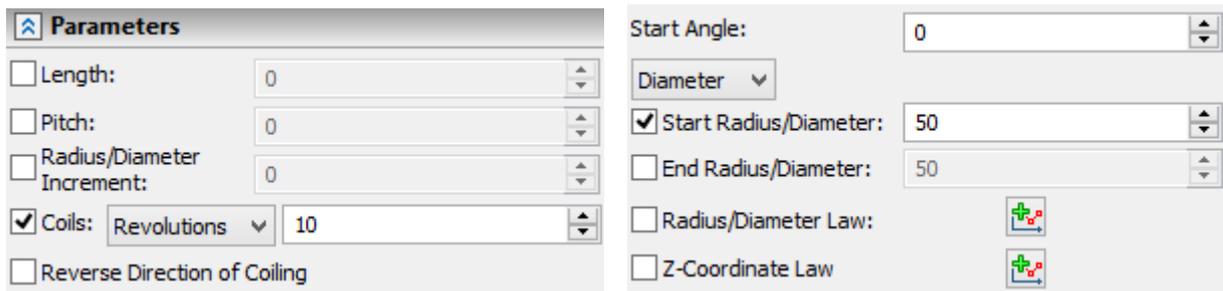
HELIX 3D PATH

The command **3PB: Helix 3D Path** was added. The operation creates a helix path that you can use to translate a profile of any shape.



You can specify a graph for a helix path that will control its parameters.

You can specify the length and pitch, as well as other options in the operation parameters.

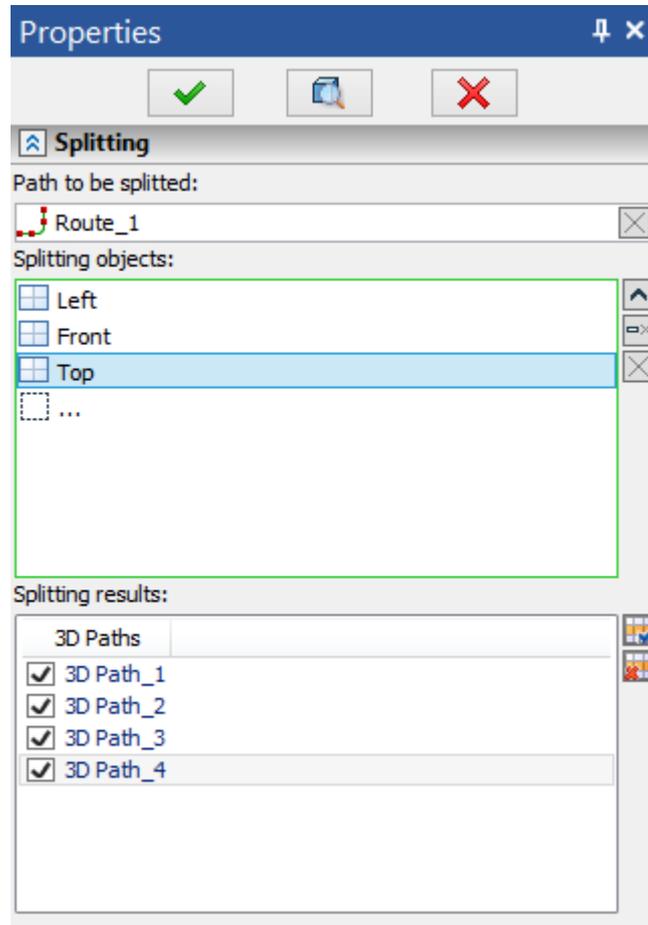


It is possible to build cylindrical, flat and conical spirals.

In addition, you can use graphs of side generating line and pitch changes to create helices.

SPLIT 3D PATH

The option for creation of 3D paths by splitting of the existing 3D path using other model elements was added. As the result of operation, copies of the selected path are created. The copies corresponds to the sections between the elements used for separation.



3D AXIS

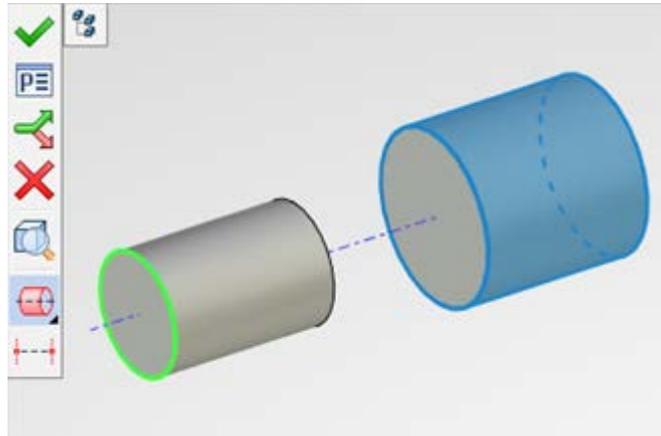
A possibility to create 3D axis was added to solve the following tasks:

- ✓ Possibility to use the axis for specifying an axis of rotation or a direction upon operations creation;
- ✓ Possibility to preliminary create all axes of the model for the following projection creation;
- ✓ Possibility to create 3D dimensions: between axes, from axis to the workplane, etc.

You should select the **Convert to axis** command in the context menu of the selected geometry to create an axis.



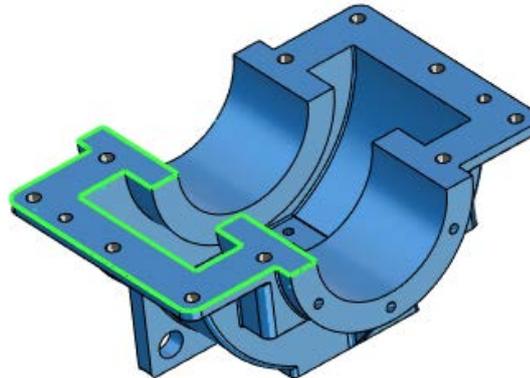
A new command **3D Axis** was added for simple creation of 3D axes on the basis of surfaces of rotation and edges. The command allows to create rectilinear axes based on the elements of rotation - cylindrical, conical, toroidal, as well as circular edges. As additional elements for specifying the dimensions of the created axis, you can select any other elements of the model.



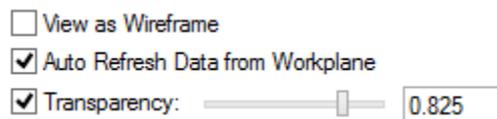
3D PROFILE

A possibility to create a profile using wire geometry was added. The **Construct 3D profile by wire geometry**  automenu option is used for it.

You can also select edges of the already existing models for quick profile construction using the option.



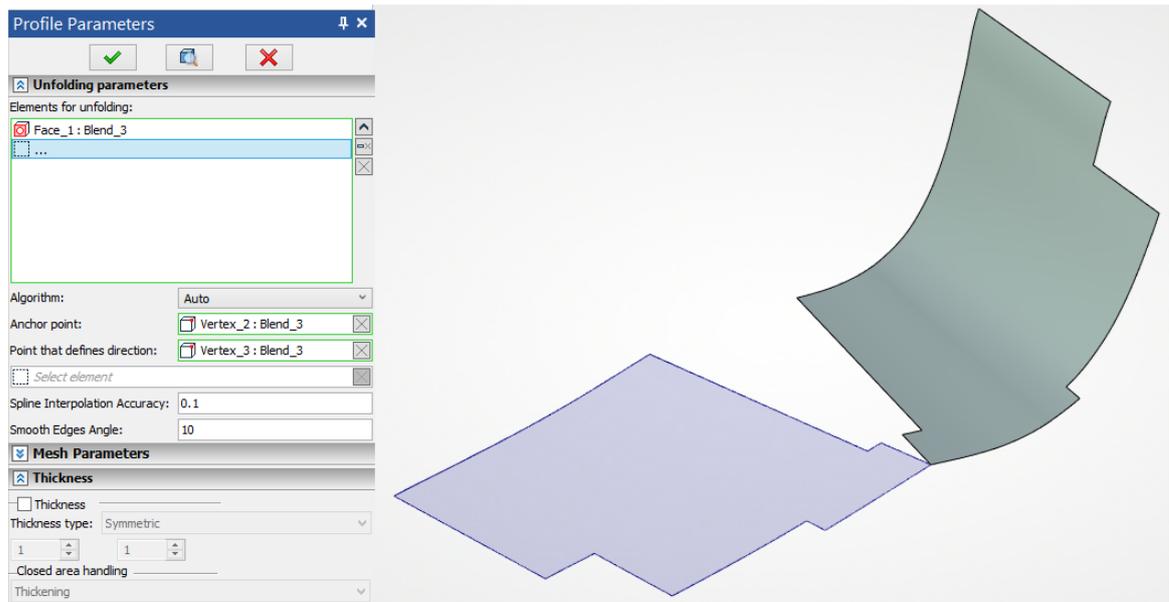
Now it is possible to change the transparency of 3D profiles when drawing in 3D window. To do this, the "View as Wireframe" flag and transparency controls were added to the profile parameters.



BUILDING 3D PROFILES BY UNFOLDING

Essential improvements were made in functionality of creating 3D profiles by unfolding. Instead of several ways to unfold faces, one mode was added, **Unfold Surface**, which unifies all the methods of unfolding faces. When several adjacent faces are selected, the **Delete redundant edges** flag appears, which simplifies the topology of the resulting profile.

When unfolding surfaces, it is now possible to select non-rotating surfaces. To solve the problem of development of such a surface or set of surfaces, an iterative algorithm is implemented.



For this algorithm, the accuracy parameters of the intermediate computational grid used for iterative calculations can be specified.

MODELING

SHEET METAL

Interface was improved and intuitiveness of commands implementation was increased for all sheet metal commands.

All dialogs of the commands were updated and unified. Now it is easier for user to find parameters and less clicks are required to create operation.

All commands are added to a special Ribbon tab.

A separate command is available for each sheet metal operation. It increased speed of operations creation and model recalculation.

A separate prototype with all required parameters was added for sheet metal parts to accelerate modeling process.

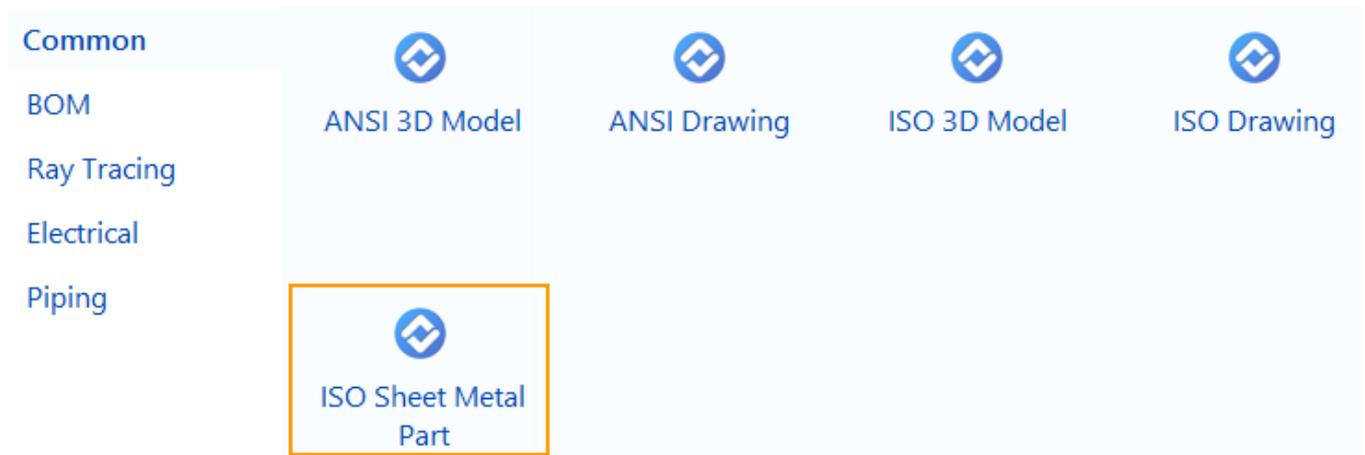
Commands for solving the following tasks were added: corner creation, normal cutout creation, gusset creation, creation of flange using user profile, bridge bend creation, etc.

Suggestions from users were taken as a basis for refining the functionality of sheet metal.

Sheet Part Prototype

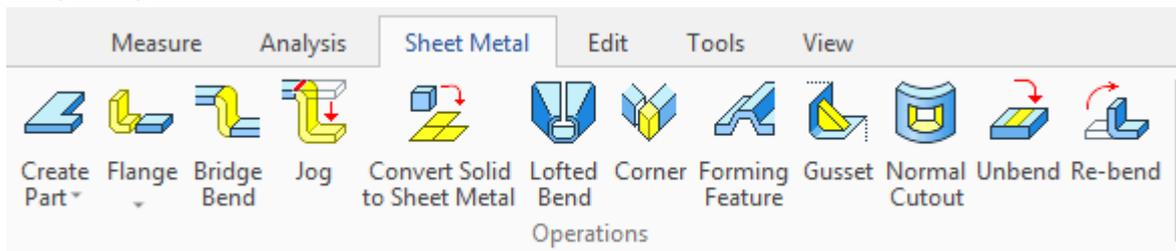
A new "Sheet metal part" prototype was added.

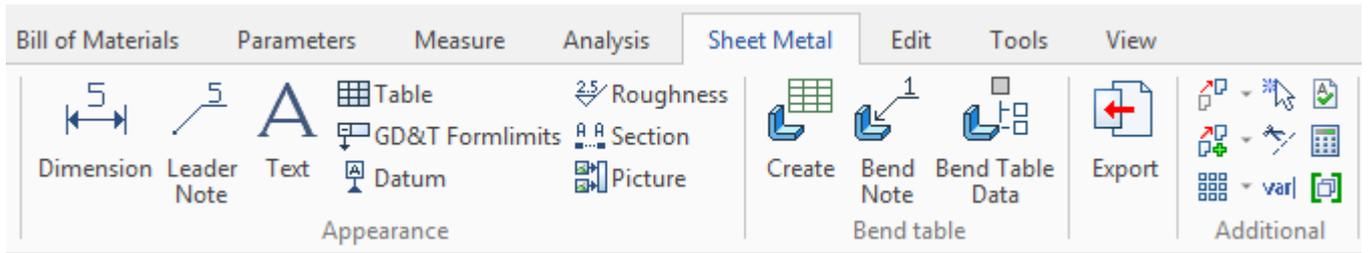
Create New Document



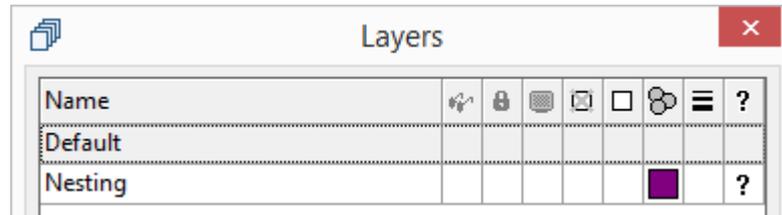
A "Sheet metal" tab is active for the prototype by default. New commands for designing sheet metal parts were added on it.

The "Sheet metal" tab on the Ribbon is divided to 2D and 3D parts with set of commands for modeling and drawing design.

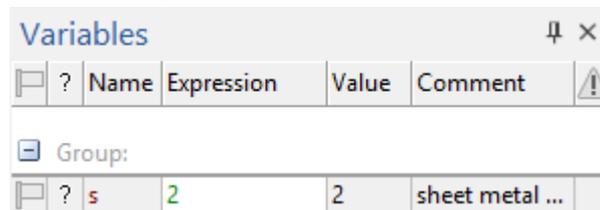




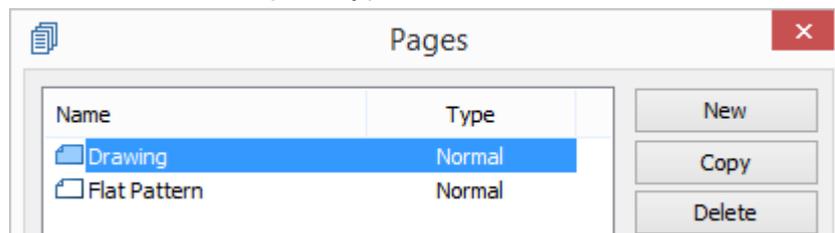
A special "Nesting" layer is created for integration with T-FLEX Nesting. Contours from the layer can be automatically added to the nesting project.



A special variable "s" was added to manage thickness of the part.



A "Flat pattern" page was created in the prototype.

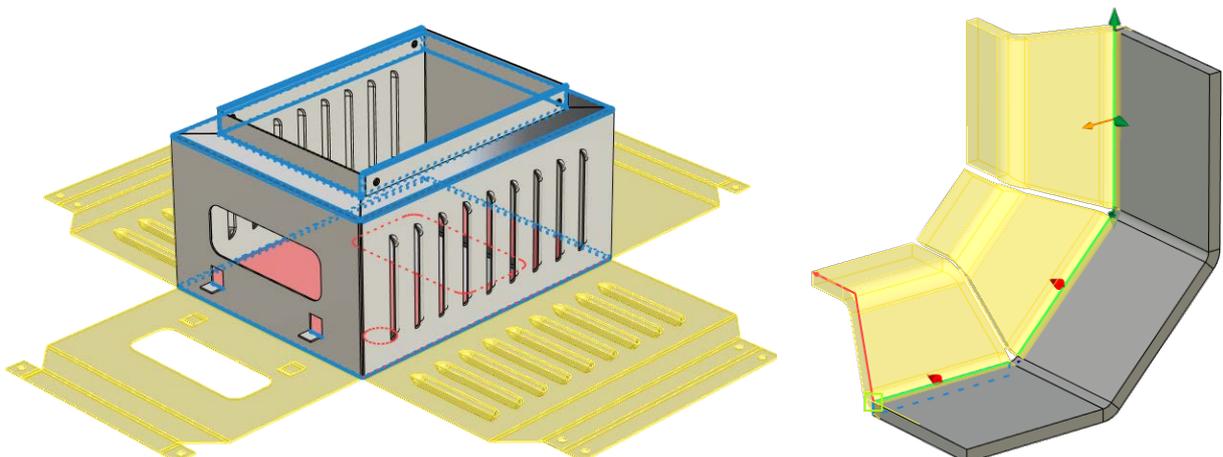


Commands Interface

The **Bend** command is divided into separate operations – "Flange", "Hem", "Contour flange", "Bend", "Cut and bend".

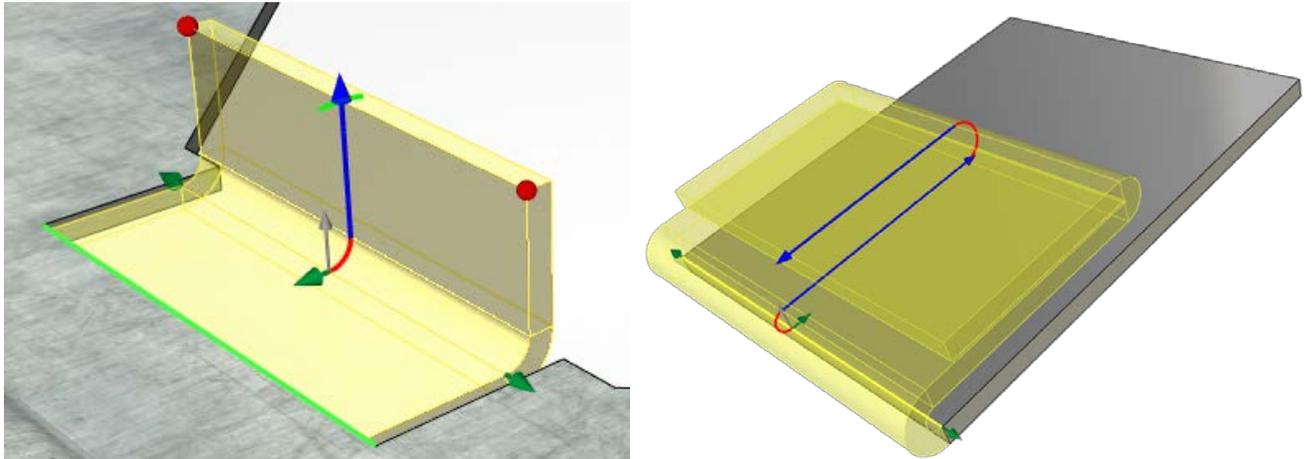
Interfaces of the commands have been redesigned to make it easier to create operations.

A translucent dynamic preview now available for all sheet metal operations. User immediately sees the result of the operation and can correct it.



Graphic draggers were added to all command of the sheet metal group to specify operation parameters (draggers for length, angle, intends, direction of bends).

Object snaps can be specified for draggers. It means that you can point other objects in the scene, according to which the operation parameters will be set, and snap to the objects. You can delete and reset snaps.



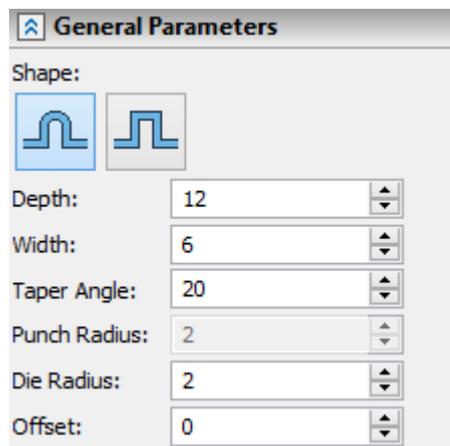
The **Operands** group appeared in all operations. It allows to manage the model geometry. All operations command are updated. It is easier to find operation in the 3D model tree.

"Gusset" Operation

A possibility of stiffness ribs creation for sheet metal was added. For this purpose, use the command **Gusset**. The command solves the problem of strengthening the construction.

Each of the ribs is created separately.

User should select a bend for which a gusset should be created and specify the gusset parameters. You can specify accurate gusset position. To do so, create a workplane and then select is upon operation creation using the  option.

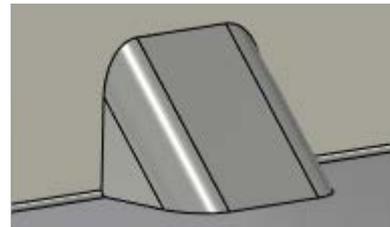
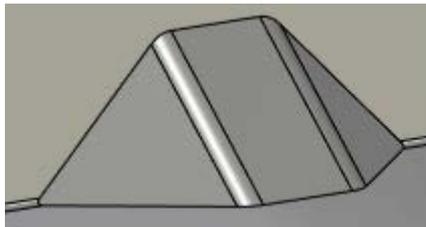


Shape. Specifies shape of the gusset - round or square.

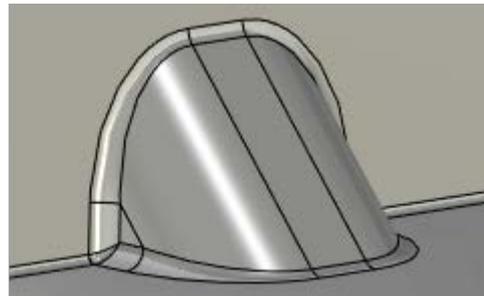
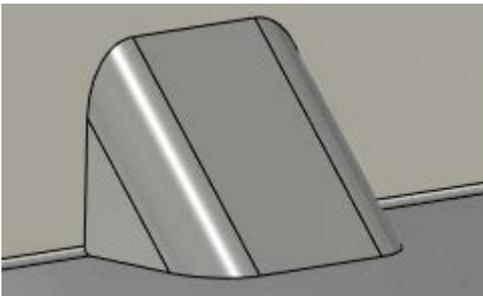


Depth and Width specify main gusset parameters.

Taper angle specifies an angle of gusset sidewalls.



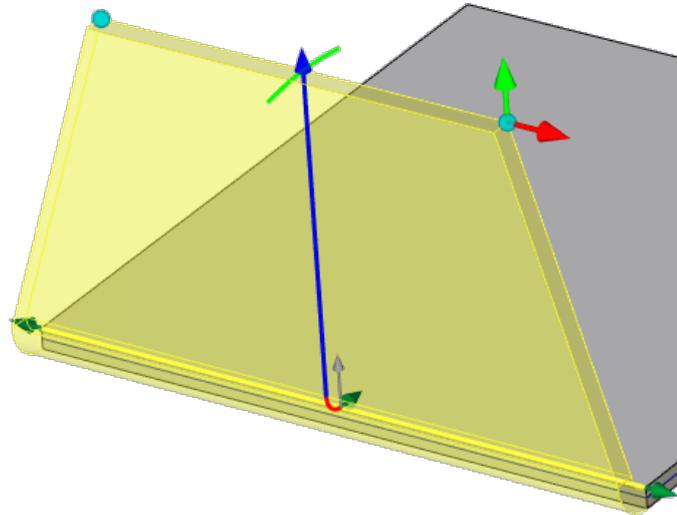
Punch radius and **Die radius.** Allow to take technological radius of die and punch into account when creating gusset.



Offset. Specifies position of the gusset. Offset is specified from the closest vertical face. If a workplane was specified upon gusset creation, the offset is calculated according to the workplane position.

Flanges of Nonrectangular Shape

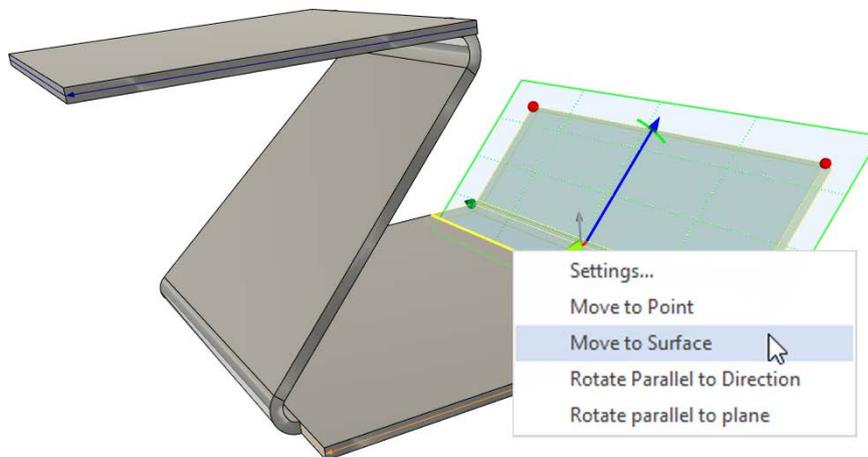
Now it is available to add flanges of nonrectangular shape to the part using graphic draggers in the "Flange" operation. The functionality solves task of trapezoidal tongues creation.



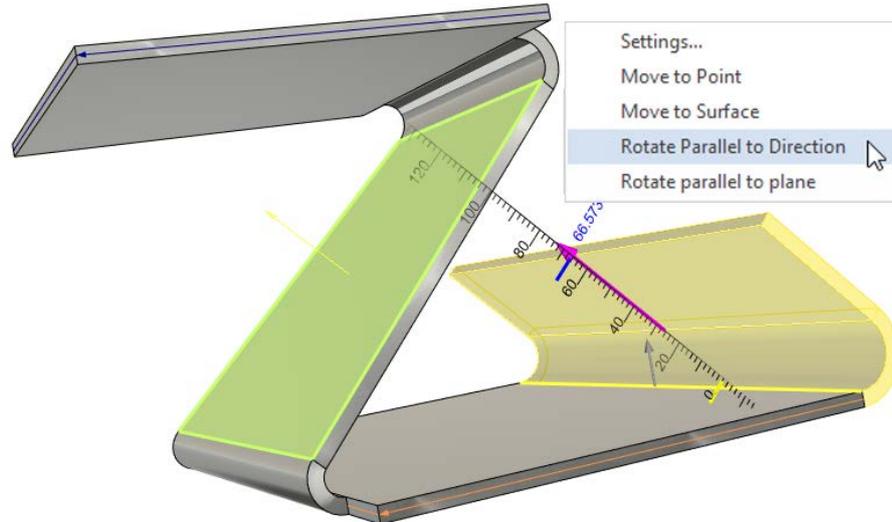
Flanges with Snap to Geometry

A possibility to create flanges with a snap and orientation according to the already existing geometry was added:

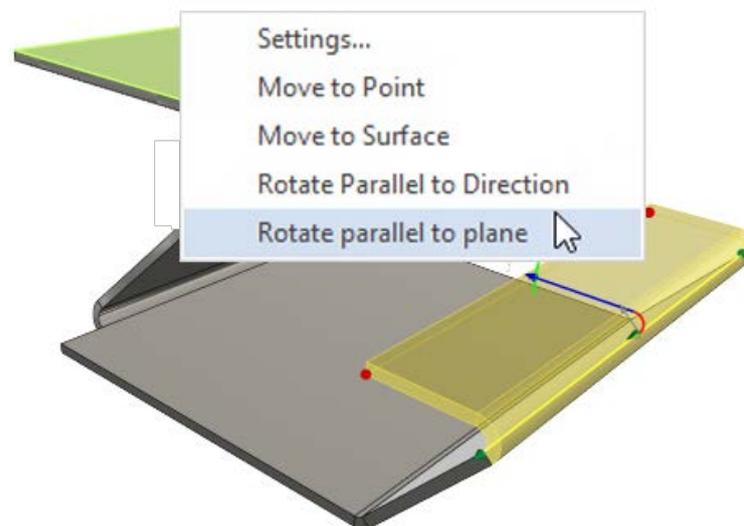
- ✓ - translation of flange to the selected plane with tilt angle that is defined by the plane;



- ✓ - rotation of flange parallel to the selected direction;



- ✓ - rotation of flange parallel to the selected plane;



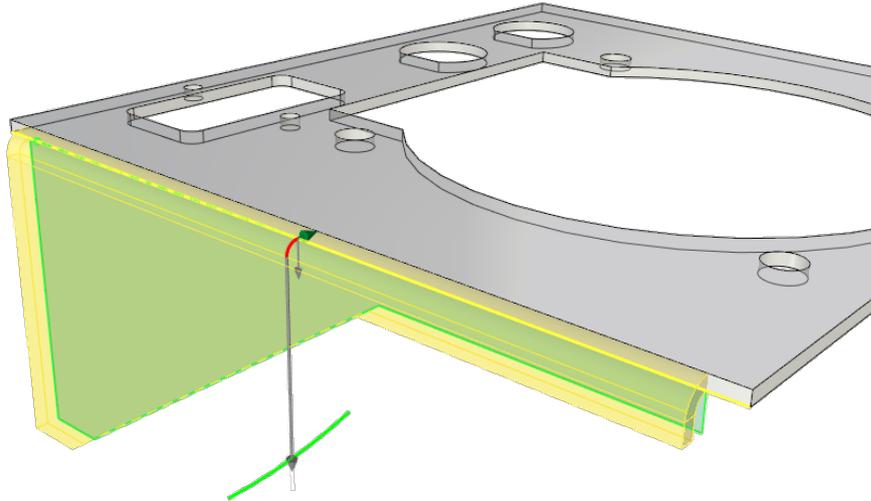
Workplanes, surfaces, body faces are used to orient flanges.

You can manage snaps using context menu, when you click on the appropriate dragger of translation or rotation.

Flanges by User Profile

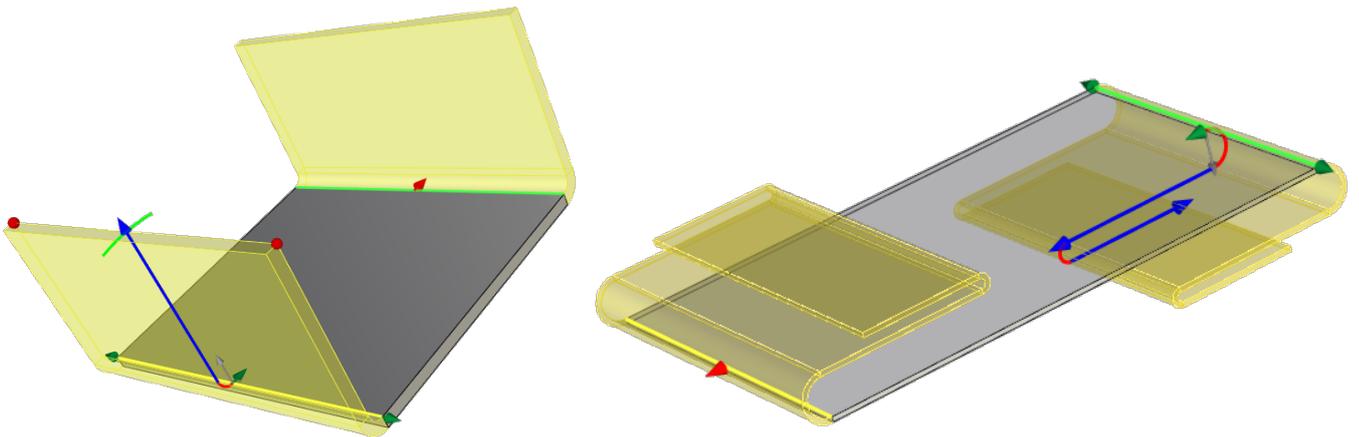
Now it is possible to create flanges using user 3D profile. You should preliminary create a 3D profile to create such a flange. The profile should lie in the plane of bend edge.

Profiles on the basis of construction lines or sketch lines can be used for the flange.



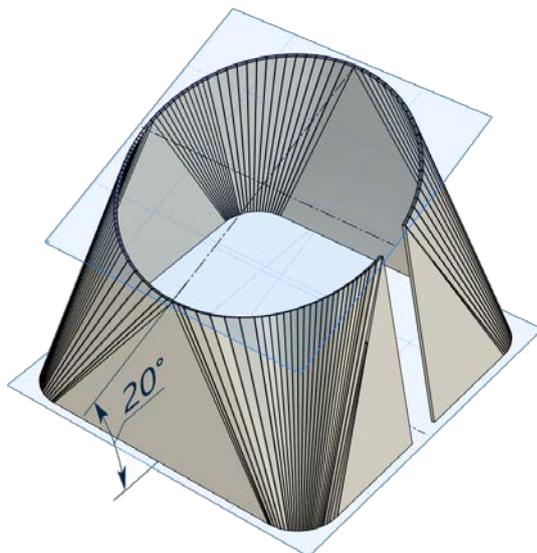
Creation of Flanges on Contiguous Edges

Now it is possible to create flanges and create hem on non-contiguous edges. Direction of each flange can be set individually using graphic dragger. Edges are processed automatically.



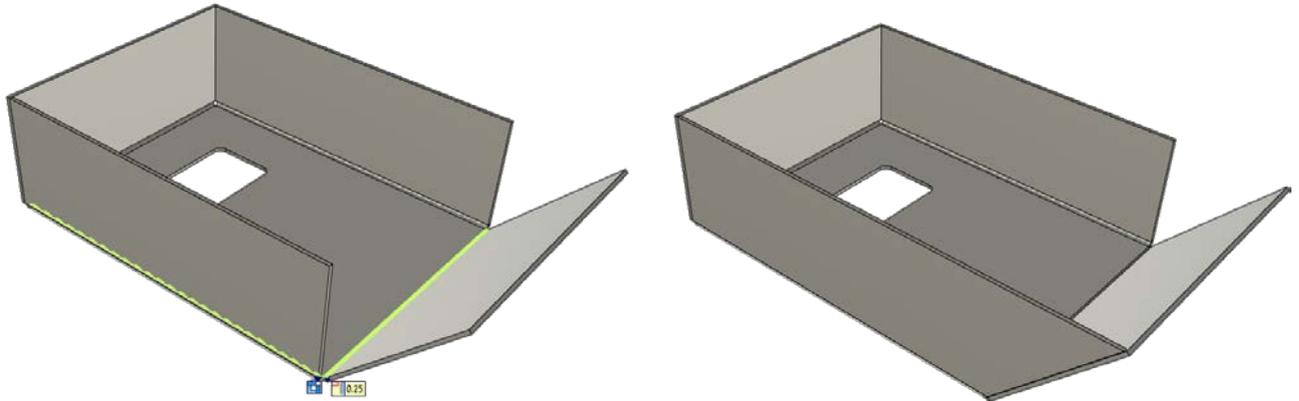
Lofted Bend

Now it is possible to use nonparallel contours in the "Lofted bend" operation.



"Corner" Operation

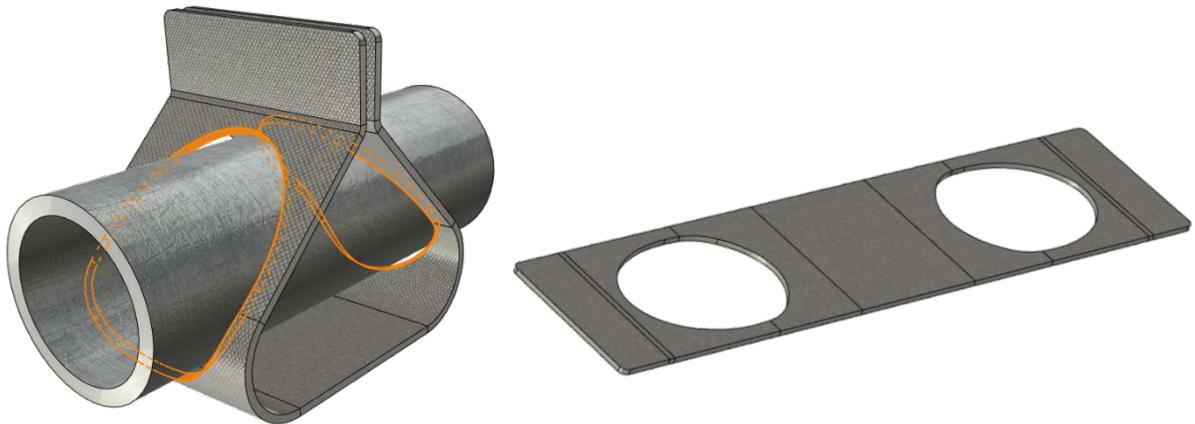
The **Corner** command allows to create reliefs and specify gaps in raw corners of sheet metal parts. Allows you to close angle for any flange configurations, including inclined flanges.



Normal Cutout

The "Normal cutout" operation allows creating cutout with a gap on the sheet body with sidewalls perpendicular to the main faces.

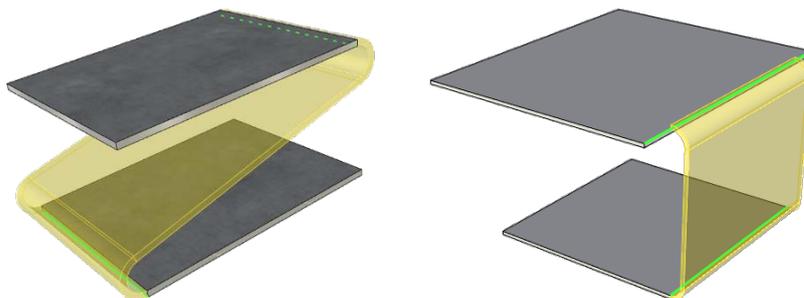
The operation should be used for cutout creation on the inclined faces. The operation allows to use geometry of the part for cutout creation and to specify gap for the cutout without additional actions.



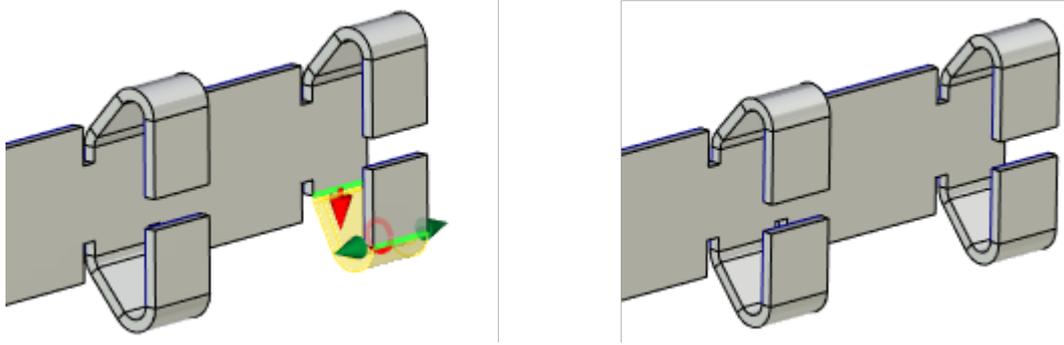
Bridge Bend

The "Bridge bend" operation allows to create a transitional geometry with the specified bend radius between the edges of two sheet bodies of equal thickness. The edges should be parallel. The result is a new sheet body.

There are two shapes of bridge bend available: Y - shape and Z - shape.

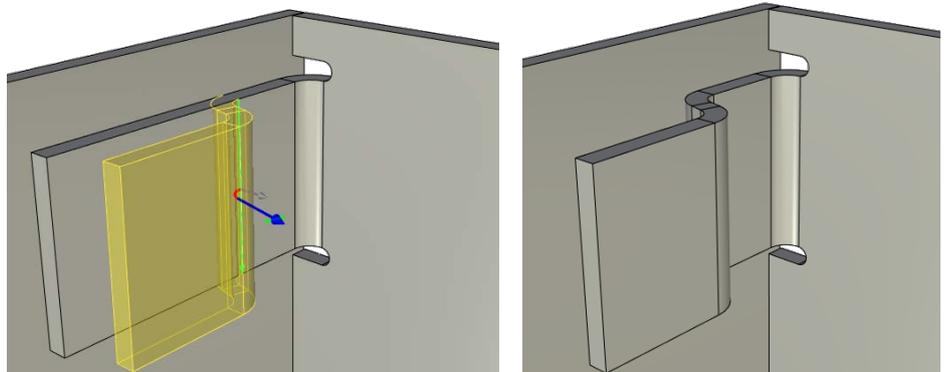
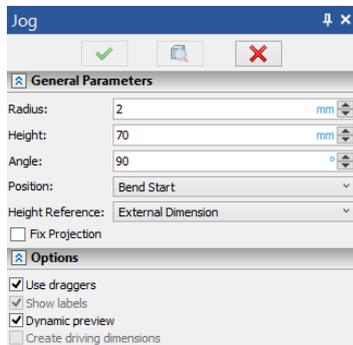


The operation allows to design assembly sheet models. Separate parts can be added to a single body using this operation.



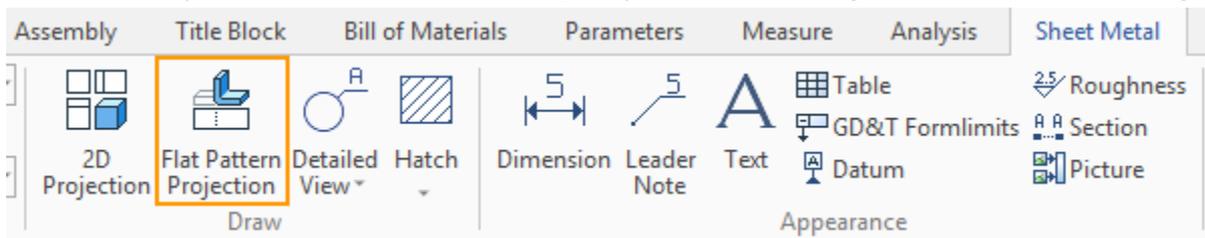
Jog

New **Jog** operation allows you to build a double fold on the sheet body at an arbitrary angle. It serves to couple two parallel flanges.



Flat Pattern Projection

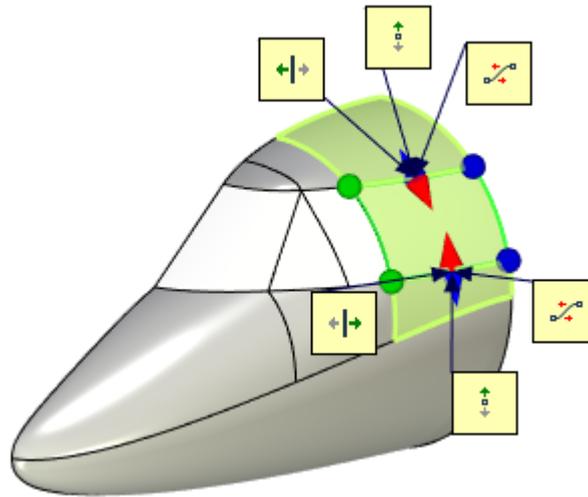
The **Flat Pattern Projection** command allows to quickly create a drawing of a sheet part unfolding.



"BRIDGE" OPERATION

A designer is often faced with the task of connection of two sets of surfaces using transitional surface. The **Bridge** command was added to solve the task with minimal number of actions.

The command allows to create a body that used for connection of two other sheet bodies or faces of solids.



You can specify a degree of continuity for the bridge: "surface (G0)", "tangent (G1)", "culture (G2)".

You can control the position of the bridge edges by using draggers in the scene or fields in the options dialog.

You can sew the bridge to the source bodies, or you can create a bridge as a separate body.

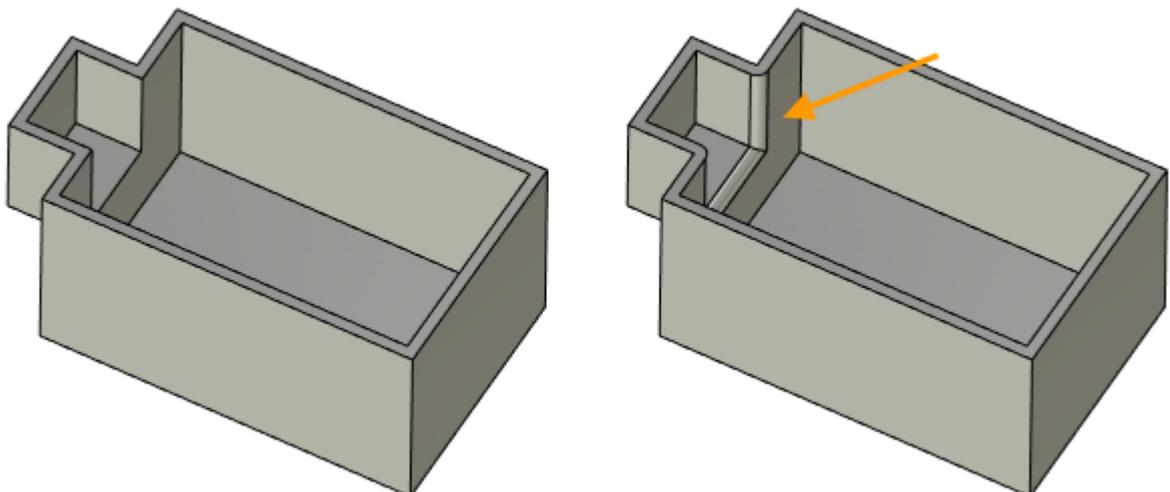
"SHELL" OPERATION

The following features were added to the **Shell** command:

- ✓ Option for automatic edge blending;
- ✓ Option for overlapping faces processing;
- ✓ Possibility to create shells on the basis of other shells with option to include or exclude additional faces;
- ✓ Draggers and dynamic preview of changes.

Blending of Convex Edges

The option **Blend convex edges** was added. The option specifies blending of convex edges that appeared in the direction of faces offset. For example, if the shell is created inside the inner convex edges will be blended as on the picture below.

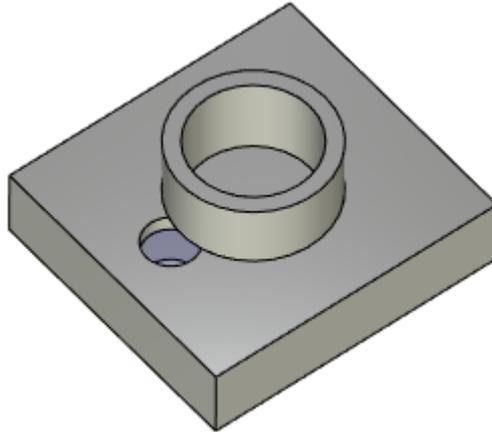


The value of blending radius should be inserted in the **Blend radius** field.

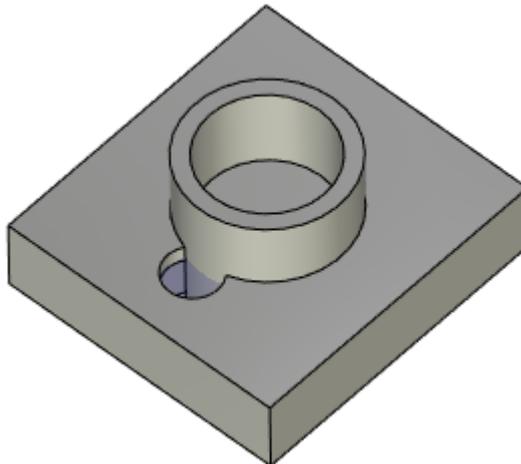
Face Extension Type

The **Face extension type** option manages processing of the overlapping faces, which appears upon shell creation.

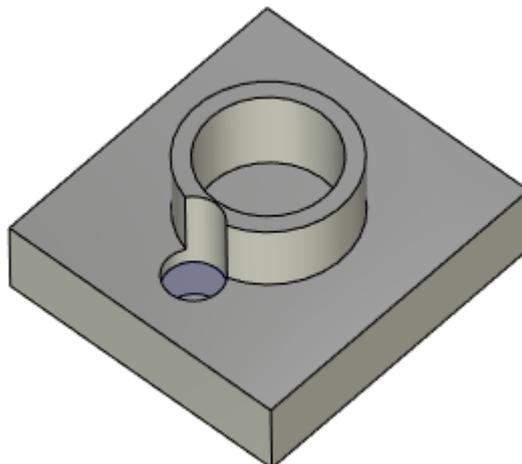
The system processes overlapping faces independently in the **Auto** mode.



In the **Extend moved** mode the priority goes to the moved faces. Fixed faces are processed according to the geometry of moved faces.



In the **Extend fixed** mode the priority goes to the fixed faces. Moved faces are processed according to the geometry of fixed faces.



Inclusion and Exclusion of Faces

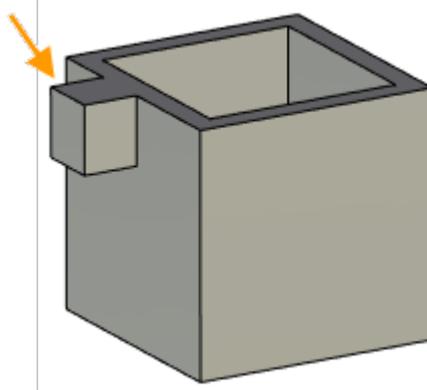
A possibility to include and exclude faces for part of the model if the main model is a thin-walled body was added.

The **Additional** tab was added to the parameters window of the command. The following options are available on the tab:

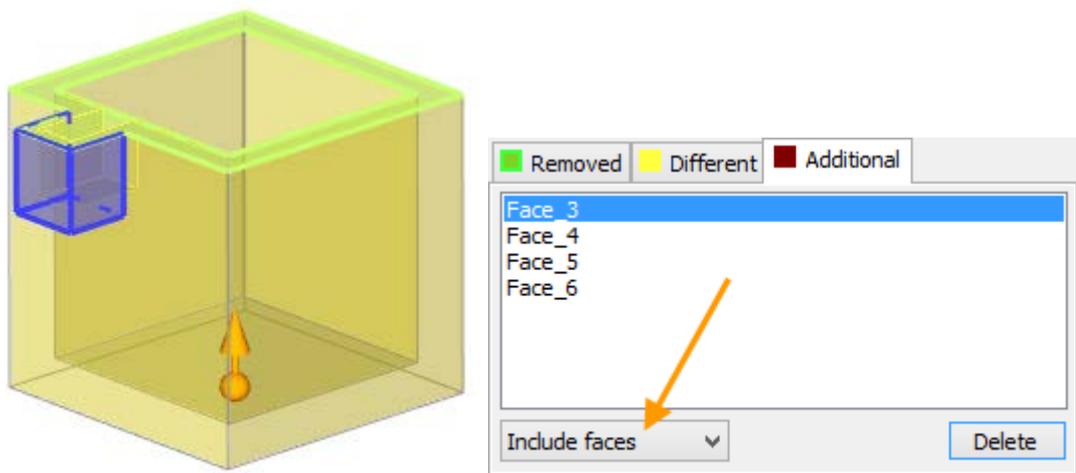
- ✓ **Include faces;**
- ✓ **Exclude faces.**

Include faces. The option allows to specify faces that will be included in the shell creation operation.

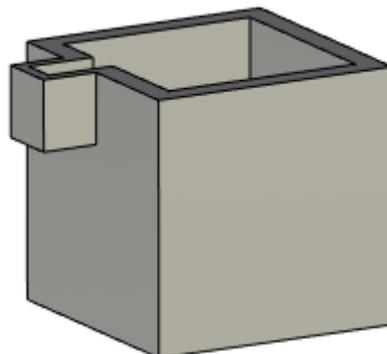
For the protruding part of the body in the figure below, you need to create a shell.



The system cannot determine the geometry independently, so you need to manually include the faces in the operation.



Result:



Exclude faces. The option allows to specify faces, for which the shell will not be created.

Operation **Shell** is sometimes applied not to all faces of the body, but only to their parts. In the example below, we need to build a shell only for the spherical part of the body.

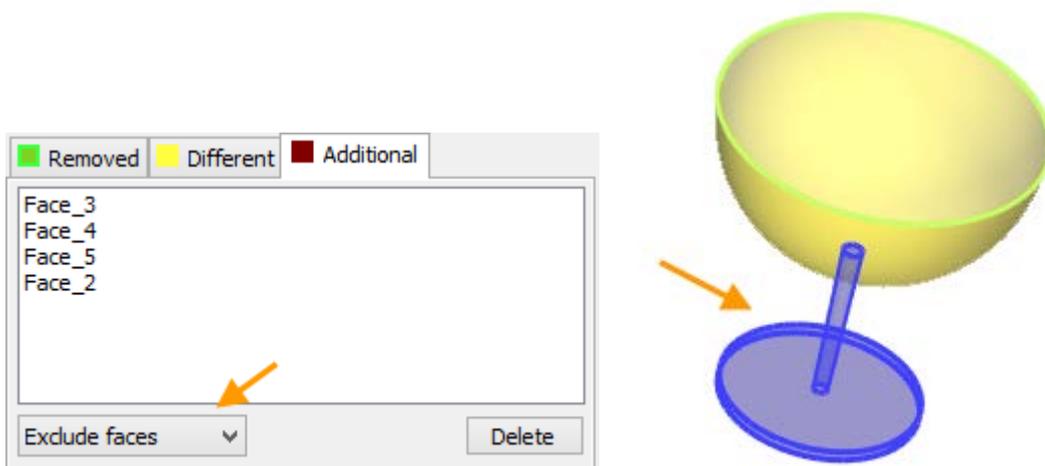


After selecting the removed face, the shell will be built for the entire body, and not just for its spherical part.



To get the desired result, you need to exclude faces from the operation.

To do this, you should specify four faces that will be excluded from the operation on the **Additional** tab.

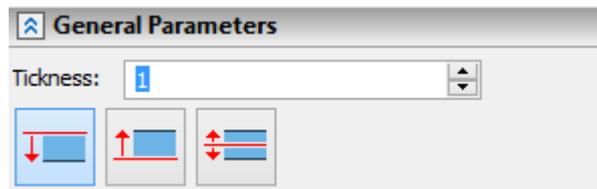


The required model will be obtained.



Other Changes in Operation

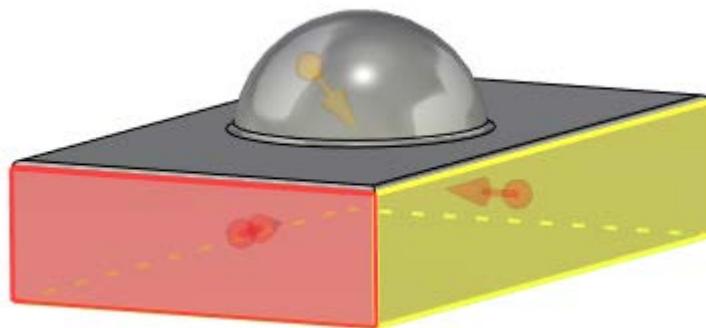
Buttons that allow to specify direction of faces offset upon operation creation appeared in the parameters window.



Dynamic preview and Use draggers options were added to the Shell command.

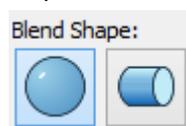
If the **Use draggers** option is active, a single dragger appears, which specifies the wall thickness and the direction of a face offset.

A dragger also appears for each face that was added to the **Additional** tab. It allows you to specify the direction of offset and the wall thickness for this face.

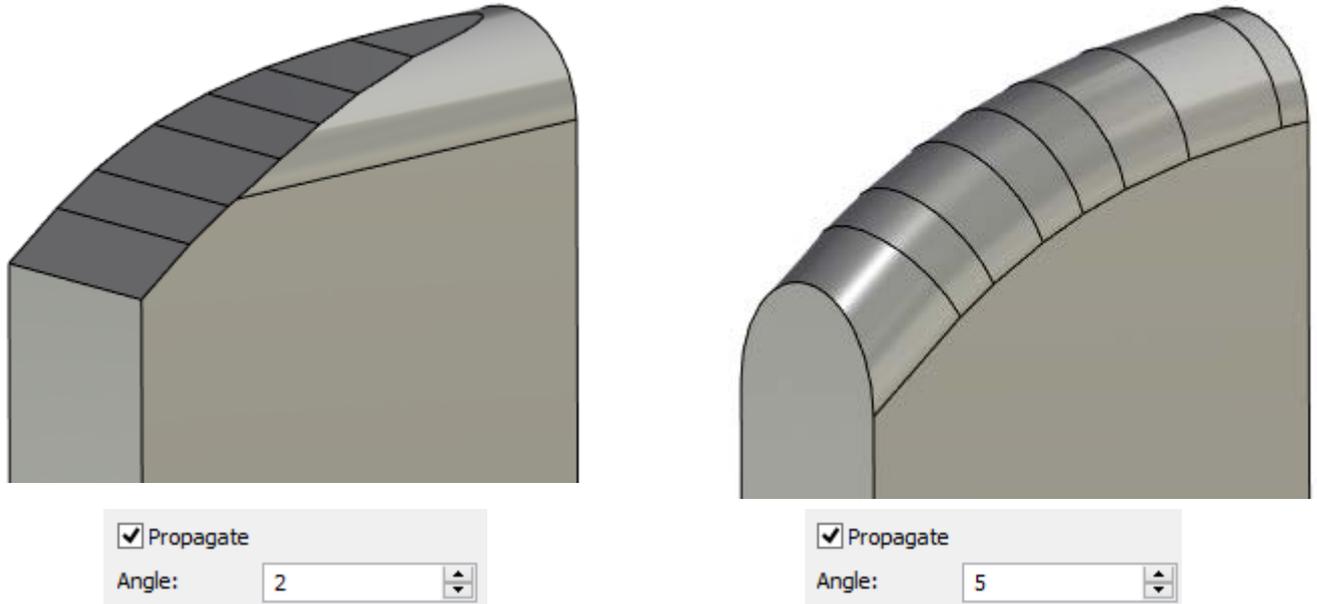


OPERATION "THREE-FACE BLEND"

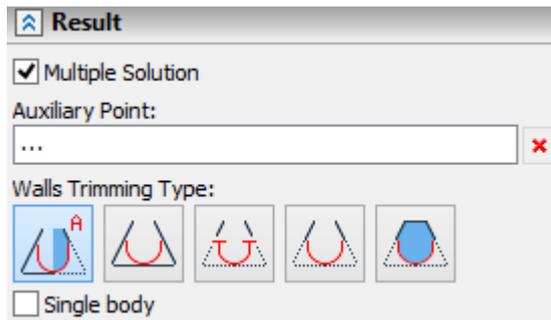
A new type of blending was added - spherical. Special icons are used to select the type of blending.



The **Angle** allows to set angle at which the conjugate faces of the center wall participate in blending. The parameter is available only when the option **Propagate** is active.

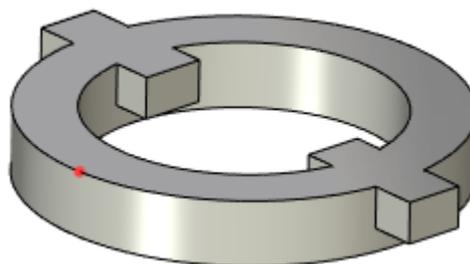


A new section was added to the parameters of the operation - **Result**.

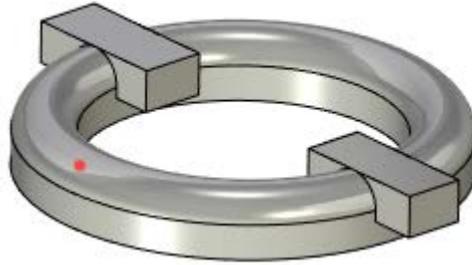


The **Multiple solution** option allows to specify exactly, which surface should be blended when performing operation. To define this surface, you need to specify a point in the scene using the **Auxiliary point** option.

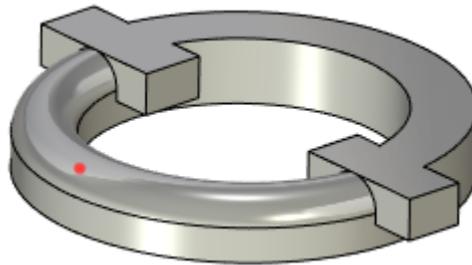
For example, we need to blend the model shown in the figure below.



With normal blending, both halves of the detail will be blended.



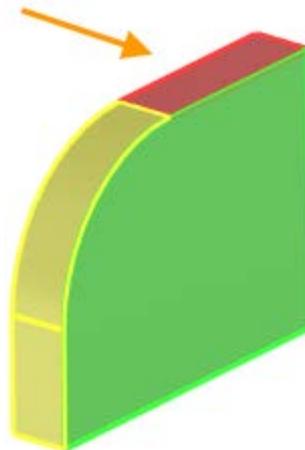
To blend only one part of the model, you need to select an auxiliary point. Only the closest to the given point faces will be blended.



When using the option the parameter **Trimming by walls** should be in the **By both walls** mode.

Automenu option **Smooth face chain selection mode**  allows you to simplify the selection of faces to create an operation.

If the option is active, then after selecting a face, the system checks for the presence of smoothly conjugate faces next to it. If such faces exist, they are also added to the operation.



Options for defining type of walls trimming were added to the command.



No Trim - blending face is created separately. Result is a separate surface.



Trim Both - Result is a separate surface.



Trim and Sew - If the blended surface lies between two different bodies, then they are united into one body.

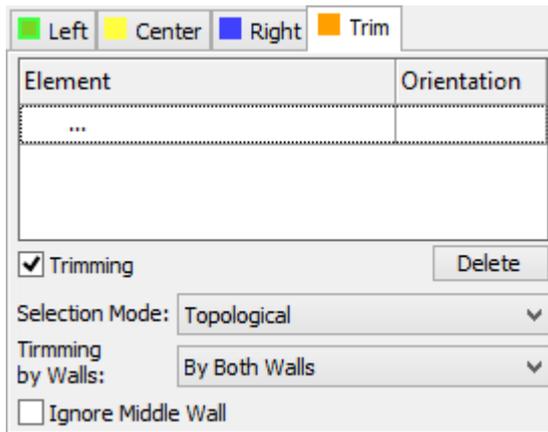


Create Solid - if the result of blending forms a closed space, a solid body is created.

Now you can create blend as a separate body. To do this, you should enable the **Single body** option.

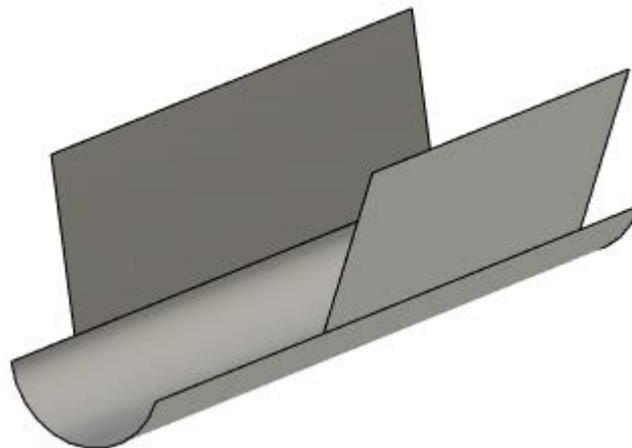
The **Trim** tabs was added to the command. It allows you to select elements in the scene to create a trimming operation, and allows you to build blending of the desired shape for bodies with different length.

The **Trimming by walls mode** is enabled by default. In this mode, you can select the mode for **Trimming by walls** parameter. The option Trimming by walls allows you to process bodies with different lengths.

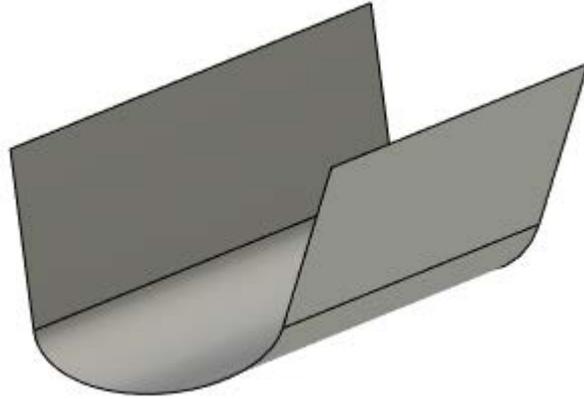


For example, the blending of three separate sheet bodies was created.

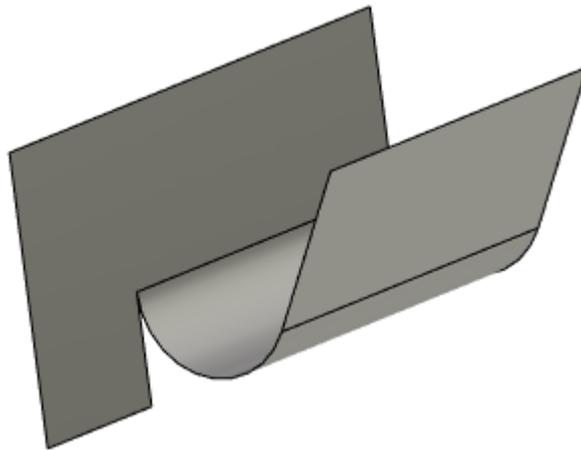
No Trim. Blending surface is not limited.



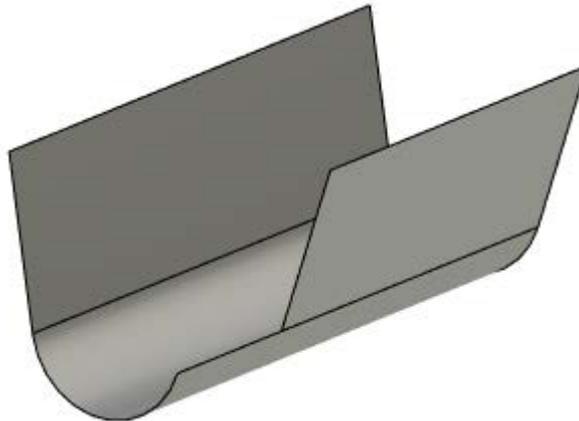
By Both Walls. Blending surface is limited by length of the both walls.



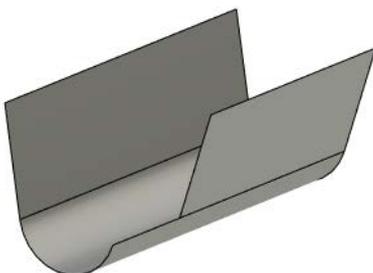
By Short. The blending will be created according to the length of the short wall.



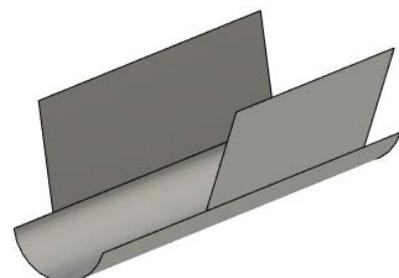
By Long. The blending will be created according to the length of the long wall.



The **Ignore Center Wall** option allows to ignore the center wall when creating a blending surface. For example, the option may be useful when you create a body with a trim in the **By Long** mode.

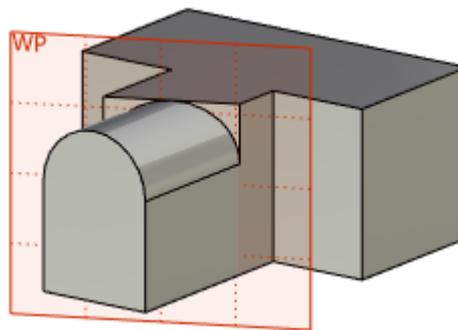
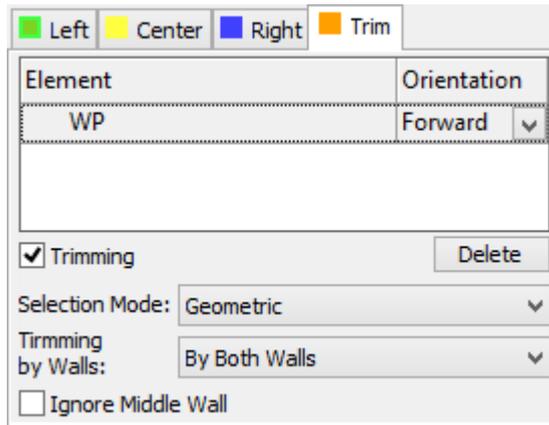


The option is enabled

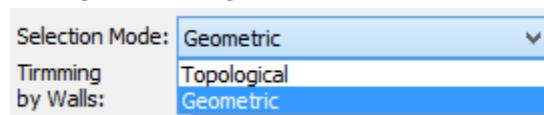


The option is disabled

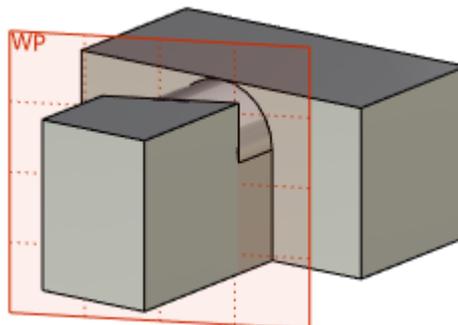
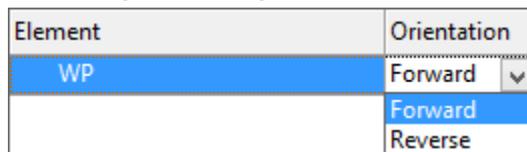
User can set element in the scene according to which the geometry will be trimmed.



You can choose topological (face, edge, etc.) or geometric (plane, surface) elements to create a crop.



You can also specify direction of trimming according to the selected element.

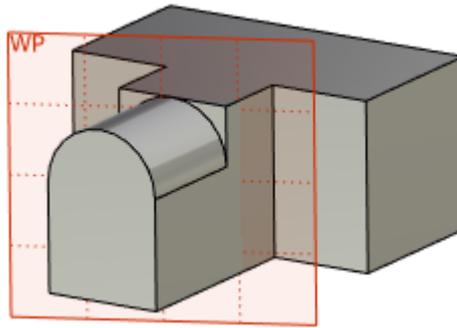


The **Trim** flag allows to specify type of the body trimming. If the flag **Trim** is enabled, the boundaries of the blending are defined by the position of selected element.

Otherwise, the system creates trimming perpendicular to the point that is formed by one of the following methods:

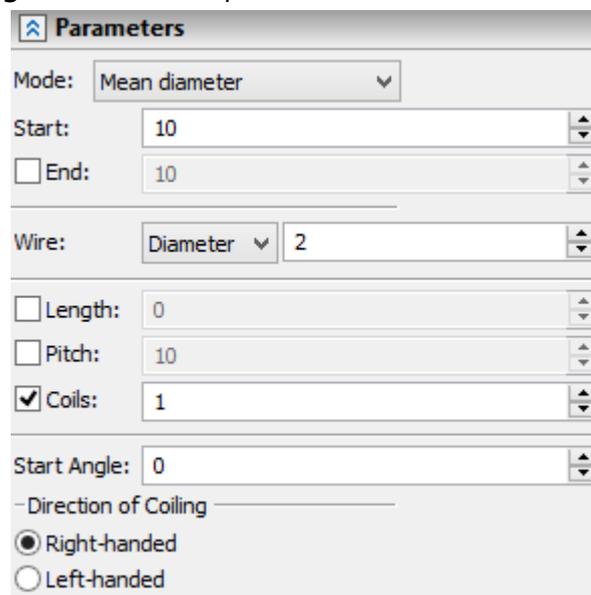
- for the spherical blending it is the intersection of the selected element and the blending vertex.

- for disk-type blending, this is the intersection of the selected element and the specified guide.

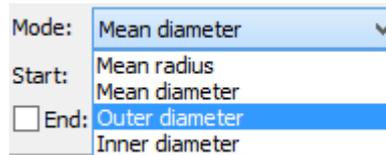


"SPRING" OPERATION

A set of options for the **Spring** command is expended to increase the convenience of the command.



The **Mode** option allows you to specify whether the radius or diameter will be used to specify beginning and end of the spring. Diameter can be specified as inner or outer.



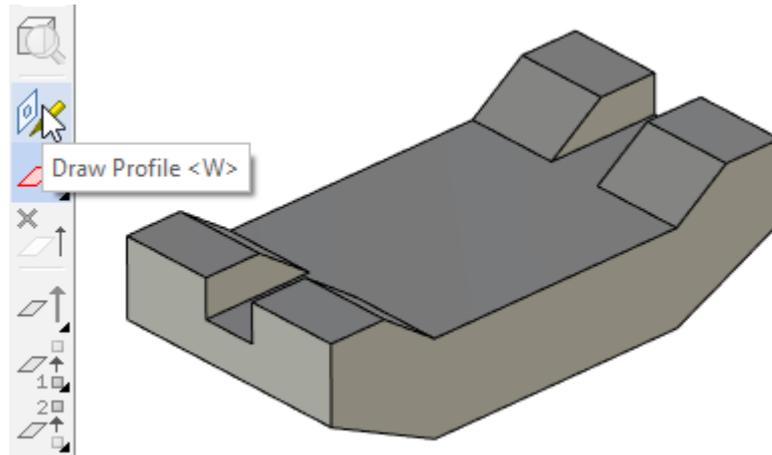
Now it is possible to specify the radius/diameter of the wire of the spring using the **Wire** option.

The spring can be created using any combination of input data: **Length**, **Pitch** and **Coils**. The system keeps track of the minimum number of parameters required to create a spring.

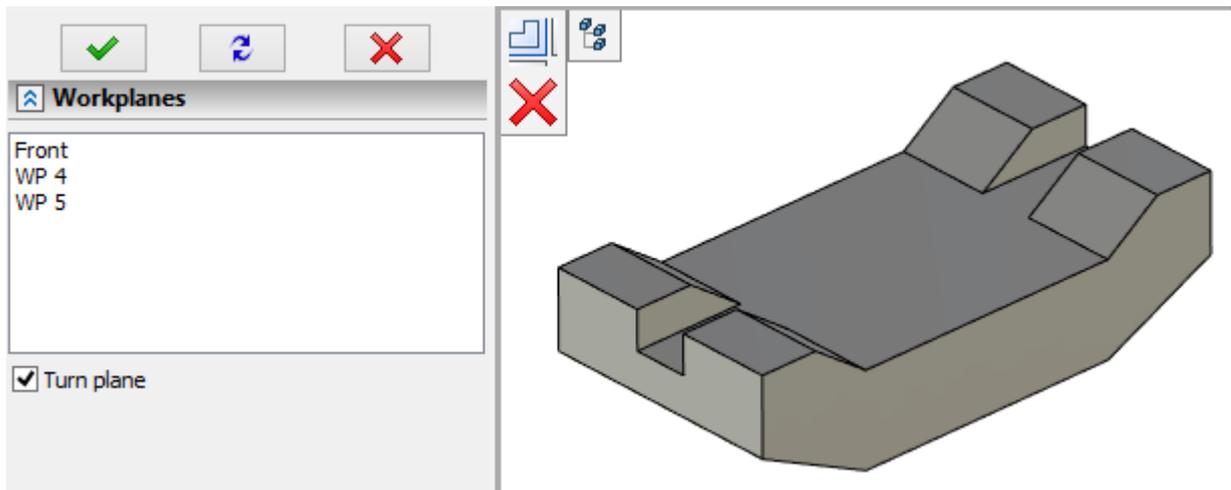
The **Start angle** option was added. The option allows you to set the start angle of the spring.

CREATION OF PROFILES IN COMMANDS OF 3D OPERATIONS

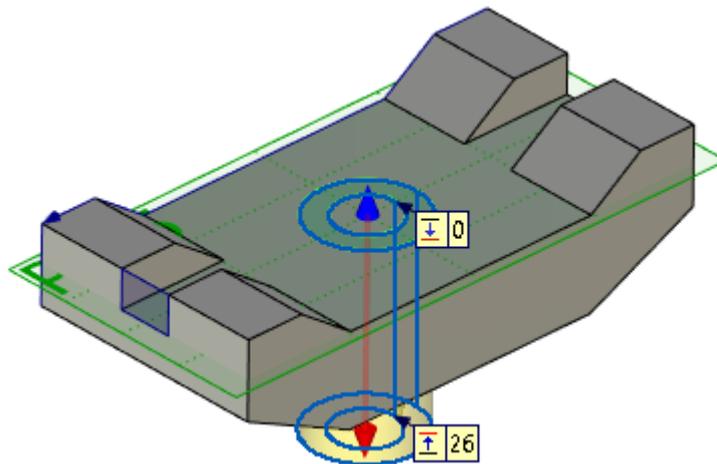
Now you can create a profile in a transparent mode from the commands **Extrusion**, **Rotation**, **Sheet Metal Part**, etc. Activate option **Draw profile** to do this.



After the option activation, the system prompts you to select the face or plane on which to draw.



After plane selection the system starts drawing on a workplane or face. When the drawing is complete, the system returns to the 3D modeling command and uses the profile to create the model.



If you do not have any profiles when you call the 3D modeling command, the system automatically prompts you to select a face or plane to create a profile.

Due to this mechanism, some 2D drawing commands are now available in the 3D scene. After they are called, you are prompted to select a face or plane to create a drawing.

MANUAL MASS-INERTIA CHARACTERISTICS CONTROL

Now it is possible to manually control the mass-inertial characteristics of bodies, 3D fragments and surfaces in the system. To do this, the "Characteristics" tab has been added to the parameters of the

elements. In order for the system to take into account the specified parameter, you need to activate the corresponding flag. To specify a value, you can use either constant values, variables or expressions.

Manually entered values are used:

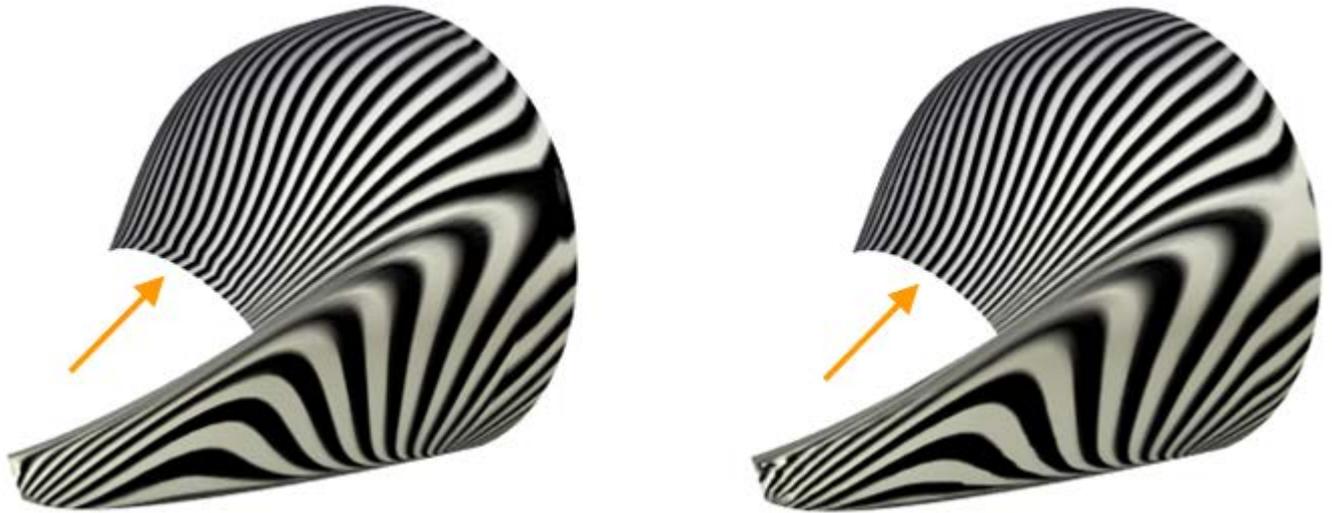
- ✓ When calculating the total mass and the coordinates of the product center of mass;
- ✓ In the **Measure** command, when receiving the mass using the appropriate functions in the variable editor;
- ✓ When forming BOM and other reporting documents.

This mechanism can be used when working with large-scale models of products, as well as purchased products, when there is no accurate model of the product, and its mass-dimensional parameters are specified in the form of requirements/data in the technical passport.

Besides specifying mass of separate bodies, it is available to specify characteristics of the whole document manually. In this case, the value is set in the **Document parameters** command. It will be useful when you create element libraries, and the fragment mass is not calculated automatically but is manually specified regardless of the number of bodies in the subassembly.

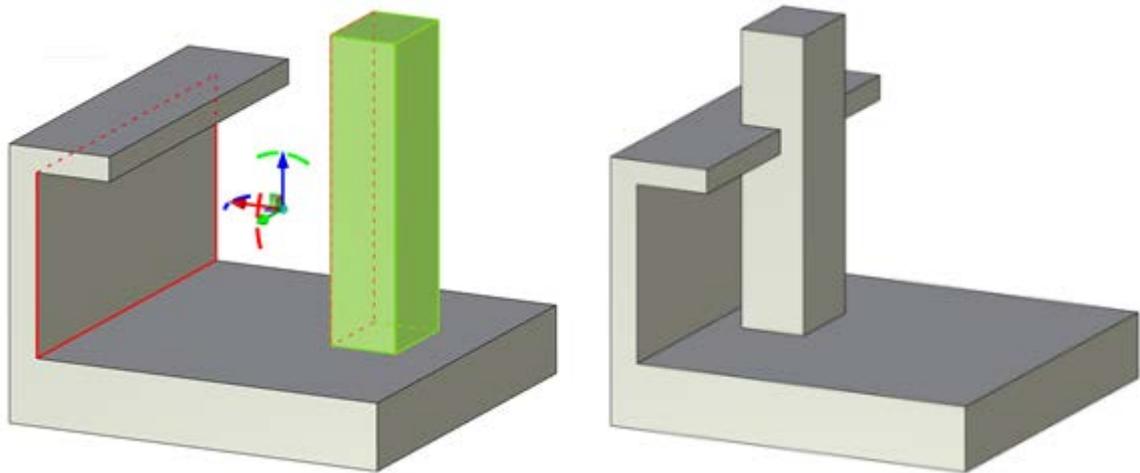
SIMPLIFY

A possibility of simplifying a spline surface was added to the command. If faces simplification cannot be done by reducing it to the standard face type, i.e. plane, cylinder, etc., then faces are replaced by one face with a smooth spline surface. This simplification improves the quality of the surface. The resulting surface is less demanding on computer resources.



FACES EDITING OPERATIONS

The basic algorithms for operating with faces have been updated. In particular, in the operations "Transform faces", "Replace faces", "Change faces" processing of the faces that are changed to other faces of the model with the correct processing of intersections was added. As a result of such processing, the composition of body elements can change (edges and faces appear or disappear). Example of the «Transform Faces» operation:

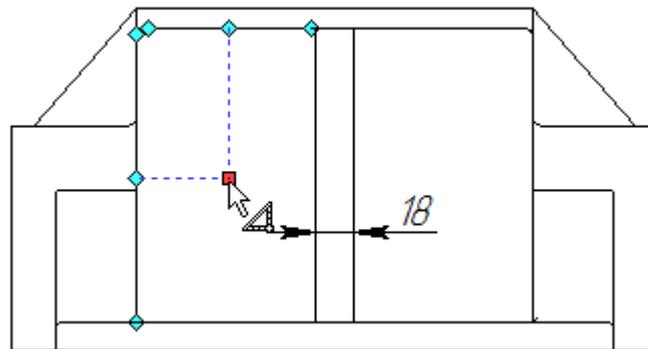


DRAWING CREATION

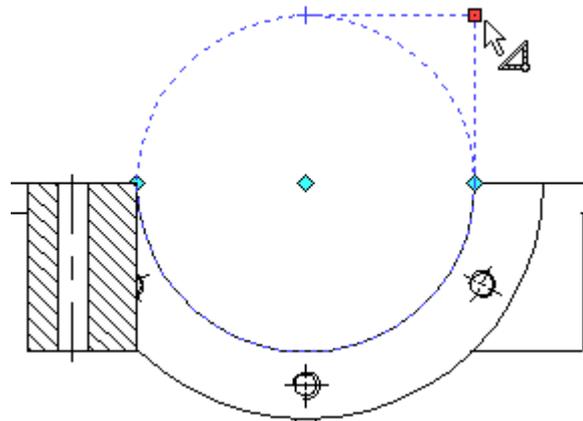
SNAPS

To improve the user experience, a new binding mechanism has been implemented. If one of the drawing commands is active, then when you hover the cursor on the object in the scene, its characteristic points are highlighted. These points can be used to associate dimensions and notations, or to create new drawing elements.

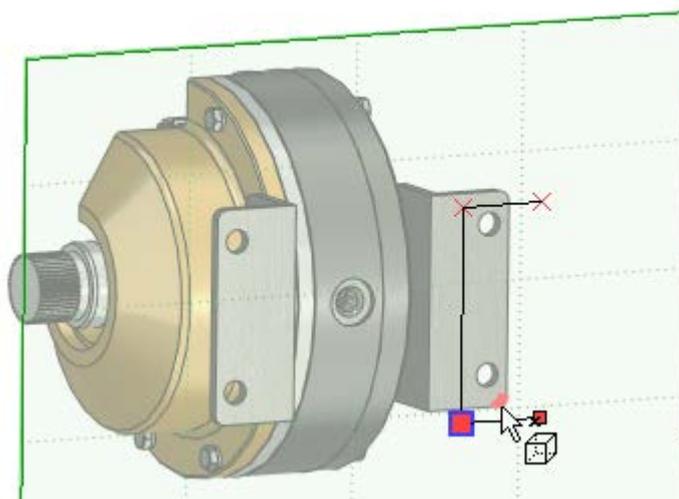
For example, the mechanism allows to create a new element relative to the center of rectangle.



In the same way, you can get a snap to the characteristic points of the circle.



A new mode of snapping to 3D elements allows you to create profiles using snaps to nodes and vertices of the 3D model.



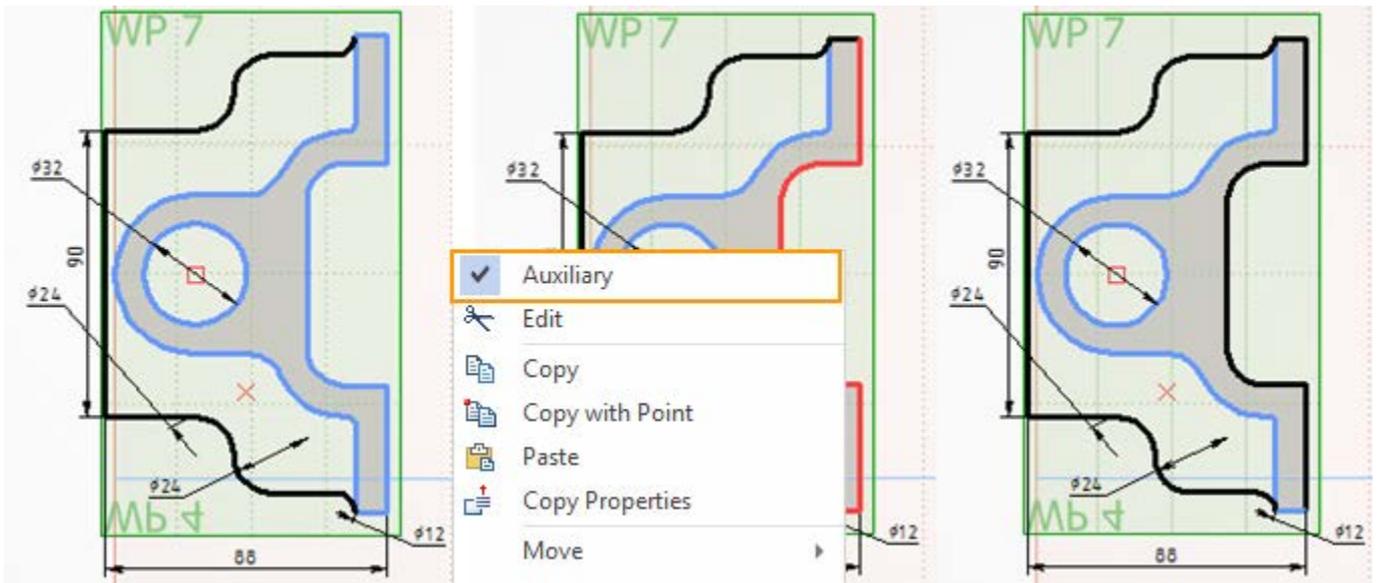
The snap is active if a special filter  is enabled.

AUXILIARY LINES

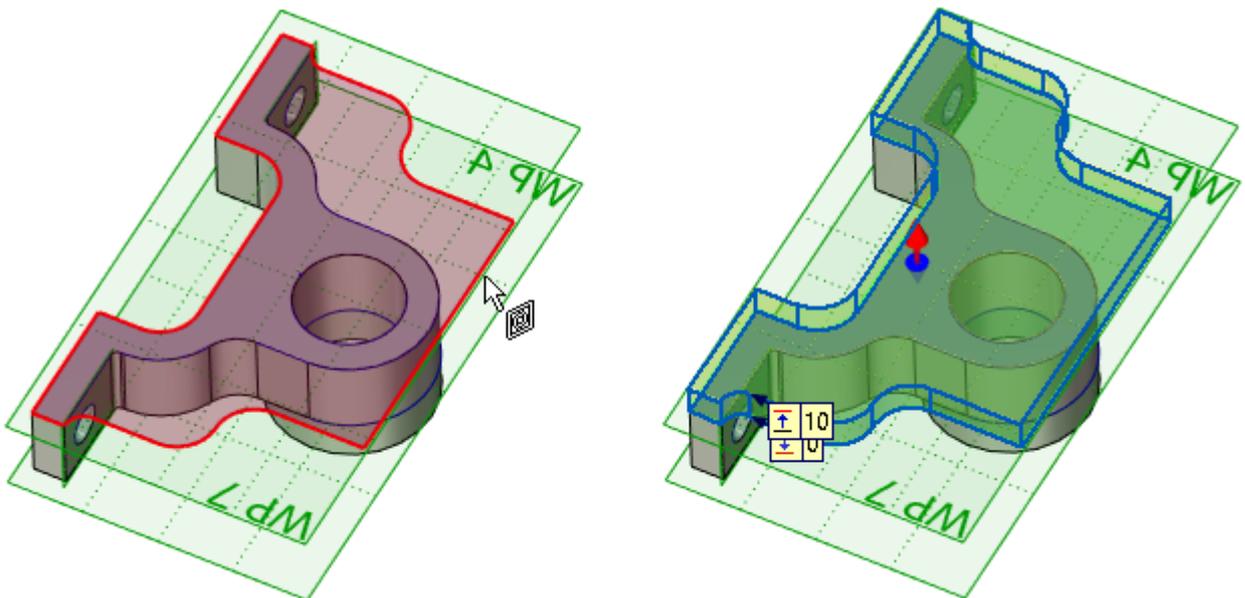
The **Auxiliary line** option was added to the lines parameters.

Now when performing command **Draw on Face** the face projection lines are displayed as auxiliary ones. You can not create a profile on the basis of auxiliary lines. But they can be converted to normal lines.

To create a profile, you need to select the necessary lines and remove the **Auxiliary line** flag in their parameters, or select an option from the context menu.

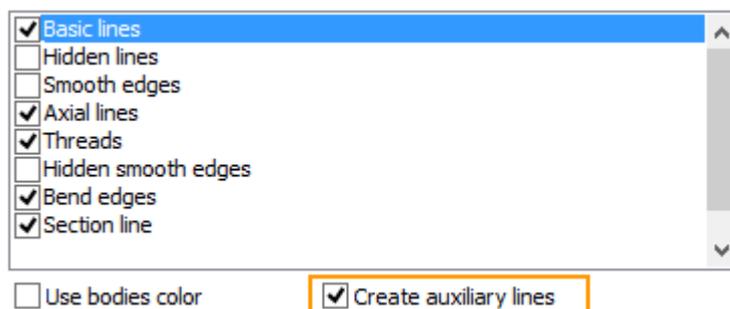


Auxiliary lines will become normal lines and can be used for the profile creation.



In addition, auxiliary lines can be used to snap new elements to them.

If necessary, when creating a projection, you can manually set the **Create auxiliary lines** flag and then all lines on the created projection will become auxiliary.



When you draw on the face of the body, this flag is set automatically.

Auxiliary lines can be hidden on the drawing in the **Hide construction** mode.

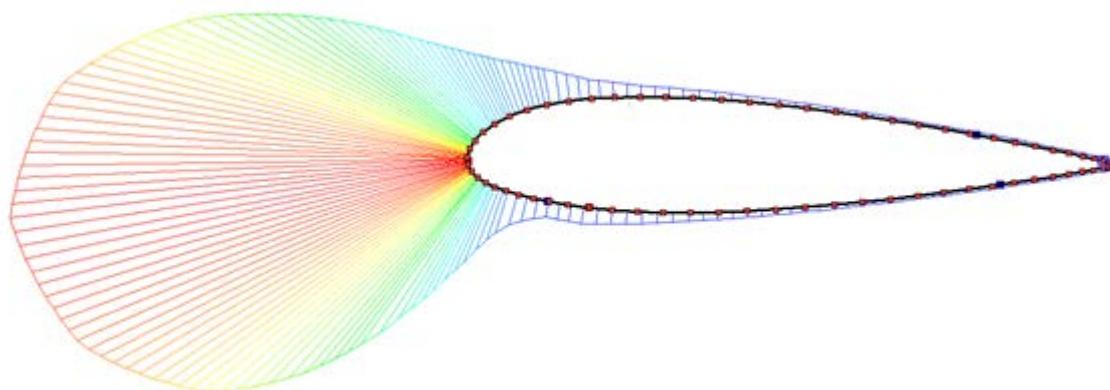
To switch the type of lines from normal to auxiliary, it is convenient to use special options in the context menu. They allow you to quickly switch the type of individual lines or all selected lines.

SPLINES

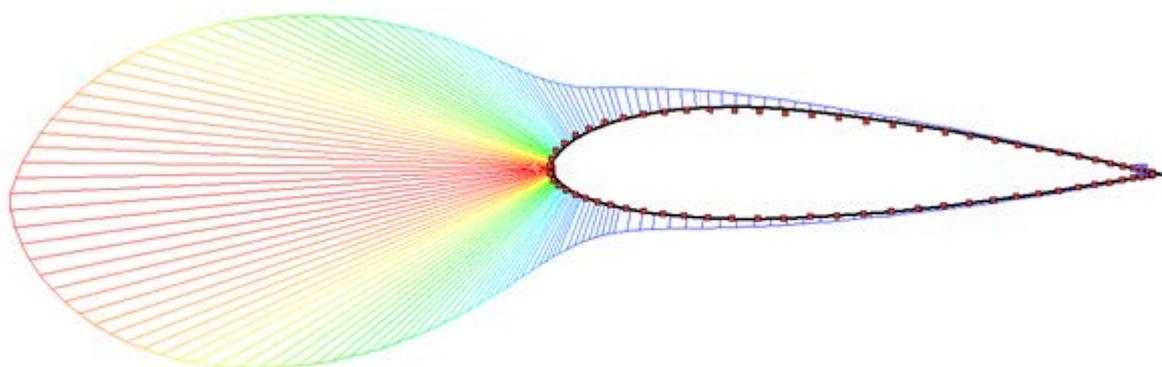
The option of spline creation in the **Sketch** command is substantially refined. The spline is constructed as a smooth curve without approximation by segments.

The smoothness of the diagram depends on the given tolerance.

value 0.0001

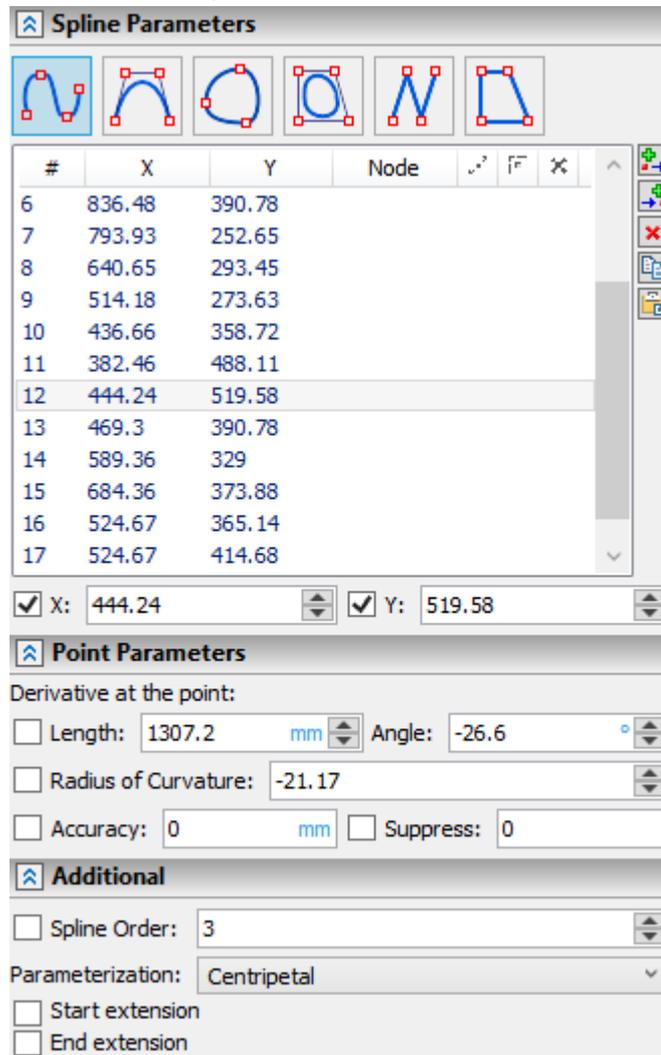


value 0.01

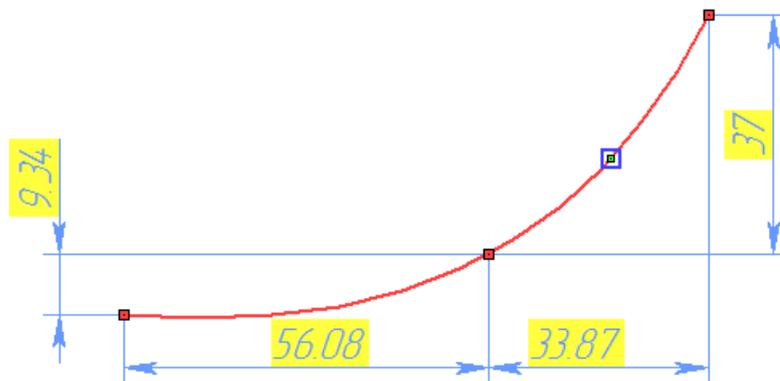


The general scenario of work was extended with:

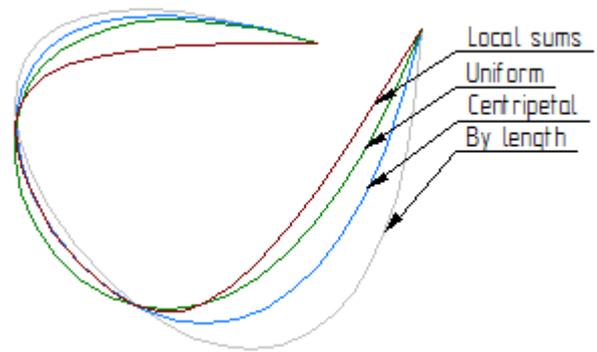
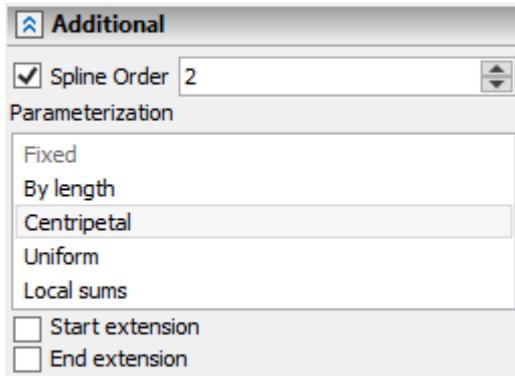
- ✓ The ability to switch the spline type is added;
- ✓ The modes for creation of closed and non-closed polyline are added;
- ✓ The list of points was added to the parameters window. The list can be edited;



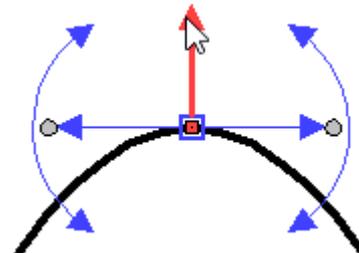
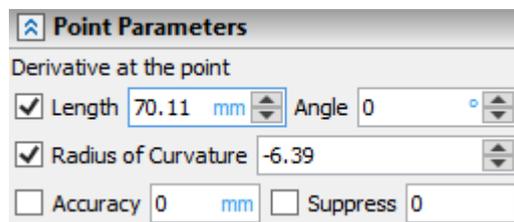
- ✓ Coordinates, snapping to nodes, boundary conditions, tolerance and suppression mode are displayed in the points list;
- ✓ The ability to transparently add new spline points and automatically set dimensions for spline points was added;



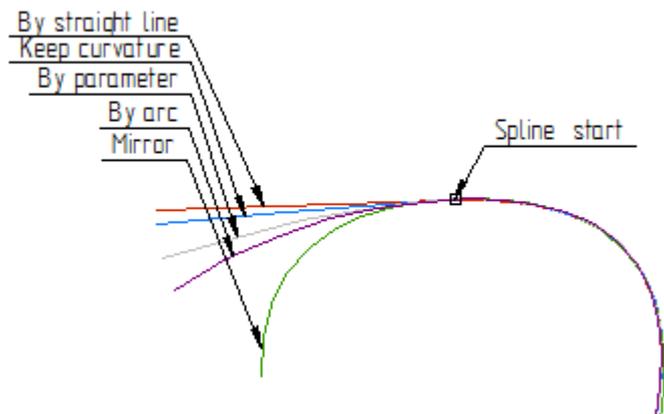
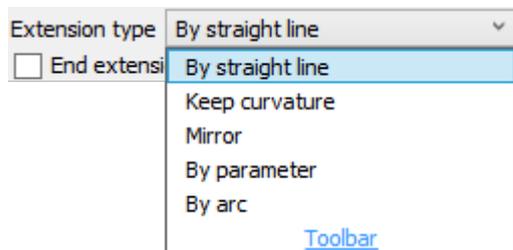
- ✓ Several new spline parameterization methods have been implemented;



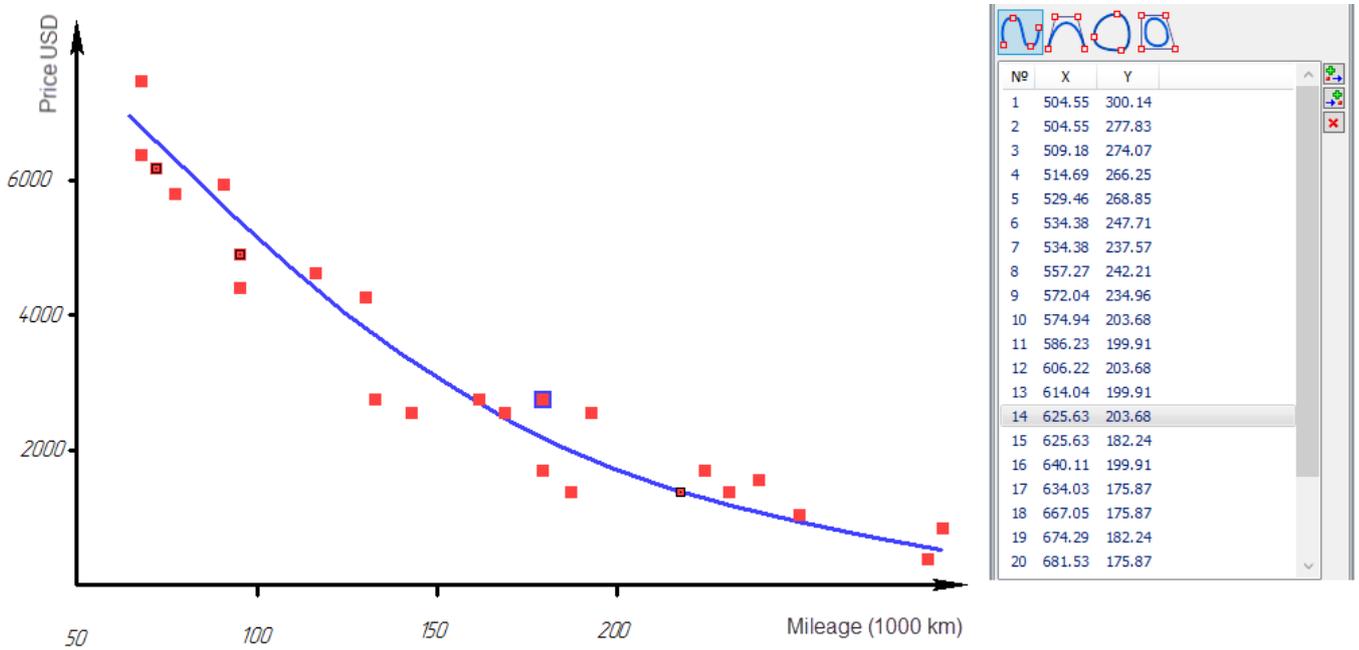
- ✓ Possibility to control boundary conditions using draggers including intermediate points of the spline was added ;



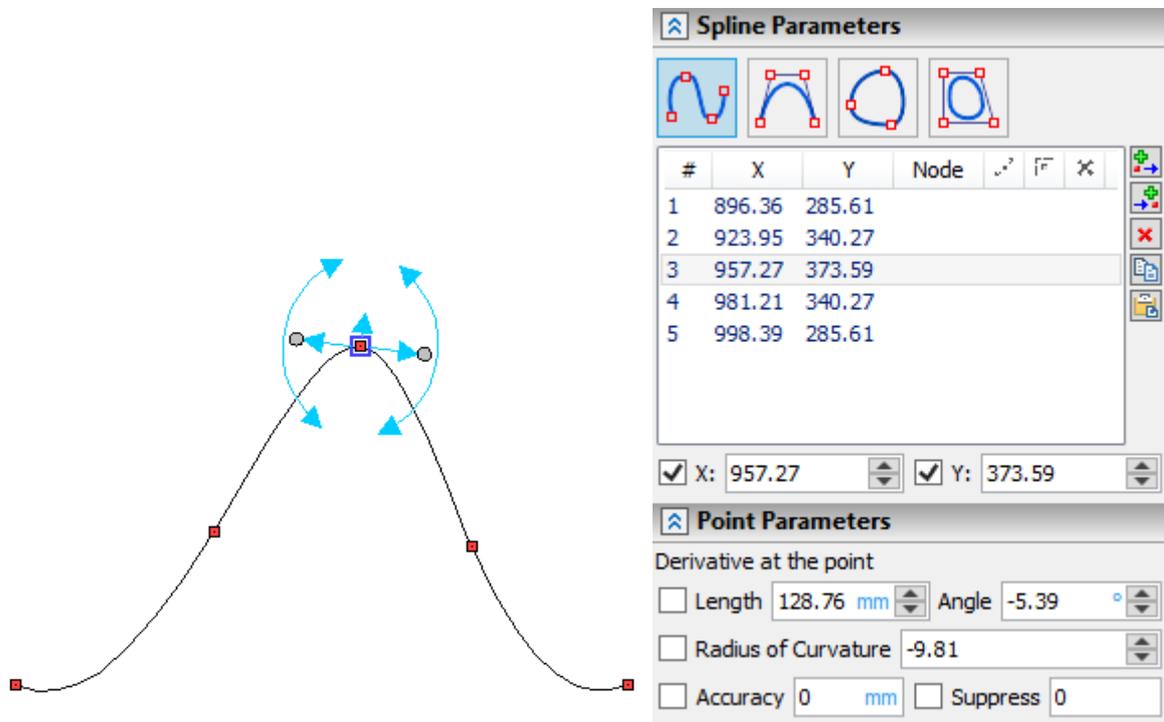
- ✓ Possibility to control of elongation/shortening of the spline using different methods is realized.

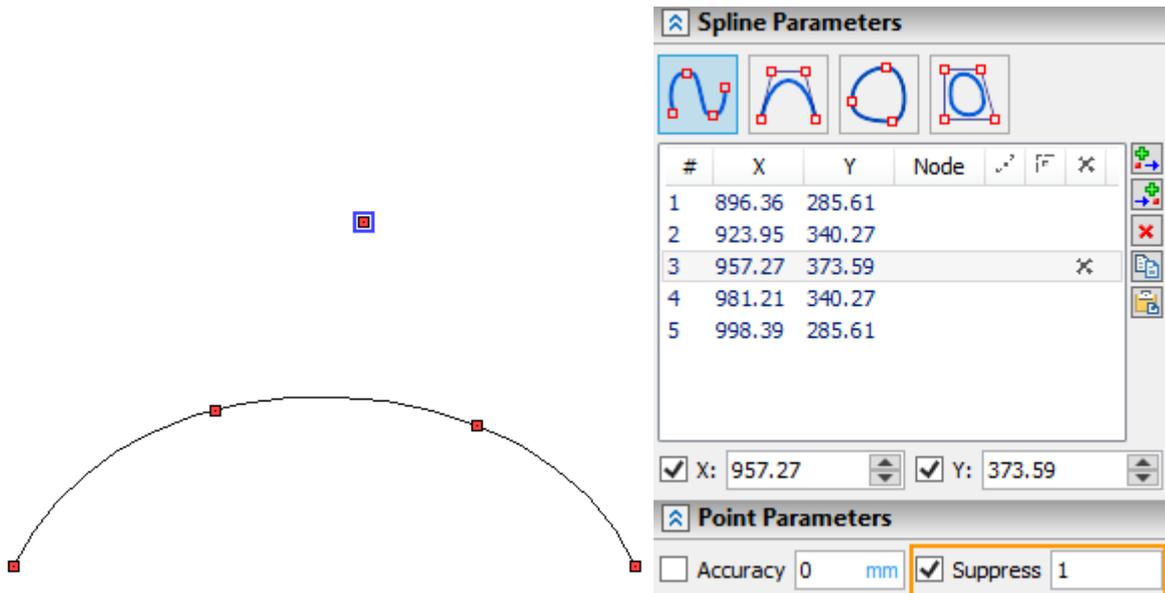


- ✓ possibility to create approximating splines with a given tolerance using the **Accuracy** parameter was added. Tolerance is specified for the entire spline, i.e. cannot be different for different points. This solves the problem of creating a smooth spline over a large array of points.

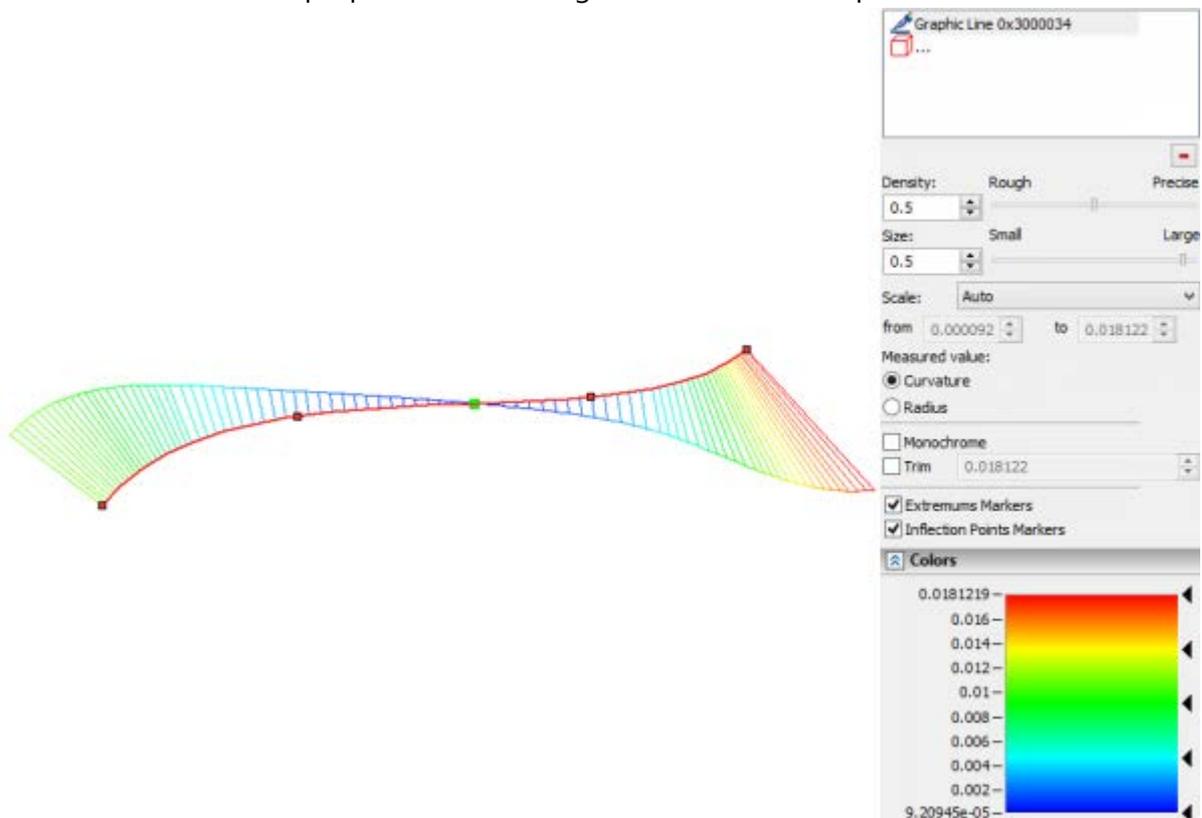


- ✓ possibility to suppress individual spline points using the **Suppress** parameter was added. You can use variables or expressions to control the suppression.

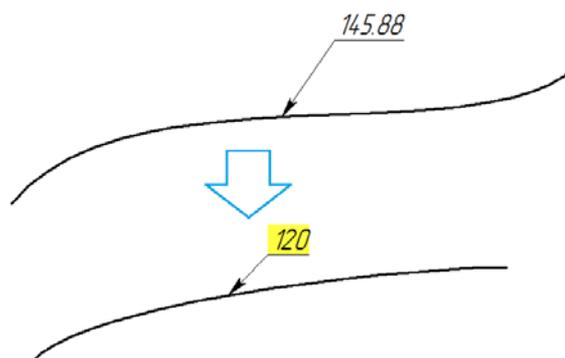




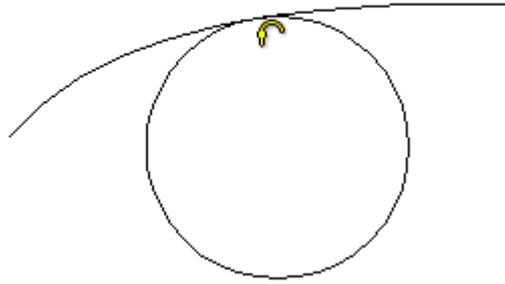
- ✓ possibility to display the curvature of the curve using the context menu command **Show curvature** and control properties of this diagram was added for splines.



There is an opportunity to control the length of the spline with the help of the leader dimension.

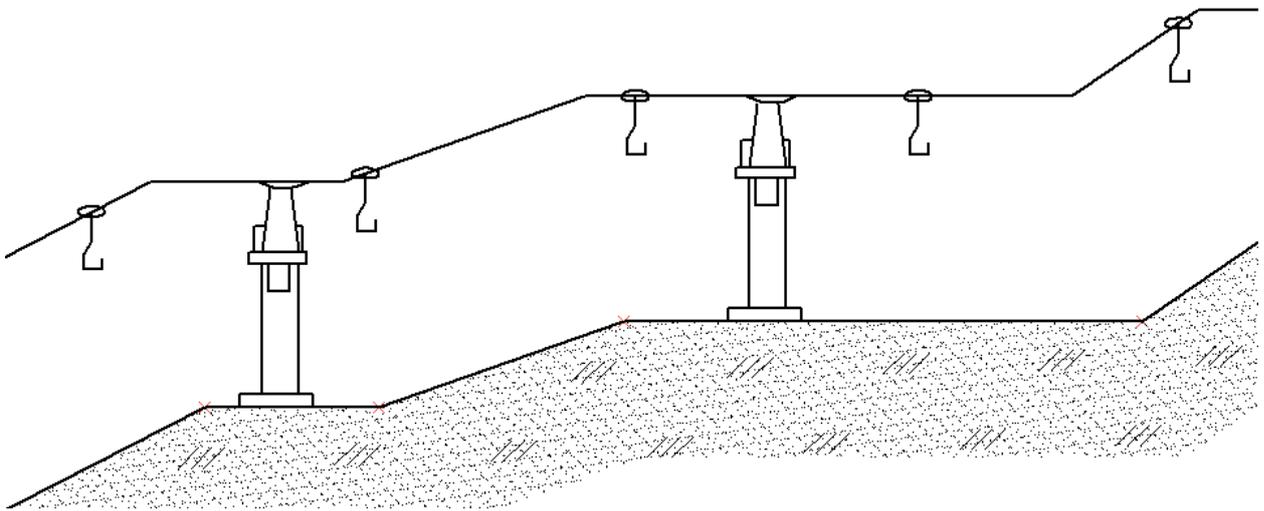


You can set constraints on other elements of the drawing. For example, when using constraints, it is easy to construct splines that are tangent and smoothly conjugate to each other or to other drawing lines.

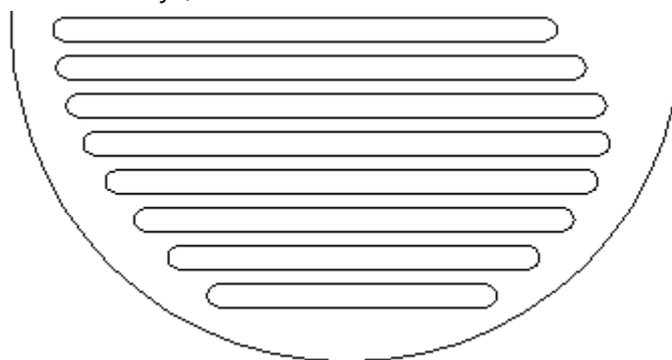


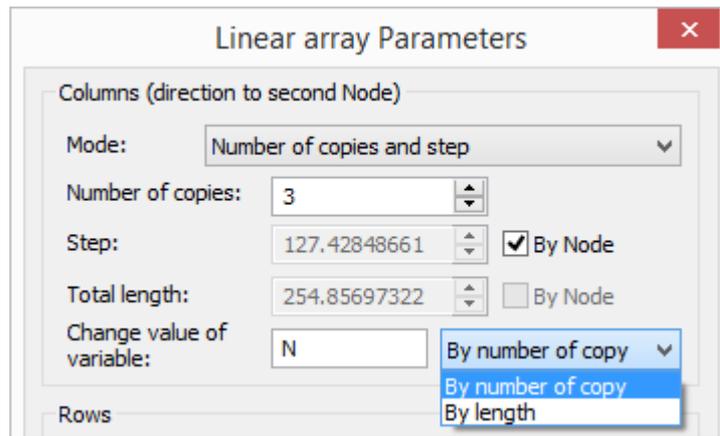
2D ARRAYS

A new type of 2D array **Array by curve** was added. It allows you to place copies of image lines along the selected path with the ability to control the orientation of the copies.

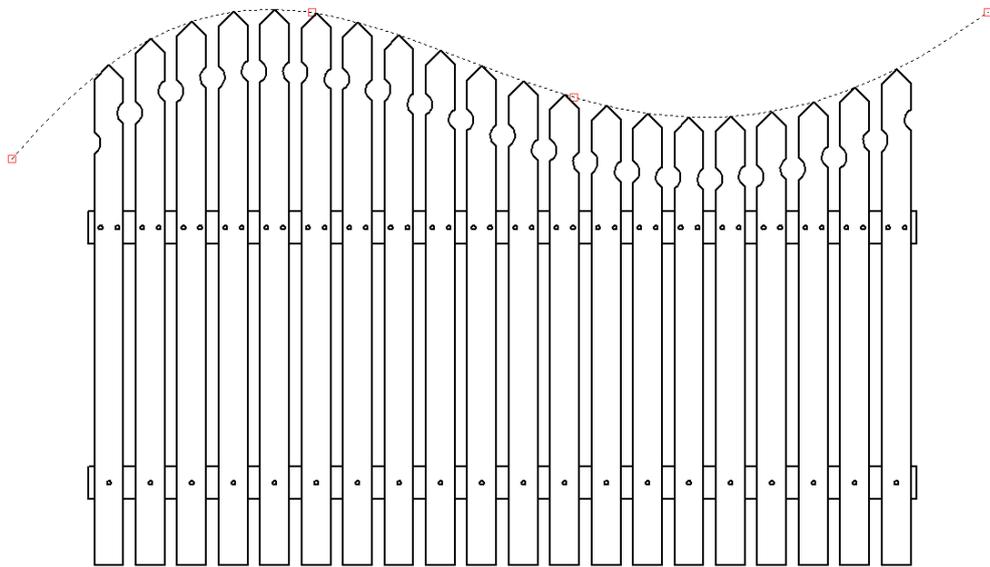


For linear and circular arrays, the ability to control variables during the generation of copies was added. To do this, the parameter **Change value of variable** was added to the parameters of the arrays. The variable takes a value either according to the copy number (**By number of copy mode**), or according to the distance from the start of the trajectory or according to the angle value (**By Length mode** for linear arrays, **By Angle mode** for circular arrays).



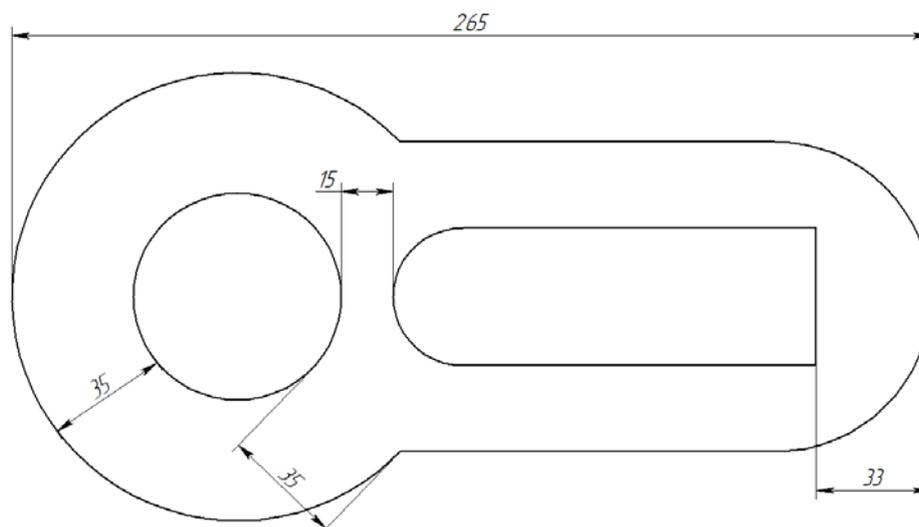


Example of a linear array: the diameter of the circle is obtained by measuring the length of the secant line, which measures the distance to the spline.

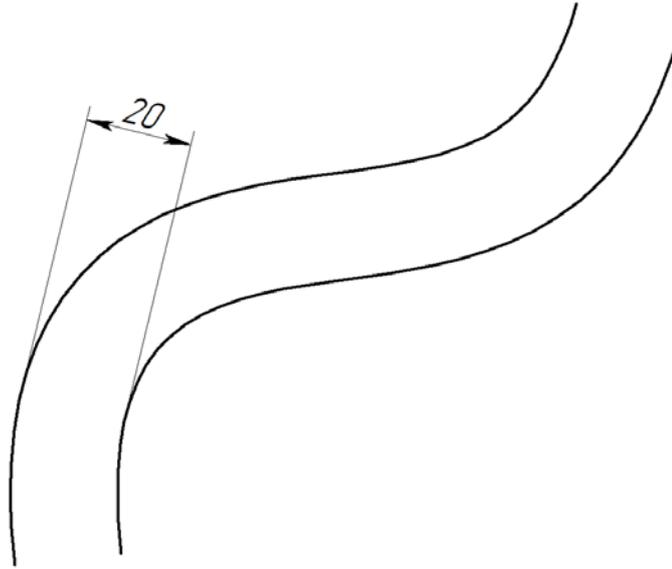


DIMENSIONS

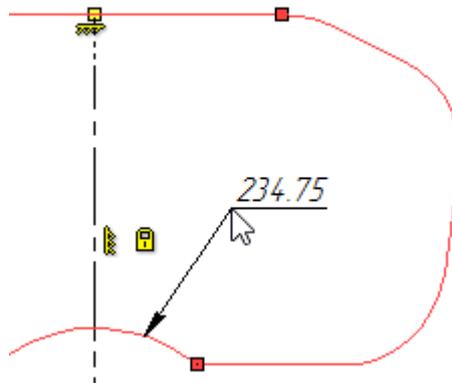
A possibility to create dimensions between circles and straight lines/arbitrary curves was added.



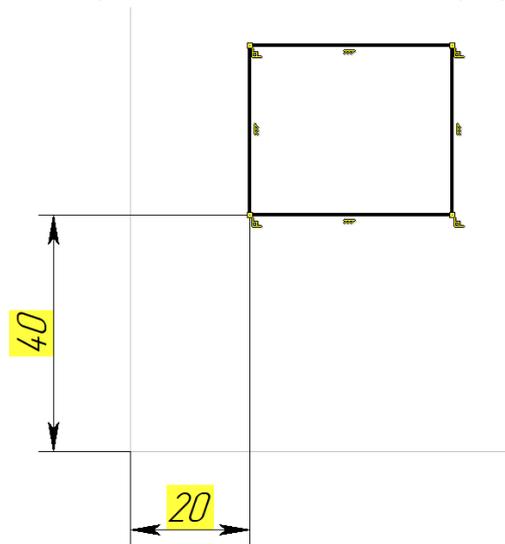
Added the ability to create dimension between equidistant curves



Now the leader dimension measures the length of the element on which it is created (graphic lines, sketch lines). You can select additional elements to create a leader dimension, after which the total length of all selected elements will be displayed.



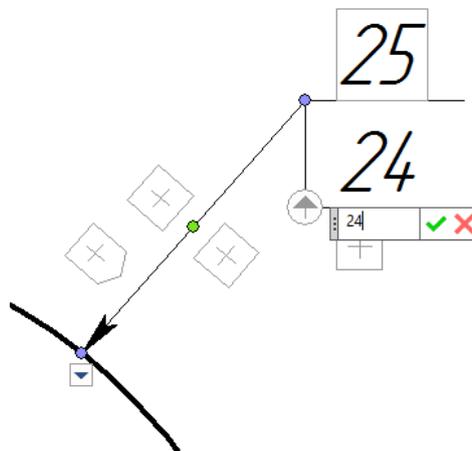
The possibility to bind the dimension to the zero point of the page was added. For example, this allows you to fix points that do not need to change their position when changing the rest of the drawing.



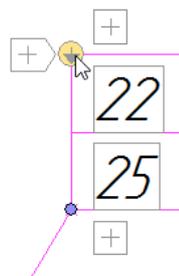
Now it is possible to create dimensions based on wire geometry (3D paths, 3D profiles, 3D axes)

LEADER NOTES

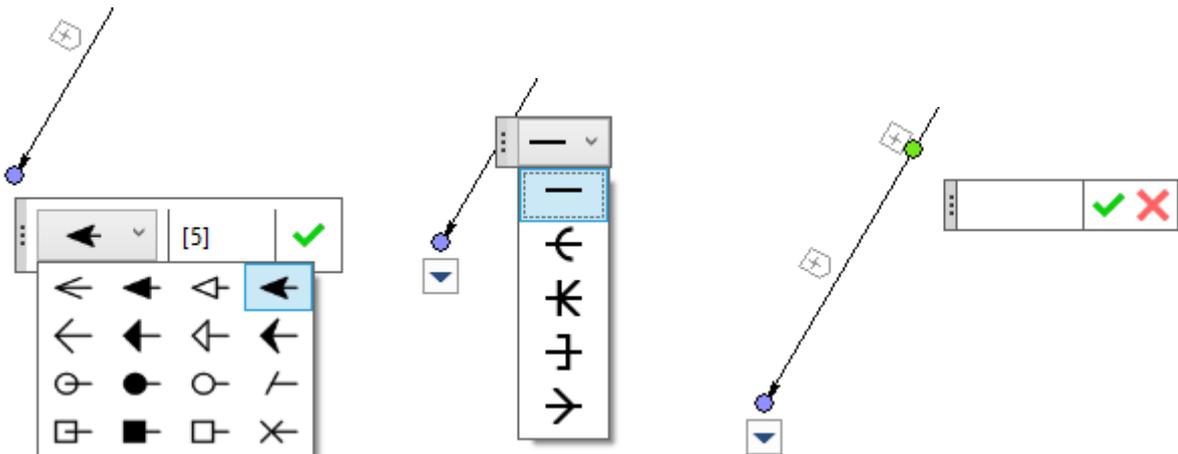
Control elements that simplify leader notes creation were added.



The control elements allow you to enter a name on the shelf of the leader note and create new jogs. A marker that allows you to change the direction of the jogs in leader notes/callouts with several jogs was added.



Markers for creating text on the jog line, selecting the arrows for the leader note and setting the symbol were added.



2D COORDINATE SYSTEM

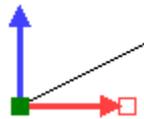
The possibility to create custom 2D local coordinate systems was added. LCS exists only for a particular page.

User can create drawings relative to his own LCS, which will allow the entire created drawing to be moved while moving the LCS.

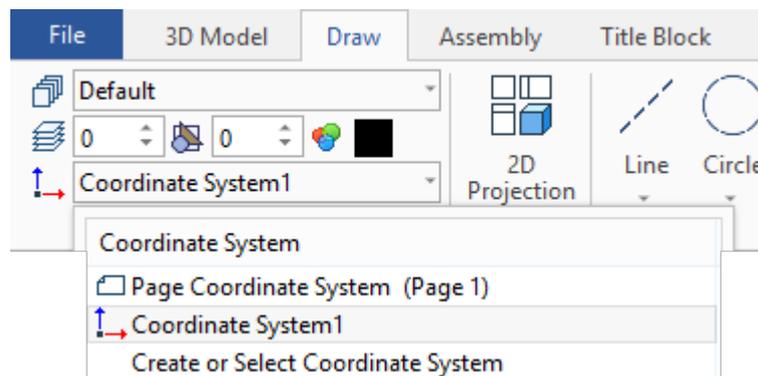
You can create a LCS using the **Create LCS**  command on the Ribbon.

You can select the active LCS, according to which all the elements will be created, from the special list.

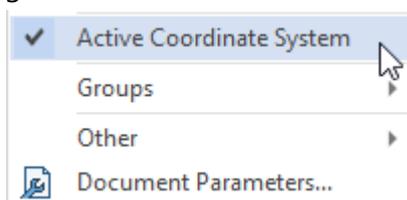
The active coordinate system is drawn by thick lines on the drawing.



The coordinates of all created and edited objects will be recalculated according to the active coordinate system.

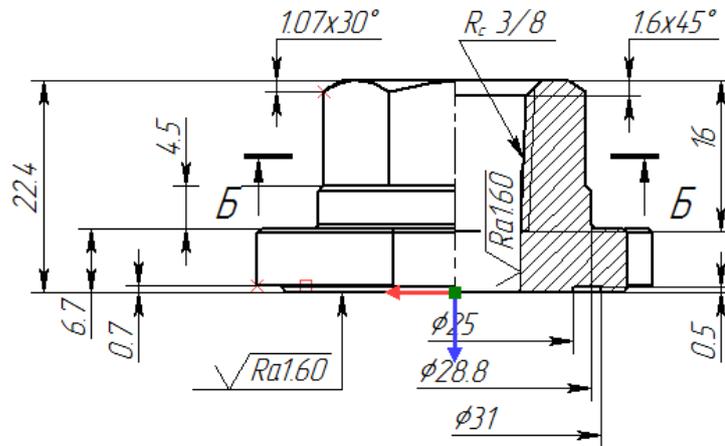


You can also select the corresponding item from the LCS context menu.

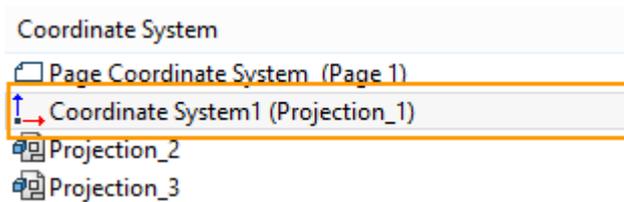


The coordinates of the 2D elements given relative to the coordinate system will be recalculated taking into account its position.

It is also convenient to bind 2D LCS to the projection. This allows you to create drawings directly on the projection.



In this case, drawings that are created manually will move along with the projection. To do this, select a projection in the drop-down list and the associated LCS will be created.

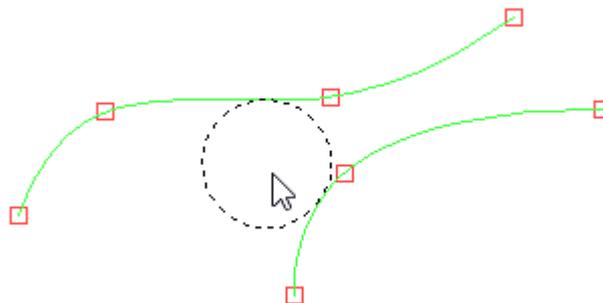


2D CONSTRUCTION ELEMENTS

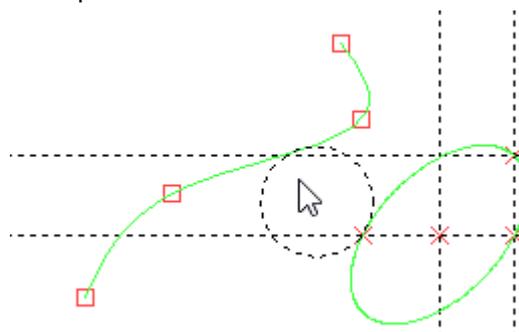
New Types of Tangent Circles

New types of tangent circles were added.

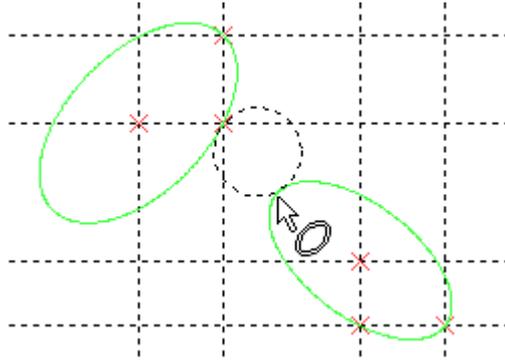
A circle tangent to two splines.



A circle tangent to a spline and an ellipse.



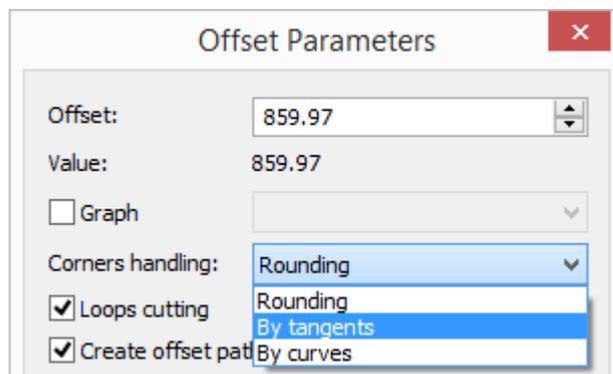
A circle tangent to two ellipses.



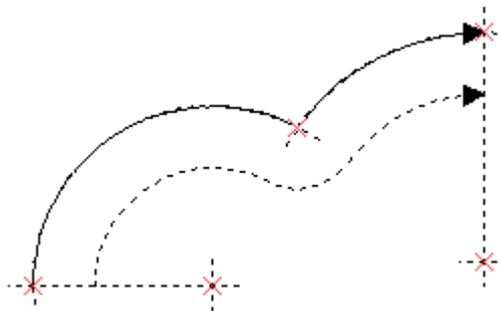
New Options for Creating Offset Curve

New options were added to the **TO: Create Offset Curve**.

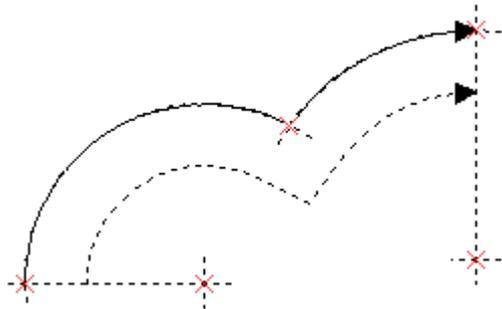
Corners handling option was added.



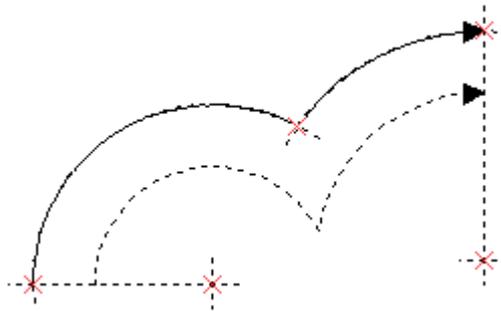
By default **Rounding** is created.



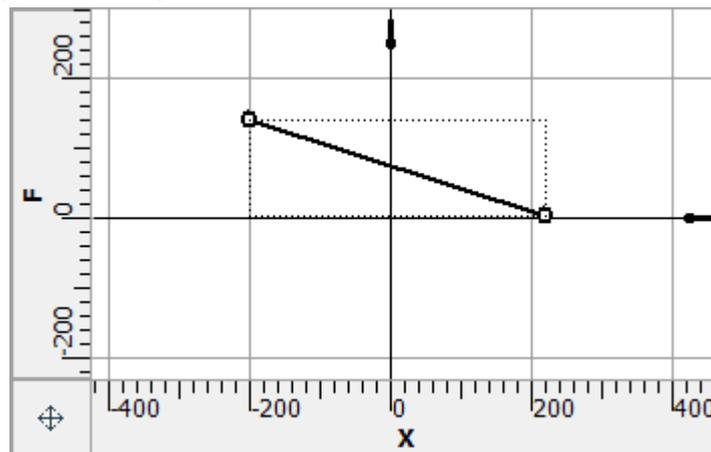
By tangents type was added.



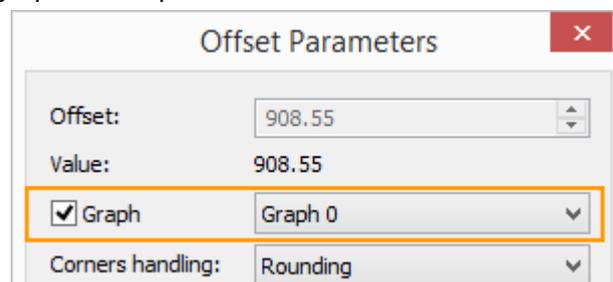
By curves type was added.



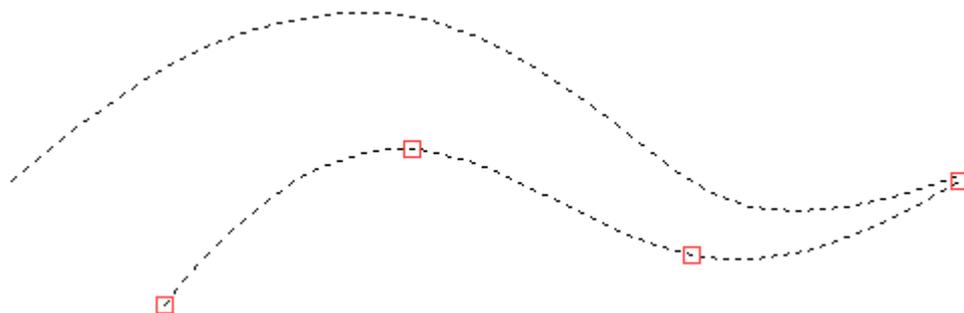
The offset curve can be constructed with a variable offset, which is specified by the graph. You should specify "Smooth curve" type for the graph.



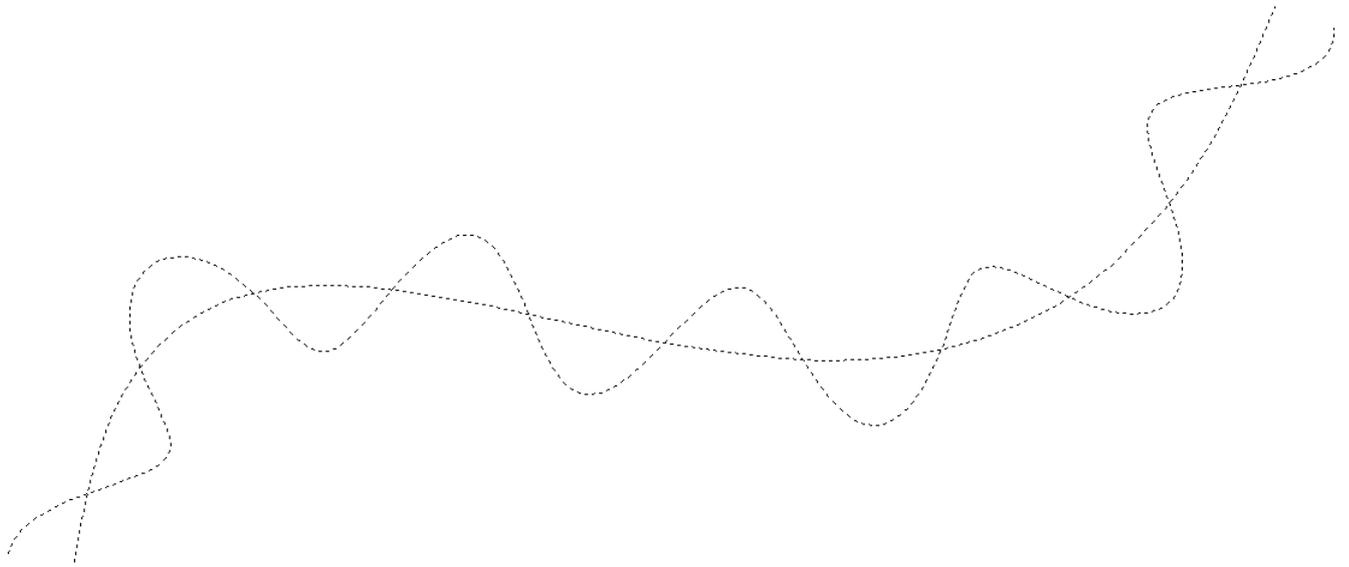
Then you should select the graph in the parameters of the offset curve.



The distance to the offset curve will be calculated according to the graph.



Example of creating an offset curve to a spline using the graph of a function constructed using the cosine formula:



Accuracy of Splines and Ellipses Calculation Increased.

Now calculation of splines and ellipses is more accurate. Previously, they were calculated using polyline and had an error.

Now the spline is created by points. The node on the spline will be located without error. This is seen with a strong zoom-in.

INCREASE AND DECREASE OF FONT SIZE

The commands **Enlarge Font Size** and **Reduce Font Size** have been added to the View toolbar. They change the default font size, as well as the default arrows size for the active page, according to a set of standard font sizes.



VARIABLE EDITOR

Mechanisms for working with variables have been improved, which greatly simplified the convenience of work and visual perception of information in the window.

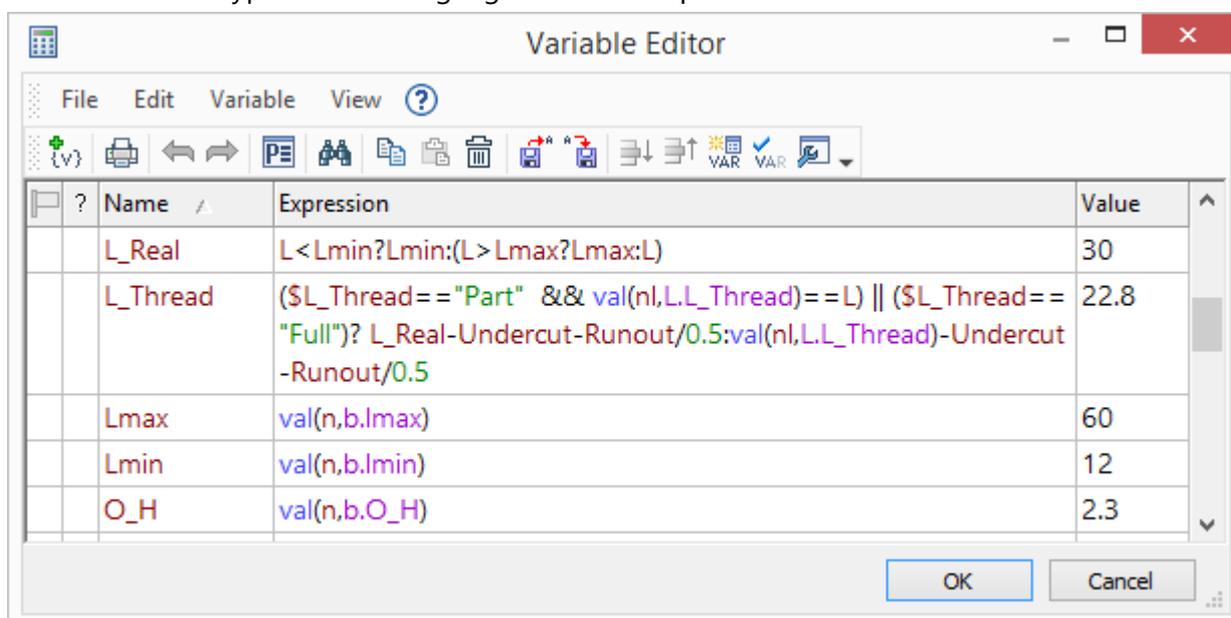
The syntax is highlighted in different colors, which saves users from confusion and allows you to better perceive information on the screen.

The variable editor displays more information. The user does not need to stretch the window to see the entire expression.

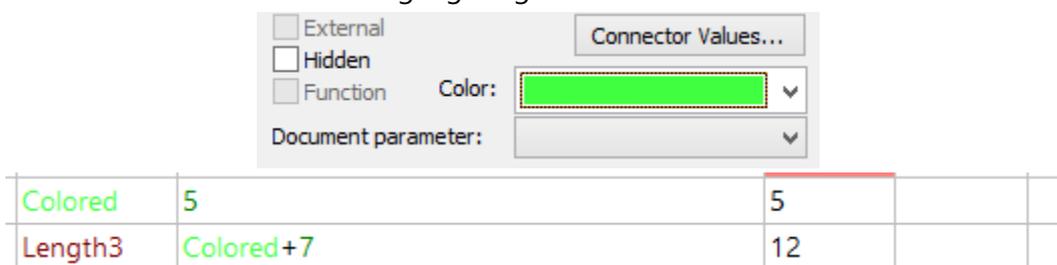
The following features were added to the editor: highlighting the use of a variable in other expressions, a drop-down list of variables and functions, tooltips and new functions.

INNOVATIONS IN VARIABLE EDITOR

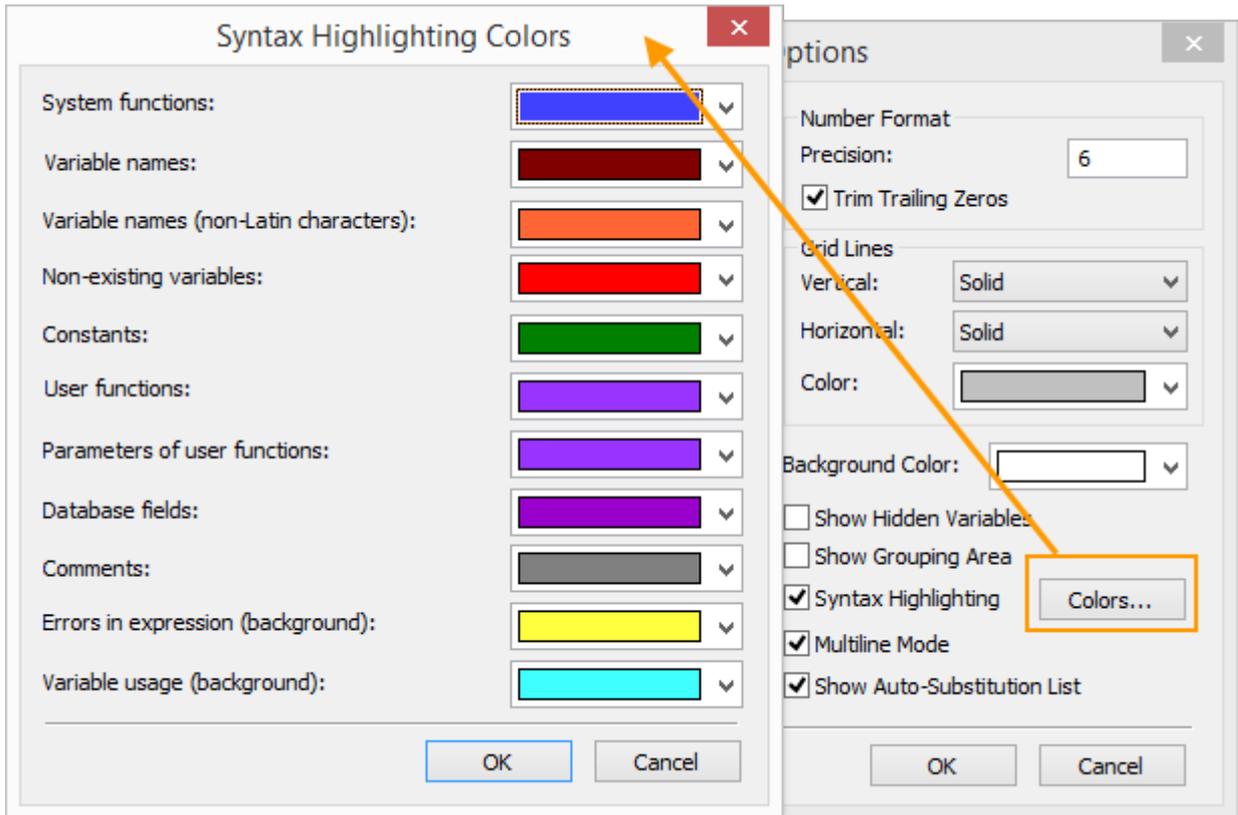
Elements of different types are now highlighted in the expressions in different colors.



All variables specified by the user are highlighted in the expressions in a separate color. The user can manually set the color for the variable's highlighting.



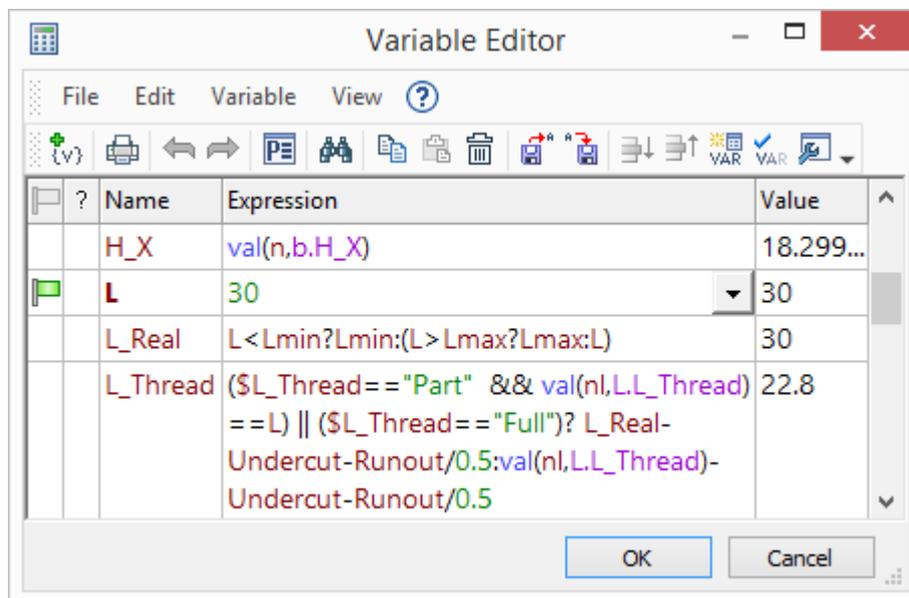
In addition, the user can change the colors for the syntax highlighting.



If there is an error in the expression syntax, the error expression will be highlighted.



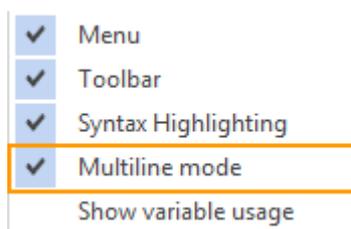
Added multi-line mode to the variable editor.



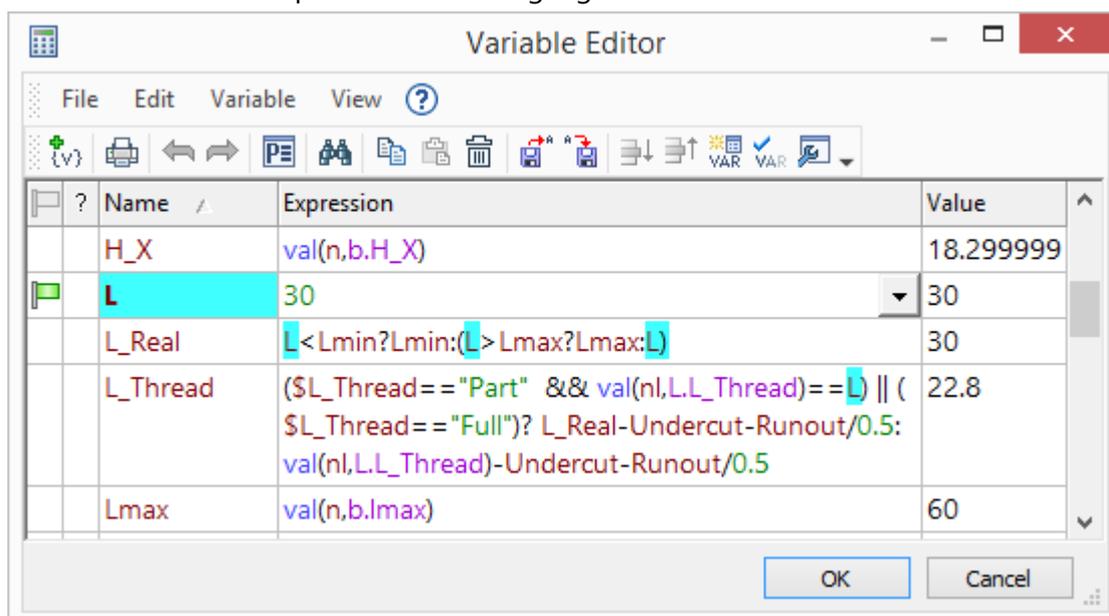
It is possible to add a line break with the combination of keys "Ctrl" + "Enter" to format the expressions in a convenient way in multi-line mode.

nl	dd==8?frec(18.L,L): (dd==10?frec(110.L,L): (dd==12?frec(112.L,L): (dd==16?frec(116.L,L): (dd==20?frec(120.L,L): (dd==24?frec(124.L,L): (dd==30?frec(130.L,L):0))))))
np	frec(P.d,dd)

You can enable multi-line mode in the context menu of the variable editor.



In addition, you can call the option **Show variable usage** in the context menu. When this option is enabled, all references to the expression will be highlighted.



Tooltips are implemented in the variable editor window.

When you hover the cursor over a variable, a comment is displayed.

nl	rec(L_Real==L.L&& dd==L.D)	88	
nn	frec(Metric_Thread.Pitch,Pitch)	11	
r	val(n,b.r)		
res	0		

Variable: Pitch
Comment: Specified Pitch
Value: 1.5

When you hover the cursor on the function, a brief description and an example of using the function appear.

Lmax	val(n,b,lmax)	60	
Lmin	val(n,b,lmin)		
Pitch	val(n,b,Pitch)	5	Specified Pitch
R_B_D	val(n,b,R_B_D)	3.861	

If the field belongs to a database, then the database and fields names as well as comments are displayed.

Lmin	val(n,b,lmin)	12	
Pitch	val(n,b,Pitch)	.5	
R_B_D	val(n,b,R_B_D)	.861	
R_D	val(n,b,R_D)	.8	
R_DEP	val(n,b,R_DEP)	5.64	

For text constants, a search for the model object by the appropriate name or identifier. If the text constant specifies the name of the object, then information about this object is displayed for it.

? Length	get("0x300001C","LENGTH")	134.65...	
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Element: 0x300001C
 Type: Graphic Line
 ID: 0x300001C
 Graphic Line 0x300001C
 Page: "Page 1" (0)
 Type: Graphic Line
 Layer: "Default" (0)
 Level: 0
 Priority: 0
 Color: "Black" (0)

Tooltips are active. When you click on a tooltip, you perform an action that is different for different types of data:

- ✓ **Variable name.** You are moved to the variable line;
- ✓ **Function name.** Help with a section containing the function description is displayed;
- ✓ **Access to the database field.** The Database Editor window appears;
- ✓ **An identifier or name of the model element.** The Info dialog for this item is displayed.

When you enter an expression, a drop-down list of suitable functions, variables, databases and database fields appears. This list contains descriptions and comments for the list items.

? Var1	D	0	
--------	---	---	--

Name	Value	Comment
fx distance(,)		Returns the distance between elements
fx dbfwin(,)		Returns the value from DBF-file
fx dbf(,,)		Returns the value from DBF-file
fx db_sum()		Returns the sum of database column
fx db_min()		Returns the minimum value of database column
fx db_mid()		Returns the average of database column
fx db_max()		Returns the maximum value of database column

To call a list of all available variables and functions, use the combination <Ctrl> + <Space>.

When user selects a function from the drop-down list, the parentheses and delimiters necessary for the selected expression are automatically inserted.

Now users can create variables whose names are the same as the function names.

NEW FUNCTIONS

The **select()** function is used to select a value using several conditions.

The terms and results that will be specified when the condition is met are listed through a comma.

Select (Condition1, Result1, Condition2, Result2, ..., Result by default)

The value of the variable will be the result of the first condition met, or the default result.

Numb1	1	1
Numb2	20	20
Select1	select(Numb1 == 3, 100, Numb2 < 10, 200, 300)	300

In case that none of the above options are appropriate, and the default value is not set, the value of the variable will be zero.

The **switch()** function is used to select a value by one condition.

A result whose value corresponds to the condition will be set as a variable value.

Switch (Condition, Value 1, Result 1, Value 2, Result 2, ..., Result by default)

Numb3	2	2
Switch	switch(Numb3, 1, 10, 2, 20, 3, 30, 40)	20

The **root()** function allows you to calculate the root of a given degree for a number.

RootFind	root(27,3)	3
----------	------------	---

The **reverse()** function allows you to reflect a string or a number in reverse order.

You can directly put any number or string in the reverse function.

\$Reverse	reverse("abc")	cba
-----------	----------------	-----

You can also specify a variable whose value will be reversed.

\$Reverse	reverse(\$Word)	txet weN
\$Word	"New text"	New text

It is important that the type of the variable corresponds to the type of value.

The **random() function**. Returns a random number in the specified range.

RandomValue	random(0,14)	5.889767
round	round(RandomValue, 1)	6

The **angle()** function generates a decimal angle value from *degrees, minutes, seconds* or *from degrees and minutes*. Function takes three or two parameters.

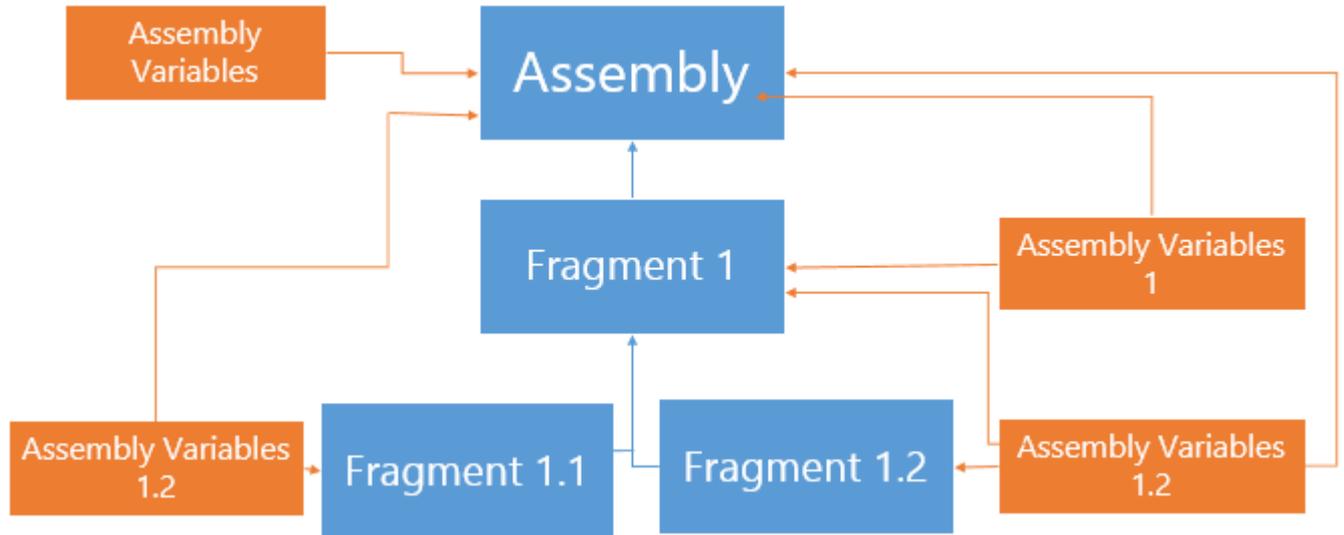
Hyperbolic functions **sinh, cosh, tanh, asinh, acosh, atanh** were added.

Functiong(). Returns the acceleration of gravity constant.

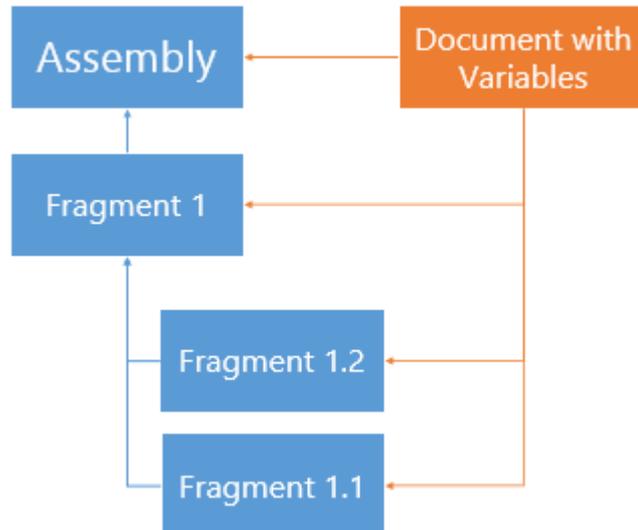
Functione(). Returns E constant.

ADOPTING VARIABLES AND DATABASES

Managing the variables of large assemblies has become easier. In previous versions of T-FLEX CAD user performed several actions to copy the fragment variable to the assembly with maintaining of the link. To change this variable in the assembly and in the fragment, the user had to change the variable in the particular fragment file.

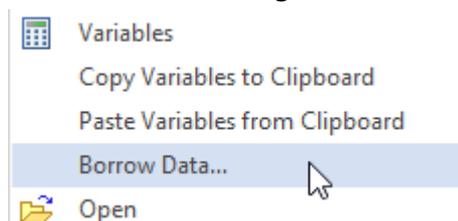


Now you can create a document in which all the variables and databases will be stored. This document should be added to all documents in which variables and databases will be used.

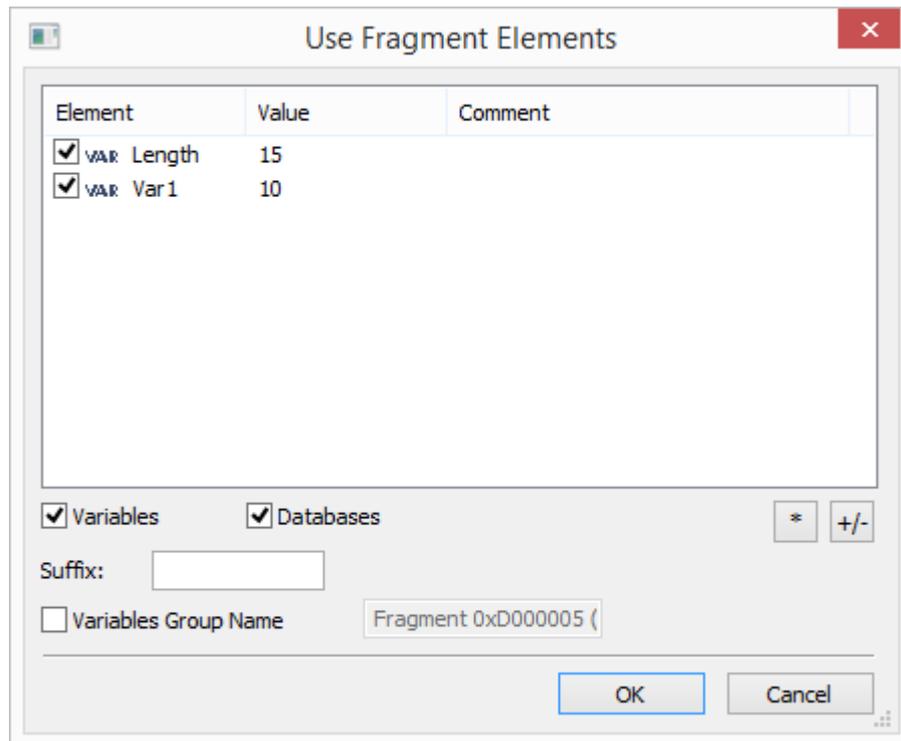


The user can then choose which variables to borrow for each file. These variables can be used as normal variable fragments to create geometry. In this case, you will need to change the variables in one single file.

A special command is used to borrow data from the fragment.



You can specify which variables and databases will be used in the assembly in this window.



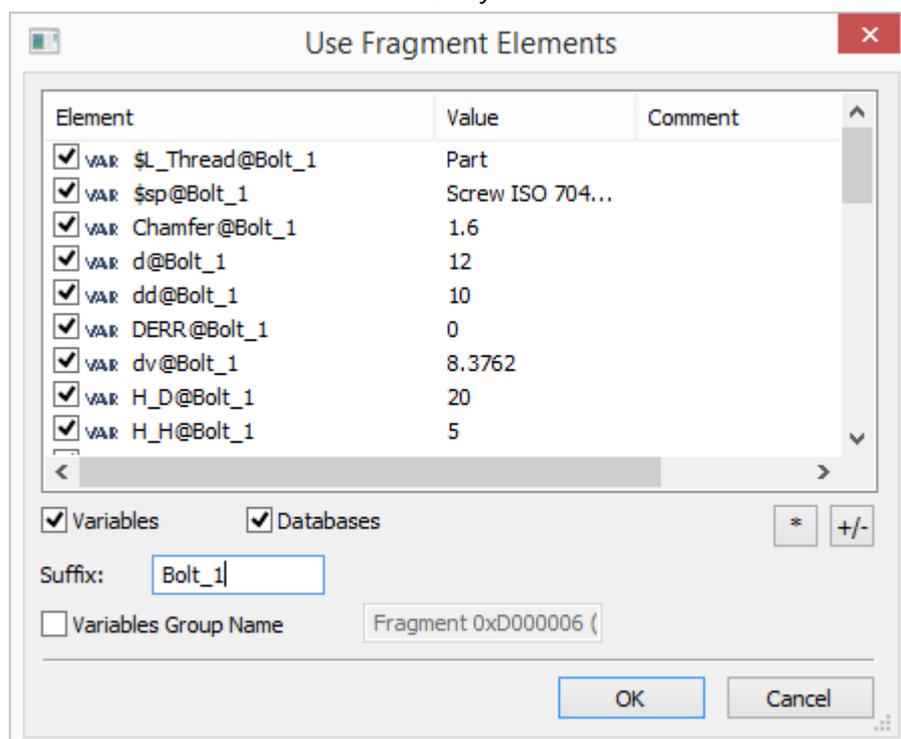
All selected variables will be added to the variable editor.

[-] Group:

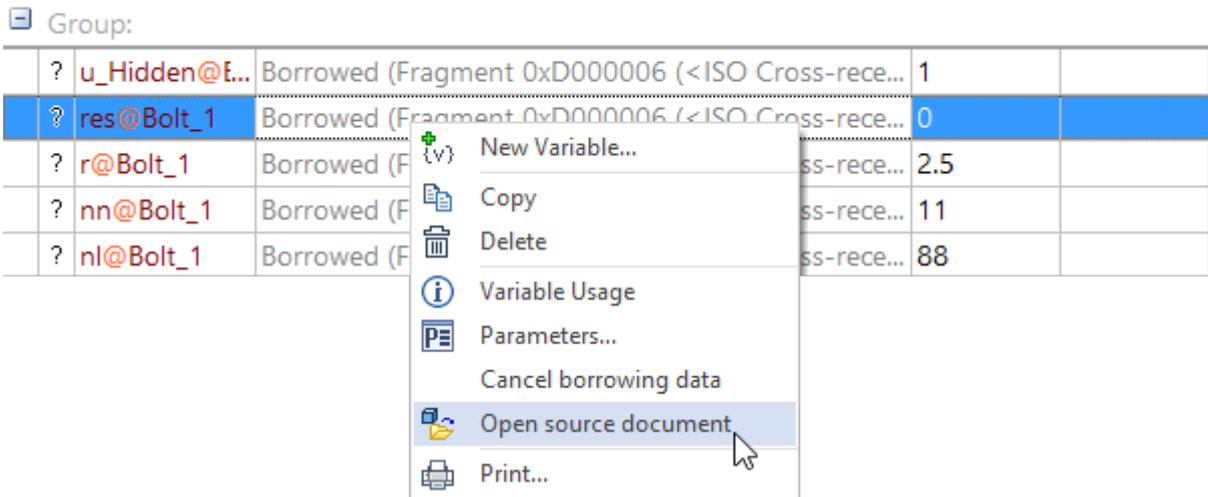
? Var1	Borrowed (Fragment 0xD000005 (ISO Drawing ...	10
? Length	Borrowed (Fragment 0xD000005 (ISO Drawing ...	15

Variables can be borrowed from the same fragment with different values of external variables. For example, if the same bolt fragment is inserted in the assembly with different diameters and lengths.

For such borrowing, you need to specify a suffix when borrowing variables from a fragment. This suffix will be added to all borrowed variables after the "@" symbol.



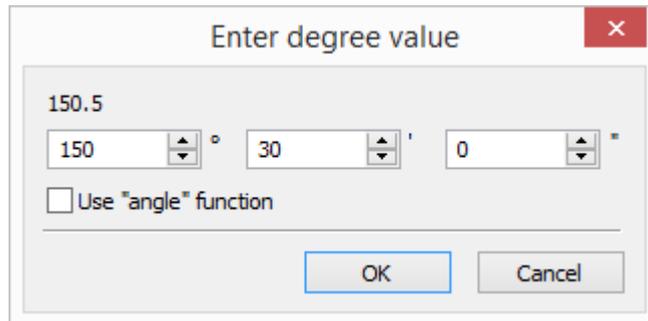
Using the **Open source document** option you can open a fragment from which the variable was borrowed.



If you call the **Cancel borrowing data** option, you can break the link of the variable with the source document. In this case, the variable will become a normal assembly variable.

USING VARIABLES IN ENTER ANGLE DIALOG

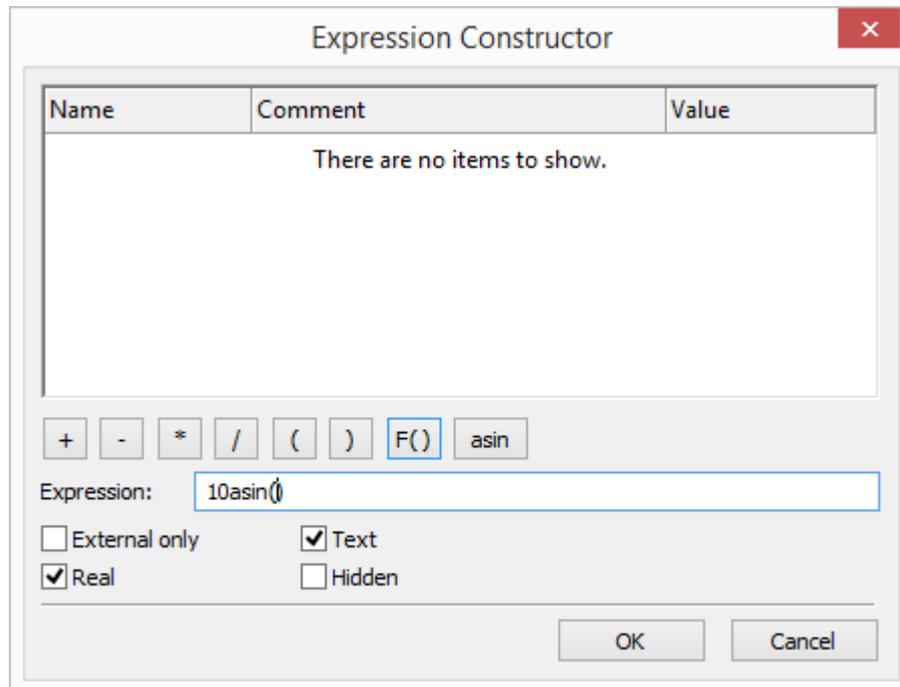
The ability to use variables was added in the dialog for entering the angle for the value fields.



If the **use "angle" function** is set, the entered values are not used as a constant, but are converted to a function call. In the example above, the resulting string is "angle (A1, A2)".

INSERT VARIABLE

The **Insert variable** dialog is converted to the **Expression Constructor**, which allows you to enter an expression using the buttons.



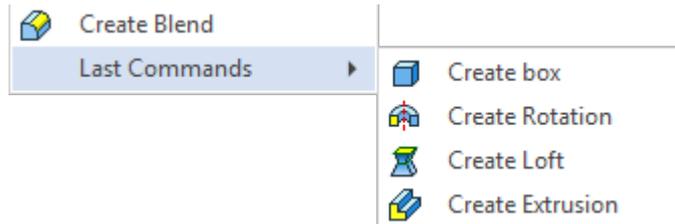
The function selection button shows a full list of functions. The selected functions will be added to the dialog.

abs	distance	log10	setg	warn
acos	e	ltot	setv	
acosh	error	max	sin	
angle	find	mdb	sinh	
asin	fixnodename	measure	sltot	
asinh	floor	message	sqrt	
atan	frec	min	strlen	
atanh	ftoa	racos	switch	
atof	ftoa2	random	tan	
atot	ftot	rasin	tanh	
ceil	g	ratan	tfind	
check	get	rcos	tget	
cos	get_lcs	rec	tgetg	
cosh	getg	reverse	tgetv	
db_max	getv	root	tmgetv	
db_mid	graph	round	tpart	
db_min	hypot	rsin	treplace	
db_sum	int	rtan	tsetg	
dbf	isfragment	satot	tword	
dbfwin	log	select	val	

IMPROVEMENTS IN INTERFACE AND USABILITY

REPEAT LAST COMMAND

The possibility to re-execute one of the previously executed commands was added. The corresponding items were added to the context menu, which is called in the working window of the system. Some of the last commands are displayed in the list itself. Some of the commands are displayed in the submenu.



The number of commands that are added to the list to perform a re-execute call is configured in the **Options** dialog on the **Additional options** tab. The number of commands directly in the context menu and in the submenu is configured separately.

Additional Options	
Right mouse button action	Cancel in command and menu without command ▾
Invert mouse wheel direction on zoom	<input type="checkbox"/>
Enable dynamic delete	<input type="checkbox"/>
Number of commands in the Last Commands list	1
Number of items in Last Commands submenu	5

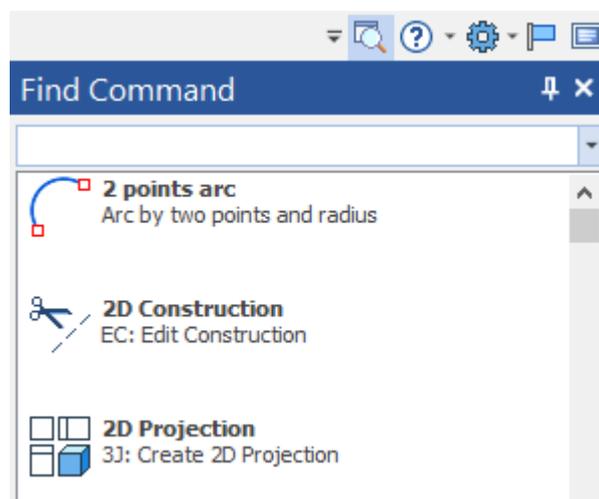
The list of commands available for re-execution is maintained separately for 3D modeling, drawing and drawing on workplanes.

You can repeat the previous command by pressing <Space> in the command standby mode.

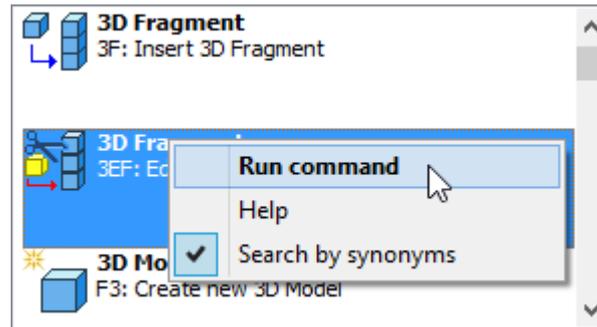
FIND COMMAND

A separate window for commands searching was added to facilitate the search for commands.

To call the window, you can use the corresponding option in the upper right corner of the main system window. In the search field you can enter not only the name, but also a description and synonyms for the command name.

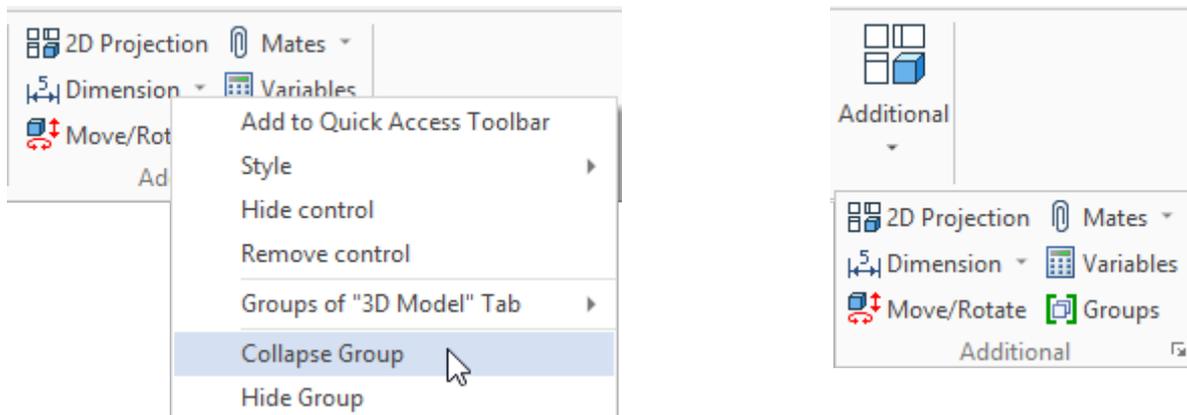


Through the context menu, you can get help on the command and activate it.

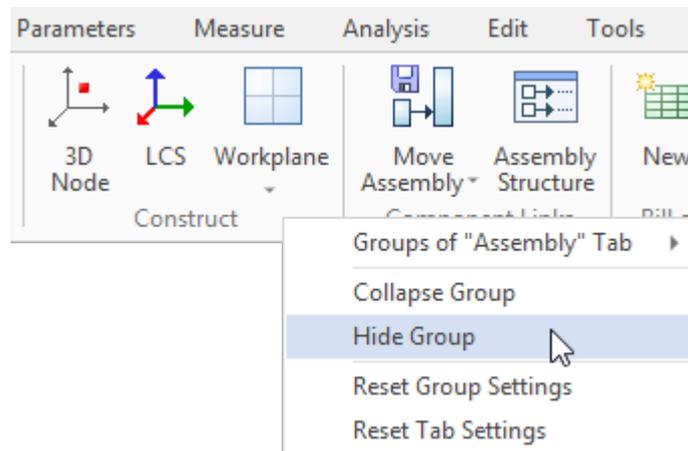


RIBBON

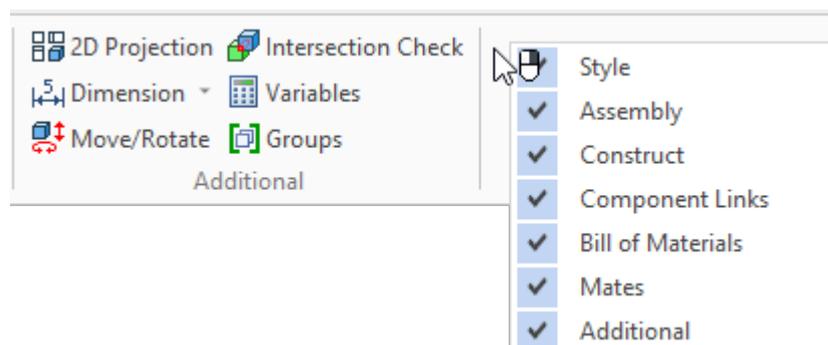
The possibility to collapse/restore groups of commands was added.



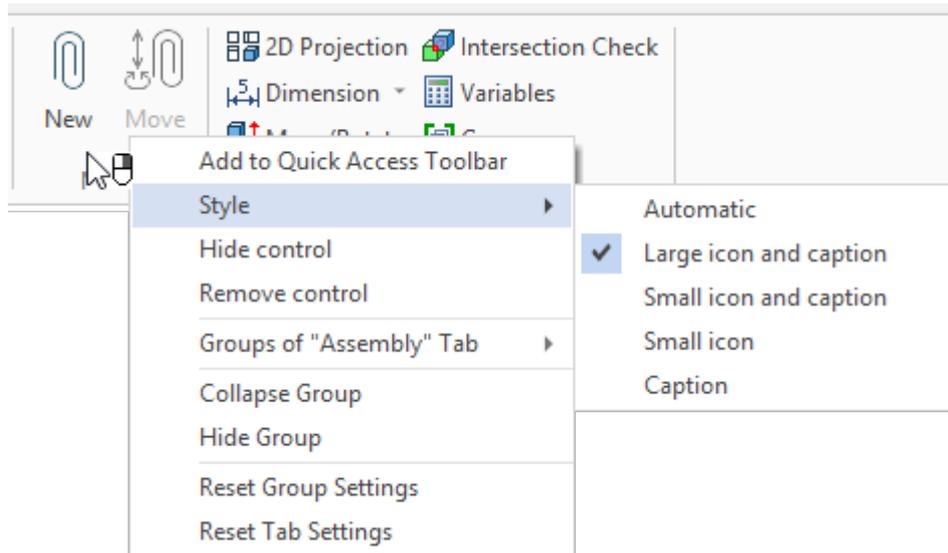
User can hide groups that he never use in the Ribbon.



Hidden groups can be restored from the context menu, which is called when clicking into an empty space of the Ribbon.

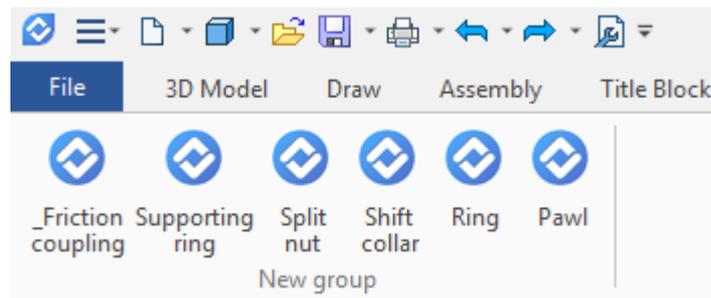


Now it is possible to call context menu where user can specify style of any button on the Ribbon or hide it without opening **Customize** dialog.



The possibility to add fragments from the library to the Ribbon was added in the command **SB: Parameters**. To do this, select the **Insert fragment from library** category.

This innovation allows you to create your own Ribbon tabs with standard elements or add them to the existing tabs.



In addition, the category **Create documents from prototypes** was added.

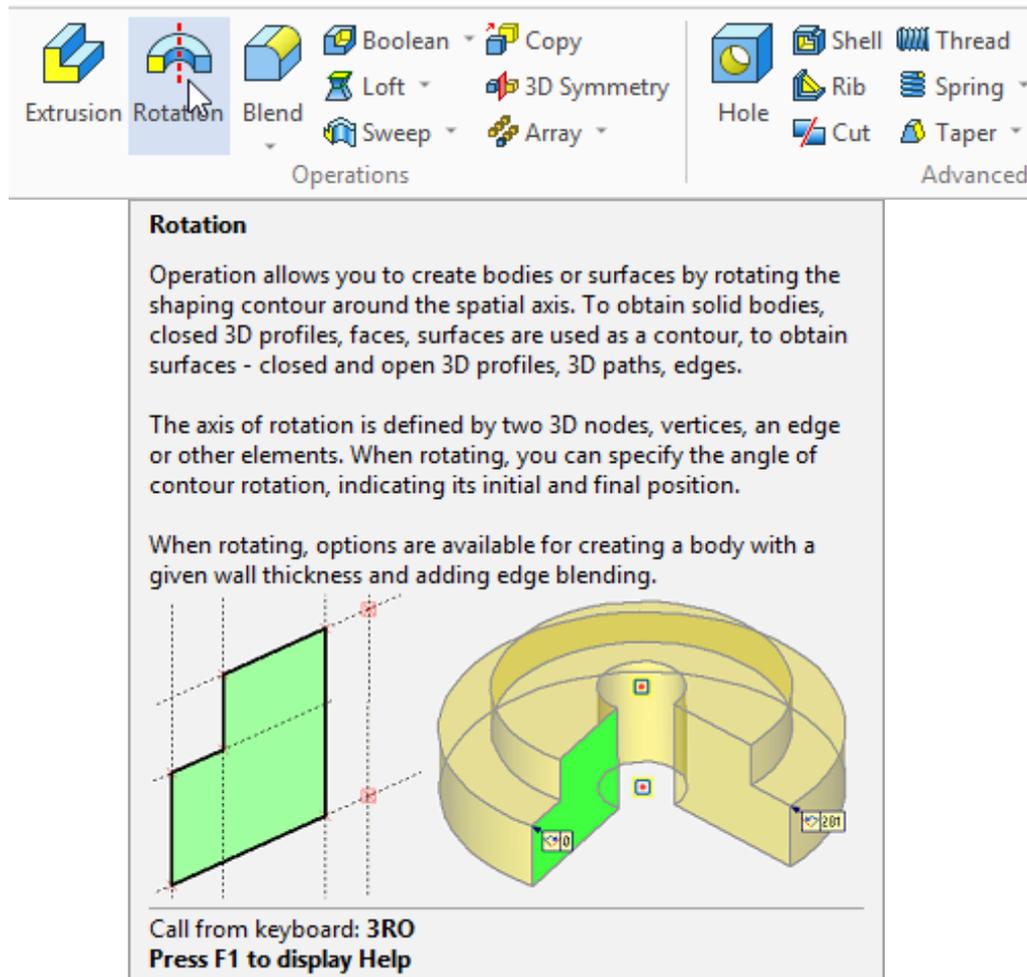
CONFIRM CHANGES WITH DOUBLE CLICK

Now user can finish input by double-clicking the mouse. After setting or changing of the parameters, it is enough to double click on any free place of the working window, and the action will be confirmed.

For example, it is convenient to use this feature when creating hatchings, splines, extrusions.

EXTENDED TOOLTIPS FOR COMMANDS

Extended tooltips for the commands were added. They work for both the Ribbon and the text interface. When you hover the cursor, a brief tooltip for the command appears. If you hold the cursor down for two seconds, an extended tooltip with an image appears.



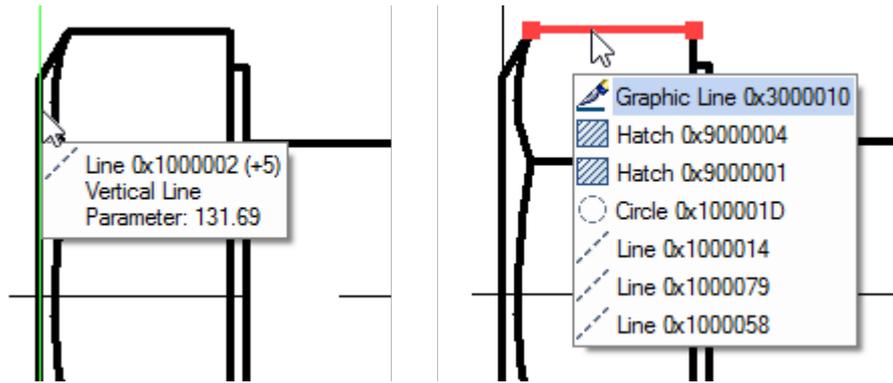
If necessary, you can disable extended tooltips in the **Customize** dialog.

- Show ScreenTips on toolbars
- Show Extended ScreenTips
- Show shortcut keys in ScreenTips
- Show Open API names of commands

EXTENDED TOOLTIPS FOR ELEMENTS

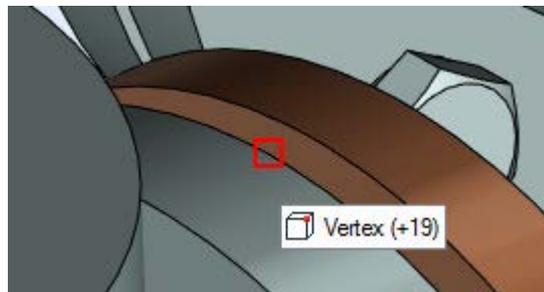
When hovering over elements (construction lines, graphic lines, nodes), extended tooltips appear. In addition to information about the name and identifier of the element, they contain additional information about the type and method of the object creation.

In addition, these tips contain icons for objects.

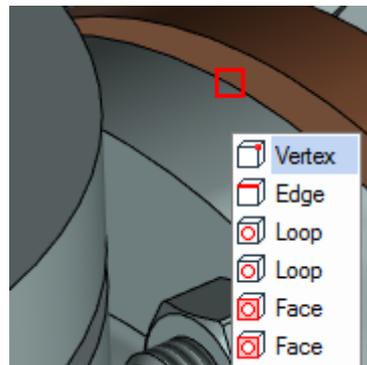


ELEMENTS UNDER CURSOR SELECTION

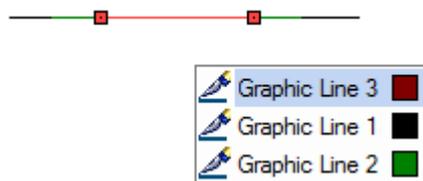
The mechanism for elements selection under the cursor was improved. In the case when there is more than a single element available for the selection under, the number of the available elements is shown. It is displayed with the "+" symbol.



A list of elements will be shown after a small interval if the cursor is not moving. You can select a required element by clicking on the corresponding item or using a mouse wheel, as in the previous versions of the system.



If elements under cursor have different colors, the colors are also displayed in the tooltip.



SCALING USING MOUSE WHEEL

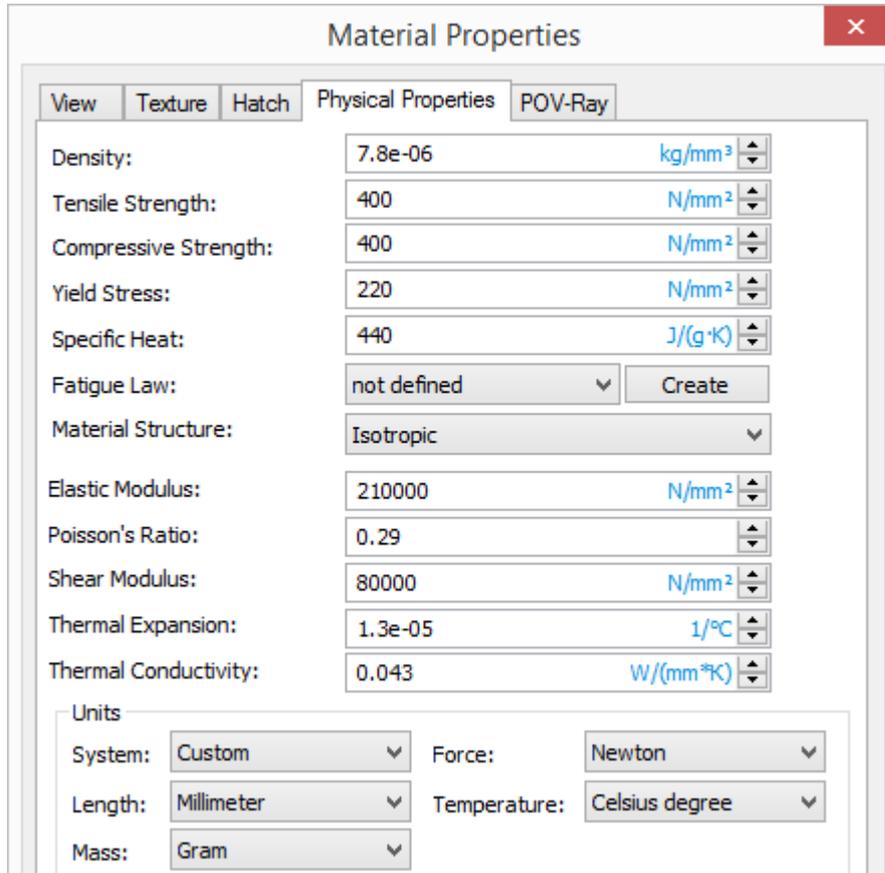
Improved control of the image scale using mouse wheel. When the <Ctrl> key is pressed, scaling is performed with an increased factor.

In the **Options** command the possibility to set the value of the scale when rotating the mouse wheel was added.

OTHER IMPROVEMENTS

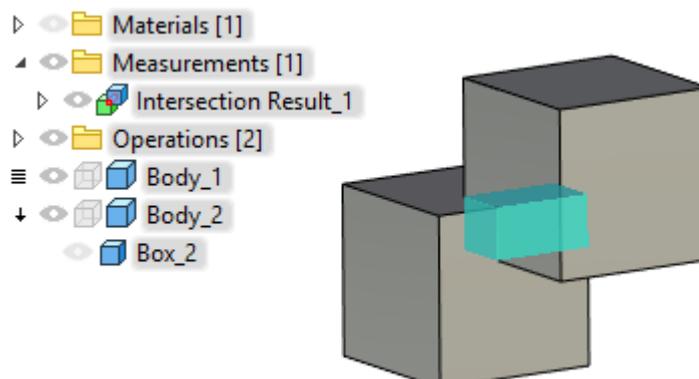
MATERIALS

The values of the physical properties of materials are now stored together with the units of measure. You can change the unit by clicking on it with the left mouse button. After changing the units of measurement, recalculation takes place.



MEASURE

The option **Create result** was added to the command **Intersection check**. It allows you to save the measurement result as a separate object of the model that is displayed in the model tree as well as in the scene. Upon model regeneration the element changes its display and indicates intersections between the specified model elements.

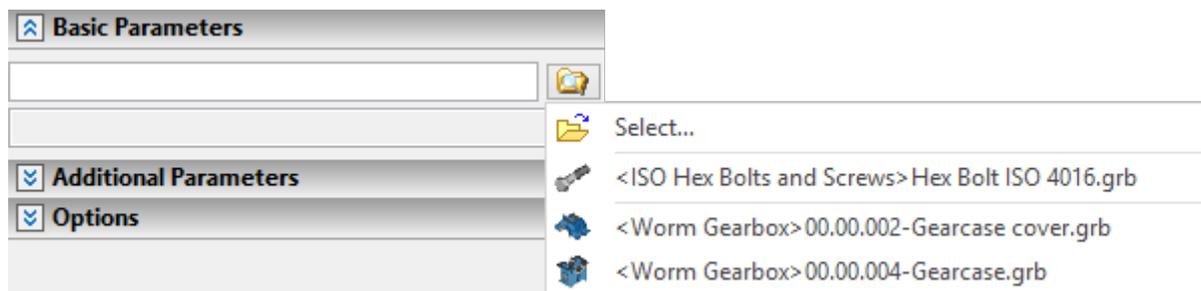


The possibility to create a curvature measurement result for 3D objects (3D paths, edges) and for 2D objects (construction lines, graphic lines, 2D paths).

The curvature data can be used for variables. You can get the data in the **Measure** command by selecting the created curvature measurement object.

INSERTING FRAGMENT, PICTURES, 3D IMAGES

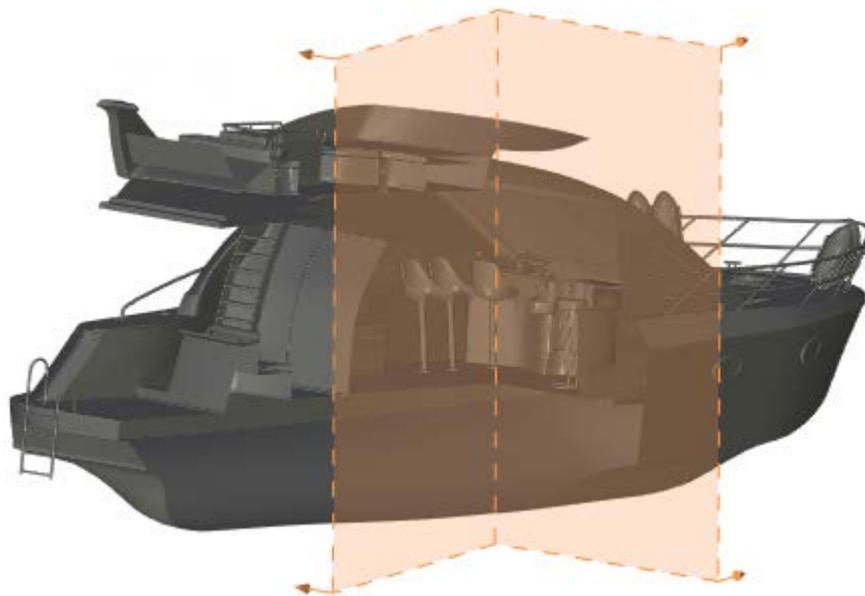
When you insert a fragment, picture or 3D image, you can see previously opened or pasted files in the special list.



The same list is available in the command auto-menu for the **Select file** option.

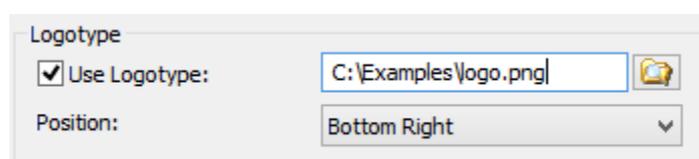
FACETED OBJECTS

Sections and clip planes can now be applied to faceted objects.



USE LOGO WHEN GENERATING VIDEOS

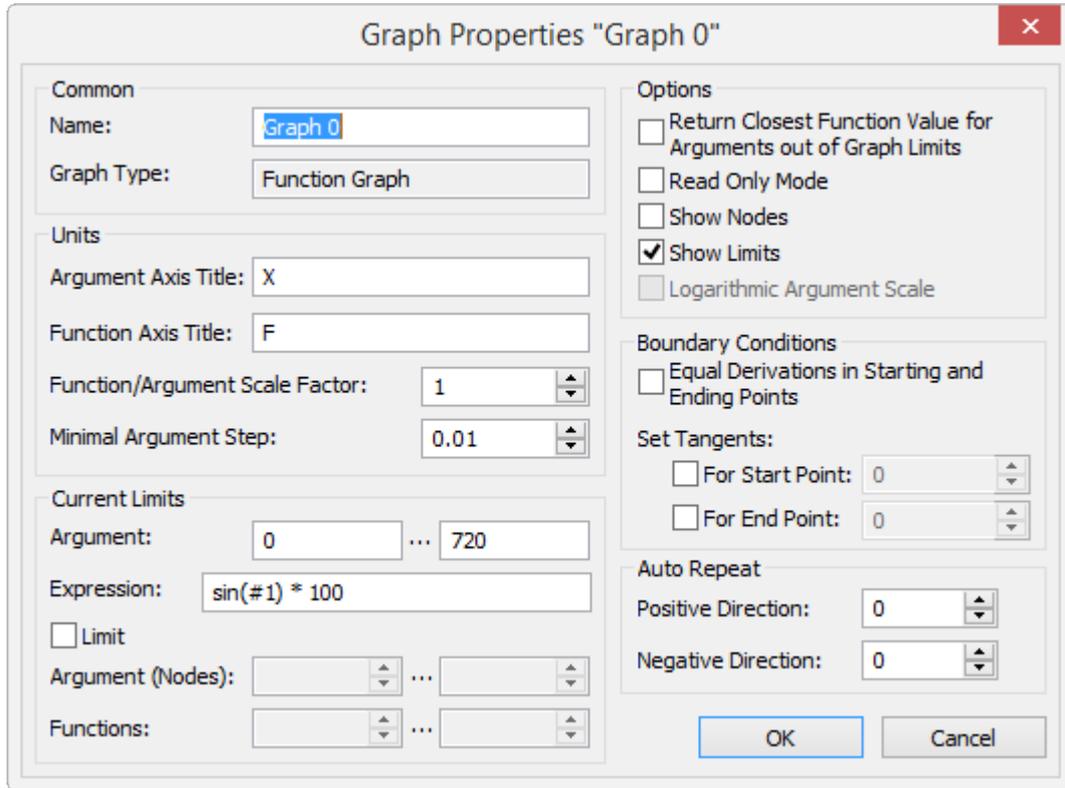
The possibility to select a raster image in the dialogs of the video recording in the **Animation** and **Record Scenario** commands was added. This image will be used as a logo for video generation.



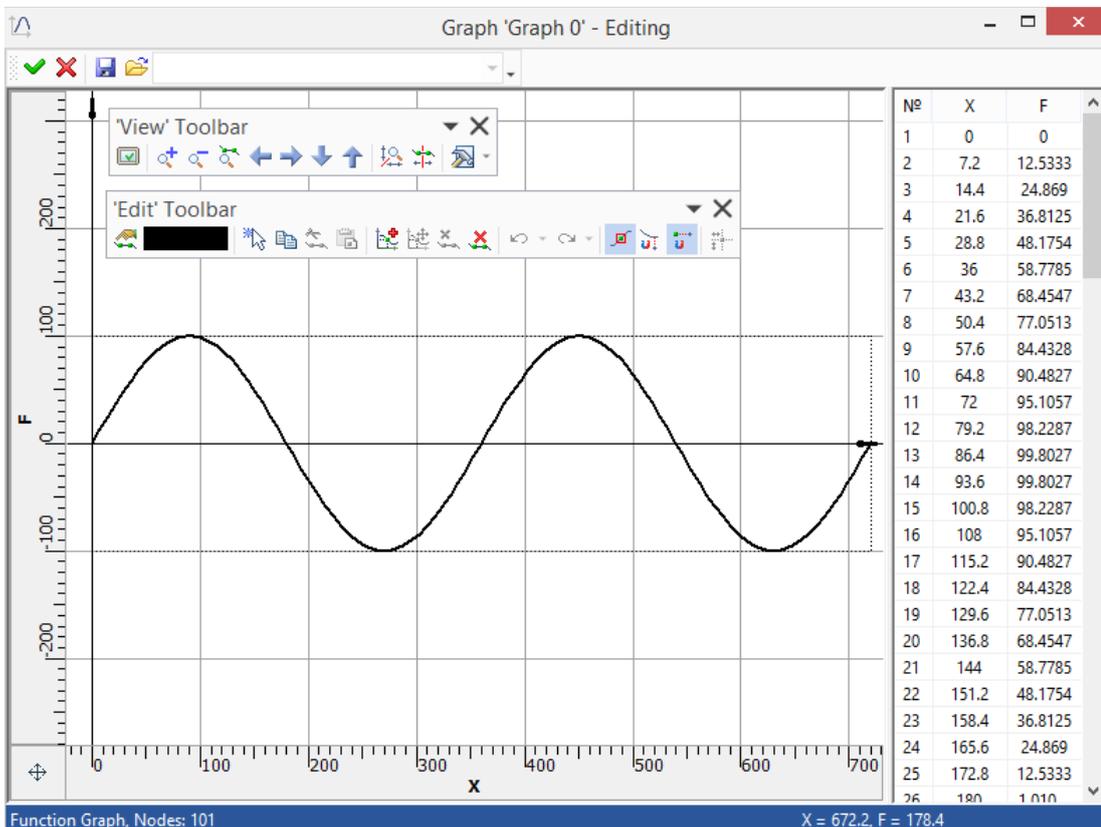
You can set the position of the logo in the dialog.

GRAPHS

It is not possible to create graphs controlled by a formula in editor of graphs.



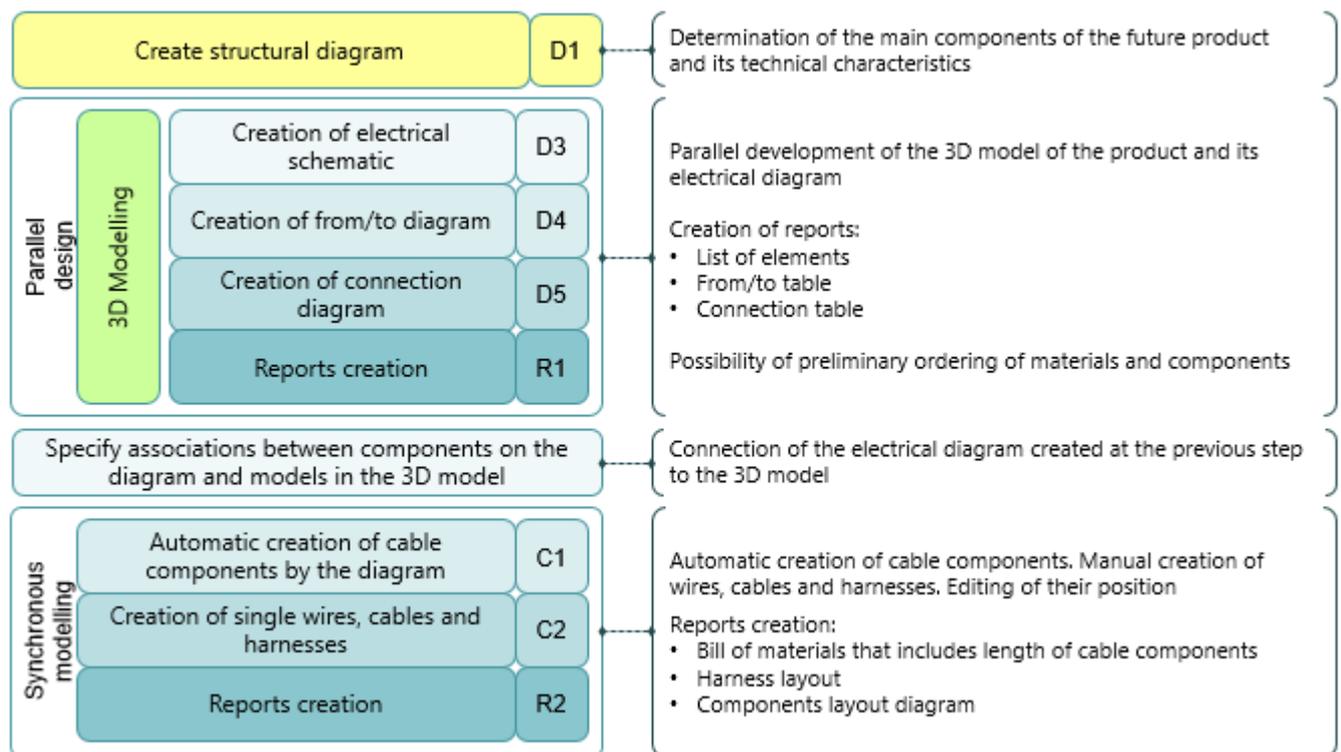
Formula specifies the dependence of the value of the graph function on the argument. At the same time, it is possible to specify a connection with other model variables. Such graphs are convenient to use, for example, to specify non-linear characteristics of materials, when creating displacement curves, etc.



T-FLEX ELECTRICAL

T-FLEX Electrical is a software product of the line T-FLEX PLM, which allows solving a full range of tasks in the design of electrical products. T-FLEX Electrical is a module of type "Add-On" built into the base interface T-FLEX CAD, along with other applications.

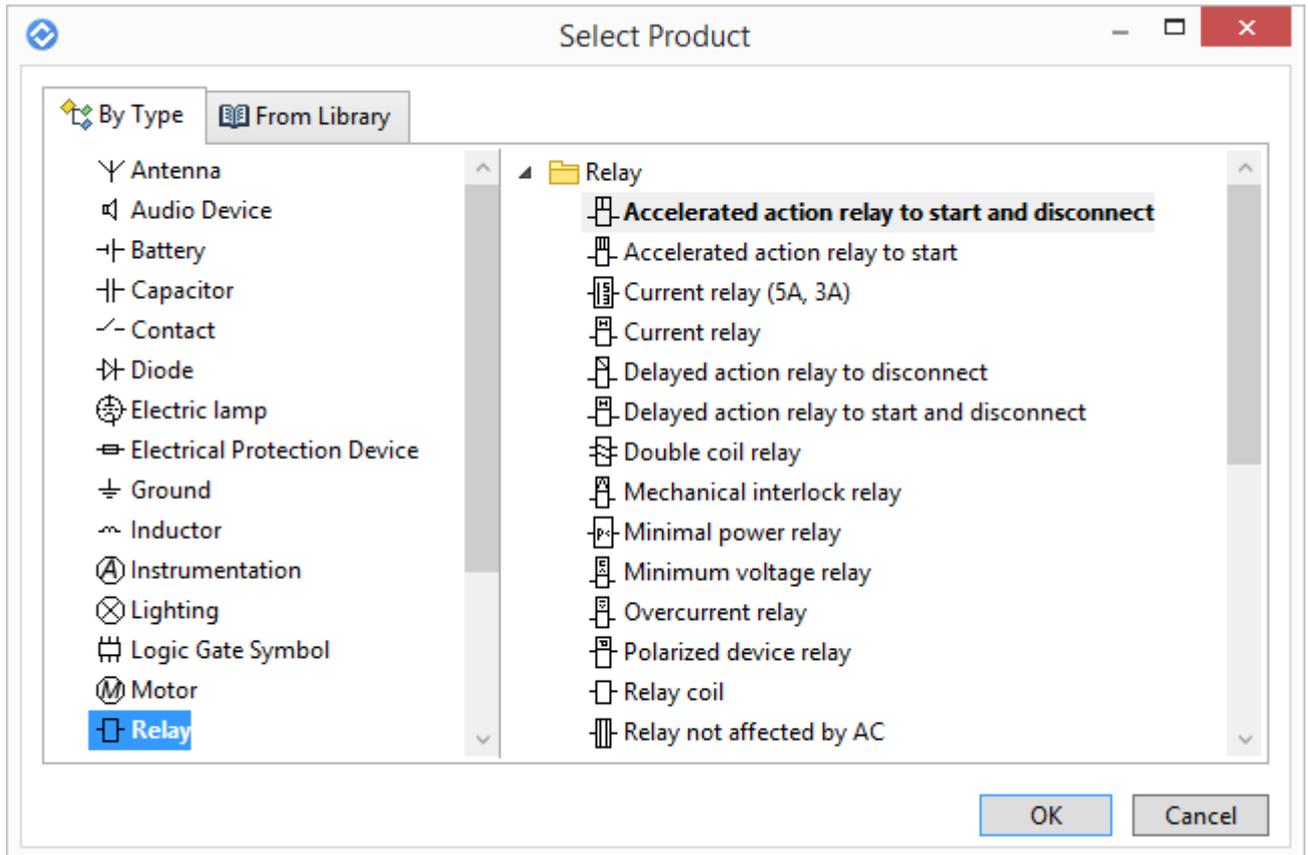
Thus, T-FLEX Electrical provides an opportunity for collective work of users within the same platform and a single information base in the tasks of designing complex electrical components. Designing of electrical diagrams and development of three-dimensional models can be carried out in parallel within single engineering project. Components on the diagram and 3D models of the components can be associated.



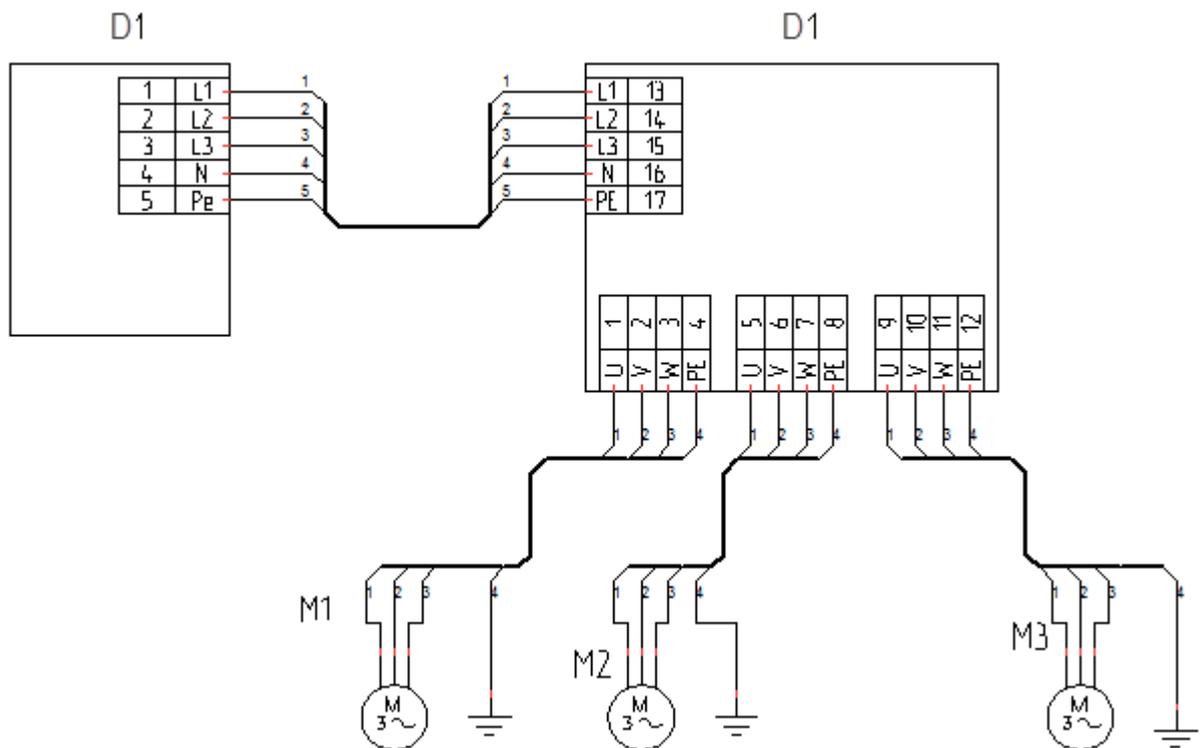
The application allows you to create any type of diagram: electrical schematic, connection diagram, "from/to" diagram, etc.

T-FLEX Electrical provides extensive opportunities for collaboration of several users on single project. Using the structural diagram one of the users works on the design of the electrical schematic, as well as the other diagrams, while the other user creates preliminary layout in the 3D assembly, and also develops the accompanying documentation (3D Modeling).

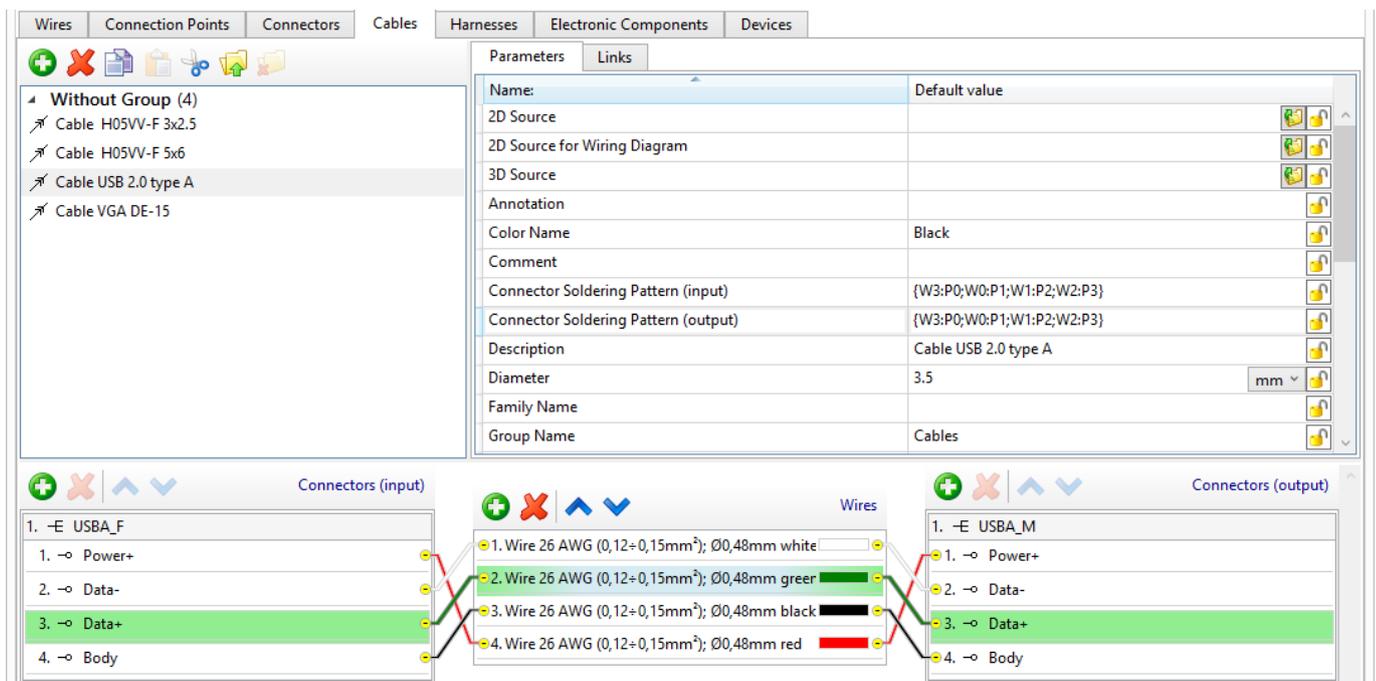
The distribution includes an extensive library of standard components. In addition, the application has tools and methods for creating your own components (connectors, devices, relays, etc.), connection points and cable components of any type.



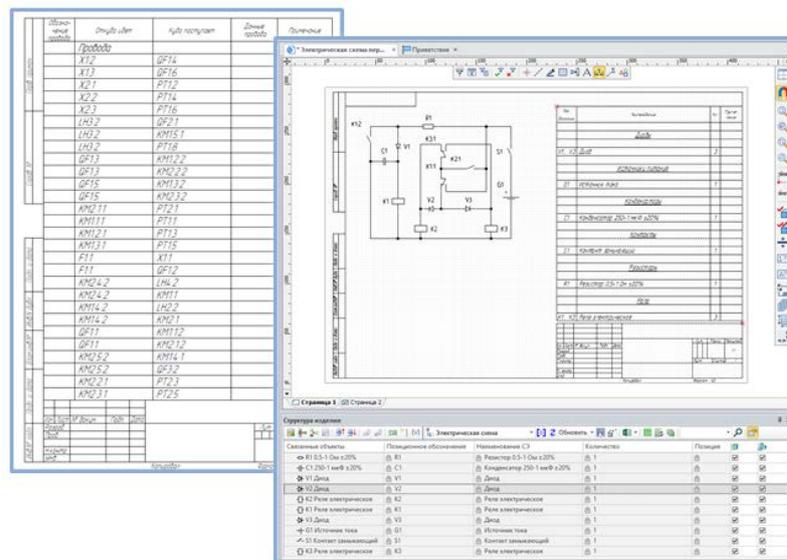
Information about all components that were added to the diagram is stored inside the diagram, so you do not need to provide them separately.



Each of the components has not only schematic symbol, but also a set of characteristics. Component characteristics can be changed manually right in the diagram. You can easily find real components based on the specified characteristics.



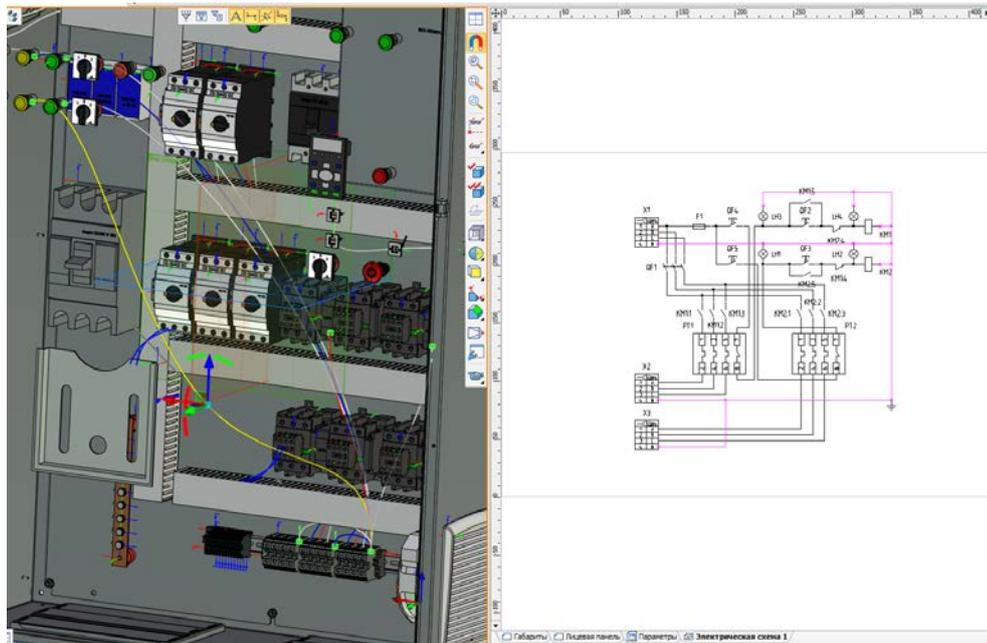
You can immediately generate ready-made documents on the basis of components in a diagram: list of elements, "from/to" table, connection table. All changes of the component characteristics in the diagram are accounted in reports.



An important advantage is creation of 3D assemblies based on 2D diagrams. You can establish associations between the components in the diagram and the 3D models of these components. This allows you to simplify the process of laying cable components and editing the finished diagram.

To work with the application user can load a 3D model from the catalog in any format available for import. User should create several LCS in the model. The LCS will be used as connection points. The models in an assembly can be connected according to the diagram.

Cable components can be laid manually or in semi-automatic mode. At the same time, the system monitors the connections of elements in the diagram, which allows to avoid their incorrect connection.

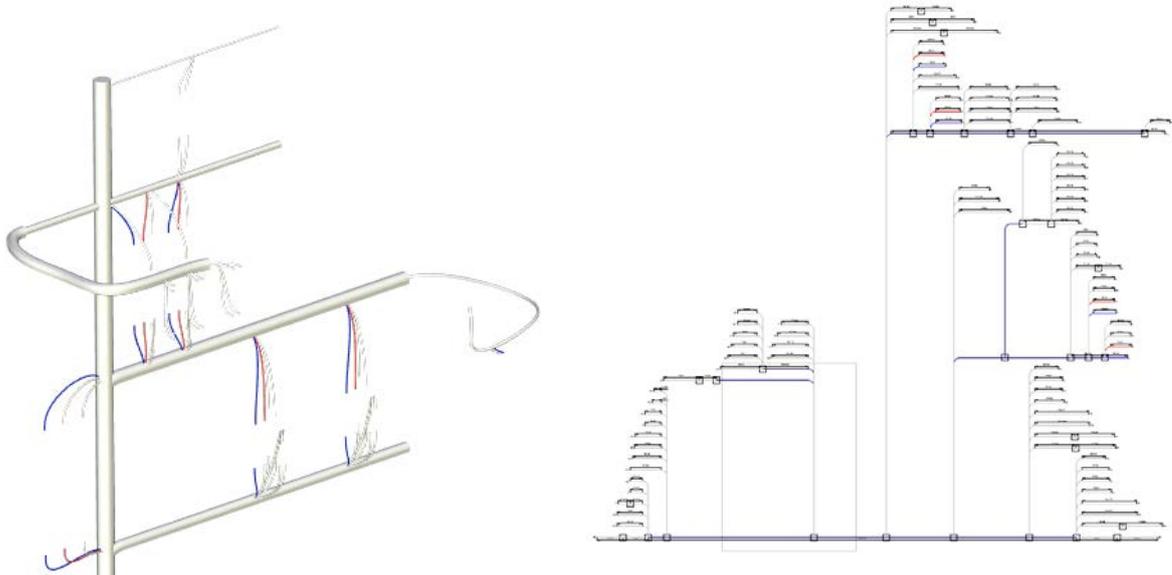


After cable components were created, you can automatically generate the following documentation:

- ✓ sheets of materials and cable components, taking into account lengths;
- ✓ "from/to" table and the connections table that are supplemented with information about cable components.

You can also create a general layout of the components in the designed assembly.

You can automatically form a flattened harness. The harness can be unloaded from the assembly into a separate file, and separate documentation can be generated for it.



The system uses a mechanism that allows you to update the diagram in the assembly, so users will always receive the latest information.

The T-FLEX Electrical module is a powerful tool for the layout of equipment of any complexity, and, when used with other products, it allows you to prepare a complete set of models, diagrams and reports for production.

VIRTUAL REALITY

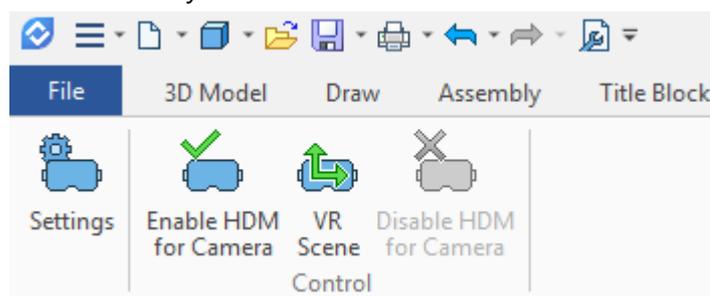
We have created a new application T-FLEX VR for users, who want to use virtual reality technologies in their work. The application allows you to explore the model in a virtual space and interact with it as if it is already a manufactured product.



The application requires a virtual reality HMD and controllers. The application works with the HMDs of the HTC Vive, Oculus Rift and other devices that use the OpenVR software interface.

T-FLEX VR is licensed as a separate product.

When the module is started, an additional tab appears in the Ribbon, containing the commands necessary to work with the virtual reality headset.

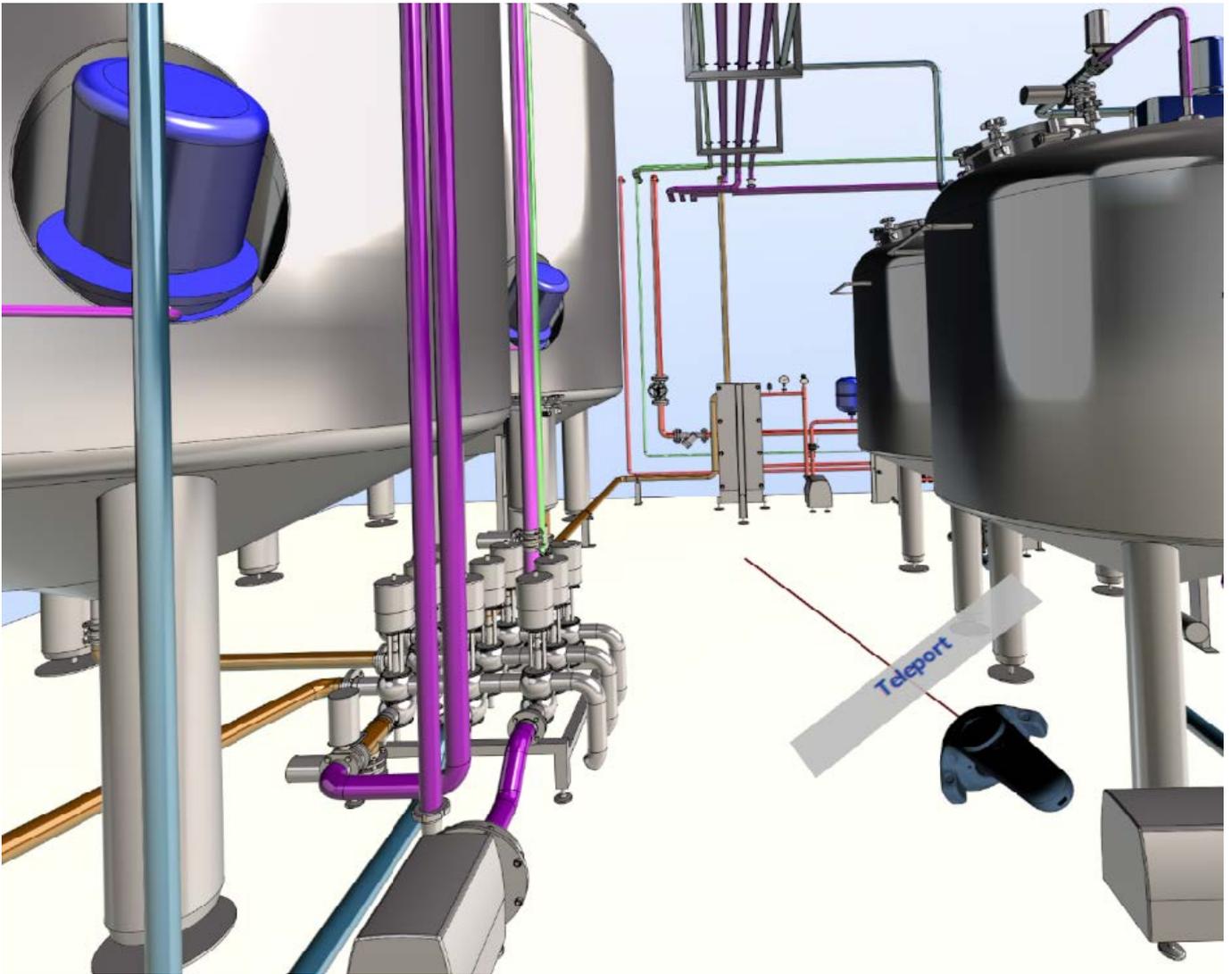


In addition, VR headset management commands are available for Camera objects created by user in the 3D model.

The controllers allow you to move objects in the scene and invoke commands, and the HMD gives the user a live view of the model. Synchronous update allows the user to see the results of his actions in real time.

T-FLEX CAD models do not require any special preparation for display in VR.

The module is built into the " T-FLEX CAD", so to display the open model in VR you only need to create a camera and start the virtual reality mode.



The user can adjust the work in virtual reality in a convenient way, thanks to a wide range of settings. For example, you can rotate around a vertical axis, or you can move the camera only horizontally by selecting corresponding options. In a scene, you can specify comfortable lighting sources and set the floor display. You can save the settings to a separate file.

Controllers in the user's hands are also displayed in the scene. Above the controllers is a line with a hint that shows the currently active command.

Configuration of each controller is separate. You can disable unnecessary commands for the controller. A special menu is used to select the active command.



Specialized T-FLEX CAD commands and standard controller commands are available in the list.

Commands for navigation in the scene:



Teleport. Moves user to the specified space point. This navigation mode is convenient upon working with architecture models when user want to "walk" inside virtual objects.



Move to object. In this mode, the user points to the part of the 3D-model, which he wants to see closer, and "teleports" to it. This mode can be useful when you need to visually evaluate industrial 3D models with a large number of small parts that are at some distance from each other.



Flight. In this mode user specifies a point on the 3D model and then moves at a specified speed to that point.



Setting the image scale to 1:1. This option is useful when the scale of the 3D model must correspond to the scale of the real part.



Camera selection. There can be several cameras in the scene. The option allows to quickly switch between them. This can be very convenient in those moments when you need to quickly switch between pre-known points of view.

Commands for working with the model:

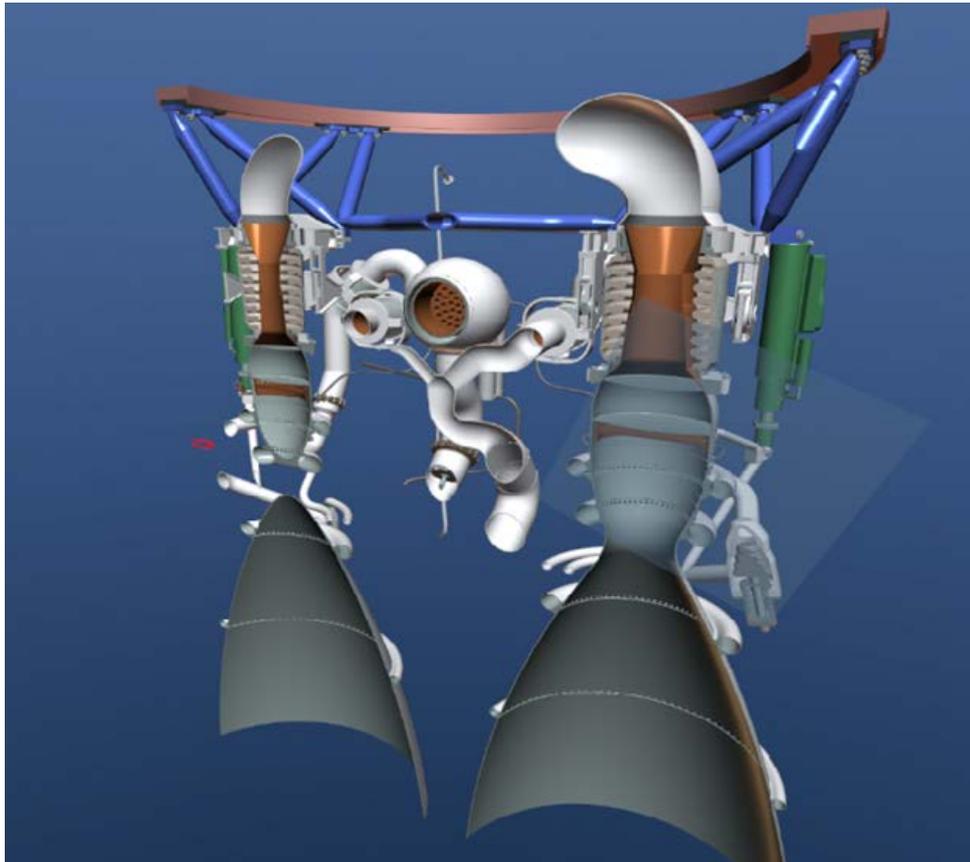
The user can rotate the scene and change its scale with the help of manipulators.



Pick up. The command allows to select an object in the scene. This object can be rotated and scaled using the second controller. Any part of the model can act as an object.



Section. A clip plane appears in the scene, and can be moved dynamically or set to the desired position. You can define multiple planes at once.



 **Exploded view.** The command allows you to run the selected scenario of disassembly and watch it from any convenient angle. This allows you to use the application for training of service personnel, which is important for those cases where the equipment is expensive, dangerous, or exists only in the form of a virtual prototype.

The T-FLEX VR application includes commands that are not available in other CAD systems with virtual reality.

 **Mates.** If mates are created in the model, the user can move the model elements according to the mates that you specify. Mates act the same way as in "T-FLEX CAD", i.e. if parts are connected to each other, then when one part of the mechanism is involved, all the others moves. It allows to simulate the operation of gears in the machine.

 **Manipulators.** If there are manipulators associated with the model variables in the scene, these manipulators can be used in the virtual space and thereby change the configuration of the model directly in the eyes.

T-FLEX VR can be used at all stages of product preparation: planning, design, approval and acceptance, after-sales service and training. You can also use the application to present finished or prepared for release products. At the same time "T-FLEX VR" gives users the opportunity not only to conduct a visual analysis of the product, check its ergonomics, evaluate the design, but also to conduct a real design in the virtual space. In addition, T-FLEX CAD users now have the opportunity to use virtual reality as an environment for training service personnel.