T-FLEX CAD 16

Three-Dimensional Modeling

User Manual

TOP SYSTEMS

T-FLEX Parametric CAD

Three-Dimensional Modeling
User Manual

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Basic Terms and Concepts of 3D Modeling

Basic 3D Terms and Concepts of Modeling with T-FLEX CAD

Introduction to 3D Modeling

T-FLEX CAD is a parametric solid and surface modeling system. It is equipped with most up-to-date tools for creating models of various complexities. Supported exporting and importing geometric data in common formats facilitates interoperability with most other CAD systems. T-FLEX CAD also includes a complete line of instruments for 2D modeling and drawing compliant with various national and international drawing standards.

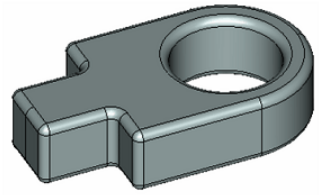
Before beginning with 3D modeling, it is recommended that users familiarize themselves with 2D drawing techniques in the appropriate sections of the documentation. This will help embracing the general modeling principles with T-FLEX CAD.

This chapter presents a classification and a brief description of all elements in T-FLEX CAD, and basic methods and techniques of 3D modeling. The following chapters of this volume will describe each of these elements in detail.

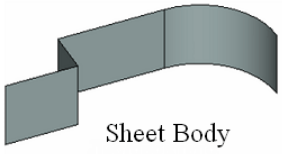
Basic Topology Elements

A model in T-FLEX CAD is a set of connected or disconnected geometrical components. Shown below are the basic topology elements that form any geometrical object in T-FLEX CAD.

Solid Body (a solid) is a set of geometrical objects – vertices, edges and faces that comprise a closed connected volume. A simplest solid can be defined by sweeping a bounded surface or a contour.

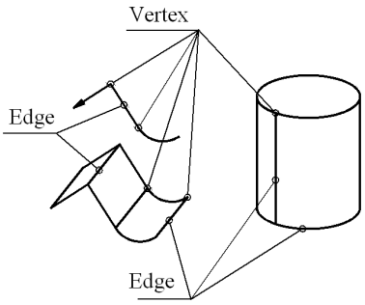

Solid Body

Sheet Body (a surface) is a set of geometrical objects – vertices, edges and faces that comprise a closed connected area and do not constitute a volume.

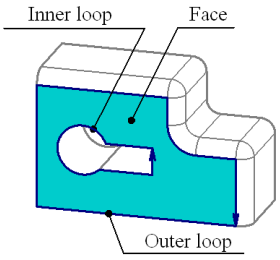

Sheet Body

Loop is a set of edges making one closed contour. A loop is an element defining the boundary of a face. Each vertex of a loop connects no more than two edges.

Edge is a curve segment bounded by two vertices. A closed edge may contain only one vertex.



Face is a bounded patch of a surface. The boundary is represented by loops. One face may have an unlimited number of loops. A face may be without loops. This is the case of a full face comprising a closed volume, such as a full sphere.



Basic Geometrical Terms in T-FLEX CAD

All geometrical objects in T-FLEX CAD can be classified into four general groups by type of the object geometry:

- The simplest object in the three-dimensional space is **3D point**. A point has only one property, namely, the placement coordinates. It can be defined by a 3D node, a 3D vertex, a "placement" parameter on a curve or surface, a characteristic location property within an object, such as on an axis of revolution, at the center of an arc or sphere, at objects intersection, etc.
- The second group of objects is "wire" geometry, the objects that possess the basic property of length. These include all linear objects, as are edges, 3D paths, loops and 3D profiles.
- The next group includes all objects that are characterized by area. These objects will be referred to as **"sheet"** geometry, and include all kinds of surfaces, sheet bodies, faces, and closed 3D profiles. A sheet object can be produced by most 3D operations.
- The forth group includes all **solid bodies**.

3D Entities and Operations

A 3D model creation implies building solid or sheet 3D objects describing a certain volume or surface in the 3D space. The creation and further modification of such objects is done by means of **operations**.

Operation is any step of creating a 3D model that leads to emergence of a new or modifications in an already existing solid or sheet geometrical object. A separate command is provided in T-FLEX CAD for each operation. The names of operation creating commands correspond to an operation's purpose.

The operations that result in new solid/sheet 3D objects will be referred to as **basic operations** (extrusion, rotation, sweep, loft, etc.). The operations that alter geometry and modify already existing solid/sheet objects will be referred to as **modifier operations** (blend, shell, Boolean, etc.).

The geometry base for most basic operations is provided by **3D construction elements**. 3D construction elements are auxiliary elements of a 3D model that are used for creating three-dimensional contours, defining spatial orientation, determining directions, vectors, axes, trajectories, etc. A separate command is provided for creating each such element.

> Some operations (for example, the operations for creating 3D arrays) can be both basic and modifier operations, depending on the source data and parameter settings.

Any solid or sheet 3D object in the 3D scene corresponds to a special element in the 3D model structure, which is **Body**. The "Body" element is introduced for user convenience: once a new geometrical object is created by the first (basic) operation (solid or sheet body), it can be handled in the future as a permanent element in the 3D model structure.

Body is created automatically upon creating a solid or sheet 3D object by a basic operation, and is maintained as long as the given object exists. The geometry of the original volume or surface may change (as a result of applying modifier operations), but it always refers to the same Body that defines parameters of this geometrical object: name, material, color, rendering settings (mesh density, wireframe display).

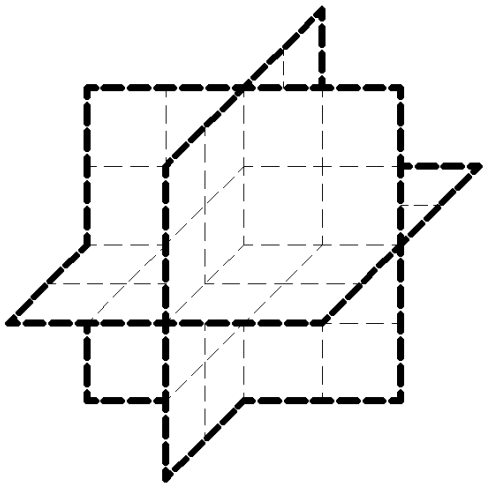
A 3D model can contain an unlimited number of Bodies.

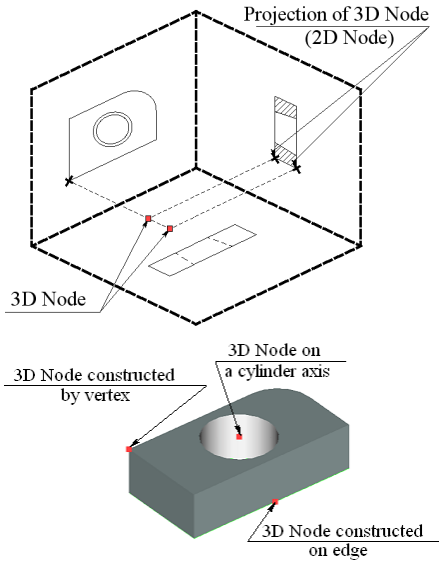
In some commands, Bodies can be used as separate elements. In such a case, the source object will be the body of the last operation in the creation history of the given Body. For example, when creating a 2D projection, a specific Body can be selected for projecting. This would be convenient, if the first design stage was making the drawing of the part's workpiece or a set thereof, while the second - modifications of this model by additional operations. In this way, the drawing reflects on all future modifications to this part.

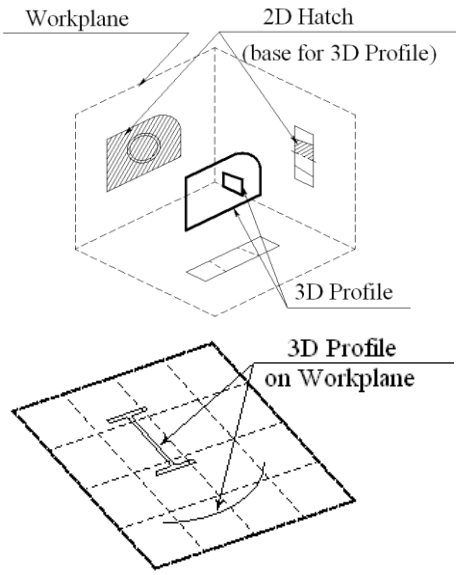
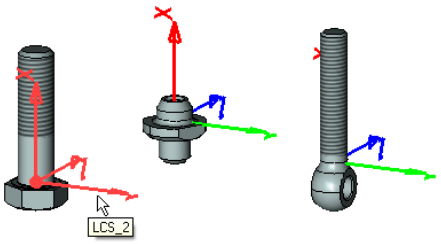
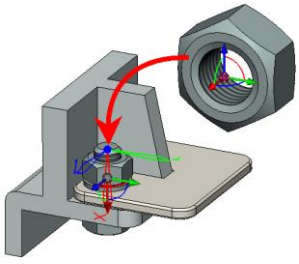
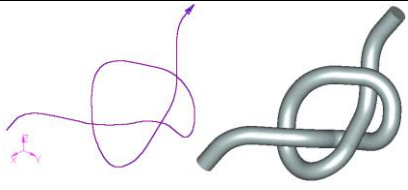
> In this manual, the term "Body" (with capital "B") means specifically a 3D model structure element. The lowercase inscription "body" will be used for a quick reference to the geometrical object, which is a volume or surface in the 3D scene.

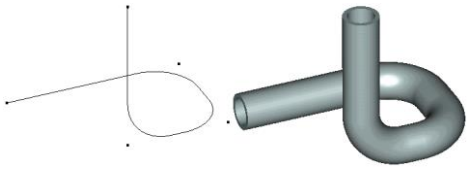
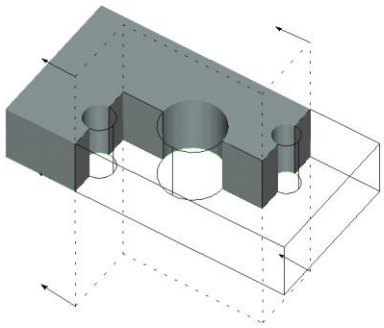
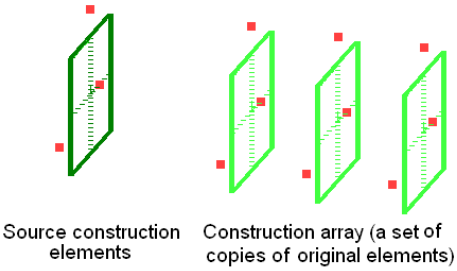
3D Construction Entities

Workplane is an entity that helps defining the required data for 3D operations, and, first of all, creating 3D profiles. A 3D model cannot be built without creating a workplane. Workplanes can be specified in 2D or 3D window based on various references, such as the 2D drawing views, projections of entities in a 3D model, or the 3D entities themselves, including other workplanes.

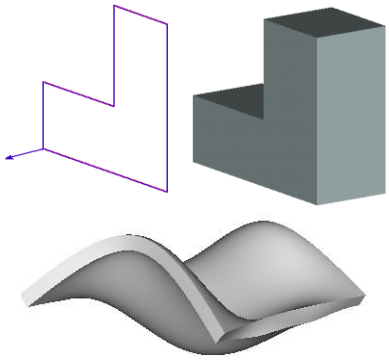
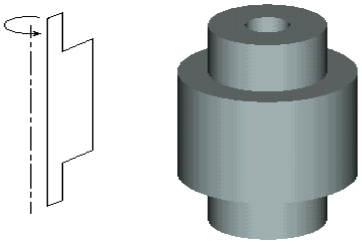


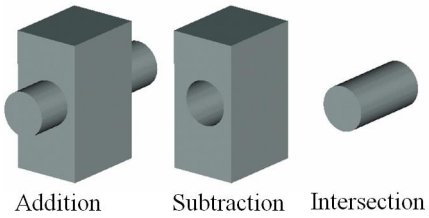
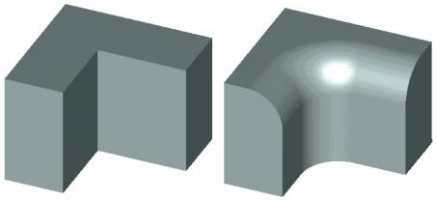
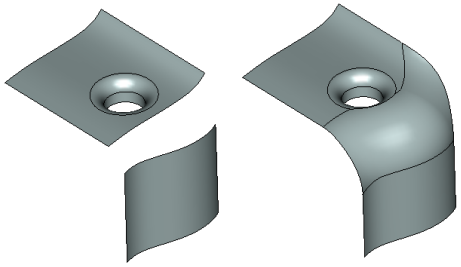
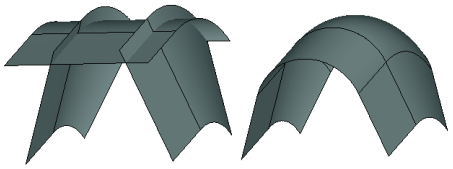
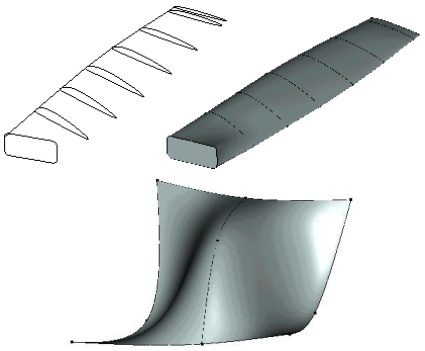
Worksurface serves similar functional purposes as the workplanes, providing a non-planar geometrical basis for further design. A worksurface can be a cylinder, a sphere, or a torus.	
3D node is one of the basic construction entities used for representing a point in the three-dimensional space. There are several ways of creating 3D nodes. A 3D node can be specified, for instance, as a characteristic point on a body, referencing a vertex, an edge, or a face. It also can be located by using absolute coordinates or offsets from other 3D nodes. 3D nodes can also be specified using the nodes from the two-dimensional drawing, and workplanes. In this way, the 3D node will be defined by selecting one node on a workplane, or two nodes on two different workplanes that are related by having a common projection.	

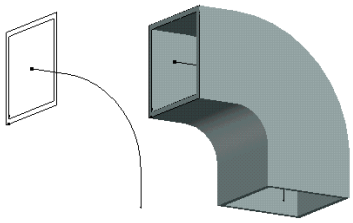
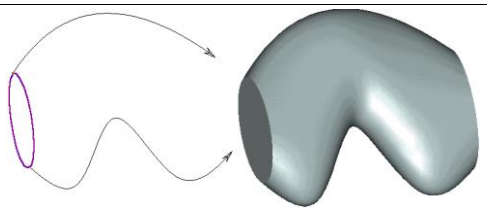
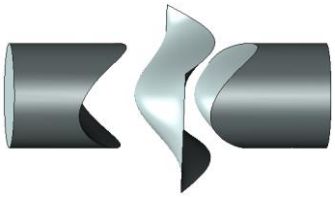
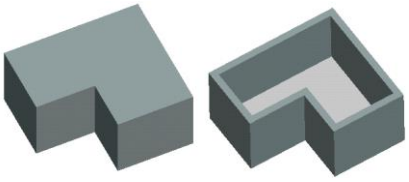
3D profile is a construction entity used for defining a patch on a surface. A 3D profile is one of the basic entities because it is used as an original reference for many operations. A 3D profile may be defined by an open or closed contour. A closed contour defines a fixed area on the surface that can be used as a base for various operations creating solid bodies. An open profile can be used as a base for creating sheet bodies only as it consists of wire geometry. There are many ways of creating a 3D profile. Thus, it can be drawn on a workplane, or get anew by a variety of modifications to an existing profile, and so on. One 3D profile may contain several contours of the same type. An example of a multi-contour profile is a text entity. A 3D profile can be obtained from hatches and graphic lines (when drawn on the active workplane).	
Local Coordinate System is an entity for referencing three-dimensional objects in space. It is used for inserting 3D fragments and other elements, copying, exploding of assemblies, etc. This entity is defined by the origin point and the axis directions. Objects snap to a coordinate system by making the object and the target coordinate systems coincide.	
3D connector – is a special type of local coordinate systems which allows a user, apart from performing the functions of snapping, to automatically tie external variables for jointed parametric elements of the 3D assembly models. This significantly simplifies positioning of parts and parameters assignment when designing the assemblies.	
3D Path is a bounded three-dimensional curve with a defined traversal direction. The 3D paths are used in the operations "Sweep", "Pipe", "Loft". A 3D path can be defined by a hatch, 2D paths, as a chain of a body edges, as a curve passing through a sequence of 3D nodes, by modifying existing 3D paths, etc. A 3D path can be closed.	

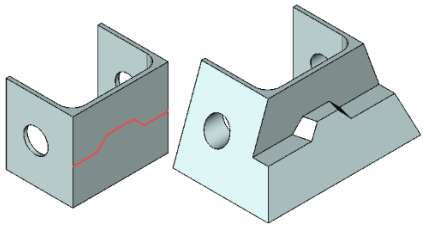
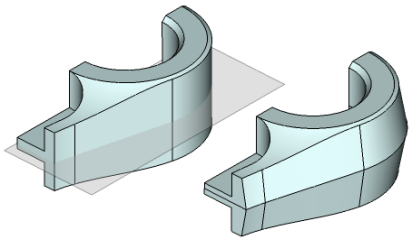
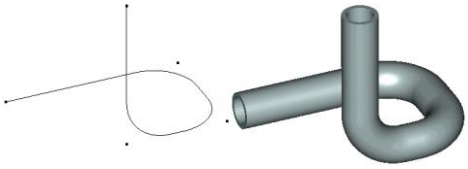
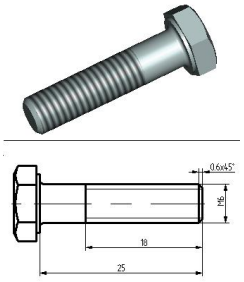
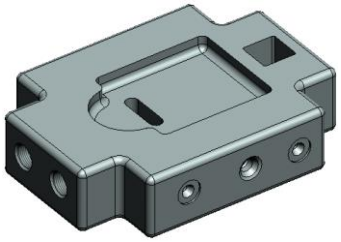
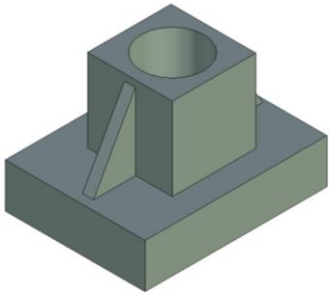
Pipe Path is a 3D path consisting of straight line segments smoothly connected by arcs. This command is mainly used for modeling pipes. A large variety of options and controls provided by this command help quickly and easily solving this sort of tasks.	
3D Section is an entity that, generally, results from extruding a planar curve or polyline through all geometry in the direction normal to the plane of the curve. In some cases, a 3D section consists of one or several planes. In this case, it can be used for creating two-dimensional cuts. A 3D section may be included in visualization of the objects of a 3D scene, and can be used in "cut" operations.	
Construction array is a special composite construction element. A construction array is a particularly organized set of copies of an arbitrary construction element, except for sections, light sources and cameras. The copies contributing to such an array can be used as conventional construction elements. Construction arrays are created and edited using any of the 3D array creation operations.	

Basic Three-Dimensional Operations

Extrusion yields a body formed by straight propagation of a contour along a specified direction. This operation creates both solid and sheet bodies. Extrusion can be performed along not only the extrusion vector, but the normal to the contour in either or both directions as well. Thus, it provides a means of thickening an arbitrary face, even a non-planar one.	
Rotation yields a body formed by revolving a contour around an axis located in space for a given angle. The original contour can be located in an arbitrary orientation with respect to the axis, but it should not intersect with the axis. This operation creates both solid and sheet bodies.	

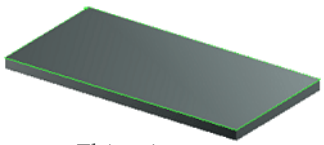
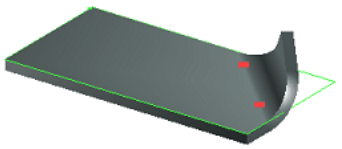
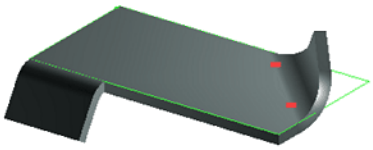
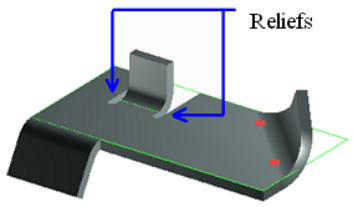
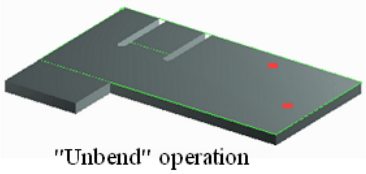
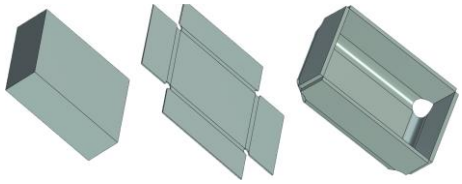
Boolean operation is intended for creating a new body by combining two existing bodies. The type of the operation is specified as addition, subtraction, or intersection.	
Blend Edge is an operation that modifies an existing body by smoothing or merging its vertices, edges and faces. The main difference of this operation from other types of blending is in that the new surface is created from the selected edge and merges with the adjacent faces only. The operation permits creating chamfers, edge rounding with a variable radius and elliptic rounding.	
Blend Face-Face is an operation for creating smooth transitions from one set of smoothly connected faces to another. The sets of faces to be blended may not have common edges (intersect). The command has numerous options for controlling the blending surface shape, trimming conditions, bounds, etc. This operation should not be considered a substitute for blending edges. Each of the two approaches has their own advantages and well complement each other.	
Blend Three Faces is a special case of face–face blending separated into another command. The operation creates a transitional surface between a "right" and a "left" wall while tangent to a "middle" wall.	
Loft is an operation for creating new bodies of complex geometrical shapes. The resulting spline surfaces are constructed on the wire guides in one or two directions and according to the specified boundary conditions. The base elements for spline definition can be practically any entities of the three geometrical types, "point", "wire" and "sheet". Depending on the base element type, the result will be either a solid body or a set of surfaces.	

Sweep is an operation that creates bodies whose surface is defined by an arbitrary-shape profile propagated along a space curve. The operation provides controls over scaling of the profile and its twisting with respect to the trajectory axis as it moves along.	
Parametric Sweep – this operation extends the capabilities of the "Sweep" operation. The base profile is defined in such a way that its geometry and orientation are driven by a variable. The driving variable can assume values within the specified range. The body is created as a result of re-computing the geometric shape and orientation of the profile across the whole range of the variable values.	
Spiral is an operation for creating spiral-like bodies. The generatrix contour can be defined by an arbitrary-shape profile. This operation may be used for modeling actual geometry of a thread. However, in most cases, when only cosmetic representation of a thread is needed, the "Thread" operation is recommended for use instead.	
Spring is a specialized operation for creating spring-shape bodies. It differs from the "Spiral" operation in the capability of forming the end cycles of the spring. The generatrix of a spring is defined as a circle.	
Cut is an operation that divides a body in two, or cuts a portion off a body. The cutting surface can be defined as a set of connected faces, or sections, or workplanes.	
Shell is an operation that removes the inside material of a part by removing the selected faces and leaving the rest of the faces just thickened by the specified width in the direction where the material used to be, removing all the rest of the material. A shell can also be created without removing any faces. Another special functionality allows creating new bodies offset from a given one.	

Face taper is an operation that tilts selected faces at the specified angle while automatically adjusting other affected faces.	
Body taper is an operation that creates 3D bodies by tapering faces of a selected body at a specified angle with respect to selected edges of that body in the taper direction. This operation significantly simplifies the design of cast molds. Unlike "Face Taper" command, this operation allows creating two-sided tapers and tapered body faces without having a clearly defined "fixed" edge.	
Pipe operation makes a pipe along a space trajectory (a 3D path). The user specifies the diameter of the pipe and the diameter of the outlet. The outlet itself is optional.	
Thread is an operation for creating cosmetic representation (imitation) of threads on cylindrical and conical faces of a 3D model. When the projections of the model are created on a drawing, the thread is automatically drawn according to drawing conventions	
Hole is an operation for creating standard apertures. It relies on a provided parametric library of holes satisfying current standards. The command supports creation of patterns of holes, holes through multiple bodies, and threaded holes. When a threaded hole is created on a face, a cosmetic thread is displayed.	
Rib – operation that allows us to create ribs of the solid body on the basis of one or several 3D profiles. Execution of this operation results in a solid body obtained via a Boolean operation of a union of the rib being created with the source body.	

Sheet Metal Operations

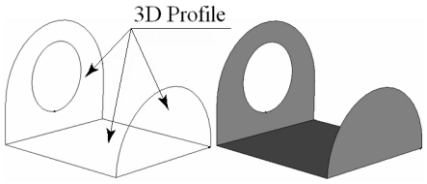
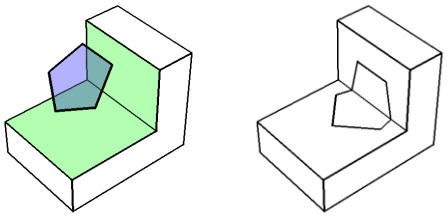
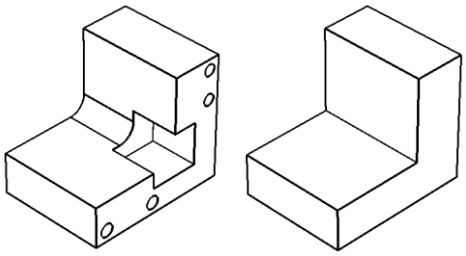
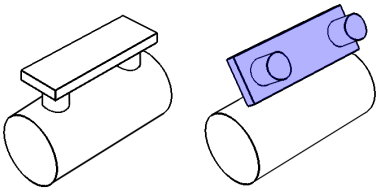
The specialized sheet metal operations make a separate group.

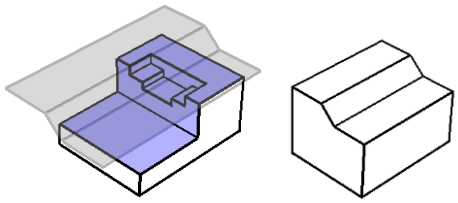
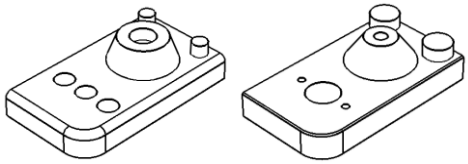
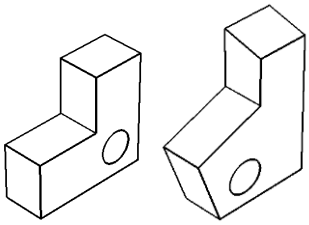
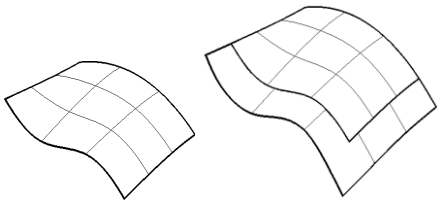
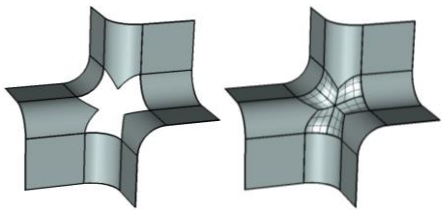
Create Part operation creates a solid body – a flat part. The base element for the operation is a flat closed 3D profile. The body is created by thickening the profile by a specified amount. The part material will be added along the normal direction to the profile at either or both sides of the profile.	 Flat part
Bend operation supports three types of bending, "Bend", "Attach Flange", "Cut and Bend". • The first type is for bending an existing body, such as a part, around an infinite axis defined by a pair of 3D nodes or a straight-line entity. • The second type supports attaching a flange to the part according to specified length, width, bending radius and the offsets from the ends of the bending axis. The third type is for bending a portion of the part around a finite line segment. This is done by making appropriate cuts in the original body. The operation allows introducing reliefs defined by their type, depth and width.	 "Bend" operation  "Attach Flange" operation  "Cut and Bend" with customized reliefs
Unbend operation can be used after a sheet metal model has been defined, to get the flat part and proceed, for instance, with its drawing. Re-bend operation repeats bending of all surfaces that were previously "unbent". It provides a means of keeping the final bent shape with the model data. This is important, for instance, when creating a drawing with both the flat part views and a final bent shape view.	 "Unbend" operation
Convert Solid to Sheet Metal. Command is used for quick creation of sheet metal bodies or unfoldings from the solid bodies.	

Forming Feature operation is specifically provided for performing common sheet metal stamping manipulations. It uses a library of typical elements for creating ribs, beads, flanges, embossings, louvers, etc. The geometry of these features is defined by 3D profiles.	

Face Handling Operations

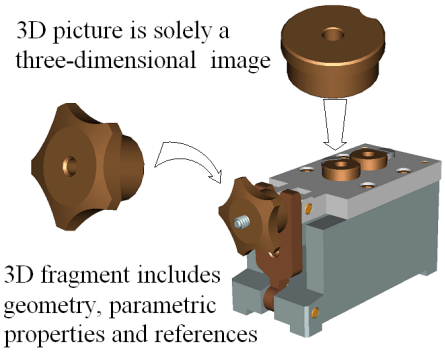
The set of commands for direct handling of body faces are united into a separate group.

Sew is an operation for creating solid bodies or sheet models from a set of disjoint surfaces adjacent to each other, as faces, 3D profiles, sheet bodies. When sewing into a solid body, the operation may add simple surfaces as necessary.	
Imprint Elements is an operation that forms patches of a specified shape on already existing faces. Depending on the dividing option, the shape of the new patch is determined either by the shape of the dividing element or by the geometry of the element being divided.	
Delete Faces is an operation that allows deleting one or several selected faces. Deleting faces breaks the solid body topology. Gaps are introduced that invalidate the closed volume. If necessary, the system may attend to mend those by various means.	
Separate Faces is the command for excluding selected faces from an already existing body and using those for creating a new body. The gaps in bodies caused by this process can be closed by one of several means.	

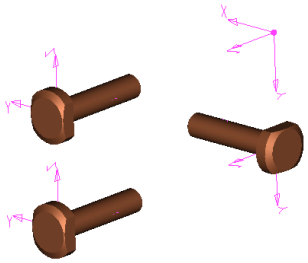
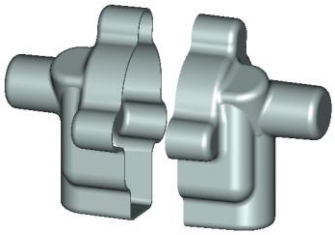
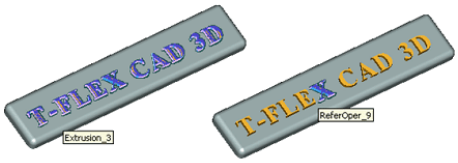
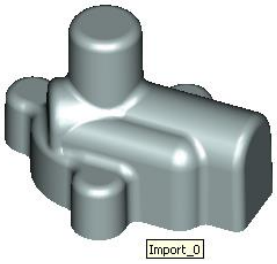
Replace Faces is the command for substituting a geometrical surface underlying the selected faces by different surfaces. Sheet bodies can be used as a replacement surface.	
Change Faces is the command for modifying face parameters when the underlying surface is analytical (a cylinder, cone, sphere or torus), as well as parameters of faces created by the blend operation.	
Transform Faces is the command that applies a transformation to one or multiple selected faces.	
Expand Faces is the command for expanding the selected face (or multiple faces belonging to a sheet body) in the specified direction by the given length. The direction of growing the face is defined by selecting side edges on the faces being expanded. 3D profiles can also be expanded.	
Fill Hole is the command for creating one or multiple faces closing an area bounded by a closed loop of edges. Depending on the initial geometry, the system may fill the area by an analytical or a ruled surface, or by an explicitly specified sheet body.	

Copy and Insert Operations for 3D Elements

Insert 3D Fragment is an operation for using the geometrical data of a standalone 3D model in creating an assembly model. Any T-FLEX CAD document containing a 3D model can be used as a 3D fragment.



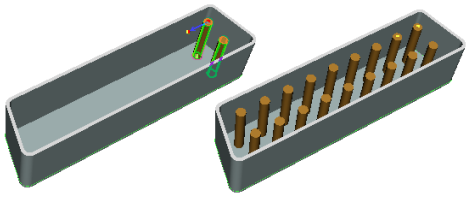
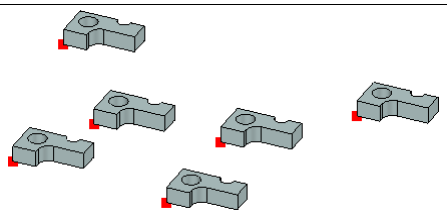
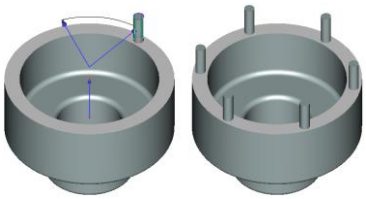
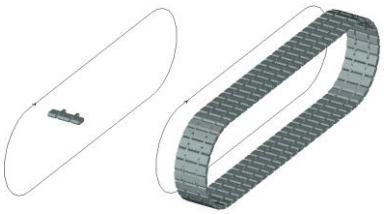
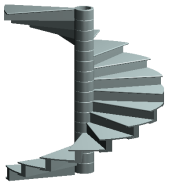
Insert 3D Picture is similar to inserting a 3D fragment, except that the 3D picture has neither associativity among elements nor parametric modification capabilities. It is merely a shaded 3D image of a part that appears exactly as a 3D fragment. 3D pictures are handy at a final stage of design when the element modifications are no longer expected. Since 3D pictures are not subject to regeneration, the overall model regeneration time is reduced. 3D pictures cannot be used as references for other elements. Thus, faces of such objects can't be selected, and projections cannot be constructed.

Copy is an operation for creating a transformed copy of a body defined by various transformation parameters. Copy operation utilizes a "local coordinate system" entity.	
Symmetry is an operation for creating new bodies as copies of the original of bodies by reflection at a specified symmetry plane.	
Divide is an operation for separating numerous bodies obtained from various operations. The resulting bodies can further be processed separately. For instance, the bodies created by an "Array" operation, will be separated into standalone elements. This command can also be used with an imported from other systems model that consists from several bodies.	 An extrusion of a multi-contour profile based on a text (Extrusion_3) is divided into separate bodies (letters)
External Model is an operation for importing models created in other systems based on Parasolid format (*.x_t and *.xmt_txt). Just like 3D pictures, these objects do not possess parametric properties, however, their elements (vertices, edges, and faces) can be used as references for further design.	

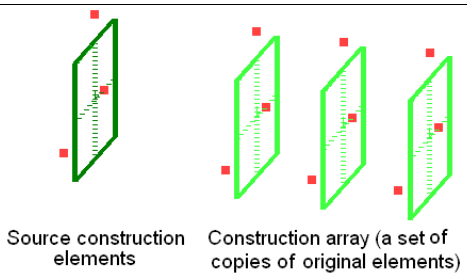
Operations for Creating 3D Arrays

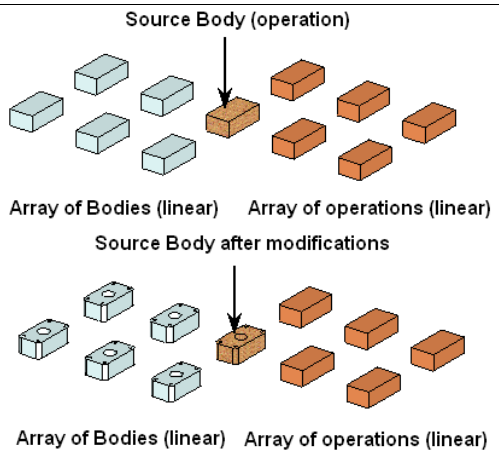
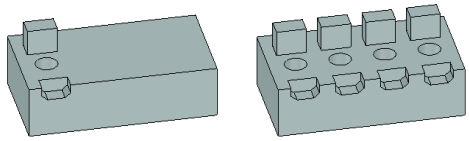
The array creating operations allow simultaneously creating multiple copies of source 3D objects. The source objects for creating arrays (objects for copying) can be not only operations and Bodies, but also 3D construction elements and faces.

Placement of the copies being created (the array elements) depends on the array type: linear, circular, array by points, array by path, parametric array.

Linear array is the one in which the copies of the source objects are placed along one or two direction vectors at a specified step. The copies can be placed not only in the forward, but also in the reverse direction along each direction vector.	
Array by points is the one in which the positions of the arrayed copies are defined by 3D points.	
Circular array is the one in which the copies are placed on a circle around the array axis. Reverse rotation is also possible, including creation of copies simultaneously in two rotation directions. It is also possible to create copies in a second direction - either along the array axis or in the radial direction.	
Array by path is the one in which copies are placed along one or two spatial curves. One can set different ways of orienting copies on each guide curve – by the cord, by the minimal twist, by parallel translation.	
Parametric array is the one in which the spatial positioning and parameters of copies are defined by a specified parametric law.	

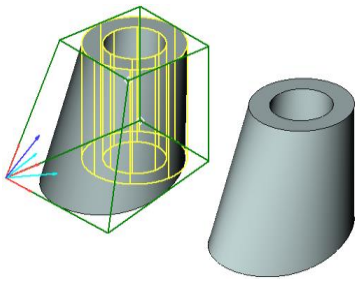
Depending on the type of the copied objects, distinguished are the following array types: arrays of construction elements, arrays of operations, arrays of Bodies and arrays of faces. All arrays of one element type, regardless of their properties, share the specifics of creating and editing.

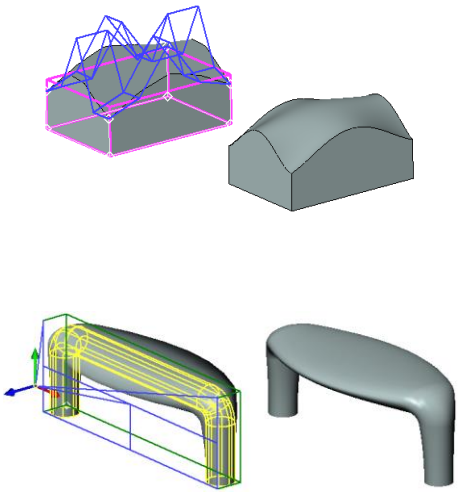
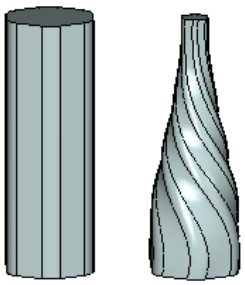
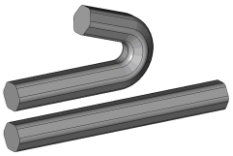
Array of construction elements makes copies of any 3D construction objects, except for sections, light sources and cameras. It results in a special 3D construction element – a construction array.	

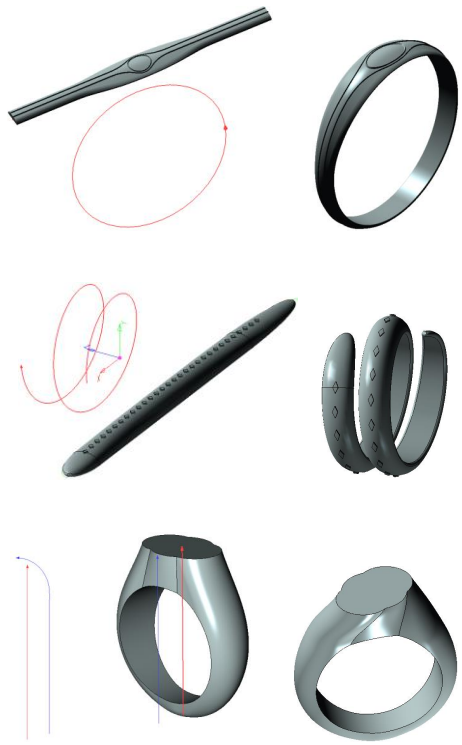
Array of operations copies only the result of the selected operation. If the body created by the operation is later transformed by another modifying operation, the array is not affected. Array of Bodies copies a whole Body. If that Body is modified in the future, the array will regenerate, accounting for the new operations added in the Body's history.	
Array of faces is used for adding holes and protrusions repeating existing design elements, to Bodies already existing in the 3D model. Any array of faces is always based on one model Body: all copied faces must belong to this Body.	

Deformation Operations

The operations of deformation allow a user to carry out modification of solid and sheet bodies by various means. When applying these operations on the basis of parameters specified by a user, an internal function producing the volume deformation of the deformed body is generated in the model. Applying this function in a continuous manner deforms the entire volume of this body (or its part). Topology of the deformed part of the body is not changed. The number of faces, edges, and vertices, etc. is preserved. If necessary, the faces and edges of analytical type (planes, segments, cylinders, arcs of circles, etc.) are automatically replaced with spline surfaces and curves.

Skew – this type of deformation assumes existence of the original body and the coordinate system in which a bounding parallelepiped is calculated. Deformation law is specified by displacing the vertices of this parallelepiped in different directions. The displacement can be realized along any of the axes of the coordinate system, along the edges of the bounding parallelepiped, and along the diagonals of the faces of the bounding parallelepiped.	

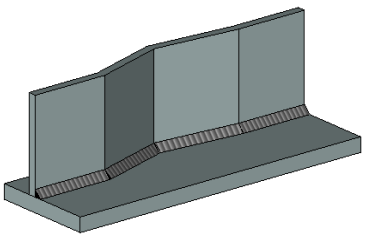
Sculpt deformation – in this type of deformation a regular mesh of points is defined on one of the faces of bounding parallelepiped. Any point on this mesh can be displaced relative to its initial position by a specified value. As a result, a flat face of virtual bounding parallelepiped is transformed into a space spline surface which forms the required transformation law for the body. The operation of sculpt deformation has 3 modes: • One side – only the points located on one face of the parallelepiped are displaced; • Both sides – the points located on opposite faces of the parallelepiped are displaced. The points on the face opposite to the selected one are displaced in the same direction by the same distance; • Symmetric – points on the opposite face are displaced symmetrically with respect to symmetry plane of the parallelepiped.	
Scale/Twist – this operation allows a user to specify various scales and angles of twist in different sections along the axis of the selected coordinate system. This deformation can be carried out either for the entire body or within the borders of the user-defined region. In addition to scaling and twisting sections, it is possible to entirely stretch or compress the deformation region in the direction of the selected axis of the deformation. For sections, the scales along different axes can be different.	
Bend – this operation allows a user to bend the selected body by a specified angle with respect to the selected axis. In addition to bending angle, the distance defining the location of the «neutral» surface is specified.	

Deformation by curve – for the deformation by curve, the overlap of the source curve, associated with the deformed body, with the target curve takes place. The deformation function constructed as a result of this is applied to the deformed body. A user can control the location of the body with respect to the curve. Upon deformation, one of the three body orientation control algorithms and the ways of using the source curves can be chosen. •**3D curve – 3D curve.** This method uses one source and one target curve. In most of the cases in practice, the source curve is a line. As an example, consider one of the typical problems solved by the present algorithm – wrapping the source body into a ring. • **Curve – Spiral.** The spiral-like curve is chosen to be a target one and the axis of the spiral is specified. • **Pair of curves – Pair of curves.** The source pair of curves and the target pair of curves are specified. Additional curves perform the function of controlling the twist of the body with respect to the main curve.	
Deformation by surface – this type of deformation forms the transformation law of certain single surface into another surface and applies this law to the source body. As the initial data from the source and target sides, the surfaces and the three points on each surface are chosen for the subsequent overlap. An additional displacement of the resulting surface from the source surface can be also specified. This transformation works in two modes – «By parameters» and «With minimal distortion». In the former case, the exact overlap of the parametric surface spaces by the selected points is carried out. In the latter case, the proportions of the geometric distances measured between certain points on the source surface are preserved.	

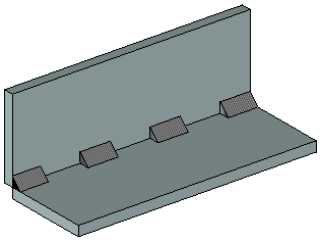
Commands to Create Welds

A group of commands in the **Tools** > **Weld** menu is provided to design welded parts. Those serve to create various types of standard and nonstandard welds on a 2D drawing or in the 3D model. Once a weld is created, its symbol can be automatically applied, and tables of welds can be built.

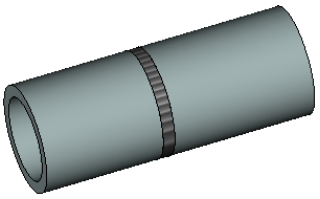
There are the following 3D weld types: fillet, intermittent fillet, butt, composite. Fillet, intermittent fillet, and butt 3D welds are denoted in the 3D scene with a special "cosmetic" body with a characteristic texture.



Fillet 3D weld



Intermittent fillet 3D weld



Butt 3D weld

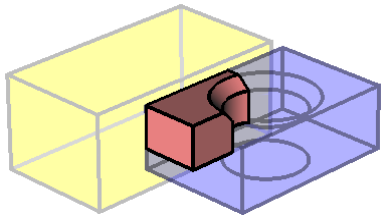
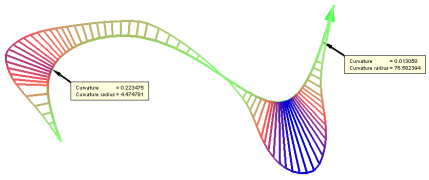
The composite weld is a variation of the 3D weld. It can be used to designate some 3D model elements (bodies, edges, 3D profiles, 3D paths, or a combination of several existing 3D welds) as a 3D weld. In this case, no image of the weld will be created in the 3D scene.

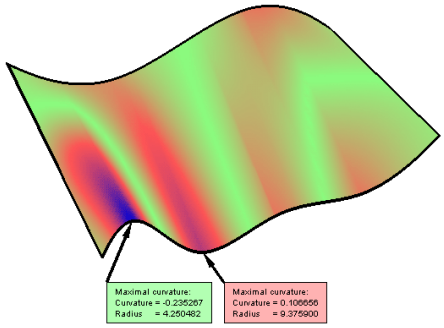
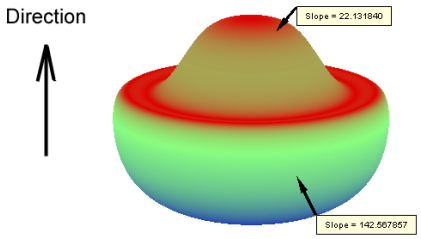
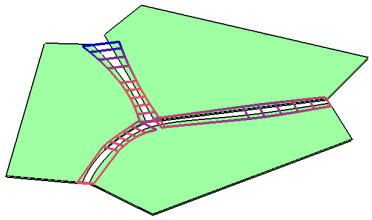
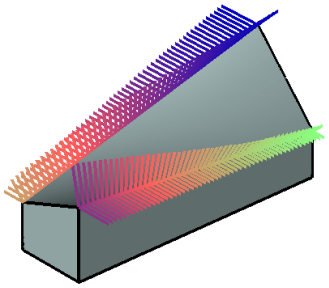
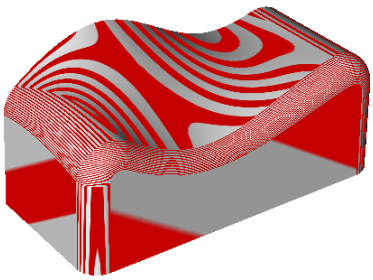
Geometry Analysis Commands

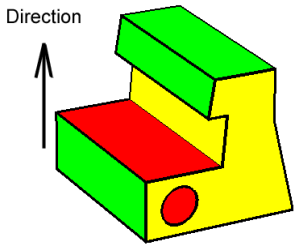
Measure is a command for defining mutual situation of objects on the 3D scene, whether one body penetrates another one, what is the minimum distance between elements. Besides that, various geometric characteristics can be computed for selected elements, such as length of an edge, area of a face, coordinates of nodes, etc. One can introduce variables that will be gaining specified characteristics from the elements of interest, using special functions. Thus, these characteristic values can be used as input for further construction.

Mass-Inertia Properties is the command for calculating the mass-inertia properties of the selected operations. If necessary, the calculations can be done with respect to a specific coordinate system.

Model check is the command for diagnosing the selected body for errors in its geometry.

Intersection Check is the command for examining the model against intersections and contacts between selected bodies. The command is particularly helpful when working with assemblies.	
Curve curvature is the command for measuring the curvature and the radius of curvature on selected curves. The curvature is displayed in the form of the porcupine quill. To measure curvature, you can select edges and 3D paths.	

Surface curvature is the command for measuring curvature and the radius of curvature of one or multiple selected faces. You can watch the overall curvature distribution on the face (the model is colored appropriately for this purpose), or read the curvature value at a specific point.	
Surface deviation is the command for defining the selected face's normal deviation from the specified direction. The colored face display allows watching the deviation all over the face. You can also measure the deviation at a specific point.	
Surface gap is the command for evaluating the gap between two or multiple selected faces. The command is used for analyzing models suffering import/export deficiencies.	
Normal deviation is the command for measuring deviation between the normals of the neighboring faces evaluated on the specified edges. The quills displayed in the 3D window help watching normal deviation across the overall edge extend. Besides that, the angle of the normal deviation can be measured at a specific point.	
Surface smoothness is the command for evaluating the model regularity. On entering the command, a special pattern is applied to the body's faces, usually made by long light and dark stripes. This pattern helps usually determine whether the adjoining faces make a smooth tangency transition, or smooth curvature transition.	

Model Separation Check is the command for identifying faces of the areas preventing a die cast or injection mold from opening. Additionally, the faces are highlighted, whose slant angle with respect to the selected direction is less than specified.	

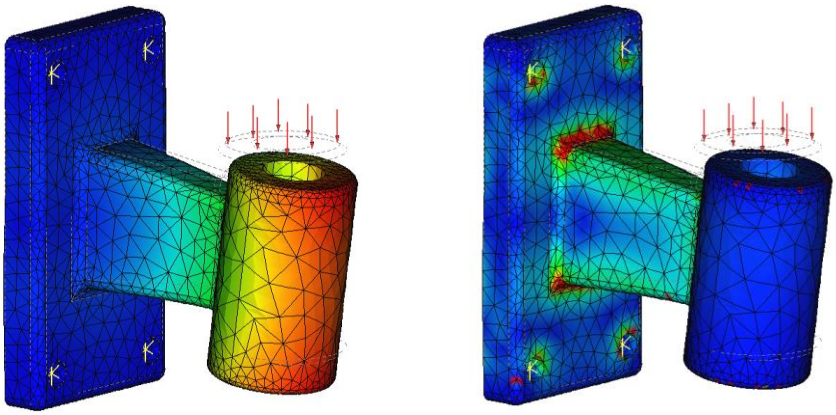
Engineering Analysis

Besides the geometry analysis commands described in the previous paragraph, the T-FLEX CAD complex provides additional modules serving to conduct advanced model analysis: **Finite Element Analysis** and **Dynamic Motion**.

Finite Element Analysis – this is a "**T-FLEX Analysis**" module that serves to perform various types of finite element calculations:

- **Static analysis** allows calculating the state of stresses and strains in a structure under the impact of constant in time forces applied to the model;
- **Frequency analysis** allows calculating natural (resonant) frequencies of a structure and the respective vibration modes;
- **Buckling analysis** is important when designing structures, whose operation implies lasting influence of loads ranging in intensity;
- **Thermal analysis** is the module providing the capability of evaluating a heated product behavior under the impact of sources of heat and radiation.

T-FLEX Analysis is oriented at solving physical problems in the three-dimensional formulation. All calculations rely on the finite element method (FEM). The product's mathematical approximation uses its equivalent replacement by a mesh of tetrahedral elements. At the same time, an associative relationship is maintained between the three-dimensional model of a part and the finite element model used in the calculations. Parametric modifications of the original solid model are automatically propagated into the meshed finite element model.



Results of finite element modeling (displacements and stresses)

The standard T-FLEX CAD distribution kit includes only a limited trial version of the finite element analysis module – **Express Analysis**. The Express Analysis is a light version of the "T-FLEX Analysis" module, specifically tuned for running simplified yet qualitatively sound strength studies. The user is provided with the necessary selection of load and restrained types. Based on the T-FLEX CAD model geometry, the automatic mesh generator creates a quality finite element discretization within the Express Analysis. Once the calculations are completed, the stress, strain, displacement and strength safety factor are output in a graphic form.

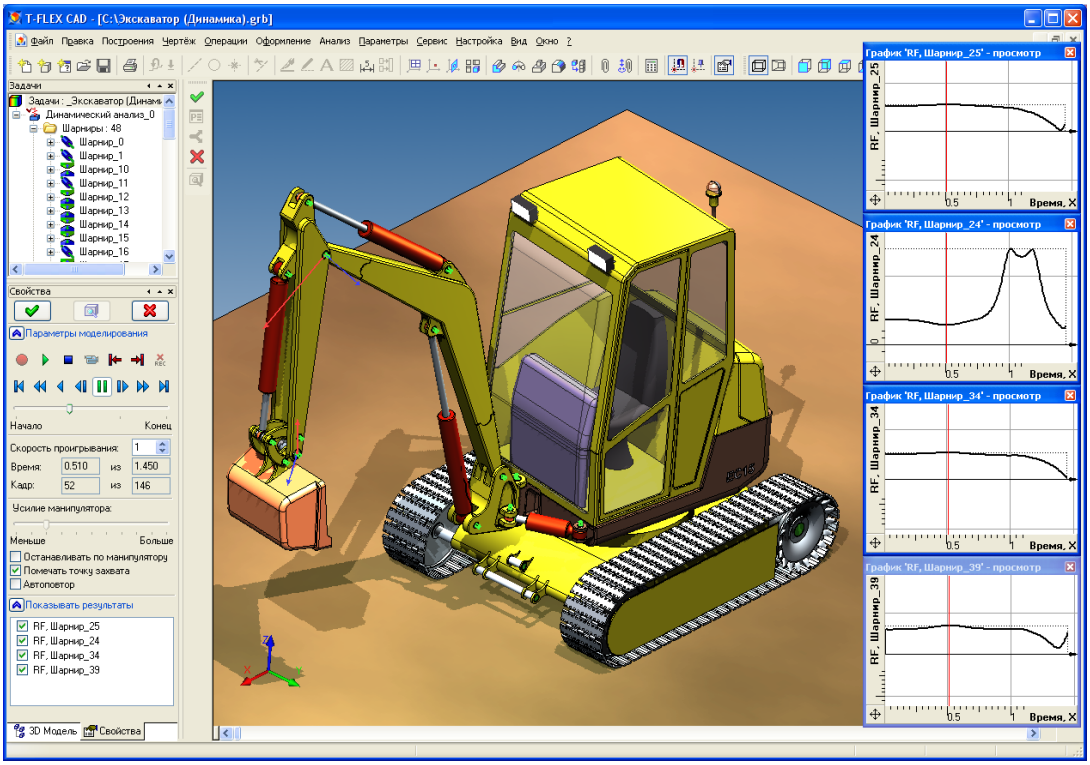
The fully functional "T-FLEX Analysis" finite element analysis module is sold separately.

> For details on working with the finite element analysis module, please refer to the "T-FLEX Analysis" manual.

Dynamic Motion – this is the module that serves to run studies on dynamic behavior of various three-dimensional mechanical systems.

The dynamical analysis module is capable of solving the following studies:

- Analysis of the motion trajectory, velocity and acceleration of any point of a component in a mechanical system due to applied forces;
- Analysis of performance times of a mechanical system (the time to the target point, the time for vibrations to settle, etc.);
- Analysis of forces building up in components of a mechanical system during motion (reaction forces at the supports, joints, etc.).



A mechanism's model is defined as a system of solid bodies, joints, and loads. The data for the analysis is automatically accessed directly from the geometrical model created in the T-FLEX CAD system. The familiar tools of T-FLEX CAD are used in modeling, with mates and degrees of freedom employed to define relations between three-dimensional bodies. The system also provides the means of modeling the contact between arbitrary solid bodies, being capable of processing a simultaneous contact of hundreds and thousands of solid bodies of arbitrary shape.

Loads on solid bodies are defined as the initial linear and angular velocities, forces, moments, springs, gravity, etc. To read the results, special sampling elements are used. Numerous values are available for the analysis: coordinates, velocities, accelerations, reaction forces in joints, forces in springs, etc. The user can observe the model behavior from any viewpoint immediately during the actual calculation. One can create animation clips from the obtained results of a dynamic calculation.

The T-FLEX CAD standard distribution kit includes only a limited trial version of the dynamic motion module – the Express Dynamic Motion Analysis. The Express Analysis has certain limitations on load types and the ways of rendering dynamic motion analysis results (the tools to obtain numerical calculation results are not available). In the commercial module, the calculation results are displayed as graphs, dynamic vector arrows and as an array of numbers (graph points).

The fully functional "Dynamic Motion" analysis module can be purchased separately whenever needed.

> More details on working with the "Dynamic Motion" module are provided in the "T-FLEX Dynamic Analysis" manual.

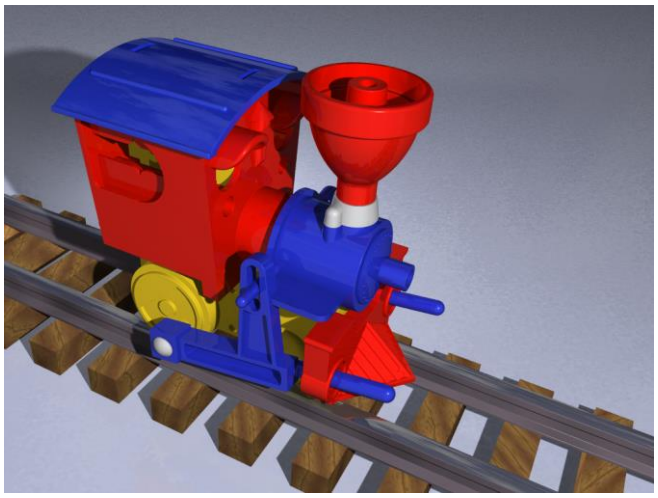
Auxiliary Elements and Operations

Material is a system element assigned as a parameter to each created body. Material helps rendering computer models so that they look like real objects. It contains a list of characteristics of the real material used in actual production. Material has the following parameters: density, luminosity, ambience, etc. The whole body material can be assigned within parameters of any operation.

Apply Material is an operation that assigns a material to specific faces of a body.

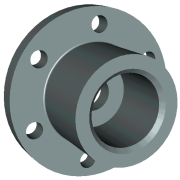
Transformation is a command for defining translation and rotation of an object that change its location and orientation on the 3D scene. This command works with all operations and most 3D construction entities.

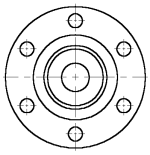
Photorealistic Imaging is a command for creating a file in BMP format containing a photorealistic image of the objects on the 3D scene. This is done with POV-RAY application that is shipped together with T-FLEX CAD. Photorealistic images are also used for creating animated clips.

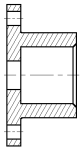


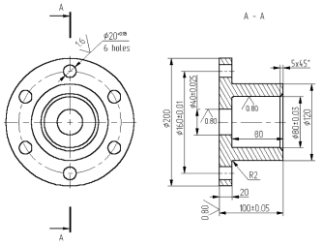
2D Projection

2D Projection projects all bodies of three-dimensional scene or selected bodies or specified elements on a plane. The resulting image is displayed in 2D window. Projection creation accounts for sections for instance when creating cuts. This approach helps avoiding additional constructions in 2D window. Simply create a three-dimensional body, and all necessary views in 2D window will be obtained by projection. Further, 2D projections can be used for laying out drawings.





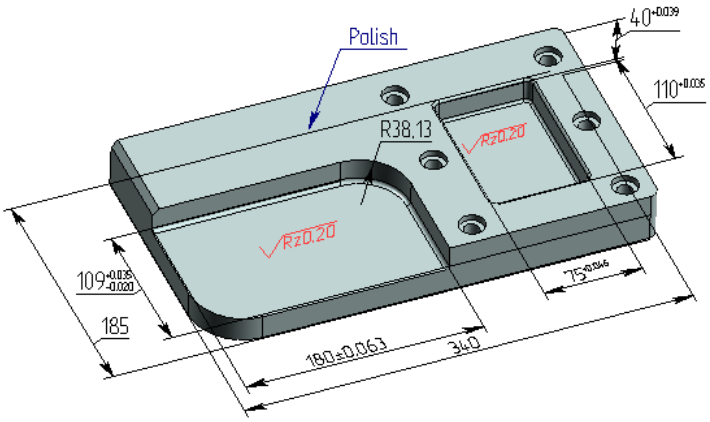




3D Annotation

T-FLEX CAD allows creating drawing annotation (dimensions, leader notes, roughness symbols) directly on faces, edges and vertices of a 3D model.

The capability of creating 3D annotation allows introducing in a 3D model not only geometrical, but also technological process and other information that can later be used when creating drawings by 2D projections, as well as in other applications, for example, in the process design modules or when creating CNC control sequences.



3D Object Rendering

View is a combination of 3D window parameters, such as view point, distance to object, visualization parameters, the type of projection, etc. Particular sets of these settings can be saved in order to quickly set up 3D scene in required configuration later.

Rendering type is a way of rendering 3D bodies in the 3D window.

First type is **Wireframe model**. This way is convenient in that the front elements do not obstruct the ones behind, and the objects inside the body can be seen as well.

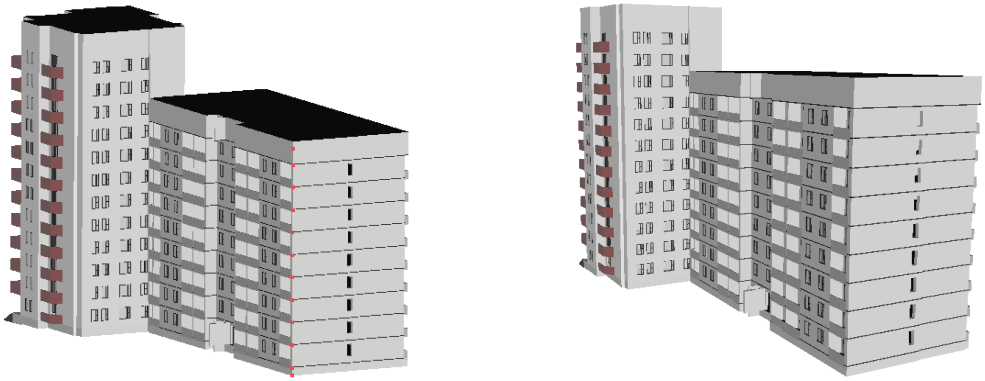
Second type is **Shading**. The faces of the bodies are displayed in a specified color.

Third type is **Shading with Material**. The faces display accounts for the material specified for the whole body, as well as applied to particular faces.

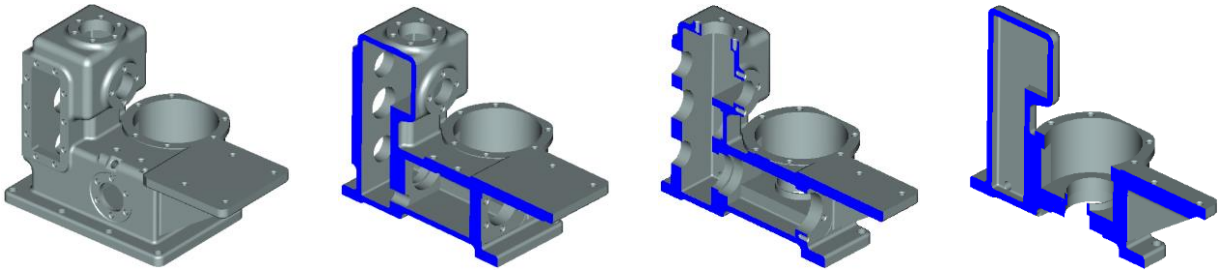
Fourth type is wireframe with **Hidden line removal**. A fast algorithm is used for determining line visibility.

Fifth type is wireframe with **Precise hidden line removal**. The wireframe is displayed with invisible lines removed per the current orientation.

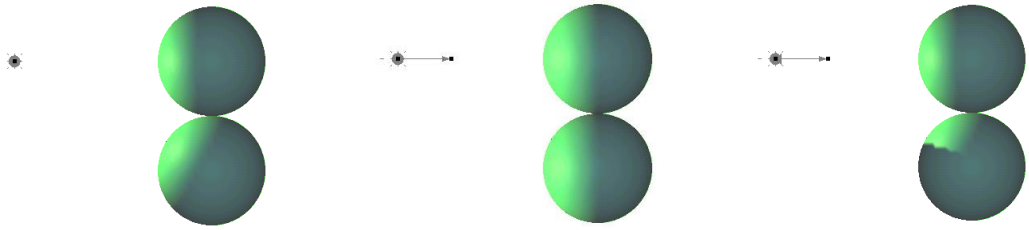
Projection method defines the way objects are represented on the 3D scene. It may be parallel projection without perspective that does not account for the distance from eye to object and the perspective angle, or projection with perspective that does account for the above parameters.



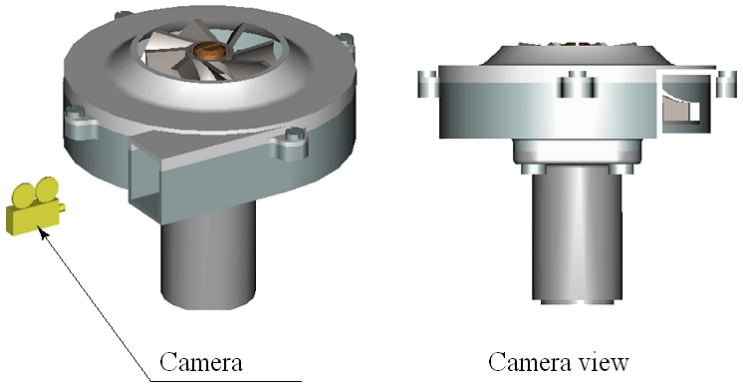
Clip plane is a plane situated parallel to the screen when originally defined. It can move back and forth along the fixed direction and clip out the portion of the objects on the scene before the plane. It is used for visual analysis of inner elements of the bodies, and for selecting objects on the 3D scene located inside the bodies.



Light Source is an element for controlling the illumination of the 3D scene. Originally, there is a single light source coinciding with the viewing point. A light source can be a Spot Light, a Direction Light, and a Projector. The user can create additional lights as needed, vary their intensity and direction, turn on or turn off any of them. Light source is used for creating a photorealistic image.



Camera is an element that defines the viewing point and direction on the 3D scene. Each 3D window has one default camera. One can create additional cameras, select an active camera. The newly created cameras can be moved using special cursor or bounded to a local coordinate system. The viewing direction of a camera can also be changed. This helps surveying inner elements of the scene and create animated clips.



Three-Dimensional Model Animation

Animation of a three-dimensional model is done by the same means as drawing animation, that is the variable-driven model modification. The user assigns the range and the step for the variable modification. Since the real-time regeneration of a three-dimensional model driven by a variable modification may be slow, the animation may not look well in interactive session. This is why a good strategy is to create a multimedia file in *.AVI format (a video clip) for watching the modification animated.

A video clip may be created using photorealistic imaging. In this case, each frame of the clip will be processed by the POV-RAY application.

Approaching Solid Modeling with T-FLEX CAD

General Recommendations before You Begin with 3D Model Creation

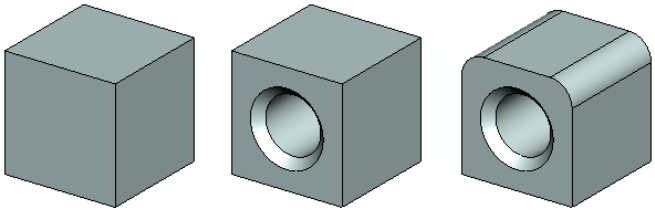
Analyze the design intent before starting 3D model creation. The degree of design automation depends on how well the engineer thought through of the model layout. Parametric design requires additional time investment at the initial stage as compared to non-parametric approaches. However, this brings an unmatched advantage at the later stages, as, for instance, in preparing documentation for various modifications of a product.

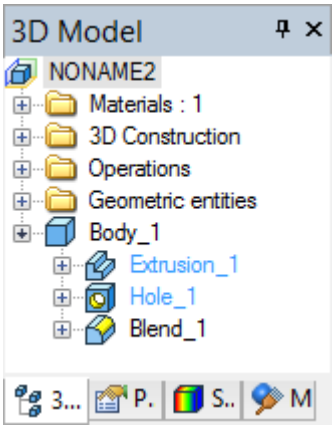
> Due to high flexibility of the system, the same result can be achieved by different means. One of the foremost goals of the designer is to find an optimum solution. Naturally, this depends on how well the designer manages to operate the instruments provided by T-FLEX CAD system.

First, decide what operations you will use for creating the elements of a part. Important is what kind of references and interdependencies will be imposed on various elements. Define the appropriate boundary conditions. Determine what model parameters should be represented by variables. Divide a complex assembly into a set of fragments, 3D pictures, copies, and library elements. Begin with a model creation after realizing a rough plan of the design layout.

Parameterization and Model Regeneration

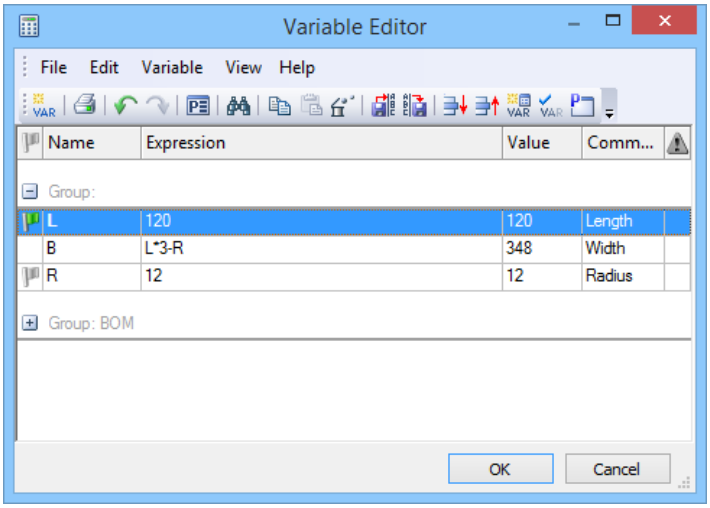
A T-FLEX CAD model is constructed as follows. First, create a new Body using 3D operations based on construction and other auxiliary 3D elements. Next, a base body is used to create other Bodies, which, in turn, undergo modifications and alterations, become subject to certain dependencies, boundary conditions, and so on. Each construction element or operation is recorded in the model history. It is possible to identify parent-child relations between certain elements. The model hierarchy is represented by a tree structure.



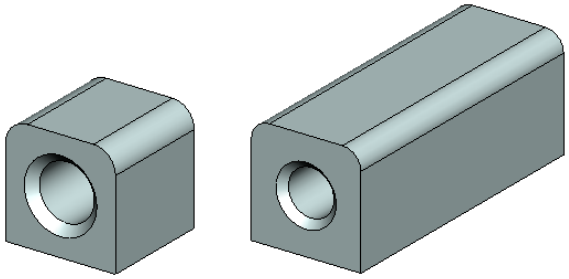


Now, suppose, some parameter of a parent element operation needs to be modified that would affect its geometrical configuration. This would cause the children to adjust their locations according to the parent geometry and the model parametric dependencies. T-FLEX CAD is capable of handling such issues due to its architecture that supports through-model parameterization.

The word “parameterization” implies a wide range of capabilities. It is possible to modify virtually any parameter of any operation at any time. Besides, parameters are not limited to numerical or textual values, as they also can be represented by variables. The variables can be driven by certain expressions, and be dependent on other parameters and variables.



Again, suppose, we changed a parameter of a parent element. Next, the model regeneration is launched manually or automatically, traversing the model structure and recalculating the model with the updated variable values.



T-FLEX CAD supports full and partial regeneration. Full regeneration is necessary for updating the whole drawing and the model. This causes recalculation of all objects from scratch. Partial regeneration saves time. The system automatically analyzes what objects were modified since the last regeneration, and recalculates these objects and their children only.

To update a model use a selective regeneration command,

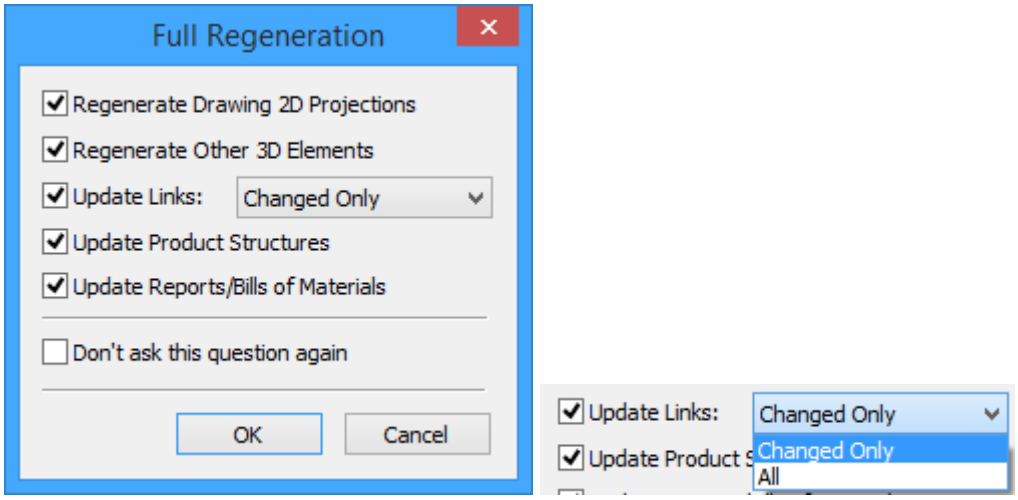
Icon	Ribbon
	Parameters → Tools → Regenerate
Keyboard	Textual Menu
<Alt><F7> or <3G>	Tools > Regenerate

The full regeneration is invoked by the following command,

Icon	Ribbon
	Parameters → Tools → Full Regeneration
Keyboard	Textual Menu
<3RG>	Tools > Full Regeneration

The three-dimensional model is fully regenerated after calling the command. This command regenerates each element, unlike the **Regenerate** command, which regenerates only changed and dependent elements. The image in 3D view window is automatically updated after the regeneration.

The following window appears after calling the command:



If the flag **Regenerate drawing 2D projections** is set, only changed 3D elements (optimal regeneration) and all projections will be regenerated.

If the flag **Regenerate other 3D elements** is set, all 3D objects are regenerated.

> If you set both flags, the full regeneration is performed. If both flags are not set, the optimal regeneration is performed.

The flag **Update links** allows to update files used in the current document (fragments, images, data bases). You can set to update **Changed only** links or **All** links. The command **UL: Update** links is indirectly performed in this case.

The flag **Update product structures** allows to update all product structures created in the document.

The flag **Update reports/bills of materials** updates all reports and bills of materials created in the document.

When you set **Don't ask this question again** this window will not appear next time after calling the command.

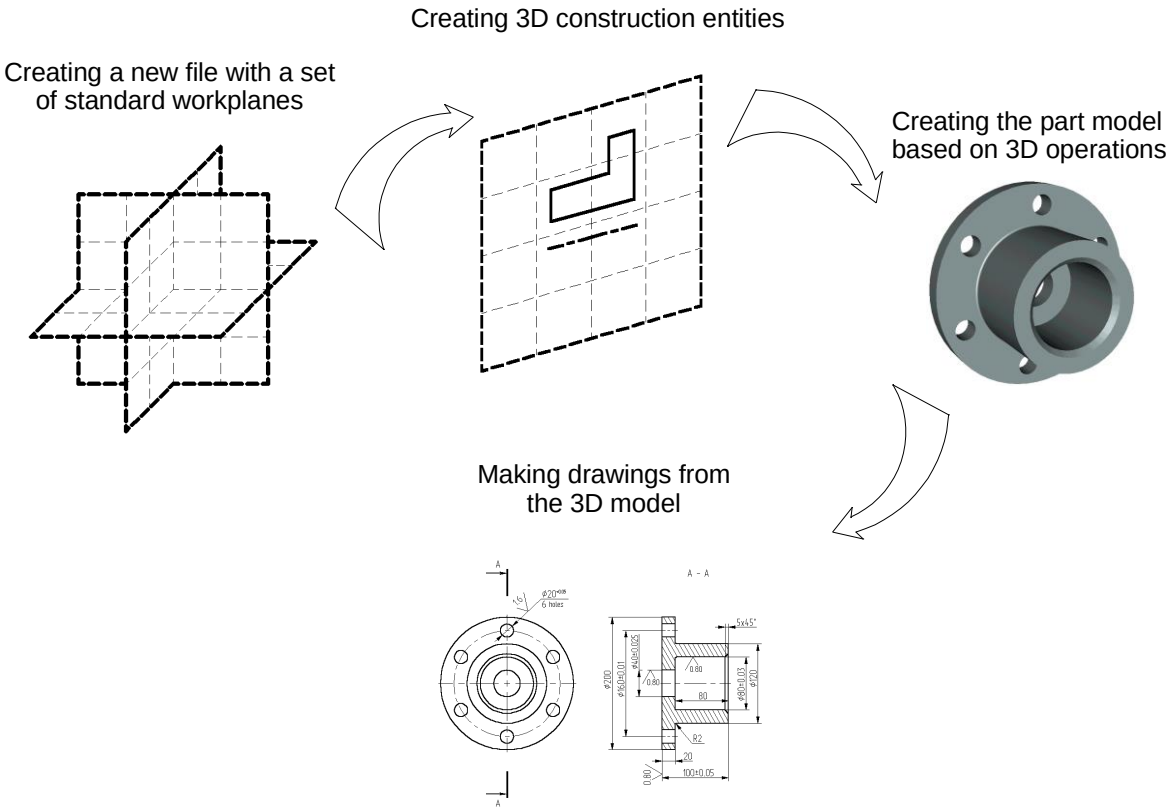
Three-Dimensional Model Creation Techniques

T-FLEX CAD supports various techniques of 3D model creation. The **mainstream** design principle implies direct approach to a 3D model creation "from scratch to workplane to sketch to model". Another approach implies use of pre-designed 2D drawings or auxiliary 2D construction. This technique can be called "**from 2D to 3D model**".

Whichever technique is used in a 3D model creation, the same operations form the body geometry. The difference is in the ways of creating the 3D construction entities.

The Mainstream Approach

Charted below is the mainstream design approach workflow.



Begin a document creation by selecting an appropriate prototype of a 3D model (`3D model.GRB`). Use the command **F3: Create New 3D Model**. This brings up a 3D window with a standard set of workplanes. Now all is set for starting with a 3D model.

> The mainstream 3D modeling approach does not require the 2D window. All auxiliary elements, including profiles, nodes, 3D paths, etc., can be constructed directly in the 3D window using the same 2D drawing tools.

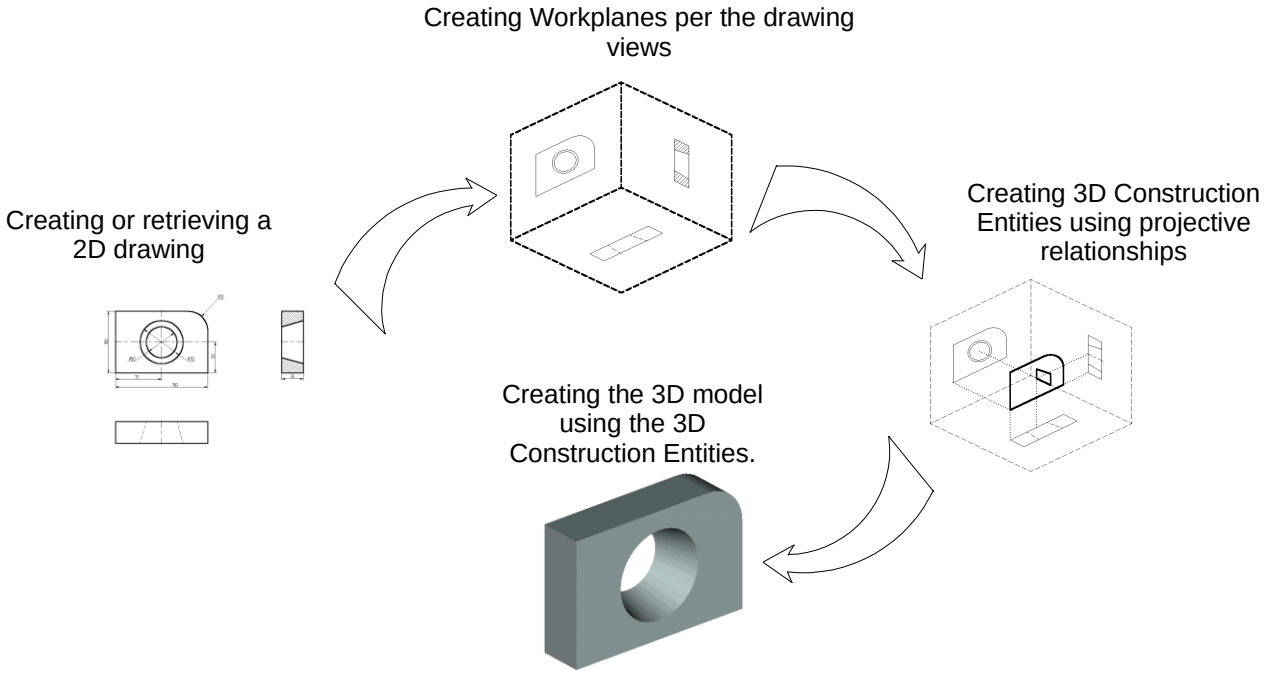
Normally, an operation requires a certain set of auxiliary 3D construction entities to be completed. To create a 3D profile, first select a workplane or a flat face as a base, then all 2D drawing commands become accessible. Just like in 2D, one can draw on the active workplane new lines, contours, etc. Parametric dependencies can be automatically introduced into the model at this early stage. A 3D operation, such as, say, an extrusion, can be invoked without leaving the drawing mode. The system would automatically create a 3D profile based on the just created graphic lines. This minimizes the number of user actions required for completing an operation. Consider, for instance, the "Rotation" operation. The required axis (a dashed line) and the contour can be created on the fly directly on the workplane. Preview is always available to evaluate the result of an operation. Preview activates automatically for the operations that do not require extended time for computations. In other operations, it is invoked by a special command.

The bodies created at an early design stage may later be used in other operations, such as in Booleans, blends, tapers, etc.

The completed 3D model can be used for generating drawings, if desired. Create required projections, sections and cuts that can be used as references for the necessary elements of the drawing layout, dimensions, etc.

"From 2D to 3D" Technique

The following chart illustrates a design technique "From 2D to 3D".



Often, a 2D drawing of a part may already be available before beginning with the 3D model creation. The proposed approach to model creation is preferable in such a case. At the initial stage, it is good to have a 2D drawing of the part that contains all necessary views. The part views are better be laid out in accordance with the projection rules, although this is not a strict requirement. Certain parametric relationships of the pursued parametric 3D model can be introduced at the 2D drawing stage. The first step in 3D model design is creation of workplanes. Often, two or three orthogonal workplane are sufficient that represent the front view (the front plane), the top view (the horizontal plane), and the view from left (the side plane).

Next, the necessary 3D construction entities are introduced, such as 3D nodes and 3D profiles. 3D nodes are used as references for 3D profiles, vectors of extrusion direction, rotation axes, etc. In case a workplane can't be activated, the required profile can be defined based on a 2D hatch. The hatch must reference an actual drawing.

Besides the profiles, 2D views can be used for constructing 3D nodes from the 2D nodes, and 3D paths from the 2D paths and other construction entities.

It is thinkable of combining the two techniques described above. Note that three-dimensional modeling is a creative process that does not necessarily implies a sole way towards a particular solution. The user is provided the necessary set of tools, while it is up to him to make a choice of a most effective approach to realizing the design intent.

Creating Assemblies

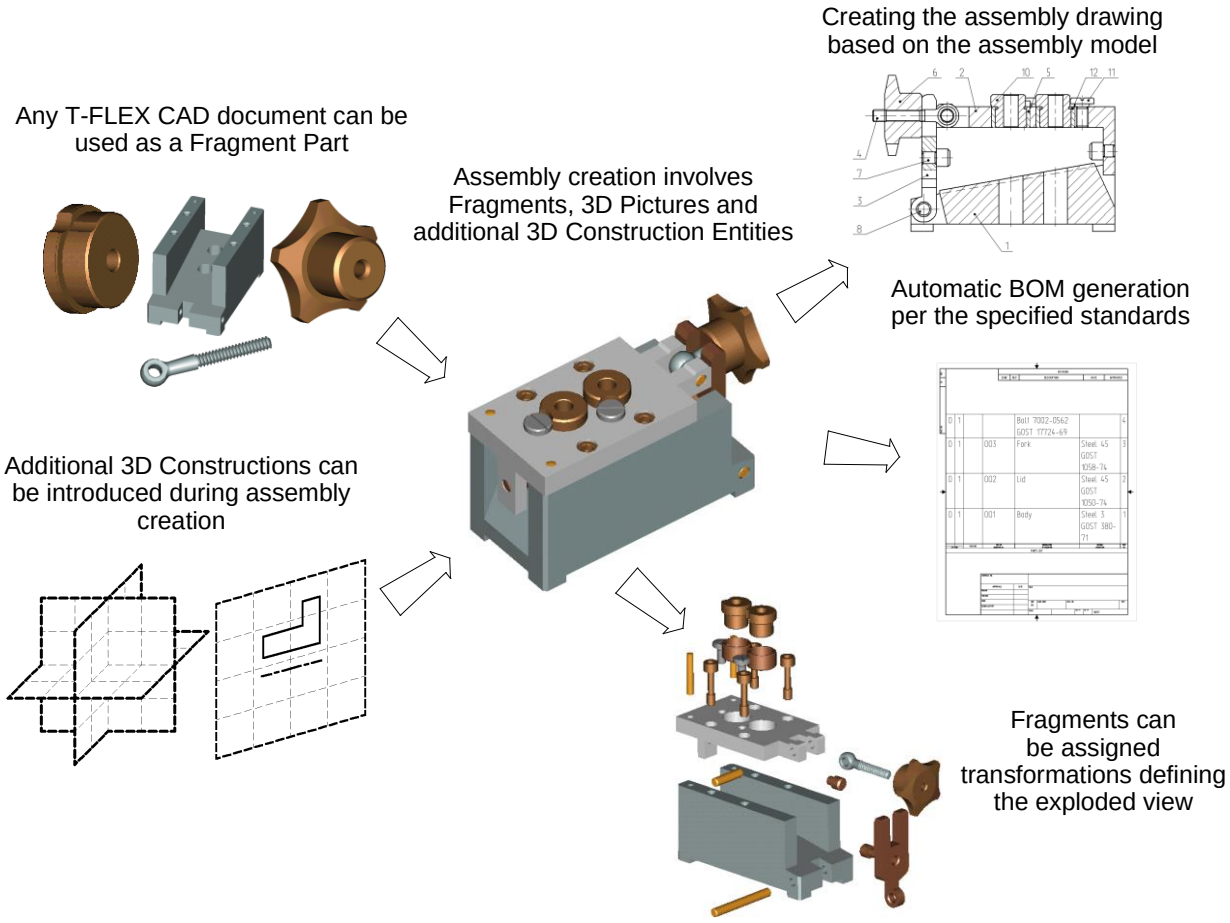
Any T-FLEX CAD document that contains a three-dimensional model can be inserted into another 3D model as a fragment. Thus composed model is called **assembly**. External models, imported from other systems in an appropriate format, can also be used in assemblies.

The component architecture of T-FLEX CAD assemblies has certain advantages. Thus, one can create libraries of parametric elements, to use them later in assembly creation.

An assembly document keeps the references to the fragment files. Once a fragment file is modified, it automatically becomes updated, causing modification of the respective assembly component. Any fragment may have external variables that drive the parametric relationships of the part. At any time, one can either modify the original part file of the fragment, or specify different values of the external parameters of the fragment. In the latter case, the part file is unchanged, while the assembly component – the fragment – is recalculated per the new external parameter values. A special functionality helps keeping various sets of the assembly parameter values thus facilitating quick loading of a desired variation.

Each fragment file provides storage for BOM data. If this data is entered in each fragment file then the assembly BOM can be output automatically.

The assembly model can be used for creating drawings by creating the necessary views, cuts, sections and applying dimensions and other layout elements.

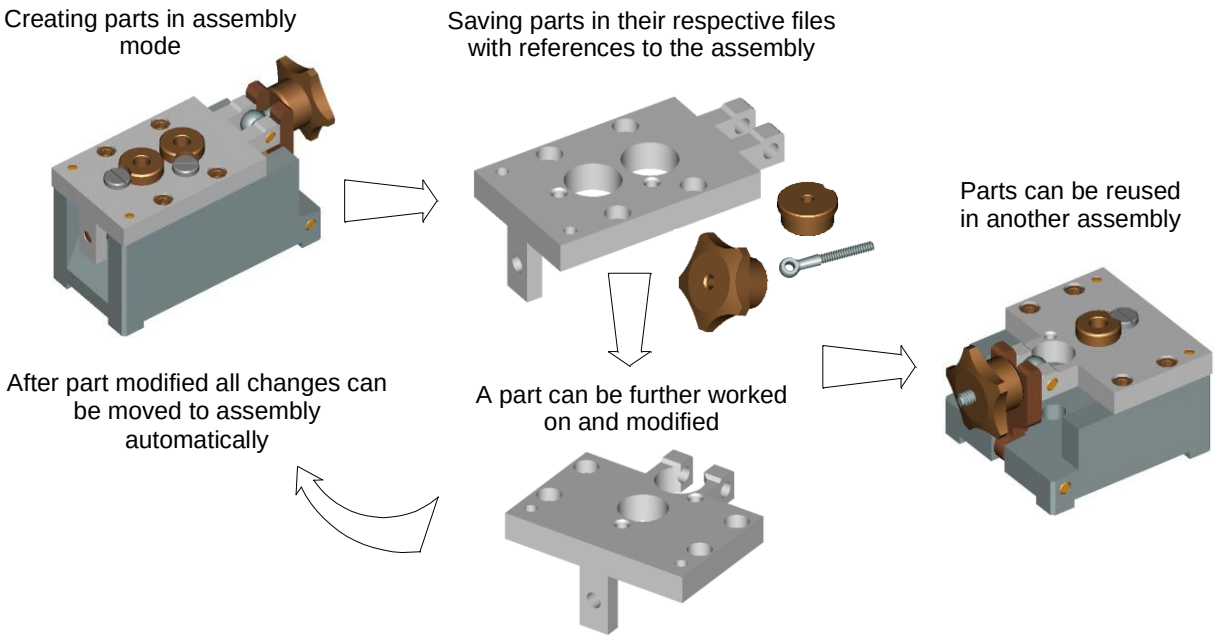


"Top-down" Assembly Modeling

There is an alternative approach to assembly creation, as opposed to the typical one briefly described above. T-FLEX CAD supports new part creation based on any geometrical and topological elements of the parent assembly. In this way, it is no longer necessary to specify placement conditions and the part mutual configuration. A part is automatically placed according to the references to elements used in the part definition. The parametric relationships among the assembly elements are saved with the model. Once a dimension or a part placement is modified, all related elements of the assembly will automatically adjust.

When working with a part in assembly mode, all elements that are not referenced by the part are shown transparent. Snapping works with all assembly elements. Any assembly element may be referenced at any moment.

The part is saved in a separate file. The file may be opened apart from the assembly and worked on separately. The assembly references are meanwhile preserved.



3D Model Rollback Mode

If you need to make a correction to an already created model, you can use the 3D model rollback mode at a certain operation level. This function is helpful in the cases when you need to do additional work in the middle of the model tree.

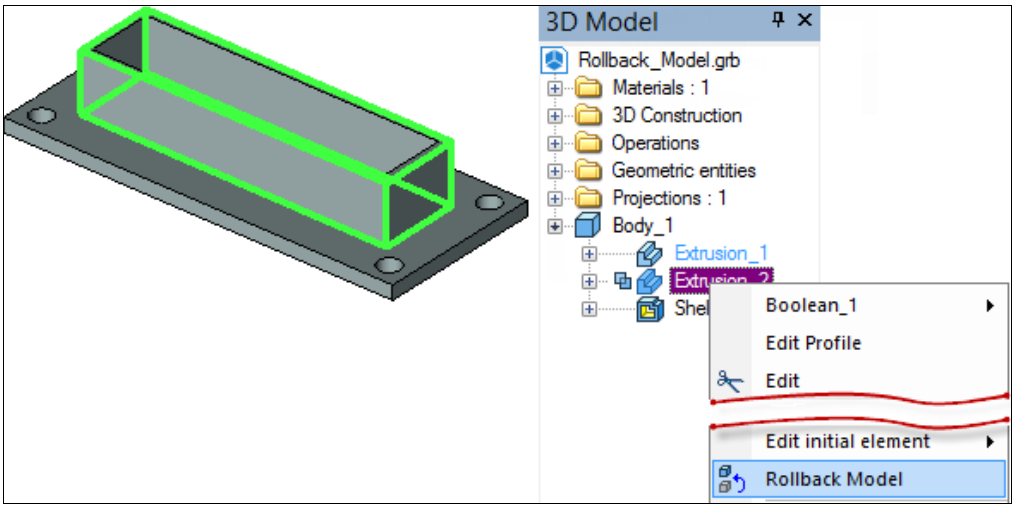
In the rollback mode, the 3D model returns to an earlier stage of its creation. Elements and operations that follow after the rollback point in the model's history become hidden in the rollback mode and are inaccessible for selection. In this state, those are marked with semitransparent icons.

All elements and operations created in the rollback mode are automatically inserted in the 3D model tree between the operation, upto which the rollback was made, with the successive operations unloaded from the scene. It is possible to do a series of rollbacks.

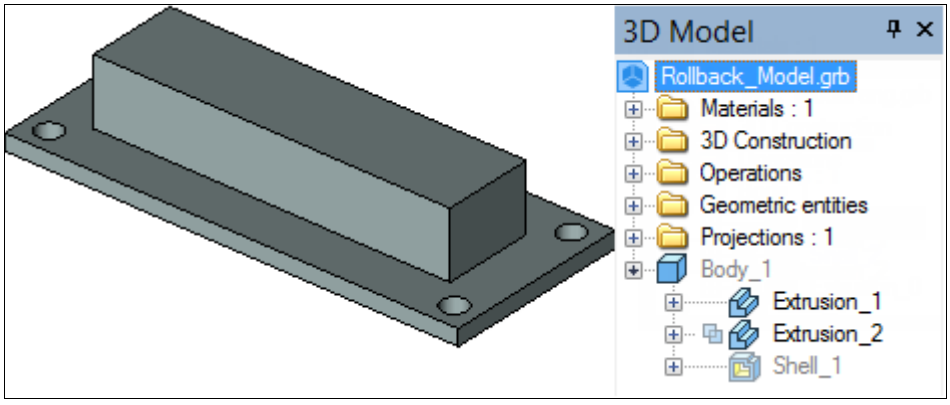
Upon finishing the rollback mode, the system automatically rebuilds internal model relations to restore the later operations. If for some reason the introduced changes result in errors in the model, then the system will offer restoring the model state before the rollback. This protects the user from introducing invalid modifications to the model.

The rollback mode is invoked and finished using context menu commands of an operation in the model tree or in the 3D view window. The rollback will be up to the level of the operation, from whose context menu the command was called.

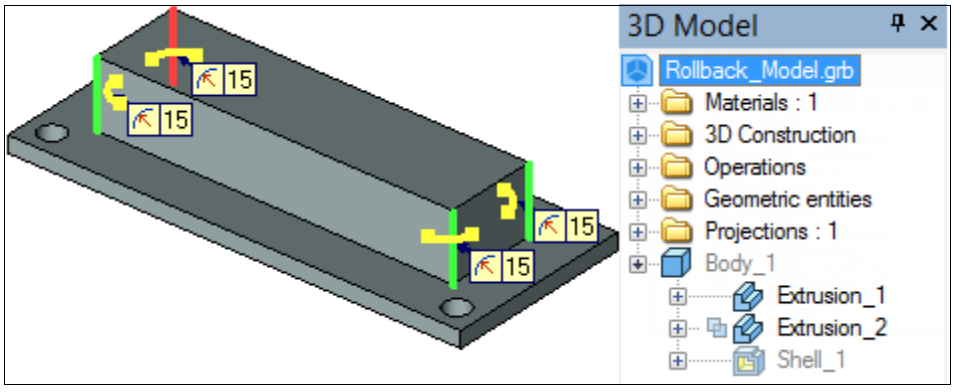
Let's review the use of a rollback on a simple example. In this model, it is necessary to add a blending on corner edges.

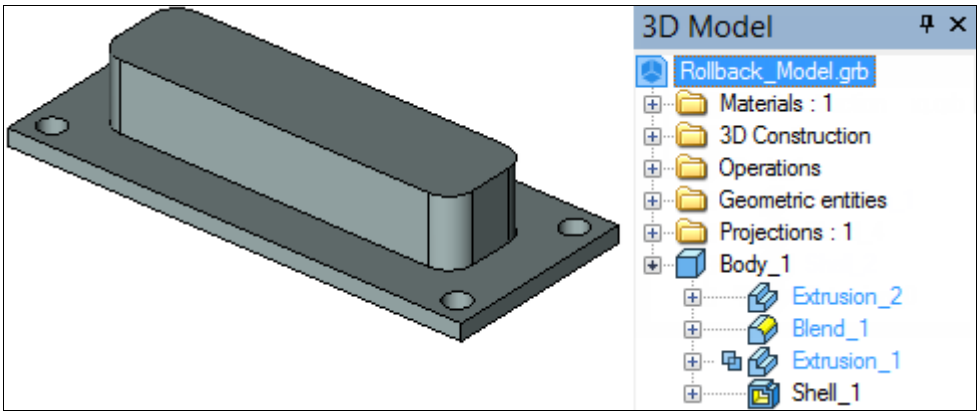


We perform the model rollback up to the base extrusion operation. In this case, the operation Shell_1 is unloaded from the scene.

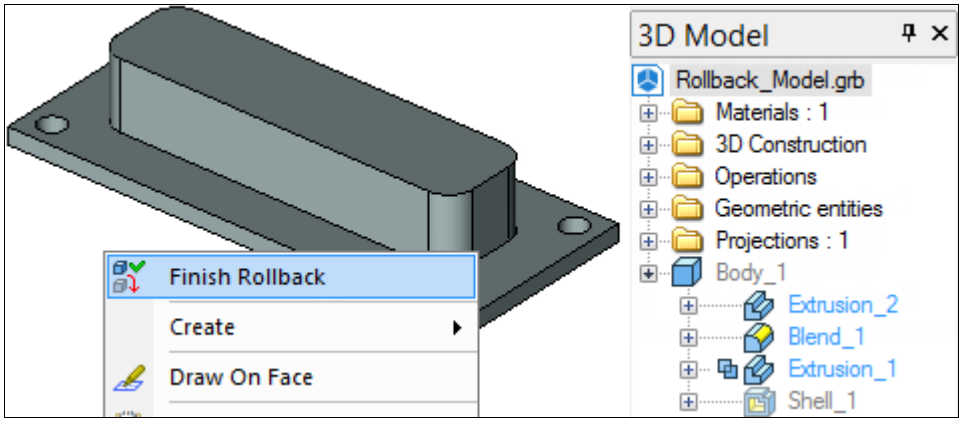


Then we make an edge blending (the Blend_1 operation is added in the model tree).

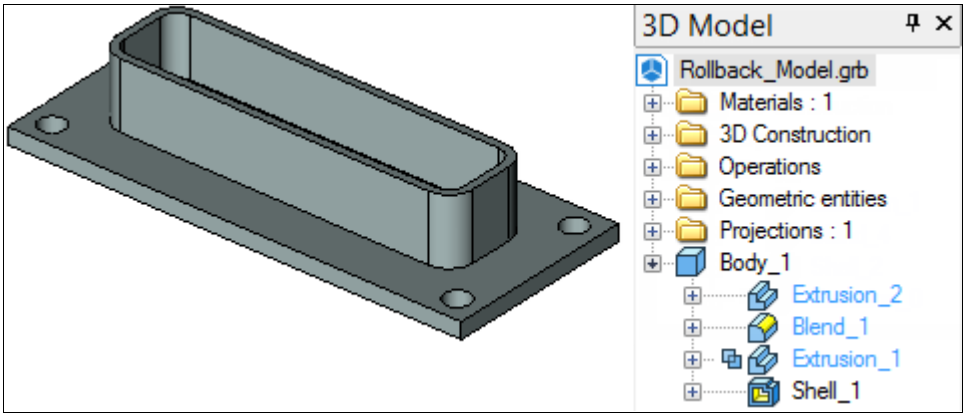




We then finish the rollback.



As a result, we get the model presented at the following figure.



T-FLEX CAD System Operation Tips

Getting Help

There are several ways of getting answers to the questions arising during the work process, as follows:

Help on the current command can be accessed by pressing <F1> key. When no command is active, pressing <F1> invokes the help.

While within a command, limited information is provided as prompts and hints in the status bar.

Pop-up help appears by the icon buttons on a toolbar and by the graphic elements when the cursor is briefly held pointing at an element or item. The pop-ups report the command or element name, the type or parent operation of the element. Pop-ups messages are duplicated in the status bar at the bottom of the system window.

Creating a New Document. Using Prototype Templates

A new project begins with setting up the new document. Depending on the design intent one can select appropriate initial options for the new model. To begin with a 2D drawing creation or other 2D construction in 2D window use the command:

Icon	Ribbon
	Get started→ Files → Drawing
Keyboard	Textual Menu
<FN>, <Ctrl><N>	File > New > Drawing

This will bring up the 2D window.

When beginning directly with a 3D model creation, use the command for creating a new file,

Icon	Ribbon
	Get started→ Files →3D Model
Keyboard	Textual Menu
<F3>	File > New > 3D Model

This brings up the 3D window with the standard set of workplanes.

The following commands are used for creation of a new assembly document:

Icon	Ribbon
	Get started→ Files → Assembly Drawing
Keyboard	Textual Menu
<>	File > New > Assembly Drawing

Icon	Ribbon
	Get started→ Files →3D Assembly
Keyboard	Textual Menu
<>	File > New > 3D Assembly

An additional record about assembly drawing exists in the product structure of the assembly document.

A new document is created using one of the prototype files whose names are defined via the command **Customize** > Options. These files may contain some useful elements and settings that will be created or set in the new document. In the case the new document settings need to be changed, edit the prototype file (such as, for instance, the ANSI drawing prototype file, named `ANSI.GRB`, or a 3D model prototype `ANSI 3D Front Right Bottom Workplanes.GRB`). These prototype files must be present in the folder *…T-FLEX CAD\Program\Template*.

It is possible to create any number of prototype files and use them as desired. For creating a new file from a prototype, use the dialog "New From…" invoked by the command **FP: Create New Document Based on Prototype** (**File > New from prototype)**. In addition to that, this dialog is available in the window "Start Page".

Each folder in the directory *…T-FLEX CAD\Program\Template* becomes a tab in the dialog box. All the prototype files that exist in a respective folder are displayed on the selected tab. Thus, the prototype files should be placed in the directory *…T-FLEX CAD\Program\Template\<The respective folder of the prototype file>*.

When you create a new document, you can choose one of four prototypes.

To save a file as a prototype use the menu command **File > Save as Prototype**. Use the appearing on screen dialog box to assign a name to the would-be prototype file. One can create new folders (turning into tabs on the dialog box) for prototype files as desired. One can also delete any prototype files and folders, except the folder "Common".

Mouse Interface. Context Menu

Part modeling in T-FLEX CAD is performed mostly by mouse. The keyboard is only used for inputting numerical values, names and command accelerators (see below).

Using left mouse button

Pointing cursor at an icon and pressing launches the command represented by the icon.

A command can also be launched by pointing cursor at a textual menu item and pressing.

Pointing cursor at an object on the 3D scene and pressing selects the object.

Pointing cursor at a 3D construction element or operation and double clicking invokes the dialog box "Element/Operation Parameters".

Pressing and drugging spins the 3D scene. The cursor must be within 3D window bounds. The system should not be in drawing mode.

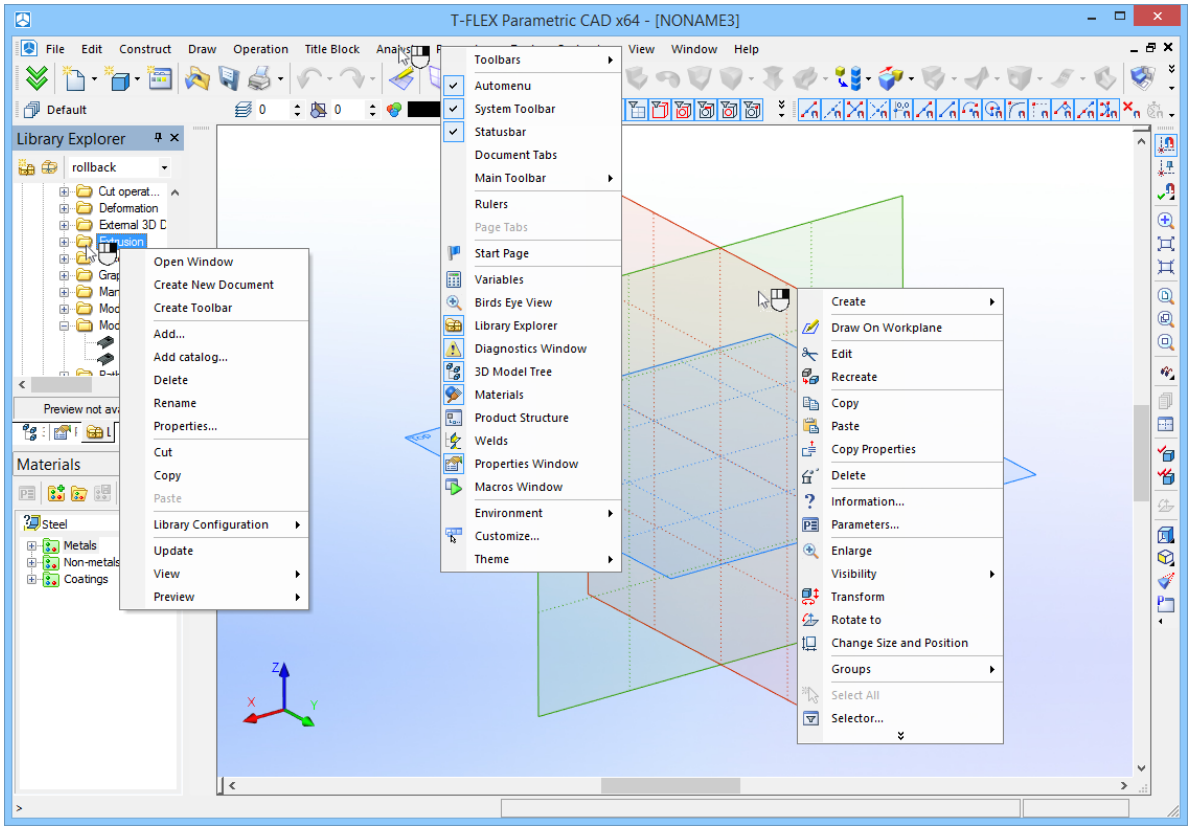
Drag&Drop can be used for managing libraries and customizing toolbars and dialog boxes. Point the cursor at an element, press and hold, and drag the element to a desired location while holding the button. For more information, refer to the appropriate sections of documentation.

Using right mouse button

While within a command, pressing cancels the last active command.

Pressing while selecting elements on the 3D scene unselects the last selected element.

If no command is active, pressing right button invokes the **context menu**. The menu is composed of the commands currently available for the particular element. The content of the context menu will be different depending on where the cursor is pointing. These could be the area of the 3D scene, a model element, the command area of T-FLEX CAD, such as a dialog box, a menu, or a toolbar. To launch a command point the cursor at the corresponding line of the context menu and press.



- Context menu can also be invoked while working with dialog boxes. See the topic “Context menu for the dialog fields”, the chapter “Customizing Drawing” in the volume “Two-dimensional design”.

The described right mouse button functions are set by default, but can be customized. To do so, call the command **Customize** > **Options** (**Preferences** tab). For more information, refer to the chapter "System Customization" in the volume "Two-dimensional design".

In "3D drawing" mode mouse manipulations are the same as in 2D window.

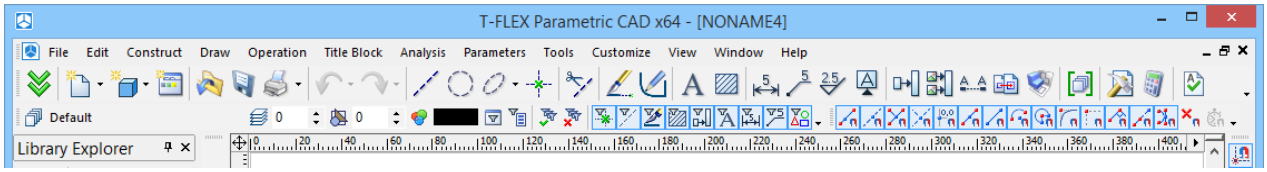
Using mouse with wheel (IntelliMouse)

- Spinning the mouse wheel zooms in and out.
- Dragging the mouse with the wheel depressed pans the 3D scene.
- The mouse wheel can also be used for alternative selection from a set of objects under the cursor. Alternative selection mode activates after two-second delay after pointing cursor at the object. Scrolling through the list of objects is done by spinning the wheel.
- Spinning the wheel performs the standard functions of scrolling in the appropriate field of dialog boxes.

Calling Commands from Keyboard, Using an Icon, from Textual Menu

There are various ways of calling T-FLEX CAD commands. Firstly, a command can be selected via an icon on a toolbar using the mouse.

A command can also be launched from the textual menu. All T-FLEX CAD commands are grouped in a certain way. Each group has an entry item in the menu.

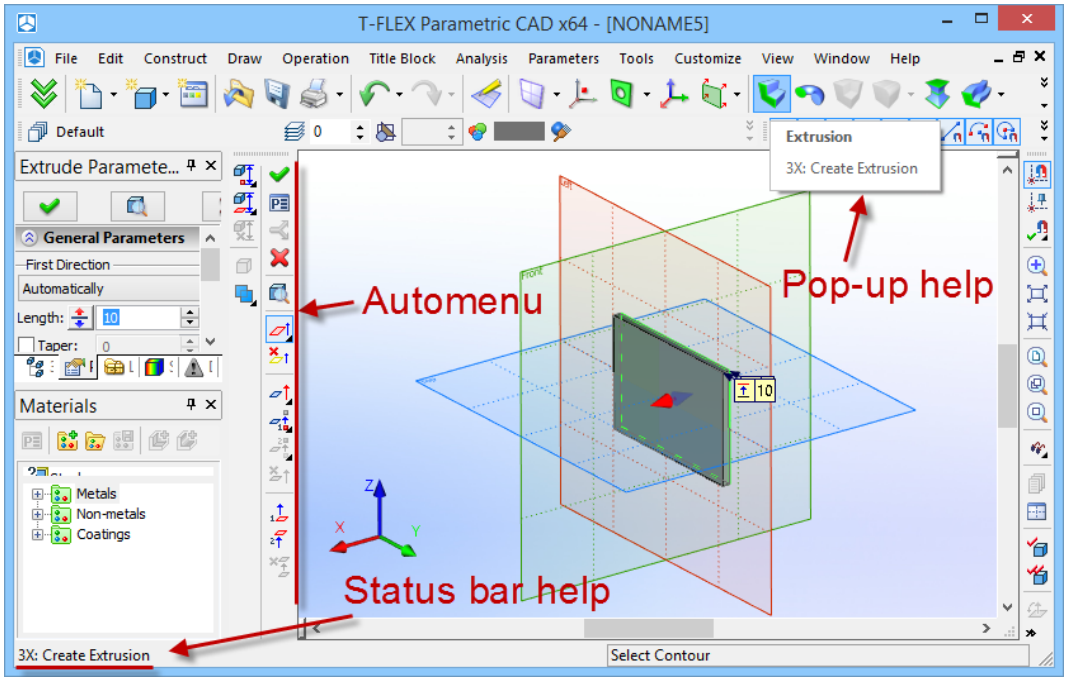


Thus, for instance, the commands that create 3D operations are grouped under the entry "Operation". The drawing commands are combined under the entry "Draw". The 2D and 3D construction commands are united in the "Construct" group. The file managing commands are united in the menu "File". Other groups include the editing commands ("Edit"), the variable managing commands ("Parameters"), utility commands ("Tools"), the system and the model customization commands ("Customize"), the visualization controls ("View"), the window management ("Window"). The system reference and information group of commands is under the "Help" entry.

Most T-FLEX CAD commands are bound to function key combinations (simultaneously pressed), or keyboard accelerator sequences (pressed in order). The textual menu item buttons contain keyboard accelerator sequences for the commands next to the command name, whenever defined. The keyboard accelerator can be changed for any command. For detailed description, refer to the "System Customization" topic of the chapter "Customizing Toolbars and Keyboard", "Keyboard" tab, in the volume "2D design".

As said before, some commands can be launched by typing the keyboard accelerator sequence. Particularly, this is supported for 3D model and drawing creation and editing commands. The keyboard

sequence and the command name is shown on the pop-up help and in the message field of the status bar.

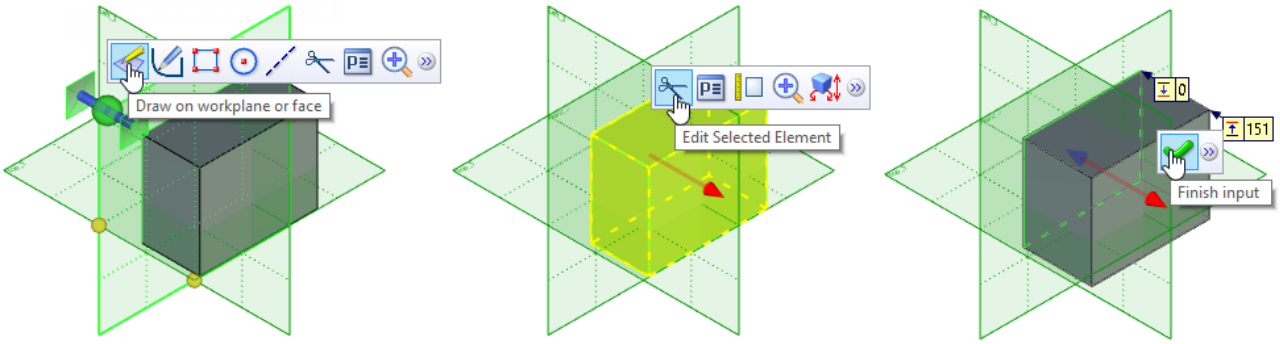


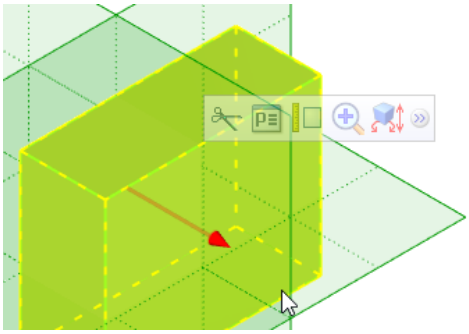
Just like the 2D subsystem of the T-FLEX CAD, each 3D command has an additional set of options and subcommands accessible via the automenu – a toolbar containing subcommand icons, and function key combinations. The function key combinations are displayed in the pop-up help widgets.

Certain commands are best accessed via the context menu. This menu is available after selecting one or several elements on the 3D scene, such as faces (see diagram below). The context menu contains the list of commands available with the selected set.

You can use **dynamic toolbar** to call the commands. The toolbar appears upon pressing .

The toolbar provides quick access to frequently used commands and options.





If you move cursor from the toolbar, it becomes semitransparent. If you move it too far from the toolbar, the toolbar disappears.

When you press button, the context menu appears. Here you can select some operations that are not included in the toolbar. When you select one of them, the operation will be added to the toolbar. Thus the toolbar consists of favorite commands.

The button of operation creation toolbar includes automenu for the current operation. When you select the automenu options, they are not added to the toolbar, because all operations have different automenu options set.

You can change dynamic toolbar display settings on the **Dynamic toolbar** tab of **SO: Set system options** command.

Confirming actions of 3D element creation

Unlike 2D elements, a three-dimensional element creation takes several steps. Not all of these steps are required. To complete an element definition, use the provided option "Finish input" by clicking on the icon or pressing <Y> key. This option becomes accessible only after the required minimum sequence of actions has been completed.

When calling a command, make sure about the command exiting option. Some commands remember the current state while the others revert to the initial settings.

Canceling and exiting a command

Exiting a command is done by pressing <Esc> key or . Alternatively, use the automenu icon.

If the system has entered a 3D command without doing anything (nothing was selected), then calling another 3D command makes the former command quit. However, if something has been selected within the former command then the newly launched command does not close the former one and becomes nested. Once done with the nested command, the system returns into the former command. This is different in 3D mode from the 2D mode operation. The above does not affect the 3D drawing mode. To return to the command-waiting mode, close subsequently all active commands. Instant exiting from all nested commands is done by simultaneously pressing <Shift><Esc>. A 3D command quits automatically upon calling any 2D command.

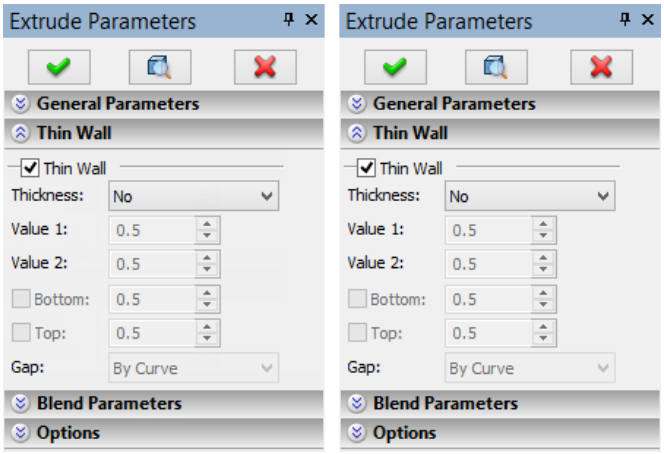
Setting up Parameters of a New Element

The parameters can be assigned to an element being created or edited at any stage of the creation or editing. To specify an operation parameters use the **Property window** that supports transparent mode, or the **Parameters dialog box** that always requires confirmation before the input takes effect.

The parameters dialog is necessary for specifying general properties of an element or operation, such as color, level, material, etc., as well as the transformation parameters. These properties are uniformly

defined for all construction elements and operations in T-FLEX CAD. The ways of handling these properties are described in details in a separate chapter of this volume. This dialog box can be used for editing operation properties without calling the operation editing command. This is especially helpful when the operation being edited is deep in the model history. Otherwise, the model rollback would be required up to the edited operation, consuming considerable time and system resources. Now, it is sufficient to select the operation of interest in the model tree and call the parameters dialog using, for instance, the context menu. The parameters dialog box may contain several tabs. The common parameters and the transformation parameters are always present and are located on their own separate tabs. These parameters are common for most of the system elements; therefore, their handling is isolated into a separate chapter of this volume. All other tabs in the dialog box repeat those in the property window. Due to this, the related chapters will describe main parameter management based on the example of the property window.

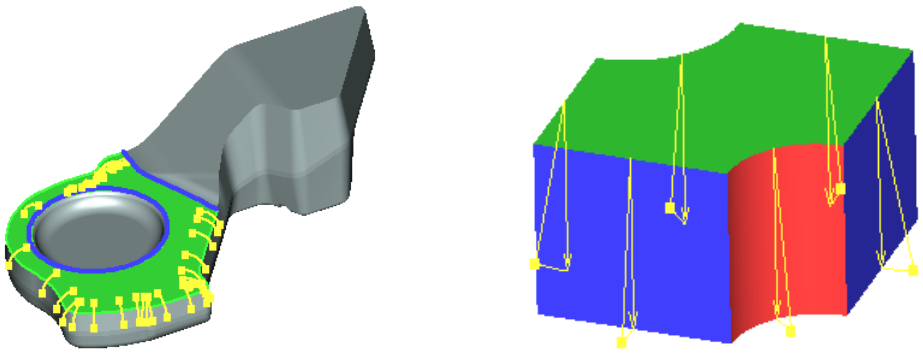
The property window is typically divided into several sections. The number of sections depends on a particular operation. Each section of the window can be expanded or collapsed using the buttons and . Once expanded within a particular command, this section state will be remembered for this particular command, and automatically accounted for in the subsequent command calls.



There is a field to the right of a parameter input box that displays the current parameter value. This is helpful in the cases when the parameter is defined via a variable or an expression, or else when defined "from status" (that is, uniformly for the whole document).

Some automenu buttons are duplicated in the property window, such as the "Accept", "Exit" and "Preview" buttons. The property window works together with the automenu. Pointing cursor at some areas of the dialog box causes activation of the respective options in the automenu, such as, say, selection options.

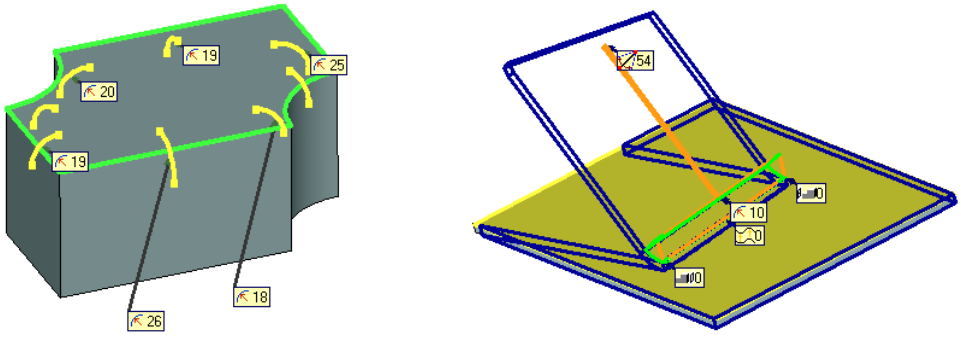
Special auxiliary graphic objects called draggers are introduced for dynamically controlling parameters of a 3D operation being defined. These appear automatically while within a command for creating or editing a 3D operation or a construction entity. Dragger manipulation is instantly reflected in the property window. The mouse-controlled dragger input defines the numeric parameter values of the operation. There can be several draggers on the scene simultaneously, providing control over various operation parameters. As an example, the "Bend" operation provides draggers for the bend angle, radius, offset and the two shifts. The "Blend" provides control over the rounding radius, etc. The values of the driven parameters are simultaneously updated in the property window of the operation being created or edited.



Examples of draggers in the commands “Blend Edge” and “Taper”

Marks with the current parameter values are displayed on the screen for each respective dragger. Besides the current value, a mark has an icon indicating the type of the displayed parameter. As the parameter value is modified via the property window or a dragger, the value on the mark changes dynamically.

If it's too many marks and those interfere with working in the 3D scene, you can hide them or move sideways. Hiding marks is done in the current 3D command parameters. To move a mark sideways, point the cursor to the mark's icon. Once the cursor turns to , depress and, while holding the mouse button, drag the mark image to the desired location. The moved mark will be connected to the dragger with a leader line.



Marks serve not only to track the current parameter values, but also to modify them. To do that, click on the parameter value displayed in the mark. The mark will go into the editing mode of the parameter it controls.

Preview

The following option is provided with each operation to see the result without completing the operation:

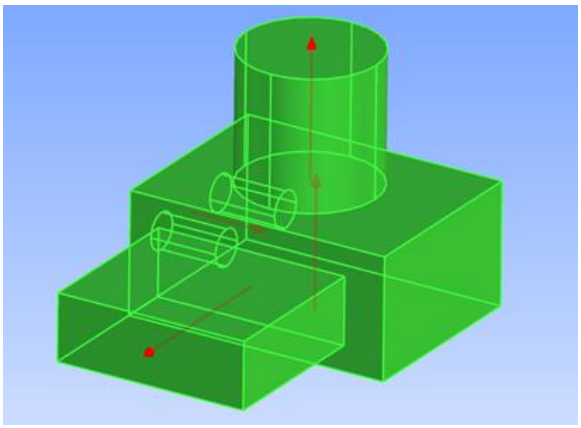
	<F5>	Preview Operation Result

This option becomes accessible once the required parameters of the operation have been defined. Once this option is set, the displayed body starts to reflect the operation per its current parameters. In the case

of satisfactory results, complete the input by pressing the icon . Otherwise, turn off the preview option by pressing on the icon once more, and adjust parameters as desired.

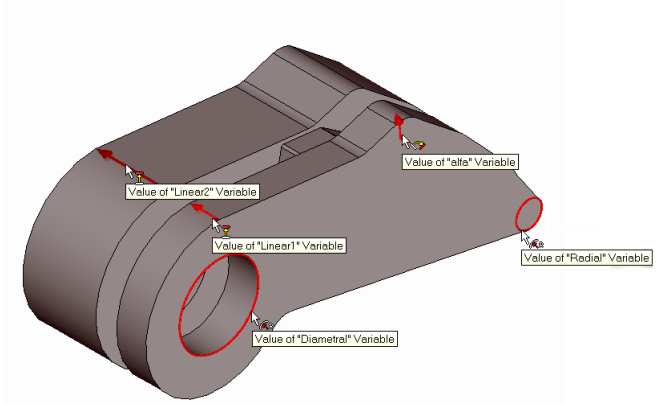
Editing Parameters of Operations with the Help of Draggers Outside of Commands

When selecting a Body in 3D window, in addition to the Body itself the draggers of the operations taking part in its formation are marked on the screen. The draggers allow us to directly edit parameters of the operations that form the Body in a transparent mode without entering into the editing command.



Draggers of External Variables

To specify the values of external variables of a 3D model, the user's draggers can be created.



Draggers of external variables can be used in two ways:

- to change parameters of the model in the scene. The command of controlling the draggers visibility in the scene – «**Show/hide draggers**» is added to the toolbar of objects visibility control in the scene;

– to change values of external variables on the fragment upon insertion of the model into the assembly.

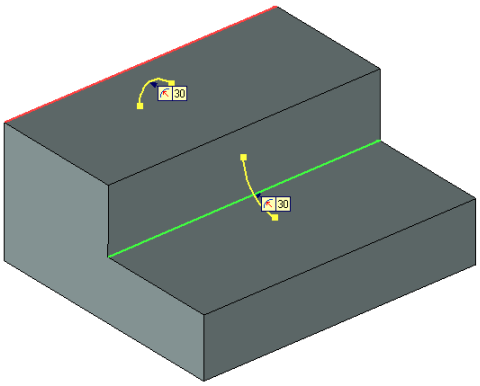
To create draggers it is required that the external variables that control parameters of the given model be present in the current 3D model.

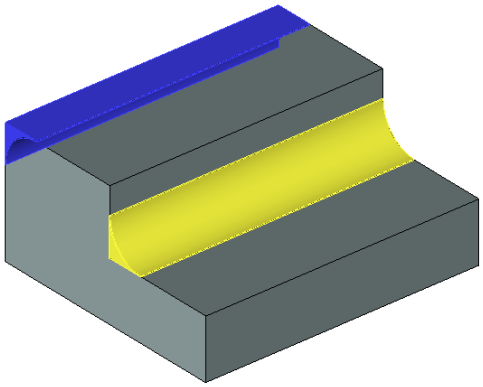
Preview of Operation Result

In the commands for creating and editing the modifier operations, there is an additional mode allowing a user to see the changes produced in the model by the current operation:

	<Ctrl+F5>	Preview Solid Changes

In this mode, the parts of the volume which will be added to the source body are shown with a yellow color, and the parts of the volume which will be removed from the source model are shown with a blue color.





Similar to the option for previewing, the option is available only when all necessary elements for creating operation have been specified.

The Groups of T-FLEX CAD 3D Modeling Commands

Workplane and worksurface management commands

	<3W>	Construct Workplane
	<3EW>	Edit Workplane
	<3SU>	Construct Worksurface
	<3ESU>	Edit Worksurface

Commands for auxiliary 3D element creation

	<3N>	Construct 3D Node

	<3PR>	Construct 3D Profile
	<3PA>	Construct 3D Path
	<3SE>	Construct Section
	<3O>	Create Local Coordinate System
	<3CA>	Construct Camera
	<3H>	Create Light Source
	<3PP>	Construct Pipe Path

Commands for auxiliary 3D element editing

	<3EN>	Edit 3D Node
	<3EPR>	Edit 3D Profile
	<3EPA>	Edit 3D Path
	<3ES>	Edit Section
	<3EO>	Edit Local Coordinate System
	<3CC>	Edit Camera
	<3EH>	Edit Light Source
	<3EPP>	Edit Pipe Path

3D model creation commands

	<3X>	Create Extrusion
	<3RO>	Create Rotation
	<3B>	Create Boolean
	<3DE>	Create Blend
	<3DF>	Create Face Blend
	<3DT>	Create Three-Face Blend
	<3SL>	Create Loft
	<3SW>	Create Sweep
	<3SA>	Create By Parameters

	<3PI>	Create Pipe Sweep
	<3F>	Insert 3D Fragment
	<3U>	Set Adaptive Fragment Parameters
	<3CT>	Create 3D Mate
	<3CP>	Create Copy
	<3SY>	Create Symmetrical Body
	<3MO>	Insert External Model
	<3I>	Insert 3D Picture
	<3AL>	Create Linear Array
	<3AR>	Create Circular Array
	<3AN>	Create Array By Nodes
	<3AP>	Create Array By Path
	<3AA>	Create Parametric Array
	<3SD>	Divide Solid
	<3SR>	Create Spiral
	<3SP>	Create Spring
	<3CU>	Cut By Section
	<3SH>	Create Shell/Offset Body
	<3TA>	Create Face Taper
	<3TB>	Create Body Taper
	<3AM>	Apply Material
	<3AT>	Create Thread
	<3H>	Create Hole
	<3ST>	Create Rib
	<SMC>	Create new Base Part
	<SMB>	Create Bend

	<SMU>	Unbend Bend
	<SMR>	Re-bend
	<SMF>	Create Sheet Metal Forming Feature
	<SMS>	Convert Solid to Sheet Metal
	<SMP>	Sheet Metal Operation Defaults
	<3SS>	Sew Surfaces
	<3SZ>	Imprint Elements
	<3ZD>	Delete Faces
	<3ZX>	Separate Faces
	<3ZR>	Replace Faces
	<3ZC>	Change Faces
	<3ZT>	Transform Faces
	<3SX>	Extend Faces
	<3ZF>	Fill Hole
	<3DRT>	Scale/Twist Deformation
	<3DRB>	Bend Deformation
	<3DRS>	Skew Deformation
	<3DRC>	Sculpt Deformation
	<3DRV>	Deformation by Curve
	<3DRF>	Deformation by Surface
	<3SI>	Simplify

3D model editing commands

	<3EE>	Edit Operations
	<3EX>	Edit Extrusion
	<3ER>	Edit Rotation
	<3EDE>	Edit Blend

	<3EDF>	Edit Face Blend
	<3EDT>	Edit Three-Face Blend
	<3EB>	Edit Boolean
	<3ESL>	Edit Loft
	<3ESW>	Edit Sweep
	<3ESA>	Edit By Parameters
	<3EPI>	Edit Pipe Sweep
	<3EF>	Edit 3D Fragment
	<3ECP>	Edit Copy
	<3ESY>	Edit Symmetrical Body
	<3EM>	Edit External Model
	<3EAL>	Edit linear array
	<3EAR>	Edit circular array
	<3EAN>	Edit array by points
	<3EAP>	Edit array by path
	<3EAA>	Edit parametric array
	<3EI>	Edit 3D Picture
	<3ESR>	Edit Spiral
	<3ESP>	Edit Spring
	<3ECU>	Edit Cut By Section
	<3ESD>	Edit Divided Solid
	<3ESH>	Edit Shell/Offset Body
	<3ETA>	Edit Face Taper
	<3ETB>	Edit Body Taper
	<3ESS>	Edit Sew Faces Operation
	<3EAM>	Edit Material Application

	<3EAT>	Edit Thread
	<3EH>	Edit Hole
	<3EST>	Edit Rib
	<SMP>	Edit Base Part
	<ESM>	Edit Bend
	<ESU>	Edit Unbend Bend
	<ESR>	Edit Re-bend
	<ESF>	Edit Sheet Metal Forming Feature
	<EMS>	Edit Sheet Metal Converted from Solid
	<3ESZ>	Edit Imprint Elements
	<3EZD>	Edit Delete Faces
	<3EZX>	Edit Separate Faces
	<3EZR>	Edit Replace Faces
	<3EZC>	Edit Change Faces
	<3EZT>	Edit Transform Faces
	<3ESX>	Edit Extend Faces
	<3EZF>	Edit Fill Hole
	<3EG>	Move/Rotate
	<3CM>	Move Mated Components

3D model visualization commands

	<3AS>	Apply Section in 3D View
	<3CD>	Show Clip Plane Position
	<3CL>	Enable Clip Plane
	<3CS>	Set Clip Plane Position
	<3RC>	Select Center Of Rotation
	<3RF>	Full Rotation

	<3RX>	Rotate about X axis
	<3RY>	Rotate about Y axis
	<3RZ>	Rotate about Z axis
	<3RS>	Rotate about global/local axis
	<3RA>	Automatic Rotation
	<3VA>	Auto-Resize 3D Scene
	<3RP>	Panoramic Rotation
	<3RL>	Fix Direction
	<3VB>	Back View
	<3VF>	Front View
	<Ctrl + Numpad 1>	Bottom Isometry
	<3VU>, <Ctrl + Numpad 2>	Bottom View
	<Ctrl + Numpad 3>	Bottom Back Isometry
	<3VL> <Ctrl + Numpad 4>	Left View
	<3VR> <Ctrl + Numpad 6>	Right View
	<3VT> <Ctrl + Numpad 5>	Top View
	<3VI>	Axonometric (front)
	<3VK>	Axonometric (back)
	<3VW>	View Wireframe
	<3VS>	View Shading

	<3VD>	View Render
	<3VH>	View Hidden Line Removal
	<3VZ>	Precise Hidden Line Removal
	<3VM>	Semitransparent View
	<3VE>	Perspective Projection
	<3VO>	Orthographic Projection
	<3VP>	Set View Parameters
	<3VY>	View Ray Tracing
	<3TE>	Create 3D Model explode
	<3VX>	Exploded View
	<3VC>	Select Camera
	<3VG>	Move Camera
	<3RV>	Open Photorealistic View Window

Geometry analysis commands

	<PM>	Measure Element or relation between two Elements
	<3MP>	Mass-Inertial Properties
	<QM>	Check Model
	<QI>	Check Model Intersections
	<QA>	Check Model Changes
	<QC>	Show Curve Curvature
	<3VV>	Show Surface Curvature
	<QD>	Show Face Deviation
	<QH>	Show Gap Between Faces
	<QN>	Show Normal Deviation
	<QZ>	Show Surface Smoothness (Zebra)
	<QS>	Check Model Separation

3D model management commands

	<3G>	Regenerate 3D Model
	<3RG>	Full 3D Model Regeneration

Commands for 2D drawing creation and editing based on three-dimensional model

	<3J>	Create 2D Projection
	<3EJ>	Edit 2D Projection

Material creation and editing command

	<3MT>	Edit Materials

Customizing List of Element Types for Selection

The user can control the element types that are subject to selection by mouse. The selection filter icons are located in the right part of the system toolbar. For detailed description of selection filter management refer to the "Two-Dimensional Design", the chapter "Main Concepts of System Operation".

To manage the filters, make sure the 3D window is active. To activate the window, click anywhere within.

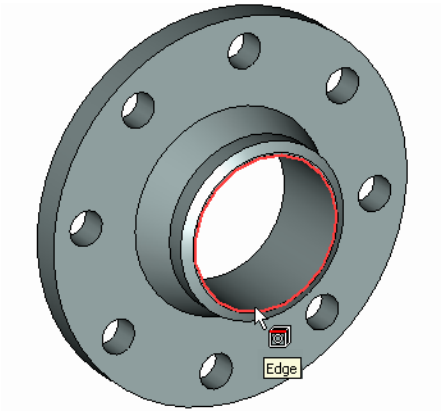
Element Selection

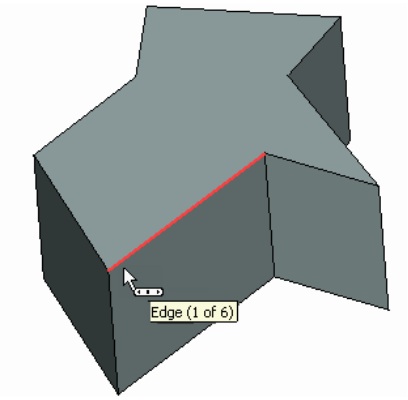
3D element selection can be done either in the 2D or in 3D window. A 3D element selection in the 2D window is possible only if this 3D element creation was based on 2D elements. For example, consider a 3D node defined by two 2D nodes. It can be selected either directly in the 3D window or via any of the two 2D nodes in the 2D window. Either way results in the 3D node selection.

The elements of a 3D sketch can be selected while in "3D Drawing" mode just the same way as in the 2D window.

Selecting a 3D element

Pre-highlighting is turned on in the 3D window. When the cursor approaches an element on the 3D scene, the element is getting highlighted, and the cursor gets an attached glyph that indicates the type of the highlighted element. If the cursor is briefly held over the element, a pop-up help appears displaying the type of the element. To select the element, press. To select a group of elements, hold the <Shift> key while selecting.

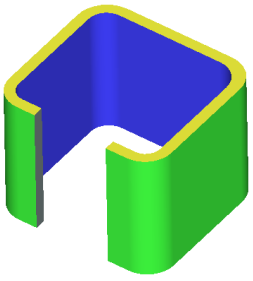




In complicated models, several elements of the same or different types can be under the cursor simultaneously. In this case, use the "Other..." button of the context menu that provides selection from list. The list is composed of the elements nearest to the cursor, whose types are admitted by the selection filter. An alternative way is to point the cursor at the desired element and briefly hold still. The cursor will be appended a "scroll list" glyph, and a pop-up widget will appear indicating the total number of elements available for selection at this point, and a current element type. The widget allows scrolling through the elements with the mouse middle wheel button (if equipped). Pressing selects the currently highlighted object.

To select a 3D profile, point the cursor at the contour line. To select bodies, set up the selector for operation selection and turn off edge and face selection. If in "Shading" or "Render" display mode, point anywhere on the body. In "Wireframe" mode, point at an edge of the body. Since the bodies may overlap on the scene, re-orient the model to bring the desired body up front.

The selected elements are marked with a color, depending on the object type and the purpose of selection. All color settings are defined in the system customization functionality.



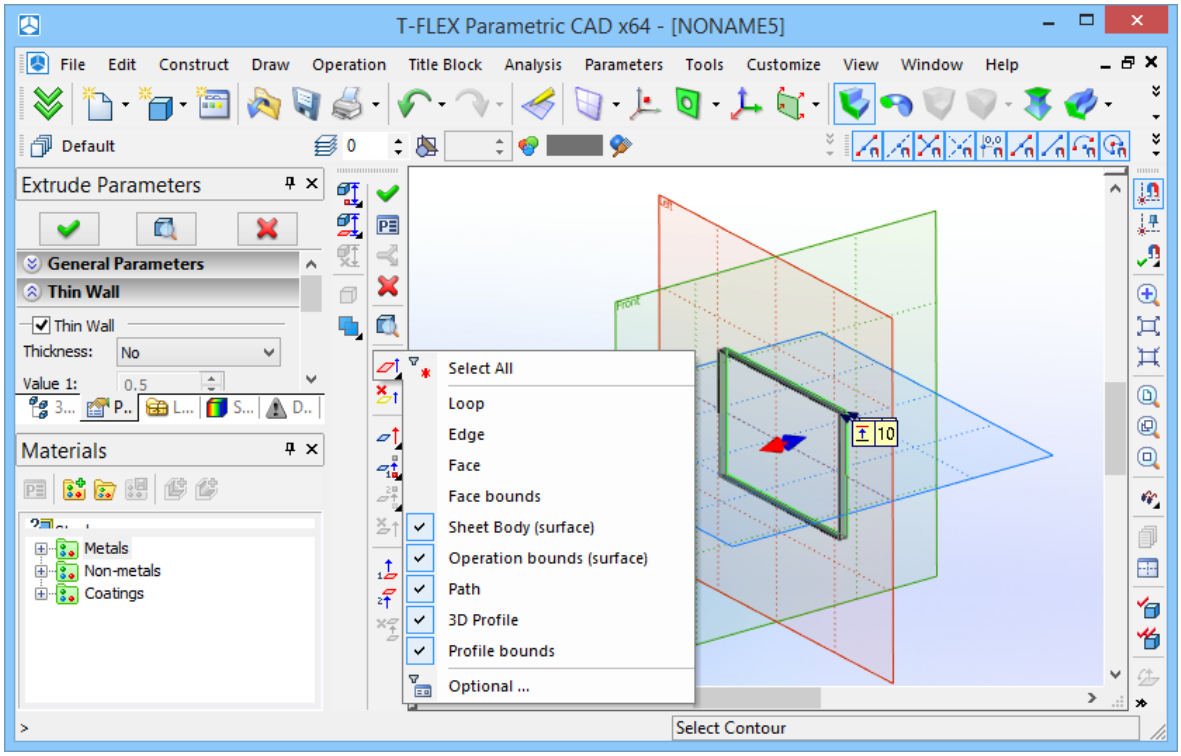
In some 3D commands, different elements of the same type are marked with different colors. For example, in the three-face blending operation, the user specifies the left, right and middle sets of faces. In this case, the faces of each set are marked with different colors. These colors also mark the respective tabs of the property window dialog box, that contain the corresponding lists of faces.

When working with large 3D assemblies containing nested fragments, only the fragments of the first embedment level are selected in 3D window. To get a quick access to the nested fragments of a lower level, it is possible to use the option found on the system toolbar.

Selecting elements within a command

The user often needs to select some existing elements on the 3D scene when defining a command input data. The system automatically adjusts the selector for the element type required by the current step in the command automenu. Suppose, for instance, while defining a blend we pushed the option "Select edge". The selector then adjusts to edge selection only. Next, reorient the scene for easy selection, point the cursor at the desired element, and press . In this case, selecting a group of elements is done by picking subsequently, without holding the <Shift> key.

Often, a required geometrical input in a command may be defined by a number of different-type 3D objects. Thus, for instance, a direction can be defined by wire objects, such as edges, paths, profiles, etc., or by a pair of 3D nodes, or by a normal to a planar surface, etc. In this case, the automenu option provides an extended list of filter settings for the selector. The availability of the list is indicated by a black triangle in the right-bottom corner of the option button. A simplified representation of the list is also displayed in the system bar.



Should a wrong element get selected (say, a neighboring one), use the automenu option,

	<I>	Select Other Element

A subsequent pick on the icon switches the selection to the next nearest element, on and on until all elements are traversed. This option is available with all 3D commands that require object selection.

Canceling selection

While within a command, element selection can be cancelled by pressing or <Esc> key. The elements are getting unselected one by one in the reversed order of the selection. Many commands allow unselecting all elements at once by using an appropriate automenu icon. (Refer to the respective topics in the documentation.)

To cancel selection of an element or a group of elements while outside any command, simply select another element or click at a blank on the 3D scene.

Element Search

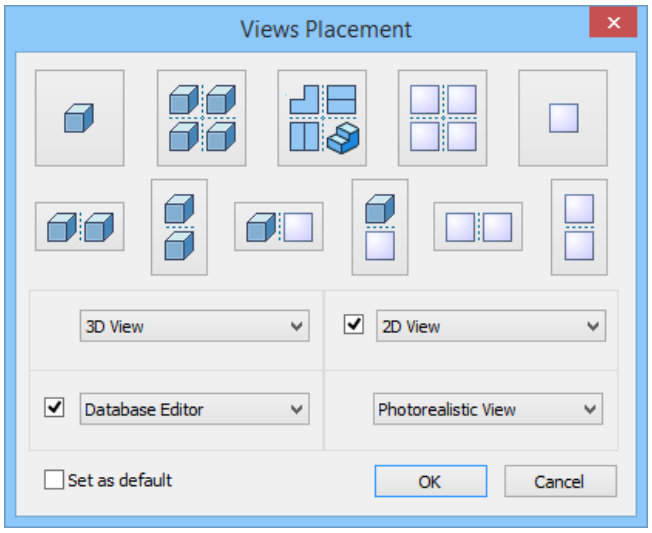
Search functionality can be invoked in transparent mode, and is available with all commands. For quick access to the search command, the button found in the set "Edit" of the main toolbar can be used.

Advanced search can be performed by the command **Edit** > **Find**.

Opening New Windows

When creating a new document from a 3D model prototype, a 3D window opens first. Later in design, the user may need to open additional 2D or 3D windows. Use the special buttons on scroll bars for this purpose. This buttons activates «Views Placement" dialog box, where you can select the location of new views.

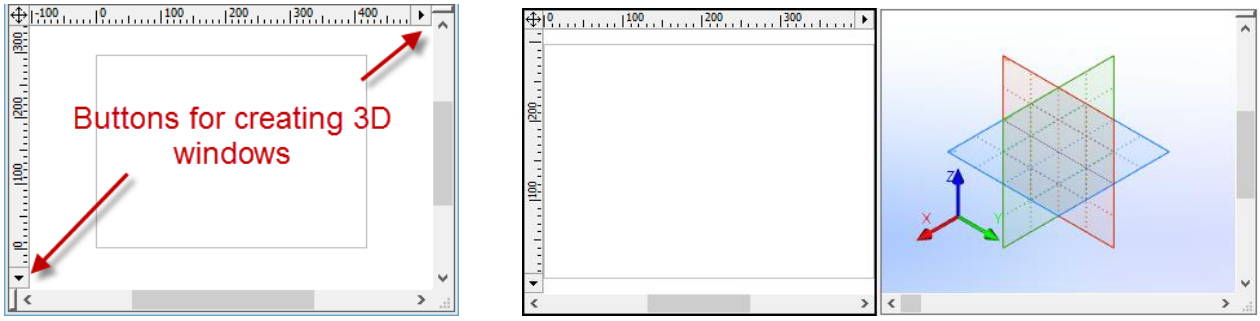
In "Views Placement" window you can select one or several 2D and 3D views using special buttons.



For custom views placement you can use the drop down lists in the lower part of the dialog box. Mark the required views with ☑. The following types of the new views are available: 3D view, 2D view, Database Editor and Photorealistic View. They can be assigned in any order and in any combination.

Use "Set as default" option to create predefined set of views when creating new windows in Window > New Window command.

If the original file was created from a drawing prototype, the 2D window was opened first. A simple way to create another 3D window is to click on the button with a triangular arrow in the right-top corner of the graphic area of the current window. As a result, the graphic area of the current window will split in two equal parts, a 2D window on the left, and a 3D one on the right.



Finally, one can call the command for opening a new window, **WO: Open New Window** by typing on the keyboard or via the textual menu **Window > New Window**.

> Depending on what kind of window is currently active, some or other commands will not be accessible. Thus, if a 2D window is active then the 3D Visualization commands won't be accessible, and neither will be 3D view controls. Vise versa, while in a 3D window, the most of 2D commands could only be accessed in a special mode - when a workplane is activated.

You can fix the particular set of views for the document window for usage in subsequent sessions by activating "Fixed set of windows" option in the **3D** tab of **ST: Set Document Parameters** command. The document should be saved to apply changes.

Manipulating Model in 3D Window

While working in 3D window a model can be spun in any way, zoomed at in and out, and panned. These actions can be done at any moment using mouse or keyboard input. When using the mouse, sometimes it is necessary to set certain options on the "View" toolbar (located at the right side of the screen).

To spin the 3D scene while working on an active workplane, use the button found on the main toolbar in the mode "Workplane" or "Workplane (Sketch)". It is also possible to spin the model with the help of the mouse simultaneously pressing the <Alt> key.

For more details see the chapter "Working with the 3D View Window".

"3D Model" Window

To open the 3D model tree window use the command:

Icon	Ribbon
	View → Window → Tool Windows → 3D Model Tree Window
Keyboard	Textual Menu
<Alt+3>	Customize > Tool Windows > 3D Model Tree Window

The window titled "3D Model" will appear at the left side of the screen. This tool window reflects the structure of the 3D model as a tree.

The toolbar in the upper part of the **3D Model** window includes: options for window status switching, a drop-down list of columns and a search field.

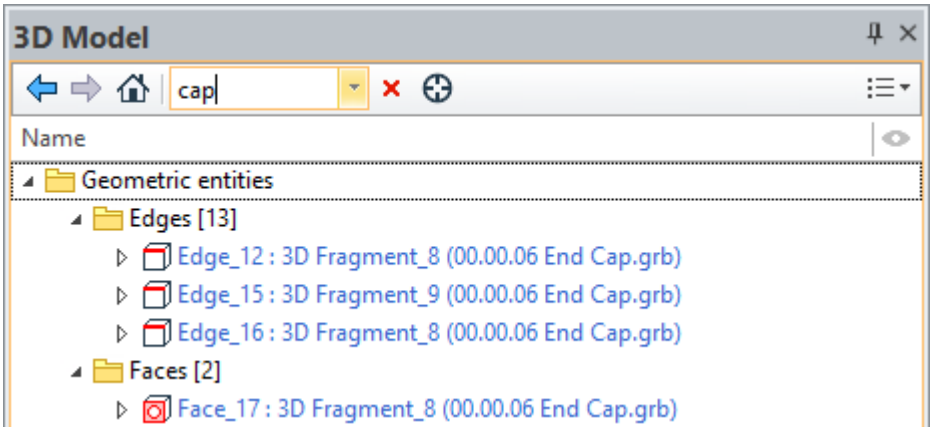
There are several ways to use the option:

If you select a 3D object in the scene and activate the option, the object will be highlighted in the model tree.

If you select edge, vertex or face and activate the option, a body/fragment, to which the selected element belongs, will be highlighted in the model tree.

If you select you select an element in the model tree and activate the option, the selected element will be zoomed in the 3D scene

A search field allows to find an element in the 3D model tree by its name or part of the name. Elements, which names match the required name, will be displayed in the tree after searching.

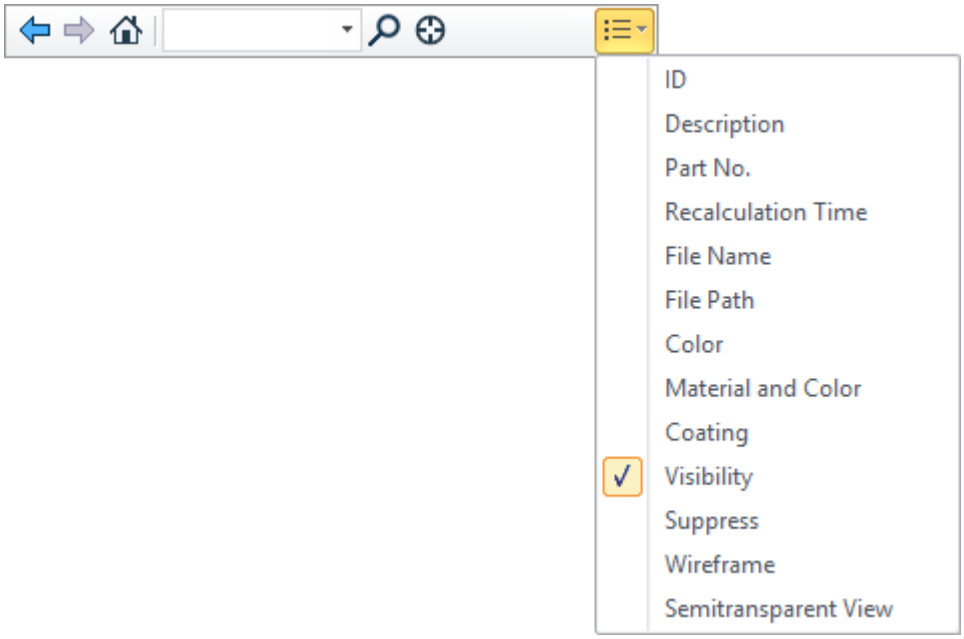


Columns allows to change elements parameters and to receive information. You can change parameters for several elements at once.

For example, if you want to hide several workplanes, it is necessary to select the planes and press in the visibility column.

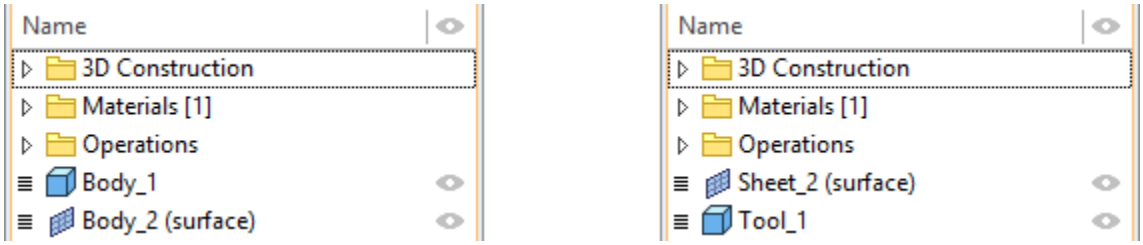
If any parameter can't be changed, its symbol will not appear in the column.

The drop-down list is used to select columns:

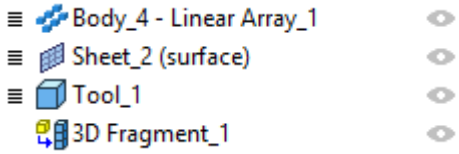


Bodies are put at the root of the model tree. Depending on the Body geometry type (solid or sheet object), the respective icon will be used for it in the model tree.

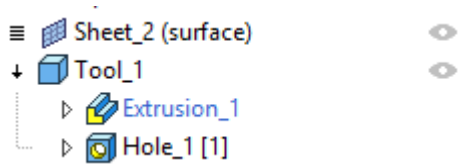
Each model Body is assigned a unique name, by default composed of the word "Body" and a number, for example, "Body_1". If desired, any Body can be assigned an arbitrary individual name in the Body parameters.



The 3D Fragment, Part and 3D Array creation operations can be displayed at the top level of the model structure, along with Bodies.



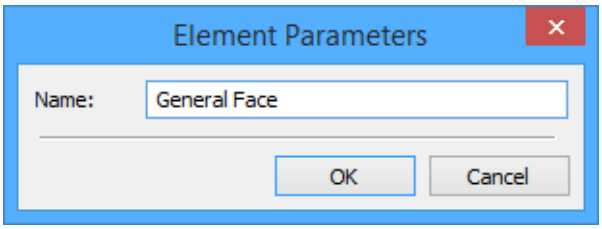
The Body knots are marked by the glyph . It allows watching the history of the given body's creation. Point the mouse at the glyph and click .



As the model tree branch with the Body history expands, the glyph changes to . The Body history consists of a sequence of their defining operations, displayed as a list. The list is formed from top down in the order of creating or using operations. To hide the Body history, point to the glyph and click .

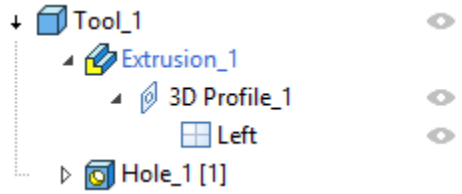
Each model element entered in the Body history has a unique name. By default, the name consists of the name of the element type and a number, for example, "Extrusion_6". If desired, the element, just like a Body, can be assigned an arbitrary individual name.

Named geometrical elements (edges, faces, etc.) can then be used to perform measurement, search of parent element used in operations. Searches can be performed via the **Edit > Find** command or **Find** section of **Edit** system toolbar. Named elements are available for obtaining information about the dependencies in "Information" command dialog. Option **Delete** that removes dependent objects is also available for them. Named elements, when used for subsequent creation of new operations based on them, retain their names. If the named geometric element is located on a visible body, it is marked in the 3D window when it is selected in the model tree.



You can change the geometric element name using the **Parameters** command in its context menu. Names may be set for vertexes, edges, cycles, faces.

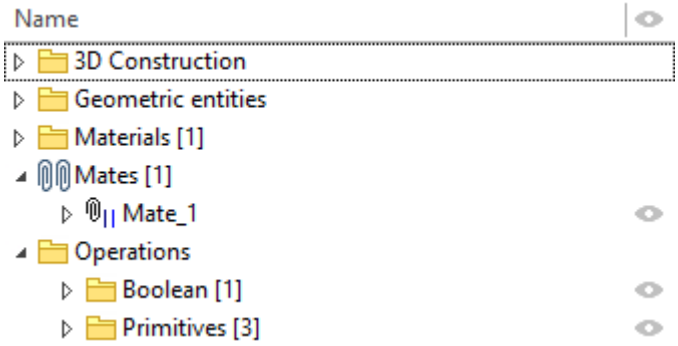
The glyph before the element means this element is based on other elements. To expand this model tree branch and view the element's parents, point the mouse to the glyph and click . After that, the glyph will change to . If there is no glyph before the element, this means the element is the last one on this branch of the 3D model tree.



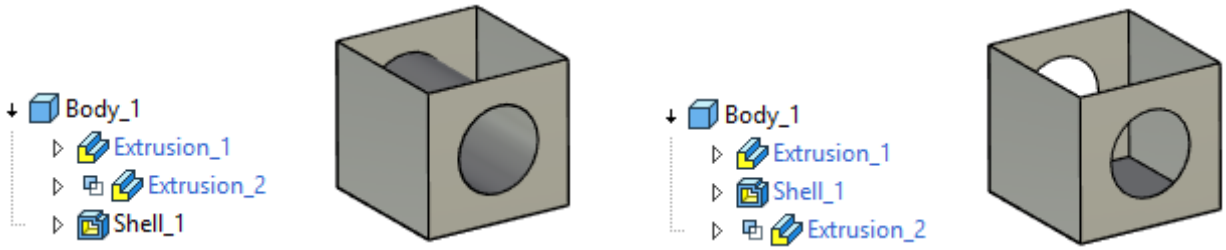
If the creation history can be built for an operation, then the history will be displayed instead of the parents. In this way, the glyph is put before the operation name instead of the glyph. An operation's history is displayed in the same way as a Body's history.

The Boolean operations are displayed in a special way in the history. The type of Boolean (addition, subtraction or intersection) is drawn before the icon of the operation that comes as the second operand of the Boolean. The diagram shows the history of the operation "Boolean_2": the body "Block_2" is subtracted from the body "Block_1".

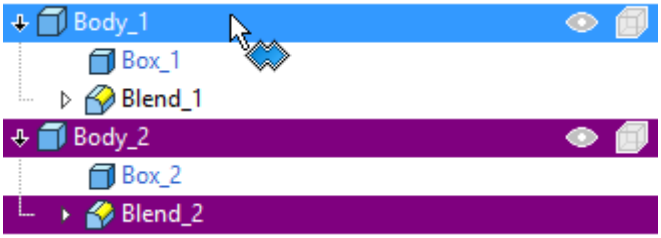
In addition to the list of Bodies, in the root of the model tree there are also special branches of the tree in which all 3D elements created in the current model are enumerated. Auxiliary 3D elements are put on the branch "3D Construction". All operations are on the second branch, "Operations". 3D annotation (3D dimensions, 3D leader notes, etc.) are put into the branch "3D service elements". The "Constraints" branch lists all constraints created in the given model. All elements are sorted into the folders by types: 3D Nodes, Workplanes, 3D Profiles, etc.; within the folders, the elements are sorted alphabetically. The number next to the folder name and the colon means the number of elements of this type contained in the model. By expanding branches of the model tree, you can access any element of the 3D model.



In certain cases, operations can be reordered by moving their labels within the model tree. This effectively changes the model structure and geometry. Press on the element label and drag it to a new location in the tree. The following diagrams demonstrate the result of reordering an element in the model tree. In the beginning, the operation Shell_3 followed the subtraction operation Boolean_2. Then, the Boolean, together with its tool body operand Extrusion_1 (a cylinder), was reordered to be after the shell operation.

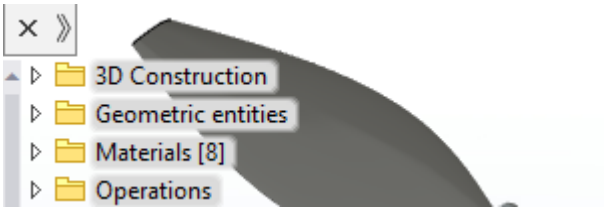


The model tree helps quickly create Boolean operations of all types. If you select a Body and drag it over another Body, the indicator of the Boolean "addition" will appear at next to the pointer. If you additionally press the key <Ctrl> or <Shift>, the type of the Boolean changes to subtraction or intersection, respectively. Instead of Bodies, you can select operations that are last in their histories.



3D Model tree in 3D scene

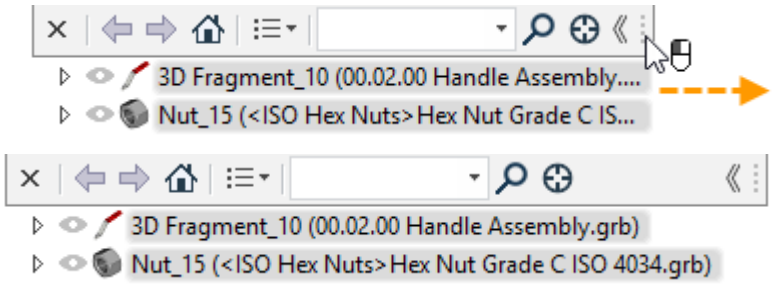
The **3D Model** window can be also displayed in the 3D scene. For this purpose press icon in the top left corner of the 3D scene.



If the **3D Model** window becomes at some point not available when working with a 3D model, the model tree is automatically expanded in the scene. This occurs, for example, when entering commands of 3D elements creating or editing. If the expansion of the tree upon commands activation is not required, it is enough to close it once using and the system will remember this status.

The model tree has a toolbar that is similar to the toolbar in the **3D Model** window. The only difference is the number of columns to be displayed. To hide this toolbar you need to click on a special button.

The tree can be expanded. To do this place the cursor on the right edge of the toolbar and move it holding the left mouse button.



Model Tree Context Menu

By pointing to an element in the model tree, you can access the context menu (by pressing).

The context menu consists of standard commands for working with element (**Edit**, **Copy/Paste**, **Parameters** and so on) and with 3D Model window.

> The context menus may vary depending on the element selected in the **3D Model** window.

Besides the common commands for managing the selected element, the context menu contains some additional commands for working with particular Bodies, operations and the model tree as a whole:

Delete Body (for Bodies). This command serves to delete all the operations which are part of this Body's creation history. In addition, all parent 3D construction elements of this Body can be deleted, or just those parent elements that are not referenced by other created Bodies.

Delete (for operations and auxiliary 3D elements). The command deletes the selected operation or 3D construction element.

Detail command allows to unload a body or a fragment from the assembly into a separate file.

> More information about detailing can be found in “Assemblies creation” chapter.

Remove mate transformations. Option allows to remove all mate transformations of the selected object and to return it to the initial state.

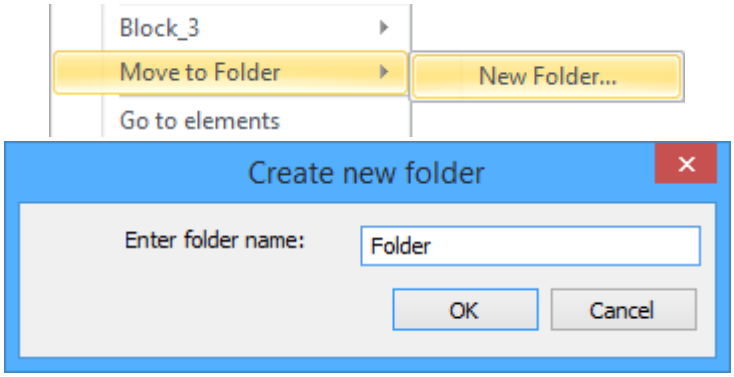
Rollback model option allows to rollback a three-dimensional model to the level of the specified operation.

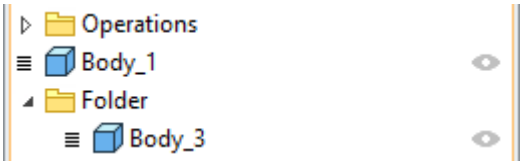
> More information about rollback model can be found in “Edit” chapter.

Retain geometry after recalculation option allows you to save intermediate geometry for the operation, thereby accelerating transition to its editing.

> More information about the option can be found in “Edit” chapter.

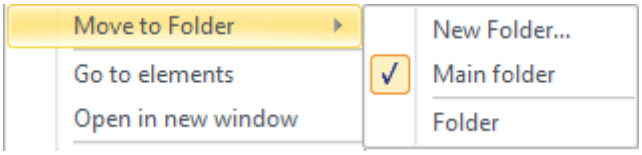
Separate branches-folders can be created for grouping Bodies, 3D fragments and 3D array. To do it, in the context menu for the Body which you want to put into a separate folder, invoke the command **Move to Folder** > **New folder** from the context menu. A window for specifying the name of the folder being created will appear on the screen. After specifying the name and pressing **[OK]**, there will be a folder having the specified name in the tree of the 3D model, and the selected Body will be put into this folder.





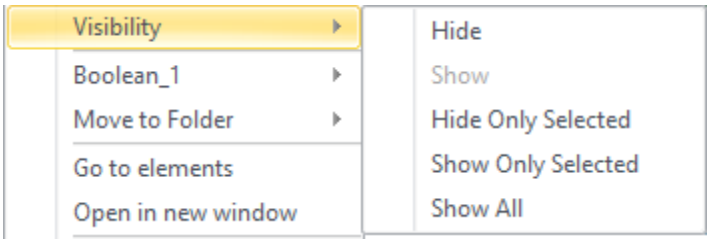
If in the tree of the 3D model at least one Body folder has been already created, in the context menu for each body in the submenu **Move to Folder**, a list of existing Body folders will be also present. The folder that contains the current body will be marked. For moving Body into one of the already existing folders, it is enough to select the folder in the list.

It is also possible to move Bodies from the root of the model tree to the folder and backwards just by dragging Bodies while pressing .



Selecting objects with the help of the model tree is sometimes very convenient in many commands, for example, when selecting this element in the 3D window is difficult for some reasons.

The **Visibility** group of commands unites the commands to control the visibility of Bodies and 3D construction elements:



Hide. This command makes the selected 3D objects invisible in the 3D scene;

Show. This command cancels the previous command, making the selected 3D objects visible in the 3D scene;

Hide Only Selected. This command makes all selected 3D objects invisible, and shows all the rest, if those were previously hidden.

Show Only Selected. This command makes all the selected 3D objects visible, and hides (makes invisible) the rest;

Show All. This command makes all objects in the 3D scene visible.

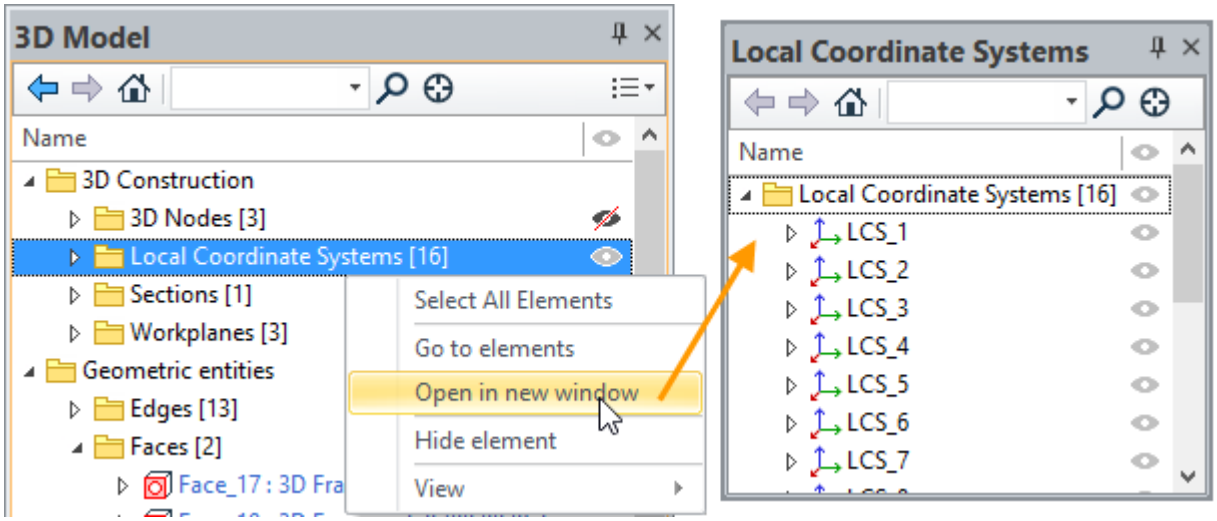
The accessibility of the commands in the group depends on which 3D objects are selected. For example, if all selected 3D objects are currently hidden, then the "Hide" command will be inaccessible.

The hidden/shown Body state can be defined using the "Hide" option in the Body's parameters dialog without the use of the described commands. This option's value can be defined via a variable. If a variable is used, then the commands of the "Visibility" group will not affect the visibility of such Body. This will be stated in the provided warning message.

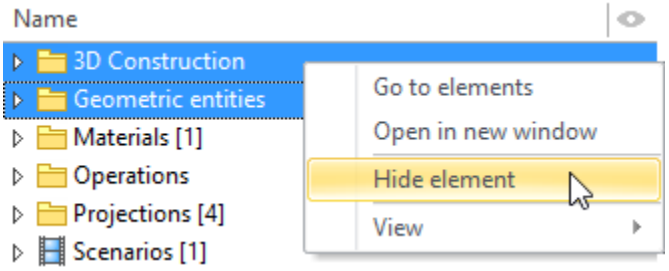
Suppress (for operations). This command excludes the operation from regeneration. This allows user to temporarily remove 3D model elements. To select a suppressed operation, use the "3D Model" window or do the element search.

Only selected elements will be displayed in the tree upon using **Go to elements** command. Status switching options , are used to return to the normal mode.

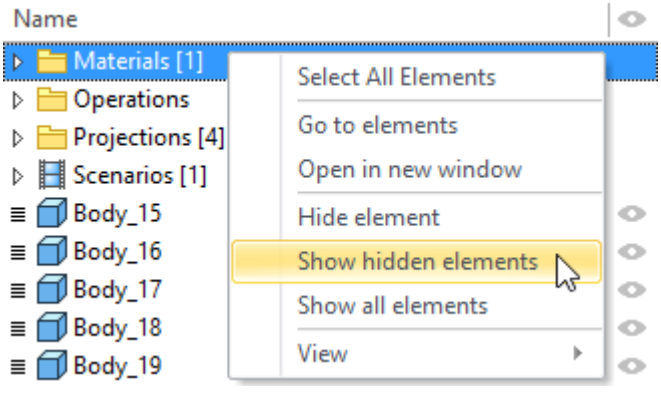
You can create a separate window for the selected elements using **Open in new window** command. This window can be moved around the screen. It retains all the functionality of the **3D Model** window. This window eliminates having to move the cursor across the screen to select an element in the model tree.



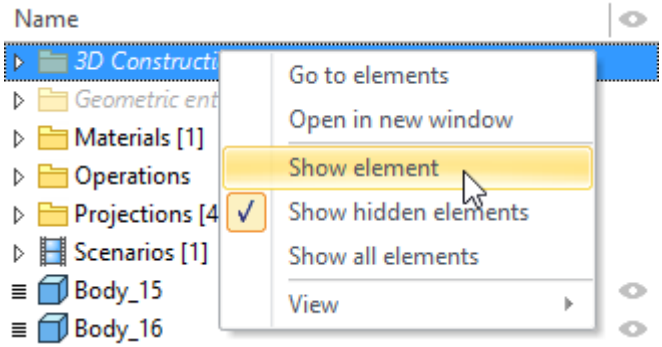
After using **Hide element** command, the selected elements will not be displayed in the model tree. For example, you can hide unused items, so as not to be distracted by them.



If the flag **Show hidden elements** is active, all hidden elements are displayed but greyed. The elements names are shown in italic to distinguish them from the suppressed elements.

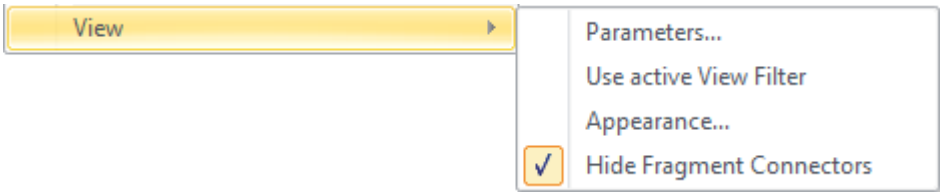


To display elements again you may use **Show element** command.



Use the **Show all elements** command to show all hidden elements again.

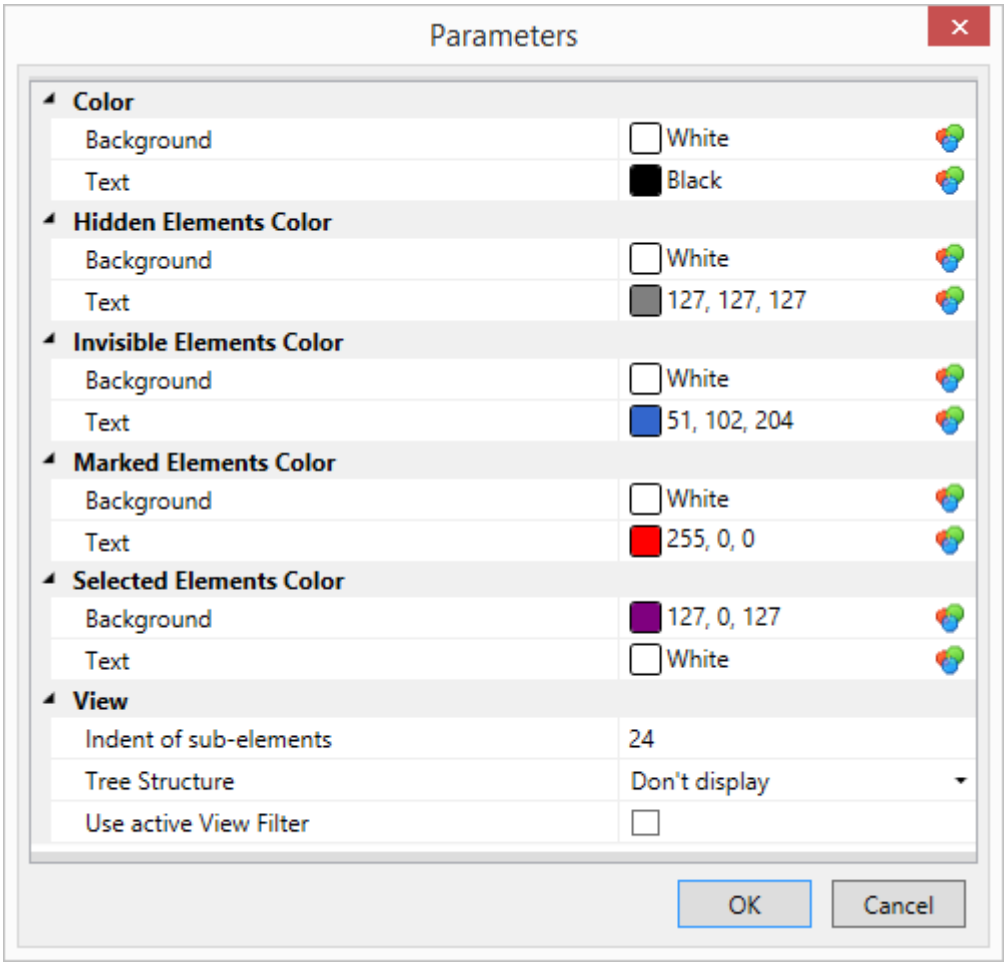
View item includes commands and options to customize the display of model tree such as **Display parameters**, **Use active View filter**, **Appearance**.



In the **View parameters** window you can set colors for: **3D Model** window, selected elements, invisible elements, marked elements and hidden elements. In each of these groups, you can set text color and background color.

View group

The **View** group in the **Parameters** dialog is different for the **3D model** window and the model tree in the 3D scene.

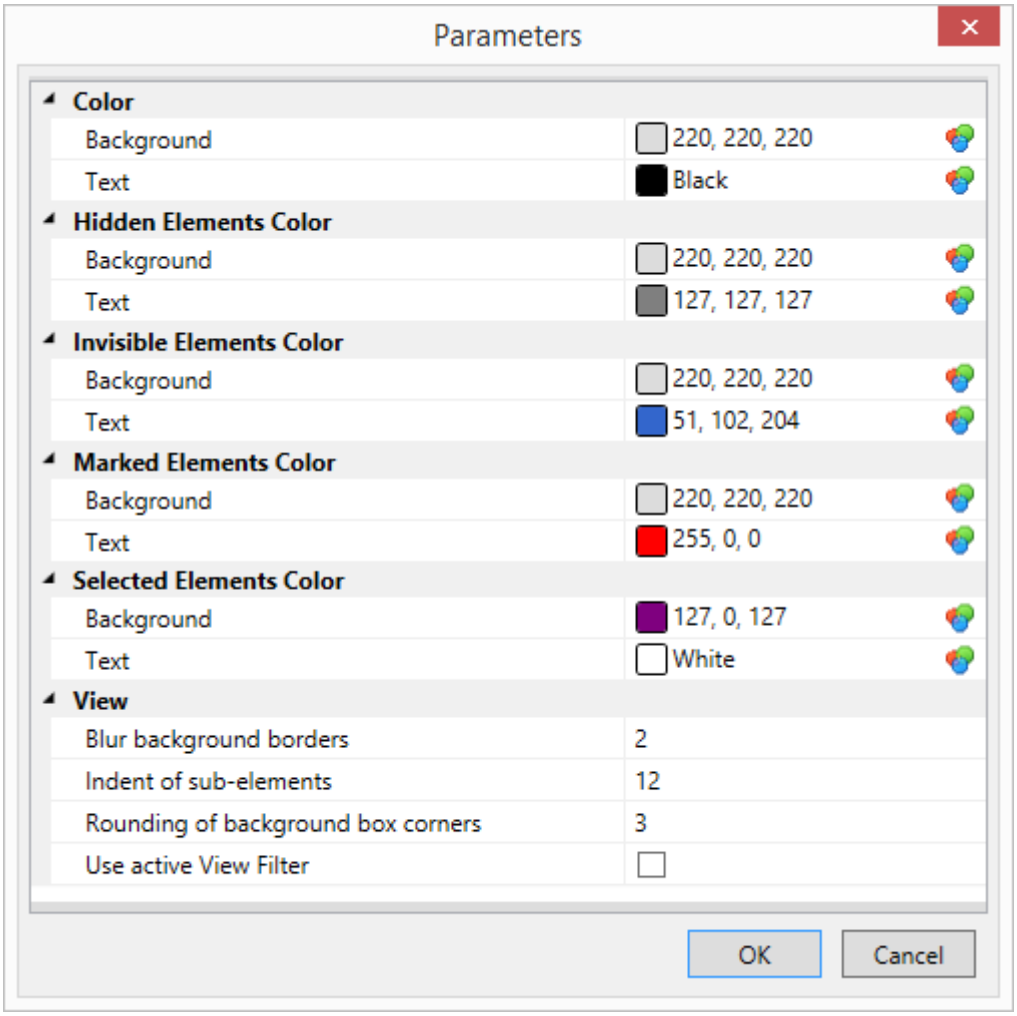


Use active View Filter. The 3D window selector can be adjusted for selecting elements in the model tree. To activate the filter, flag this item in the context menu. When you move the cursor over the items in the tree, only the elements corresponding to the active filter will be highlighted.

Indent of sub-elements parameter allows to specify the indentation of child elements in the model tree relative to the parent element in pixels.

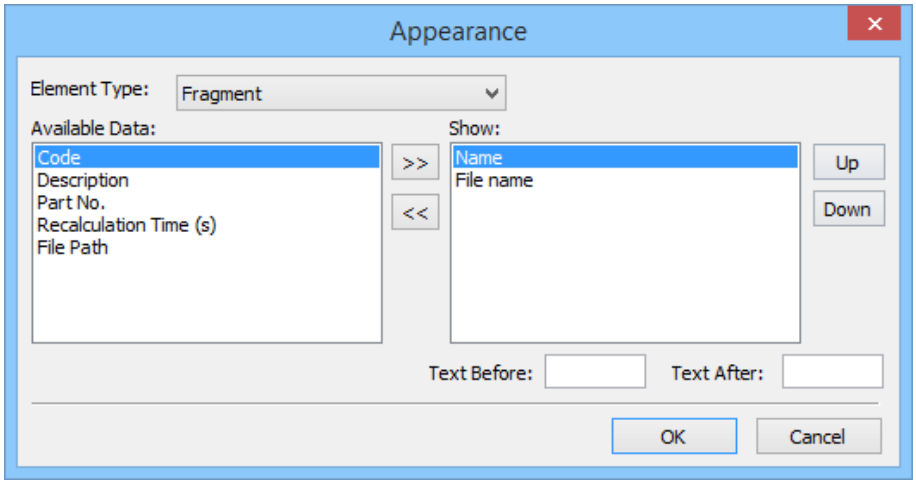
Tree Structure parameter allows to set visual representation of the tree structure. The following options are available: **don't display, lines, dots.**

Blur background borders and **Rounding of background box corners** parameters are available for **3D Model** window in the 3D scene.

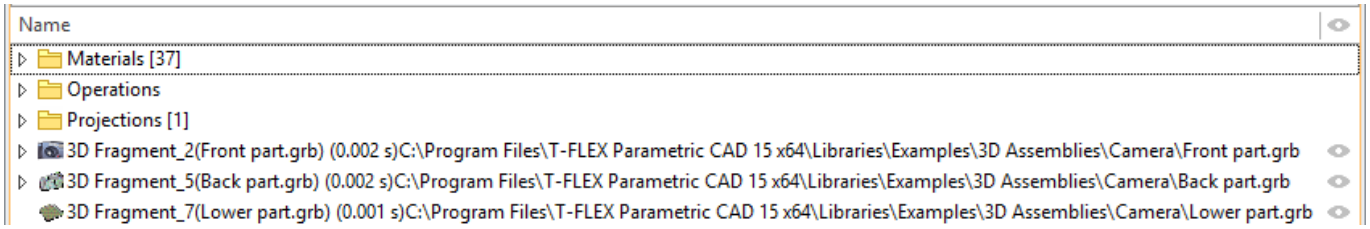


Use active View Filter - duplicate the appropriate flag from View parameters.

Appearance... This command allows a user to customize information which is displayed in the tree of the 3D model. Upon calling this command, the settings dialog window appears.



In the dialog "Appearance", it is possible to select items which will be displayed in the tree for objects of various types: time spent by a computer on object recalculation, the memory size of the geometric data for the object and the size of the data required for its displaying (mesh), etc. This customization allows a user not only to see the objects in the tree in the convenient for the user form, but also carry out the recalculation time and the memory usage optimization for the model. The information about the name and description of the assembly components can be also convenient when working with the assembly models. This information can be displayed together or instead of the filename and the path of the file for the assembly components (fragments).

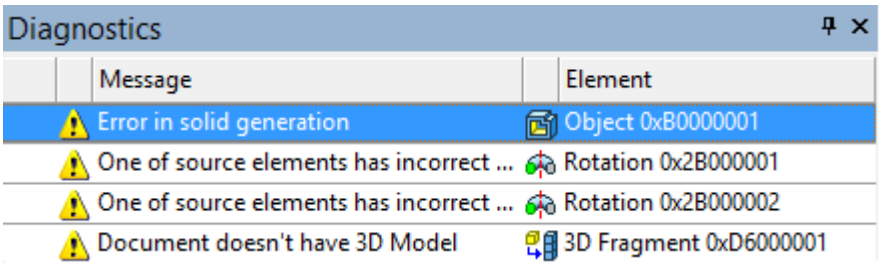


Settings specified in the dialog "Appearance" are saved in the document file and used upon its subsequent opening.

Hide fragment connectors. When the flag is active, fragments connectors are not displayed in the 3D scene.

"Diagnostics" Window

This tool window is used for displaying warning and error messages. The errors may occur due to invalid operation parameters input, an out-of-range variable value, regeneration with incompatible parameters, etc. The system attempts to identify the reason of a failure and outputs a message in this window. Besides the failure reasons, the system outputs the information about the failing element. The failing element and its dependents are marked with a red cross in the model tree.



The context menu in the diagnostics window provides, along with the standard set, some special commands, as:

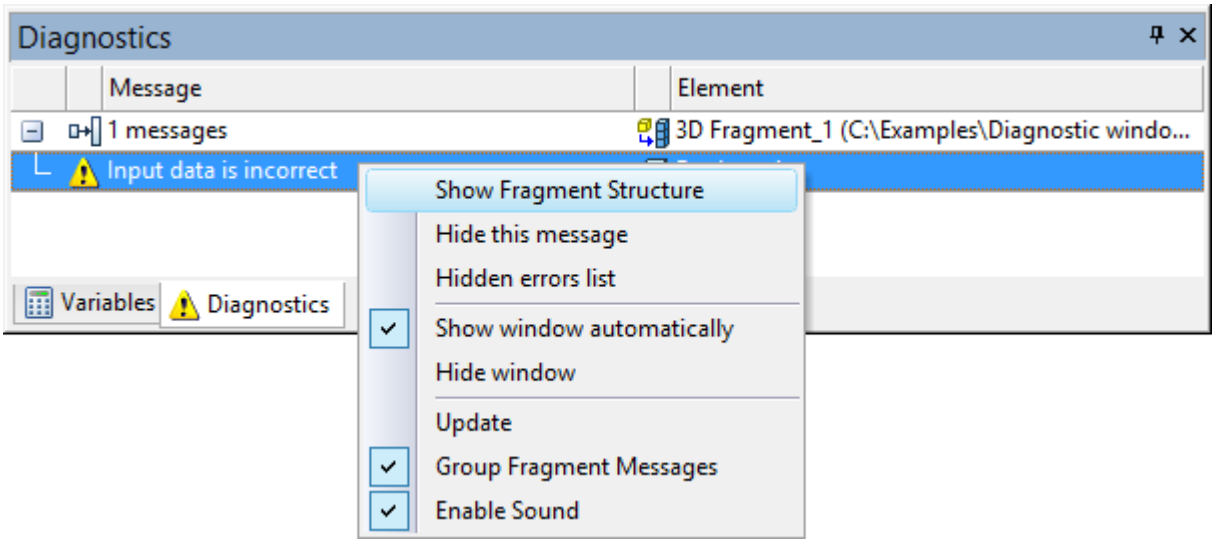
Hide this message. This command serves to hide messages of the selected type. When this command is applied, the messages of the selected type will no longer be displayed in the diagnostics window up until this restriction is canceled. The command also affects the future sessions of working with the system.

Hidden errors list. This command calls the dialog window with the list of hidden messages. To cancel the restriction on the messages of the given type, simply clear the check before a message.

Show window automatically. This option makes the window appear automatically whenever new messages are generated. It is useful in the case when several tool windows are grouped together on one console.

Clear window. This command clears the diagnostics window from accumulated messages.

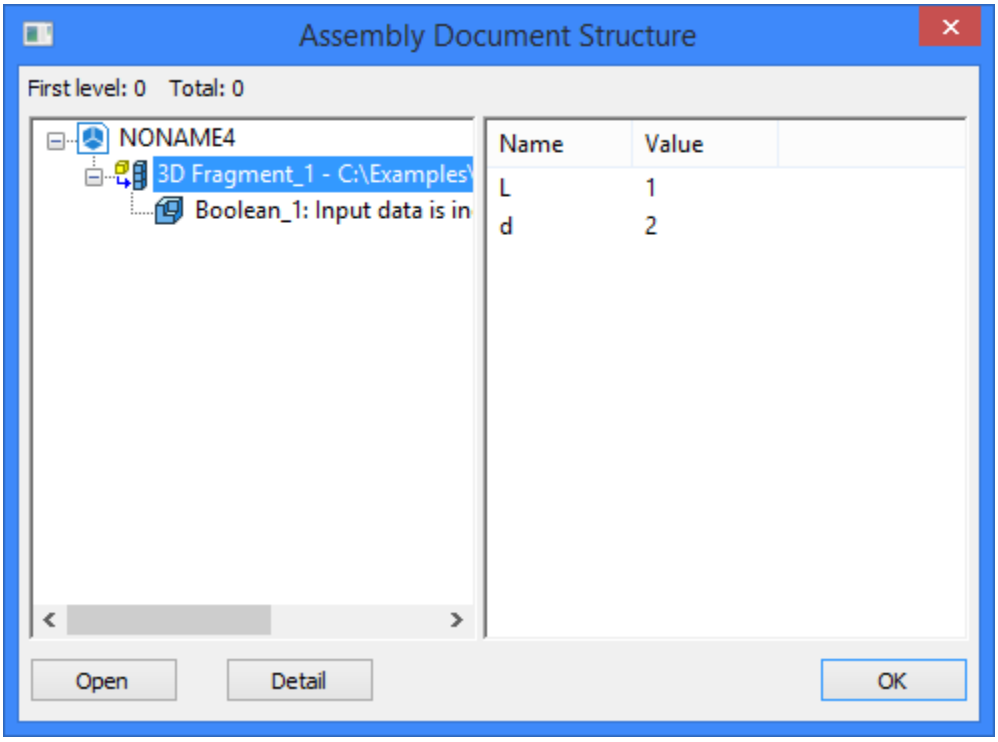
Hide window. This command removes the diagnostics window. To bring it back, use the main menu command **Customize** > **Tool Windows** > **Diagnostics Window**.



Edit. This command is accessible only when a specific message is selected. It invokes editing of the failing element or operation.

If the errors occurred within a fragment file then the diagnostics window references the operations from the fragment model that caused the error, while the context menu gains another command,

Show Fragment Structure. This command invokes the "Model structure" dialog box.



The fragment in which the error was identified is automatically highlighted. The right half of the window contains the information about the fragment variables: their names, values, and comments. To handle the error, one can open the fragment file by pressing the *[Open]* button.

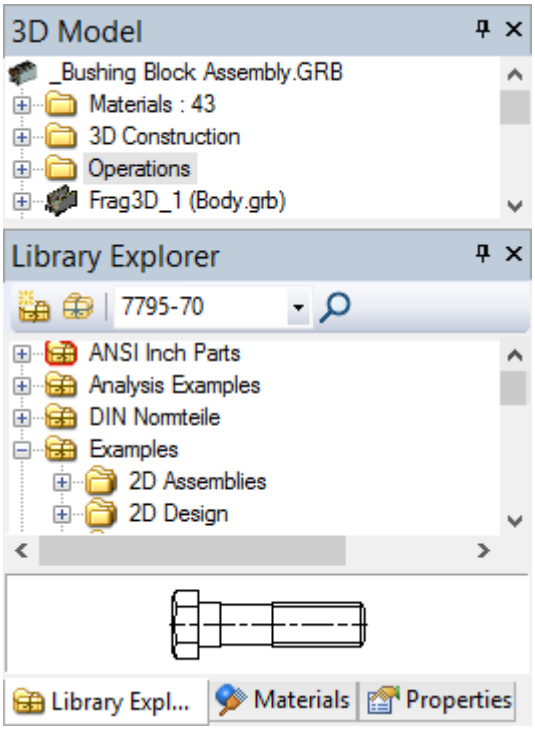
An error in the fragment model may be caused by an invalid value of a fragment variable. To open the fragment with the current values of the variables use the button *[Detail]*.

You can select multiple messages in the diagnostics window by using the combination <Ctrl>+. When several messages are selected, the following command will be available in the context menu:

Delete Related Elements. This command serves to delete all 2D and 3D elements related to the selected messages.

Arranging Tool Windows

All tool windows in the T-FLEX CAD application can be combined in a common console with tabs. This saves up the working area on screen and eases the user from constantly adjusting the windows borders while working with the application. To combine two windows, pick one of them with at the title area and drag into the title area of the other window. The two windows collapse into the common console. To switch between the windows, use the tabs that appear in the bottom of the console. The system may automatically switch between the windows during operation.



Toolbars

Toolbars provide additional convenience to T-FLEX CAD operation. Working with toolbars, their customization and access to toolbars are done in the same way as in the 2D drawing mode. Further refer to the volume “Two-dimensional design”, the chapter “Main Concepts of System Operation”.

The necessary options of the command are set using the **automenu** together with the **property window**, similar to the 2D mode. The set of icons in the automenu is specific to each command. However, some options in the automenu are common for all commands:

	<Y>	Finish input. Confirms element creation
	<P>	Set entity parameters
	<I>	Select other element
	<Esc>	Cancel selection
	<F4>	Execute Edit Command
	<X>	Exit command
	<F5>	Preview Operation Result

Customization

Certain customization is necessary for working with T-FLEX CAD. Customization settings include both the system-wide ones, and those specific to a particular document. The system settings are automatically saved in the Windows registry and are applied at subsequent starts of T-FLEX CAD. The settings specific to a particular document are saved with the document. The system customization is done via the "**Options"** dialog box, invoked by the command:

Icon	Ribbon
	→ Options
Keyboard	Textual Menu
<SO>	Customize > Options

All system settings related to 3D modeling are located on the "**3D"** tab. For more details on working with "Options" dialog refer to the volume "Two-dimensional design", the chapter "Customizing System".

Customization of document parameters is done via the **Document parameters** dialog box invoked by the command:

Icon	Ribbon
	Edit → Document → Document Parameters
Keyboard	Textual Menu
<ST>	Customize > Document Parameters

All drawing settings related to 3D modeling are located on the **3D** tab. For more details on working with "Document Parameters" dialog refer to the volume "Two-dimensional design", the chapter "Customizing Drawing".

Indicators of 3D Model Graphical Representation

When working with 3D models you can monitor parameters of graphical representation. To do this, set " Display graphics performance data" flag in **SO: Set System Options** command ("3D/Graphics Settings ..." tab). When flag is checked the upper left corner of 3D view will display the following information:

FPS - number of frames per second;

Batch - number of switching buffers when loading bodies into a 3D scene. This parameter depends on the number of bodies in the scene.

Vertex - number of vertices in the scene. Vertex in this case refers to a set of vertex data (coordinates, normals, texture coordinates, and other vertex data).

Face - number of triangular flat faces in the mesh;

Line - number of lines in the scene;

Point - Number of points.

MTr/s - speed of mesh triangles drawing. Measured in millions of triangles per second.

To improve performance, you should seek the ways of reducing the quantitative graphical representation of models. You can control this via the mesh quality parameters.

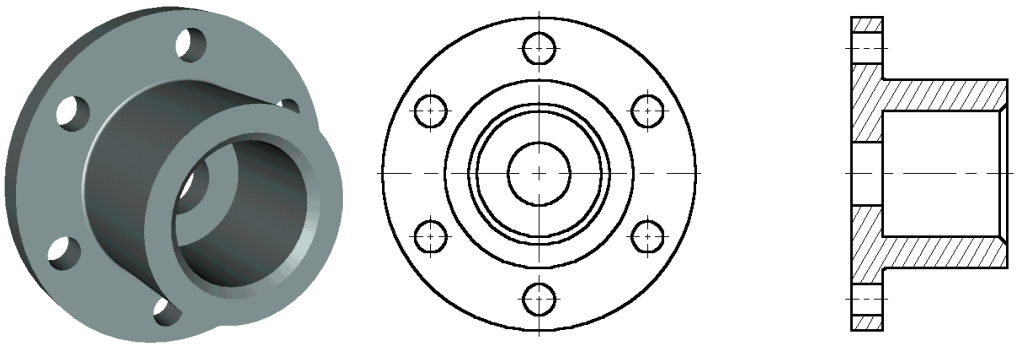
Brief Introductory Course in 3D Modeling

In this chapter, you will learn how to create a three-dimensional model using simple examples. The manual describes all necessary steps in the modeling process. Further, you will learn the basic commands and principles of creating a 3D model.

T-FLEX CAD system allows various approaches to creating a 3D model. The main method is creating most of constructions directly in the 3D window. In another approach, a 3D model is created based on already prepared 2D drawings or auxiliary 2D constructions. This construction may further be used for generating drawings by projecting the model in all necessary views and making cuts and sections. These 2D objects can further be furnished with necessary dimensions and other drawing attributes.

Main Approach to 3D Model Creation

As mentioned above, this approach makes possible creating a three-dimensional model without using the 2D window. The diagram below shows a sample part that we will be modeling. We will create the three-dimensional model first, and then automatically generate views and a section.



The model file is located in the folder *"...\Examples\Documentation examples. Introductory course\Detail with drawing detailing"*.

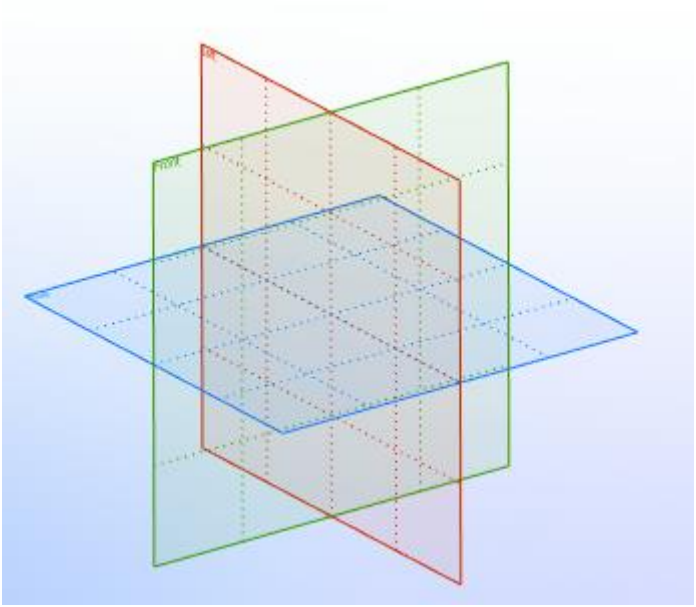
The model creation will be done in several steps. First, one has to create initial auxiliary elements. Based on those, you will be able to create a first draft of the considered part – without the holes and chamfer. To do this, use the "Rotation" operation. At the next step, add six holes in the part's body. Holes can be created by different means. We will discuss several methods here to give you a more complete idea about 3D model creation methods. Then, to get the final version of the part, you will just need to apply a chamfer via the "Blend" command.

Creating Auxiliary Elements

Let's start from scratch. Create new document.

To begin modeling, one can use an appropriate prototype file among available in the T-FLEX CAD system (**File > New 3D Model**).

You have just created a new document that already has three standard workplanes – Front, Right and Bottom. You can see that the 3D window with these elements opens automatically within the system window.



For convenience of modeling in 3D window, spinning, panning and zooming of the scene view are available at all times, even when commands are active. Note: the view manipulations while in certain commands require a specific option to be set. These commands will be described below.

To spin the scene press and hold and move the cursor as desired. One can also use the two arrow key pairs and the pair <Page Up> and <Page Down> for spinning around the three respective axes.

Zooming can be done at any moment by scrolling the mouse wheel button (as in IntelliMouse design) or via the provided commands on the "View" toolbar located along the right border of the window. Panning and zooming of the view can also be controlled via <Ctrl>+ and <Shift>+ combinations respectively.

As we move the cursor close to a workplane, the latter gets highlighted. T-FLEX CAD provides pre-highlighting on cursor over for all elements in the 3D window depending on the selection filter settings. The selection filter icons are located on the system toolbar. To pick an element among the icons, simply press .

Let's select the "Right" workplane. One can see now that 2D drawing commands become accessible. We will further use these commands for creating auxiliary elements in the 3D window.

What auxiliary elements do we need? The first body to be created is a body of revolution. To create it, we will need the contour and the axis of rotation for this contour.

Drawing in 3D window can be performed using any of the tools available for 2D drawing. Thus, one can use the sketching environment for rapid creation of non-parametric models. Accordingly, parametric modeling approach requires creation of construction lines first, then application of graphic lines. The system automatically creates 3D profiles based on drawn graphic lines, to be later used in 3D operations.

To start drawing, call the command,

Icon	Ribbon
	Draw → Construct → Line
Keyboard	Textual Menu
<L>	Construct > Line

After activating the workplane by any 2D drawing command, the main toolbar is switched to the mode "Workplane". In this mode a set of icons for working on the active workplane is displayed on the toolbar.

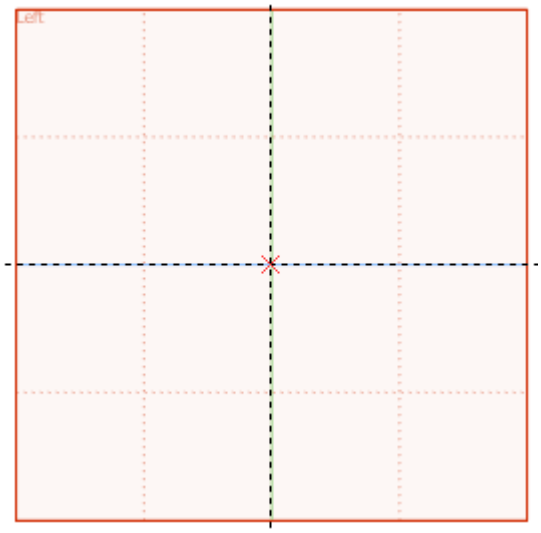
For convenience, the active workplane is reoriented parallel to the screen. However, it can be spun in any desired way, after setting the option .

It is also possible to open the 2D window and continue drawing in 2D mode. Once the 2D window is closed, all changes will be reflected on the 3D scene. The 2D window can be opened or closed using the icon.

Let's construct two base lines, a vertical and a horizontal one. To do so, push the following button in the automenu:

	<X>	Create two crossing Lines and Node

Point cursor at the center of the workplane and click .

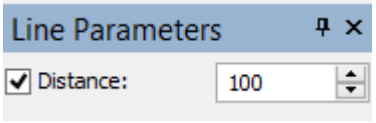


Thus, we have created two orthogonal lines and a node. These will be used as a reference for all further construction. Press to quit the last active command. We are now in the parallel line creation mode. (This mode is set by default in the command "**L: Construct Line**").

> Just like in 2D drawing, we first need to create a framework of thin lines, and then apply graphic lines along the intended segments.

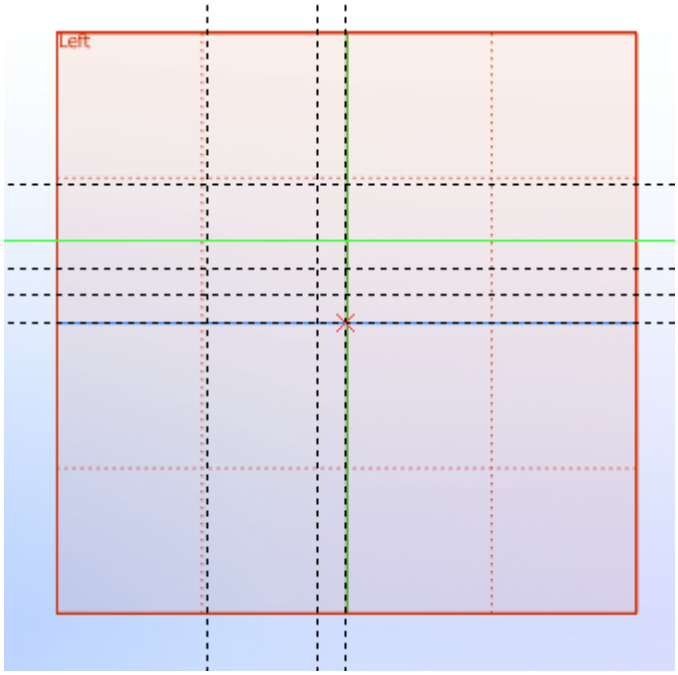
To construct a parallel line, select a reference line first, that will be used as the "parallel" reference for the new one. Similarly to 2D operation, the 3D drawing mode also supports object snapping. Thus, to select a line, move the cursor to the vertical line. The cursor will gain a glyph like this, . Click and move the cursor leftwards. One can see the cursor changing appearance again – it is now rubberbanding a line parallel to the selected.

To fix the line at an arbitrary location, just click . However, we would like to specify the distance of 100 mm, by typing it in the property window. (In future, this parameter value can be changed at any time as desired.) Press <Enter> to complete the line creation.



We have just created a line parallel to the selected, at the distance of 100 mm from it. As one can see, rubberbanding resumes at this point. The system is still in the mode of parallel line creation with respect to the original selection. Type a new value of 20 mm right away, followed by <Enter>, thus creating another line. That's all with the parallel line creation for now. Click or <Esc> key for quitting the parallel line creation mode.

Next, create four more lines in the similar way, parallel to the horizontal base line. Use the distances equal to 20, 40, 60 and 100 mm respectively. The result should look like shown on the following diagram.

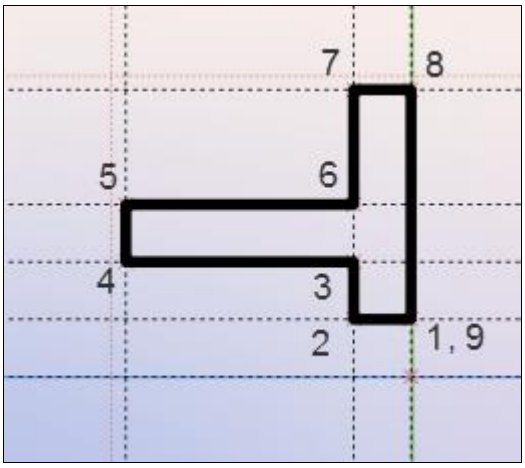


Now, apply graphic lines along the construction lines appropriately. Call the graphic line creation command:

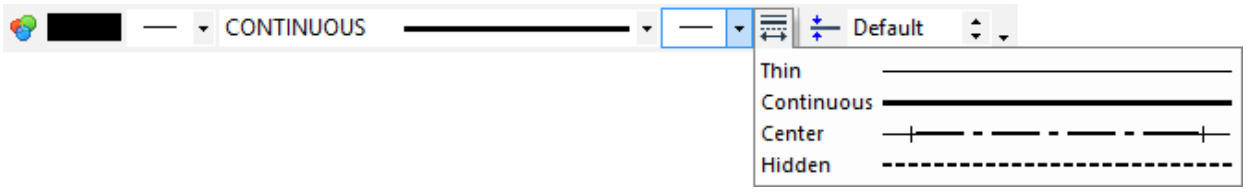
Icon	Ribbon
	Draw → Draw → Graphic Line
Keyboard	Textual Menu
<G>	Draw > Graphic Line

Graphic lines snap to the construction entities – lines, circles, nodes, etc., as well as to construction line intersections. In the latter case, a 2D node is automatically created at intersections snapped to by graphic lines. Straight graphic line segments maintain snapping at their end points. To have an end of a graphic line snapped to some entity, move the cursor to the intended location. Let the cursor snap as indicated by an appropriate glyph, and click .

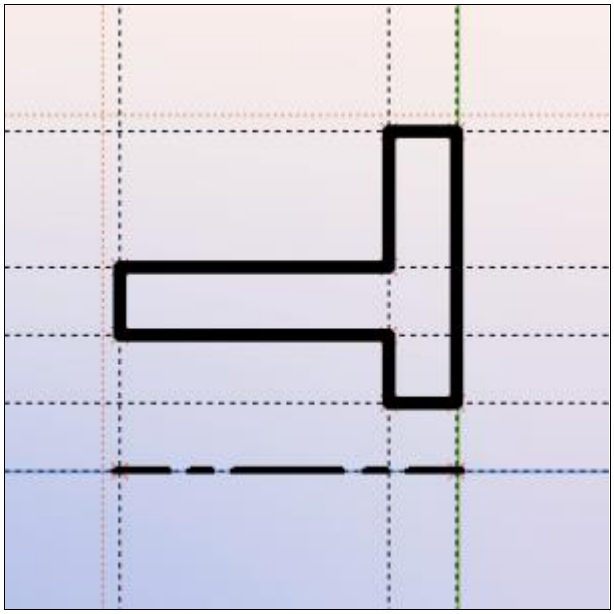
In this example, we use snapping to construction line intersections and to 2D nodes. Draw the profile as shown on the following diagram. To do so, click with at the appropriate points in the order marked on the diagram.



To quit continuous line creation mode, click . Note that the graphic line creation command is still active. Change the line type in order to draw the axis. To do so, pick the icon on the system toolbar and select CENTER type from the pull-down list.



Draw the centerline as shown on the following diagram.

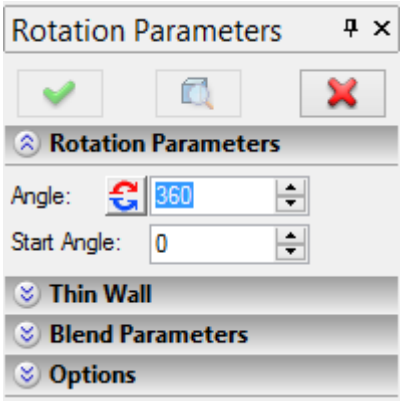


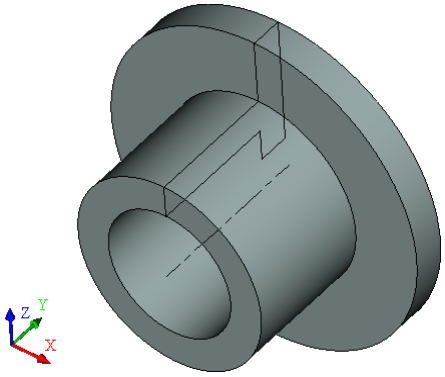
Creating Rotation Operation

One need not quit drawing explicitly in order to create a body of revolution. Simply call the "Rotation" command as follows:

Icon	Ribbon
	3D Model → Create → Rotation
Keyboard	Textual Menu
<3RO>	Operation > Rotation

The system automatically determines the rotation axis and the contour among the created lines to be used as a 3D profile. You can preview the result as a wireframe model. In our case, the rotation angle is 360 degrees. Note that the value of 360 degrees for the rotation angle is set by default in the property window. Just confirm the operation by pushing the check button , and the body of revolution is created.





Creating Holes

Next, we need to cut six holes in the part.

Holes can be created by different methods. The fastest and simplest one is using the dedicated "Hole" operation. The latter allows creating holes of standard shapes in solids, using templates provided in the T-FLEX CAD utility library. In this way, the user only needs to specify the position of the would-be hole within the solid and specify its type and dimensions.

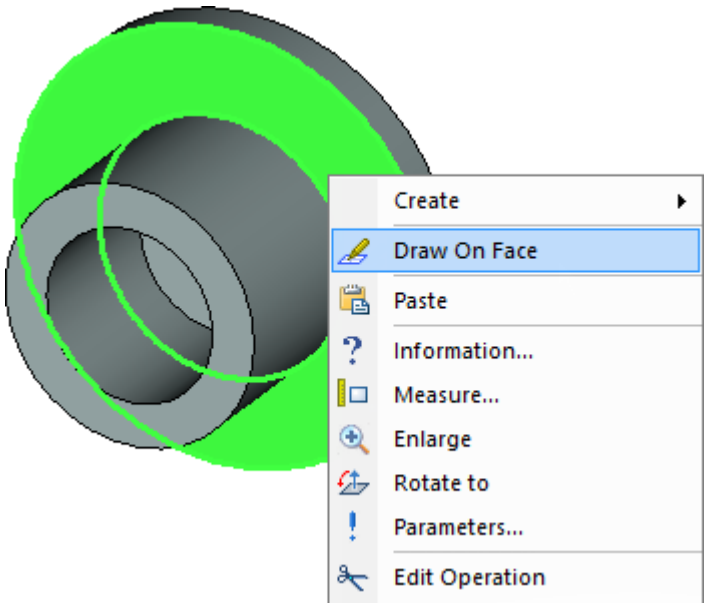
Non-standard holes and slots can also be created without using the dedicated operation. To do this, the user should create an additional solid representing the internal volume of the hole, and then "subtract" it from the main body's volume by a Boolean operation.

Let us consider both techniques.

Creating holes using a dedicated command

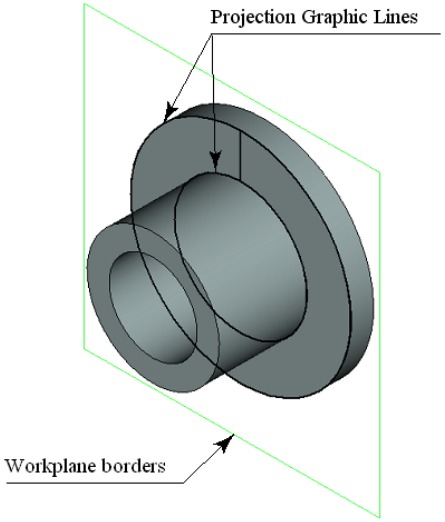
First of all, we will create 3D nodes corresponding to the centers of the would-be holes, on one of the part's faces. To create them, we will again use the 2D drawing mode for creating auxiliary 2D nodes on the respective face of the part.

To select a face, move the cursor to the desired element of the model so that it pre-highlights. At this point, press to invoke the context menu and select "Draw On Face" item in it (see the diagram).



If the desired element is not getting pre-highlighted, make sure the selector is set up for this element type. To change selector's settings, you can use selector filter icons on the system panel or select a combination of types from the list.

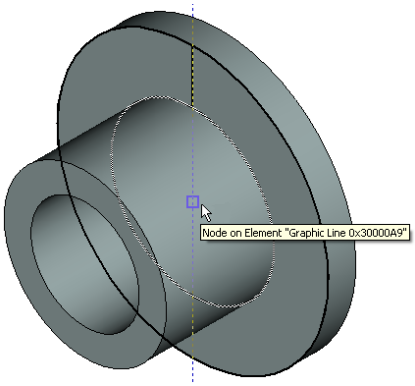
The "Draw On Face" command creates a new workplane based on the selected flat face. The selected face is automatically projected on this plane, and the mode of drawing in the 3D window activates. Further construction can use snapping to elements of the face projection.

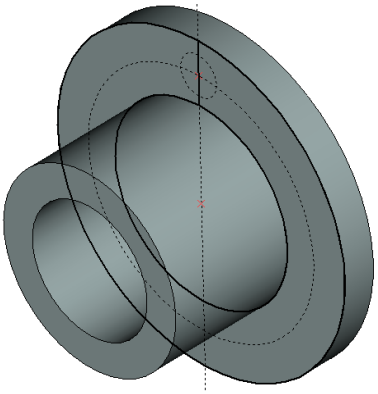


Call the command **L: Construct Line** again. Select the following option in the automenu,

	<V>	Create vertical Line

Move the pointer to the center of the circle - the circle center will be pre-highlighted. It is available for snapping to the vertical line. Click . The constructed line will be snapped to the 2D node which was automatically created at the circle's center.





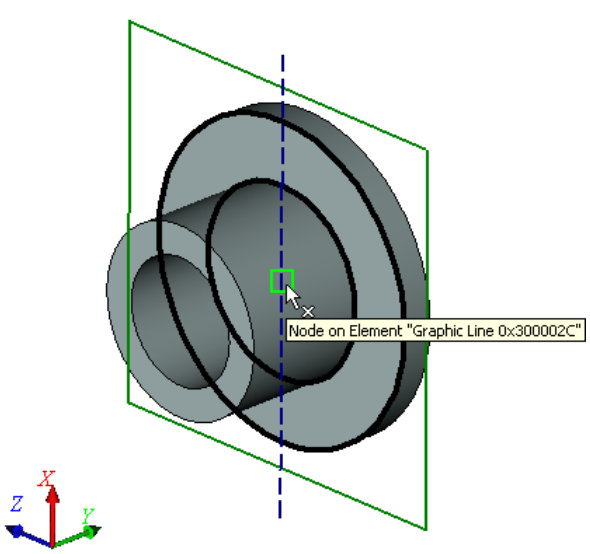
Call the command:

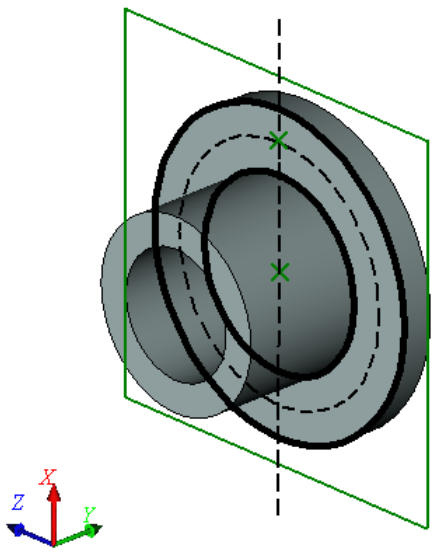
Icon	Ribbon
	Draw → Construct → Circle
Keyboard	Textual Menu
<C>	Construct > Circle

Point the cursor at the center node to select it as the center of the new circle. Specify the radius of the circle equal to 80 mm by typing in the property window.

Create a 2D node at the intersection of the vertical line and the new circle using the command:

Icon	Ribbon
	Draw → Construct → Node
Keyboard	Textual Menu
<N>	Construct > Node





Now, one can construct a 3D node based on the created 2D node. To do this, without leaving the drawing on face mode, call the command:

Icon	Ribbon
	Draw → Construct → 3D Node
Keyboard	Textual Menu
<3N>	Construct > 3D Node

When the 3D node creation command starts, move the pointer to the created 2D node and click . The 2D node will be highlighted, and the option will become accessible in the automenu. Click , and the 3D node will be created. It will be located in the plane of the selected face, while the 2D node will be its projection on that face.

The created 3D node will define the center of one of the six holes. There are two ways to define centers of other holes:

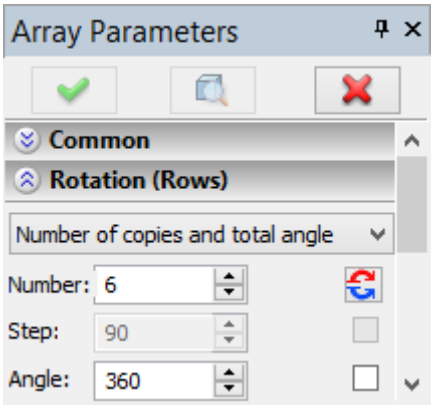
1. Construct five more 2D nodes on the same workplane and create the required 3D nodes based on the 2D ones (similar to the first 3D node creation);
2. Create the required 3D nodes by means of a 3D array based on the first 3D node.

The second method is faster, so we will use it in this case. Call the command **3AR: Create Circular Array**:

Icon	Ribbon
	3D Model → Create → Circular array
Keyboard	Textual Menu
<3AR>	Operation > Circular array

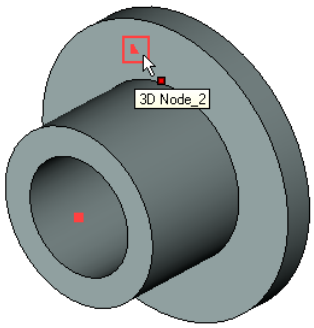
You do not have to explicitly finish the draw on face mode - it will terminate automatically upon launching the 3D array creation command.

In the command's properties window, set the array type as "Array of Construction". Then, in the 3D window, move the pointer to the created 3D node (the latter shall highlight) and click to select it. If all was done right, the name of the 3D node selected for copying shall appear in the properties window.

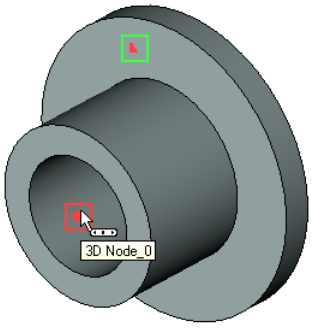


After that, you need to specify the rotation axis of this circular array. To do this, you can use a pair of 3D nodes automatically created when defining the axis for the rotation operation. Select these two 3D nodes one by one.

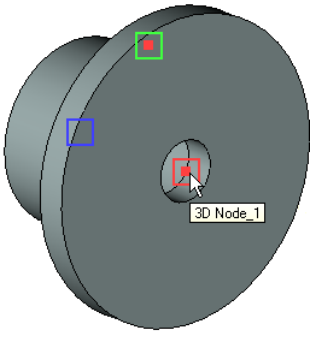
Next, let's specify what array parameters will be used ("Number of copies and total angle"), in the "Rotation (Rows)" section of the properties window, and enter the required values of the parameters (the number of copies – 6, the total angle – 360^0). To complete creation of this circular array of 3D nodes, simply click .



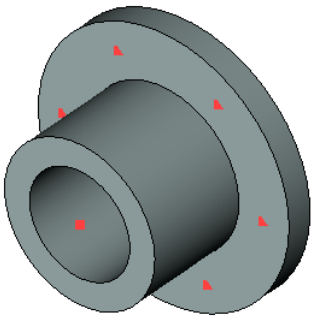
Selecting 3D node to copy



Selecting first 3D node of the array's axis of rotation



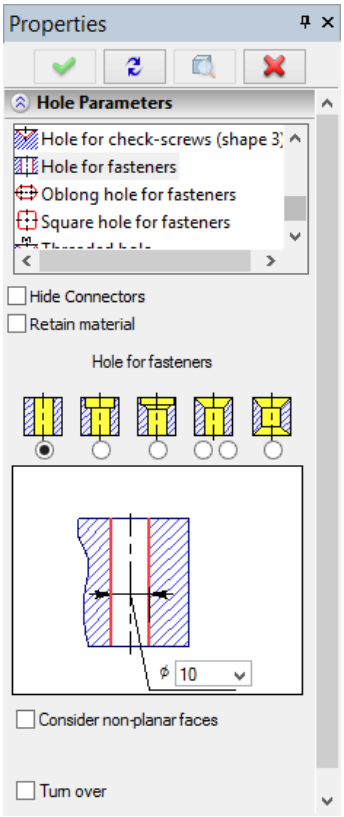
Selecting second 3D node of the array's axis of rotation



Result of creating an array of 3D nodes

After creating the array of 3D nodes, you can call the command **3H: Create Hole**:

Icon	Ribbon
	3D Model → Modify → Hole
Keyboard	Textual Menu
<3H>	Operation >Hole



Upon entering the command, you need to activate the option:

	<M>	Add Hole Array

After that, move the pointer to one of the 3D nodes of the circular array and click . In the command's properties window, select the hole type – "Hole for fasteners". The preview image of the holes being created will appear in the 3D window.

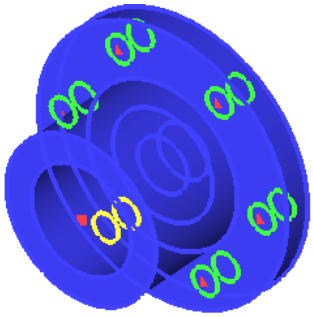
At the bottom of the properties window there is a pane with a schematic image of the hole of the selected type and fields for entering the hole parameters. Set the hole diameter to be 20 mm.

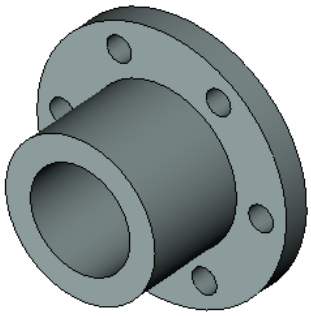
The following option will be automatically activated in the command's automenu for the holes of this type:

	<F>	Through all

When this option is set, the depth of the holes will be defined automatically by the part's thickness.

To complete creation of holes, simply click in the properties window or in the command's automenu.





Creating holes without using a special command

In this approach, initial constructions of the holes creation will be same as in the previous method.

You select a part's face and call the context menu command «Draw On Face». Call the command **"L: Construct Line"**. Select the automenu option . Construct a vertical line passing through the center of

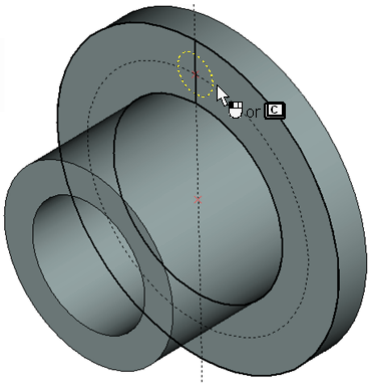
the circle (of the face's projection). Then, call the command "C: Construct Circle". Create the circle with the radius equal to 80 and the center at the central node. Similarly, create another circle with the radius equal to 10 mm at the intersection of the previous circle's circumferential and the vertical line.

Then draw graphic line over the newly created circle. To do so, call the graphic line creation command:

Icon	Ribbon
	Draw → Draw → Graphic Line
Keyboard	Textual Menu
<G>	Draw > Graphic Line

Pick the icon on the system toolbar, and select CONTINUOUS type from the pull-down list.

Move the cursor to the just drawn circle. The circle will get highlighted, and the cursor gain a circle glyph. Once so, click , that applies graphics over the whole circle. On a crowded drawing with multiple elements near the cursor, snapping may occur to an unintended element. In such cases, the circle can be explicitly selected by typing <C> key. This will select the circle nearest to the cursor.



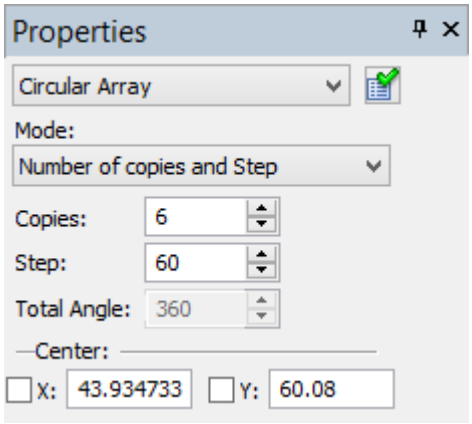
Two possible approaches are available for further construction. In one of them, you could draw five more circles, to extrude and subtract them all from the body of the part. Alternatively, extrude just a single circle, and then "replicate" the resulting hole by means of the circular 3D array.

The first approach by steps:

A handy tool for creating the required number of copies of the circle is the "Create Circular Array" command:

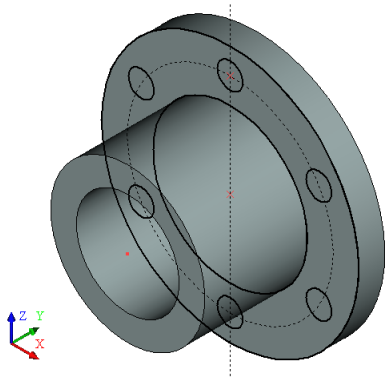
Icon	Ribbon
	Draw → Additional → Circular array
Keyboard	Textual Menu
<XR>	Operation > Array > Circular

Once in the command, the system waits for selection of a graphic entity. Move the cursor to the image of the hole and click to select the graphic line. That's all to be selected, therefore, press in the automenu.



Now the system expects selection of the array center, which must be a 2D node. Note that the default setting for the number of copies of the circular array is 4, and we need six. Therefore, specify the correct parameters in the property window, as shown on the diagram at right.

Now you can select the node. Move the cursor to the center node and click . The result will look as on the following diagram.

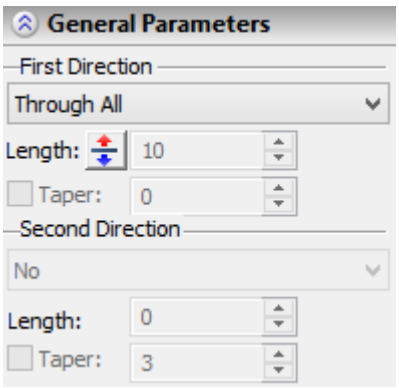


Next, we will need to call the profile extruding command,

Icon	Ribbon
	3D Model → Create → Extrusion
Keyboard	Textual Menu
<3X >	Operation > Extrusion

The system automatically makes a 3D profile from the drawn lines and sets the extrusion vector normal to the profile.

The amount of extruding, that is, the depth of the hole, can be defined in various ways (by a numerical value, by the length of the direction vector, etc.). In this particular case, we need to get a through hole, that is the one penetrating the entire width of the part.



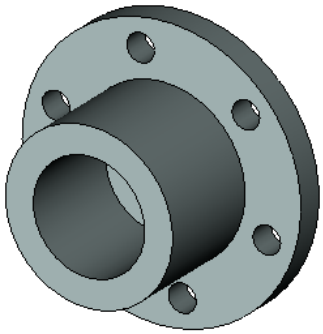
In the properties window, set the "First direction" parameter to the value "Through all", by selecting from the drop-down list. In this definition, the extrusion amount is determined by the thickness of the auxiliary bounding body. Additionally, the created extrusion is automatically subtracted from that body. This method of specifying the extrusion distance was devised specifically for quick creation of through holes in various bodies.

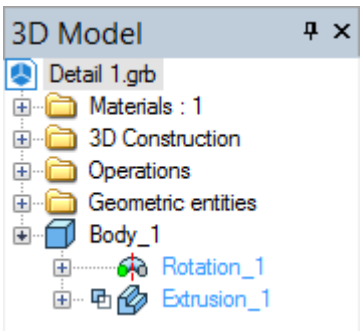
The option of creating a Boolean operation will automatically activate in the automenu:

	<Ctrl><B>	Subtraction

Since only one body is present in the scene at the time of creating the extrusion, it will be automatically selected as the bounding body and the first operand of the Boolean subtraction.

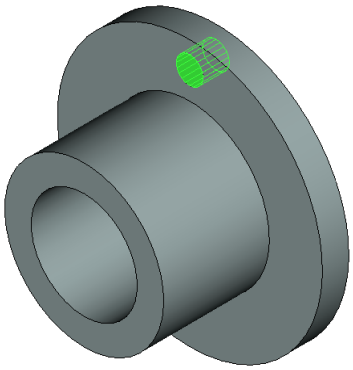
Upon clicking , two operations will be created at once – an extrusion and a Boolean operation.





The second approach by steps:

Call the profile extruding command . Specify extrusion length parameter equal to 20 in the reverse direction, and confirm the extrusion operation.



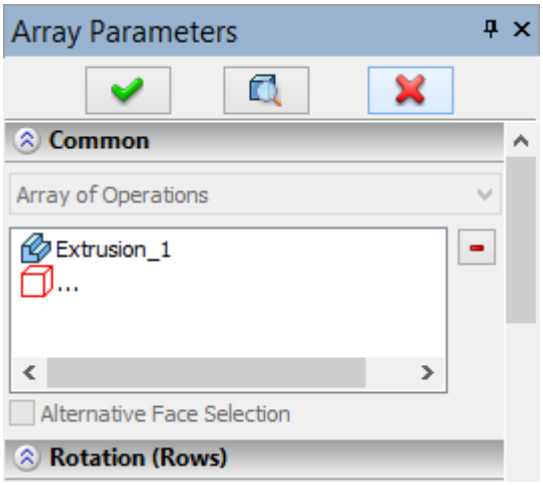
Next, let's call the circular array creation command,

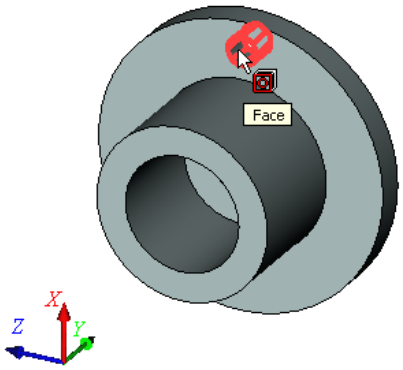
Icon	Ribbon
	3D Model → Create → Circular array
Keyboard	Textual Menu
<3AR>	Operation > Circular array

In the command's properties window, select the array type – "Array of Faces".
The following option will be automatically activated in the automenu:

	<N>	Select Faces to Copy or their boundary Edges or Loops

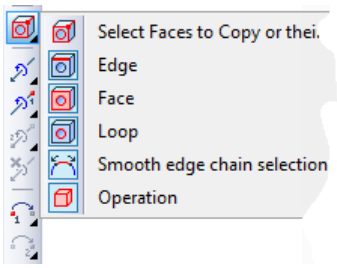
Aim the pointer at the cylindrical face of the hole and click .



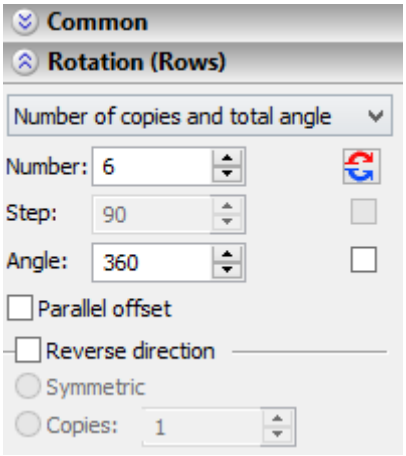


If edges or loops highlight instead of the face, then you can do any of the following:

- rest the pointer over the face until the multiple selection widget appears next to it. Then scroll through the list of elements offered for selection by rotating the mouse wheel until you get the desired face;

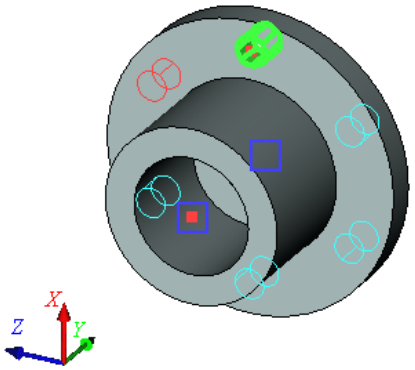


- before selecting the face, set up the list of 3D objects available for selection (banning from selection anything but faces). To do this, double-click the icon of the faces filter . The list of selectable objects can be also set up by the drop-down menu of the option : point at the option, depress and hold the mouse button for a second. In the appearing menu, disable all flags except "Faces".



Once the intended operation is selected, the system expects input of the axis of revolution for the array. The axis can be assign by the same two 3D nodes as those used in the first method of creating holes. Select these two 3D nodes one by one.

You can define the array parameters in the properties window – number of copies and total angle.



Press the button in the automenu to complete the operation. As a result, five new faces will be "embedded" in the original body, making up the additional holes.

Creating a Blend

The next step is finalizing 3D model creation. This includes creation of a chamfer and a rounding.

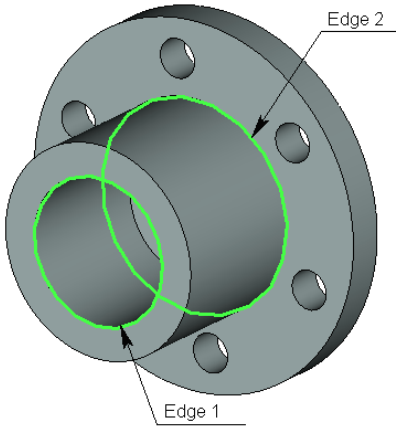
Call the command:

Icon	Ribbon
	3D Model → Create → Blend → Edge
Keyboard	Textual Menu
<3DE>	Operation > Blend >Edge

Pick the following option in the automenu,

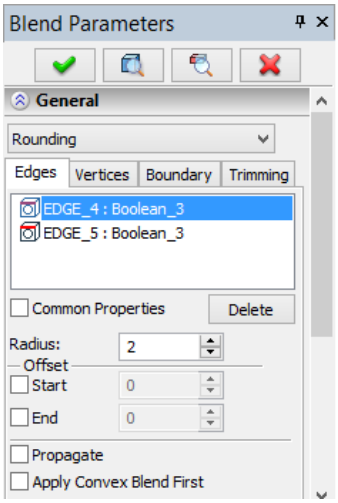
	<E>	Select Edge

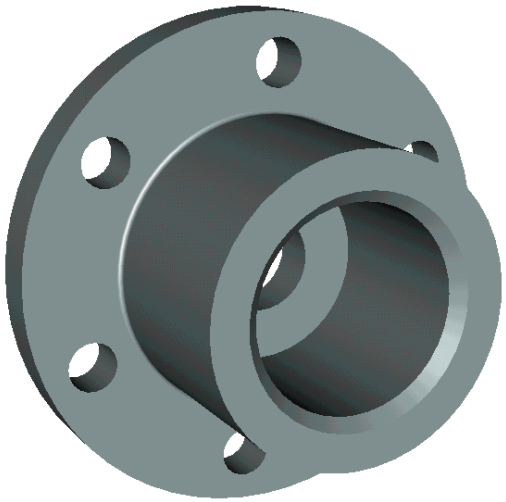
The system is waiting for an edge selection. Select two edges in the order shown on the following diagram.



Next, specify the operation parameters. Each edge can be assigned its specific set of parameters in the property window by unchecking the "Common Properties" option.

Select the first edge in the list and specify the blending type as Rounding with Radius equal to 2 mm. For the second edge, define the blending type as Chamfer (Offsets), with Offset1 and Offset2 both equal to 5 mm. Confirm the operation by pressing . The result appears as shown on the following diagram.

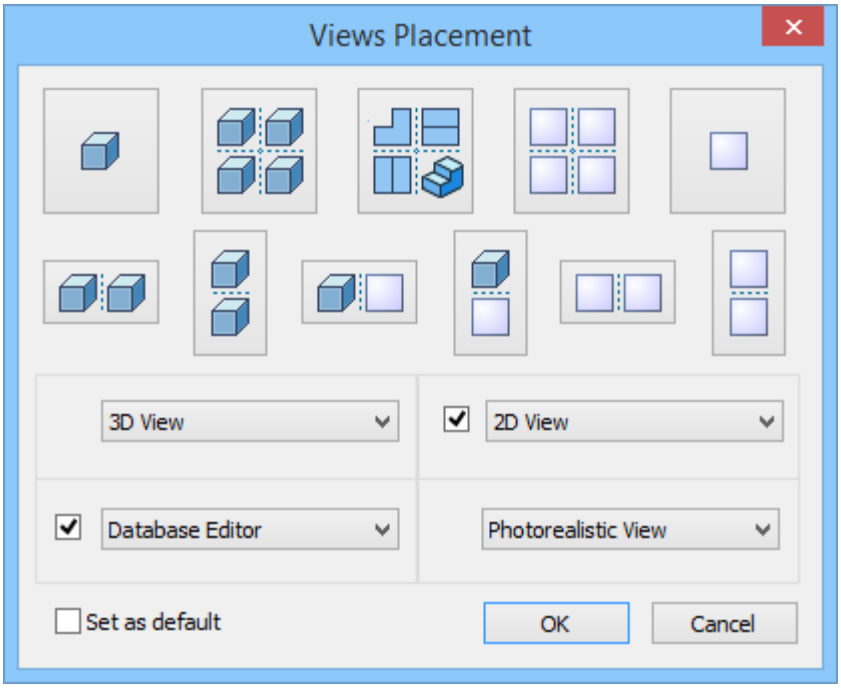




This completes creation of the given 3D model.

Creating a Drawing

Open a 2D window. Do this as follows. Move the cursor to the bottom-left corner of the 3D window and locate the split box before the scroll bar. As the cursor approaches the split box, it changes to juxtaposed arrows. Press and dialog box "Views Placement" will appear. Choose and press **[OK]**.



The 2D and 3D windows will be located side by side of the vertical split bar. The same result could be obtained by calling the command:

Icon	Ribbon
	View → Window → Split Vertically
Keyboard	Textual Menu
<WSR>	Window > Split Vertically

A new window can also be opened with the command:

Icon	Ribbon
	View → Window → New Window
Keyboard	Textual Menu
<WO>	Window > New Window

To activate the newly opened 2D window, place the cursor within and click .

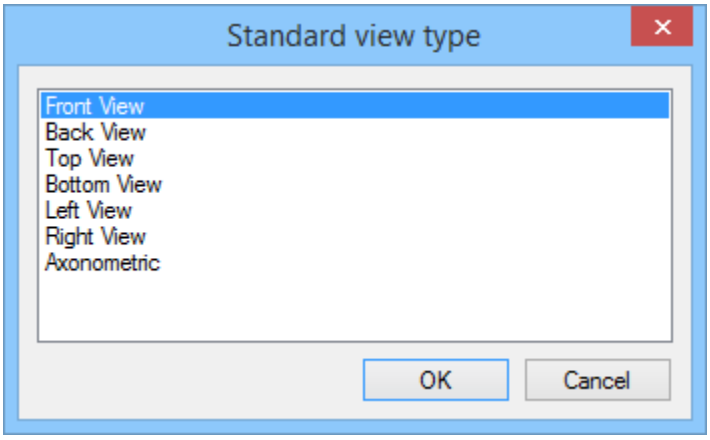
Now we can generate projections and sections. Call the projection creation command:

Icon	Ribbon
	Draw → 2D Projection
Keyboard	Textual Menu
<3J>	Draw > 2D Projection

Press the automenu button:

	<6>	Create standard projection

In the coming dialog box, select "Back View" and press **[OK]**.



A green box will be displayed on screen, marking the size of the projection to be created. Next, use the option:

	<M>	Change Projection Placement

Define the attachment point of the projection in absolute coordinates by clicking at the desired location in the 2D window. Press in the automenu to confirm projection creation.
Next, let's generate a section of the part.

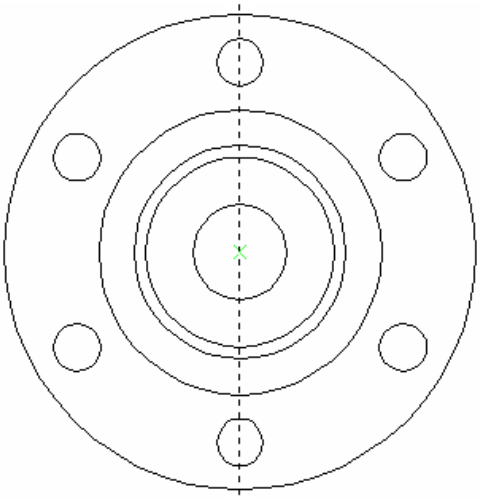
First, we need to do some auxiliary construction. Call the line creation command,

Icon	Ribbon
	Draw → Construct → Line
Keyboard	Textual Menu
<L>	Construct > Line

Pick the automenu option for creating vertical lines,

	<V>	Create vertical Line

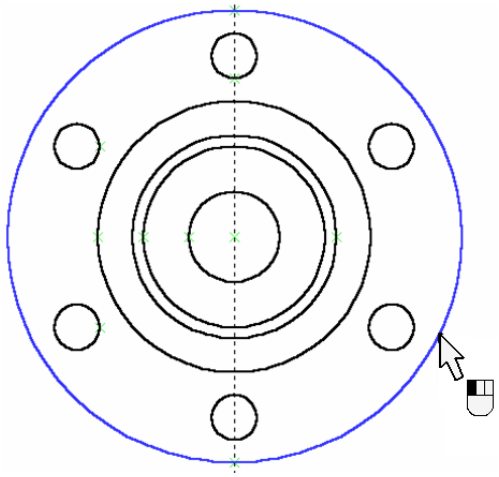
Point at the center of the circle and click, creating a line along the symmetry axis of the projection.



Then, call the circle creation command:

Keyboard	Textual Menu	Icon
<C>	"Construct > Circle"	

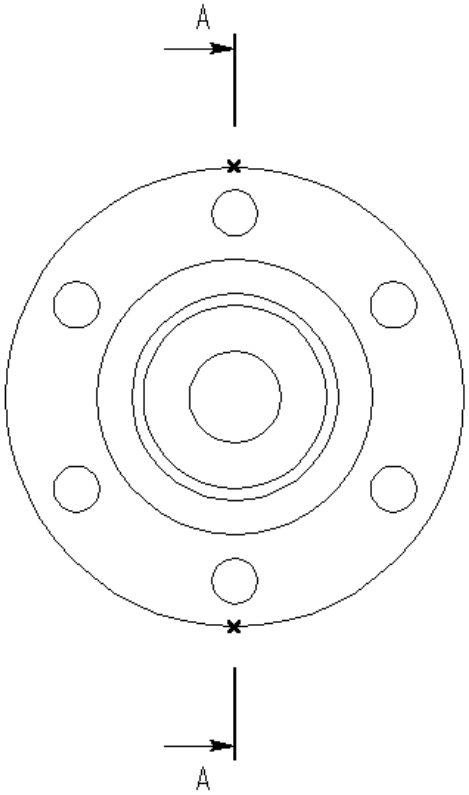
Next, we need to reconstruct a construction circle from a graphic circle on the projection. Move the cursor to the circle graphic line and click.



Now, we can create the section points snapped to construction line intersections. Call the section creation command,

Keyboard	Textual Menu	Icon
<SE>	"Draw > Section"	

Select two intersection points between the line and the circle one by one. Press automenu icon to confirm the section creation.



The next step will be creating a projection of the newly created 2D section. Call the 2D projection creation command,

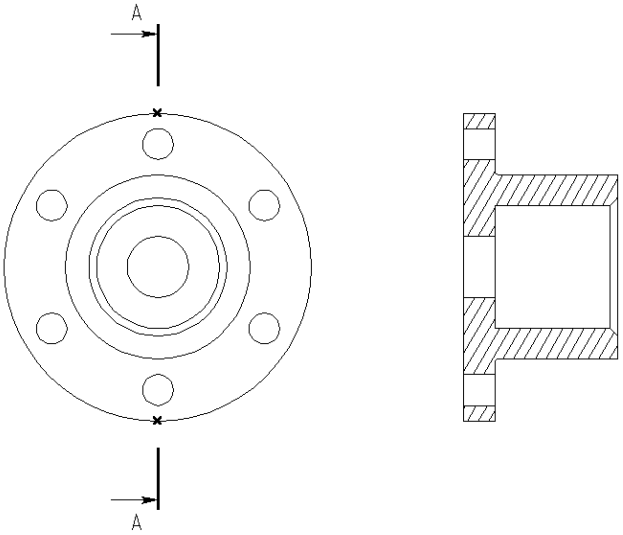
Keyboard	Textual Menu	Icon
<3J>	"Draw > 2D Projection"	

In the automenu, select subsequently the following options:

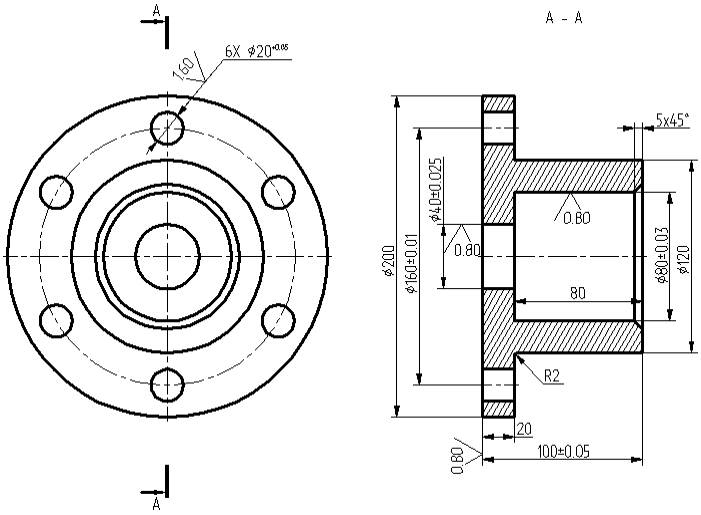
	<2>	Create section view
	<L>	Select Section defining Projection

Point the cursor at the section symbol and click . The outline of the section appears rubberbanding with the cursor. The projection outline will slide along the projection direction adjusting to the cursor movement. To fix the placement, click .

To complete projection creation, press in the automenu.

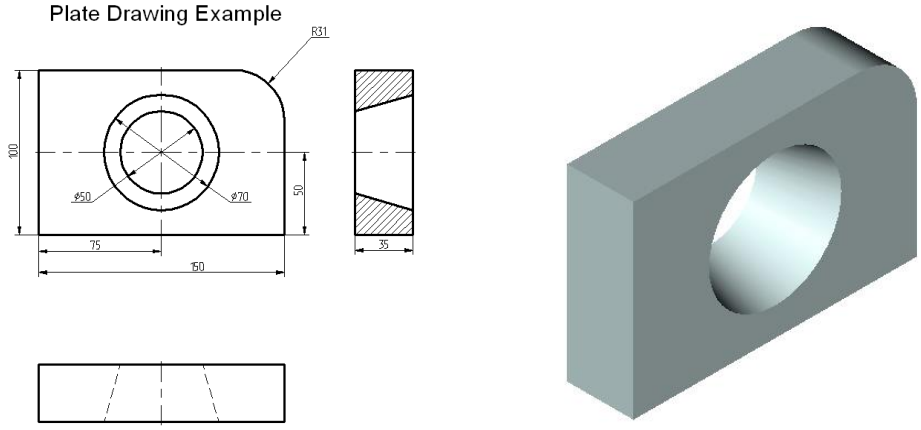


If necessary, the projection entities (graphic lines, arcs and circles) can be used as references for dimensions, auxiliary graphic lines such as centerlines, and drawing layout attributes.



"From Drawing to 3D Model" Approach

Let's create a 3D model from a 2D drawing. The model will be based on the familiar drawing of a plate with a conical hole. This drawing creation was reviewed in details within the brief introductory course of 2D design. The following diagram presents the 2D drawing and the 3D model of the plate to be created.



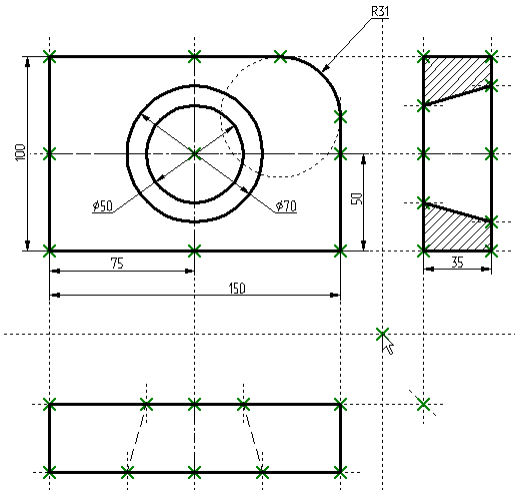
The drawing documents are located in the library "...\Examples\Documentation examples. Introductory course\ Detail with drawing detailing; Step 1. Plate drawing; Step 2. Hatch and Profile; Step 3. Extrusion; Step 4. Rotation; Step 5. Boolean operation".

We will pass through several design steps in order to create the 3D model of the plate. The first design step includes creation of the workplanes that will further be used for all the rest of 3D element construction. In the second step, we will perform extrusion of the defining contour of the plate at the specified depth in order to obtain the three-dimensional body of the plate. This will be done by extrusion operation. The next design step is creation of the tool body for the conical hole. We will create it using rotation operation. Finally, we will use Boolean operation in order to create the three-dimensional model of the plate in its final shape. We will subtract the second, tool body, created by rotation operation, from the original body created by extrusion.

The first step in 3D model creation will be creation of the workplanes, as was mentioned earlier. Note that the workplanes should be constructed in such a way to maintain projective relationship between the views.

If not explicitly defined within the drawing, the projective relationship between the views can be established by fitting the drawing with associative auxiliary construction.

The 3D model creation process requires two workplanes. Let's begin with creating a 2D node defining the separation point of the views. To make the node, first create two orthogonal lines. Place these lines as shown on the diagram below.



Then create the workplanes.

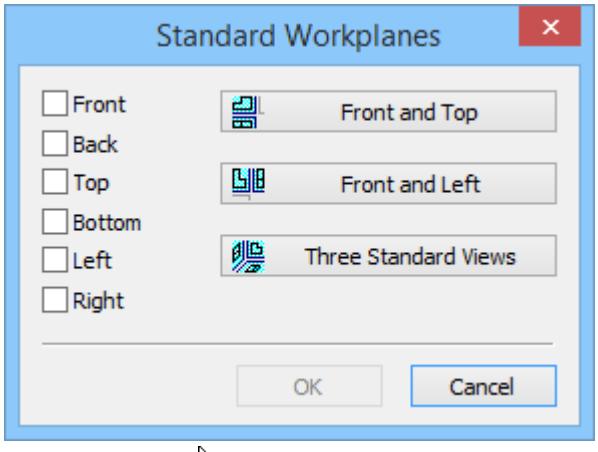
Call the command:

Icon	Ribbon
	3D Model→ Construct → Model
Keyboard	**Textual Menu**
<3W>	Construct > Workplane

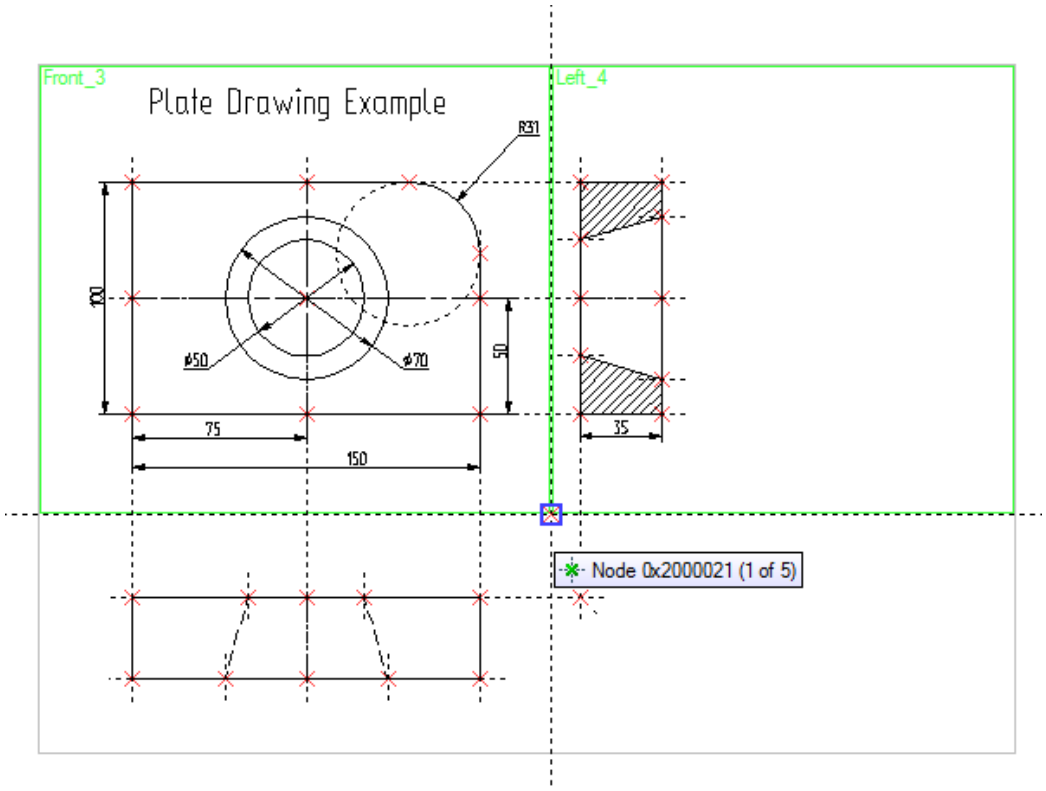
Set the automenu option:

	<S>	Construct standard Workplane

In the coming dialog box, press the button [**Front and Right]**

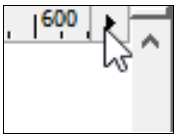


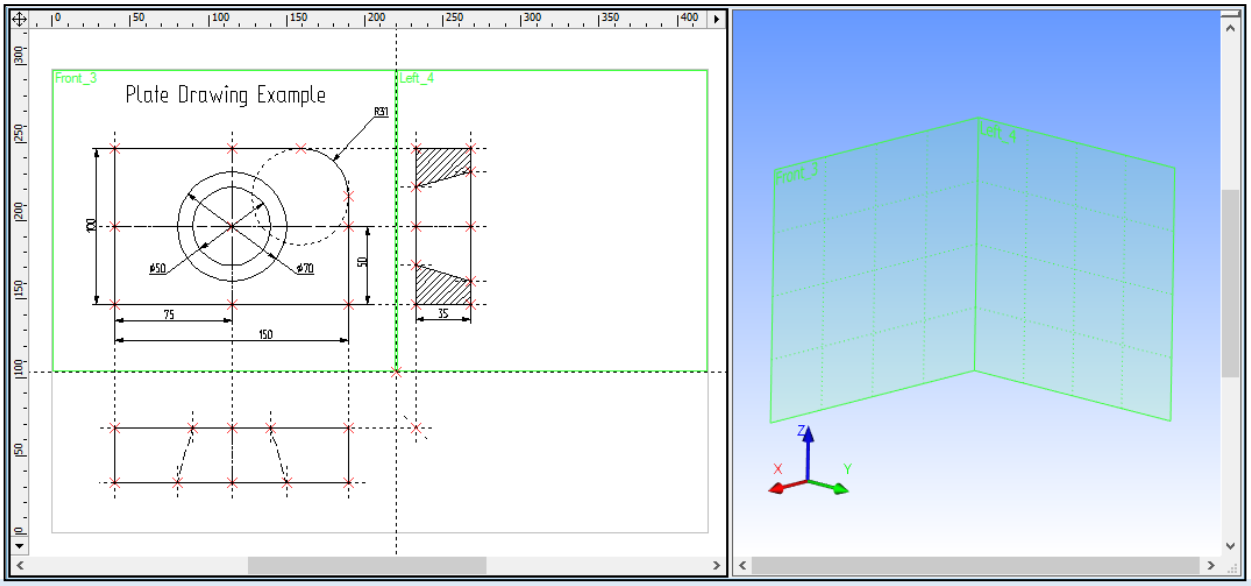
The cursor on screen gains the node mark . This mark means the system is now in selection mode. Move the cursor to the newly created node to be used for workplane snapping, and click or type <N> key. Two horizontal workplanes will be created on screen for the main (elevation) and the left side view.



Then quit the command by clicking or pressing icon in the automenu. One can also type the <Esc> key.

Now we can proceed with creation of 3D auxiliary elements. First, open the 3D view window of the T-FLEX CAD system. The 3D view can be opened by clicking on a button with a triangular arrow in either the top-right or the bottom-left corner of the current drawing window next to the scroll bars. Place the cursor over such a button in the top-right corner, and click on it with . The current drawing window will be split vertically into two windows. One of the windows will be displaying the two-dimensional drawing, while the other, 3D view one, reflect the three-dimensional elements and bodies as they appear in the process of the 3D model creation.

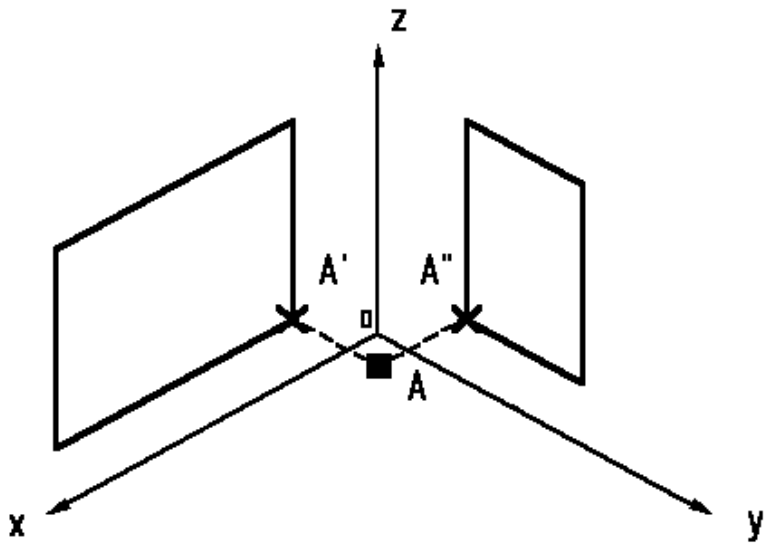




Set the workplanes property **Show in 3D View** in order to have them displayed in the 3D window. To do it, it is necessary to select these planes in the tree of the 3D model, call the context menu with the help of and invoke the command "Parameters" in it.

To create the main body of the 3D model of the plate, extrude the perimeter contour at the length equal to the plate depth. To define the extrusion based on the existing 2D drawing, we need to create auxiliary 3D elements: a 3D profile and 3D nodes in a special way.

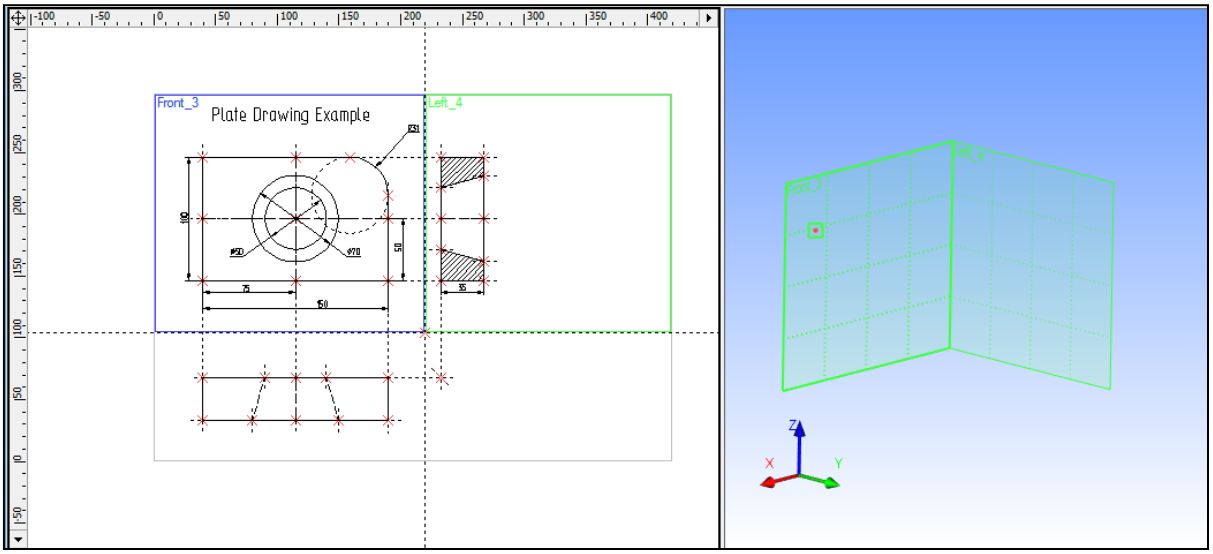
Let's create 3D nodes using existing 2D nodes and the workplane mechanism. To create a 3D node, one can specify either one node on one workplane, or two nodes on two different workplanes. In the latter case, the two nodes must be relatively located as two projections on their respective workplanes of one would-be 3D node.



Call the command:

Icon	Ribbon
	Draw → Construct → 3D Node
Keyboard	Textual Menu
<3N>	Construct > 3D Node

Next, proceed with selecting 2D nodes. Move the cursor to a node of the first projection of the would-be 3D node and click as shown on the diagram below. The 2D node and its workplane will get highlighted.



Normally, upon entering a 3D command, the system expects further user actions. Depending on these actions, the system assumes one or another specific mode. This mode usually depends on what kind of object was selected by user. The necessary options may automatically be set in the automenu depending on the user actions and the specific mode. The user can simply proceed with a predefined sequence of actions following the prompts in the status bar.

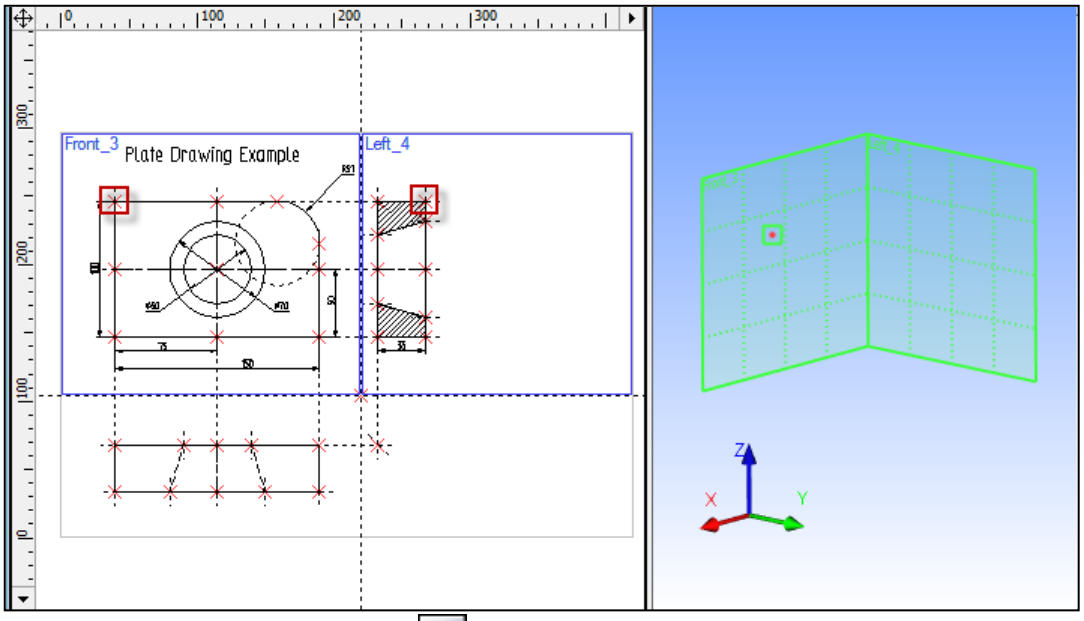
Once the first 2D node is selected, the system automatically turns on node creation by two projections:

	<J>	Create 3D Node by two projections

Once this mode is on, the user will be subsequently prompted to select the first, then the second projection 2D node, on and on, as he keeps selecting:

	<F>	Set 3D Node 1st projection
	<G>	Set 3D Node 2nd projection

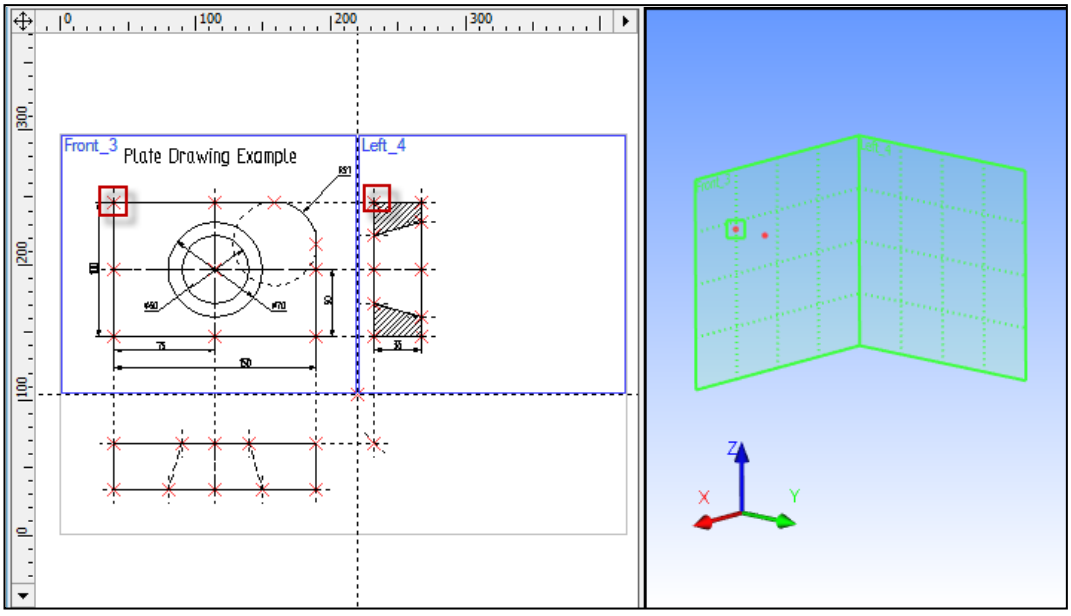
Move the cursor to the second node, representing the second projection of the 3D node, and click . The 2D node and the respective workplane, where the selection occurs, will get highlighted.



Confirm the 3D node creation by pressing icon in the top portion of the automenu. The 2D drawing elements get de-highlighted, and the newly constructed 3D node appears on the 3D scene.

> Remember that some 3D commands are inaccessible while the 2D view of the current drawing is active. To make these commands accessible, activate the 3D view window. To do so, click anywhere within the 3D view window. Switching from 3D view back to 2D view is done in the same way.

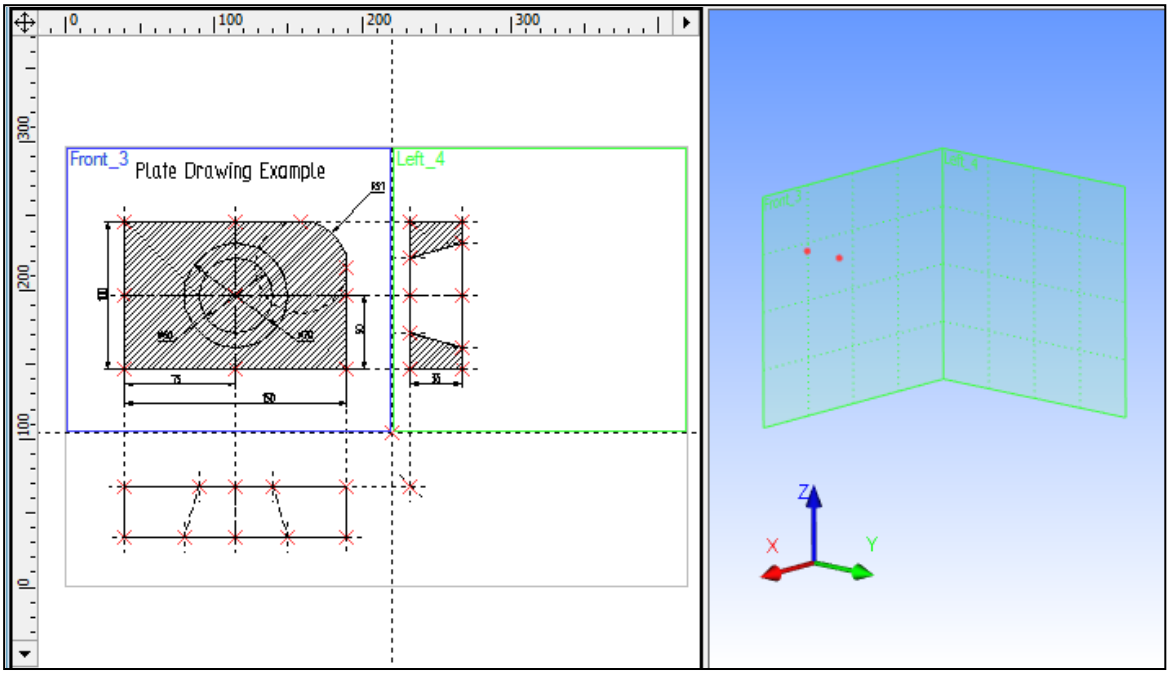
Let's create the second 3D node. We are still within the command **3N: Construct 3D Node.** Construct the second 3D node. To do so, we need to select two nodes as shown on the diagram below.



To confirm the second 3D node creation, press the icon in the automenu and exit the command. Two 3D nodes will be displayed in the 3D window.

The next step is creation of a 3D profile. We need to establish a relationship between the profile and the drawing. Drawing on the active workplane is not available to us since there is more than one workplane on one drawing page. Therefore, we will proceed with the 3D profile construction based on a hatch and a workplane. First, let's create the hatch.

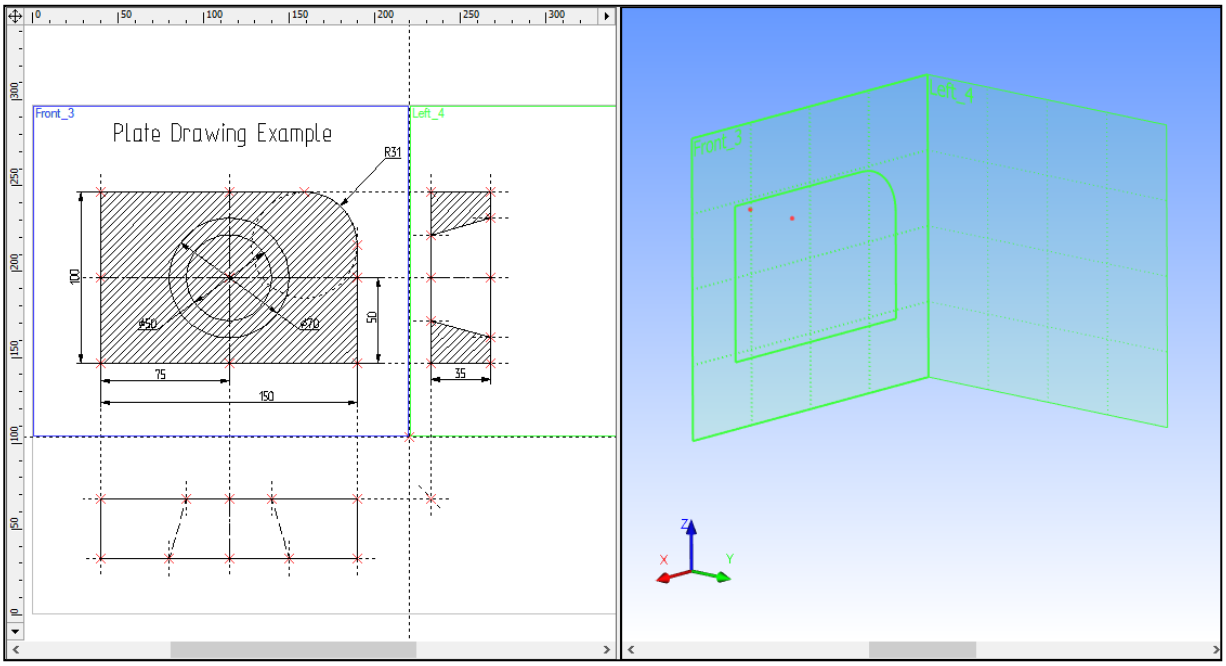
Enter the command **H: Create Hatch**. Create a hatch **A** on the elevation view. The hatch can be made invisible by setting the **Fill method** attribute of the hatch parameters to **Not Visible** mode. This is necessary for preserving the original look of the part drawing.



Now call the command:

Icon	Ribbon
	3D Model → Construct → 3D Profile
Keyboard	Textual Menu
<3PR>	Construct > 3D Profile

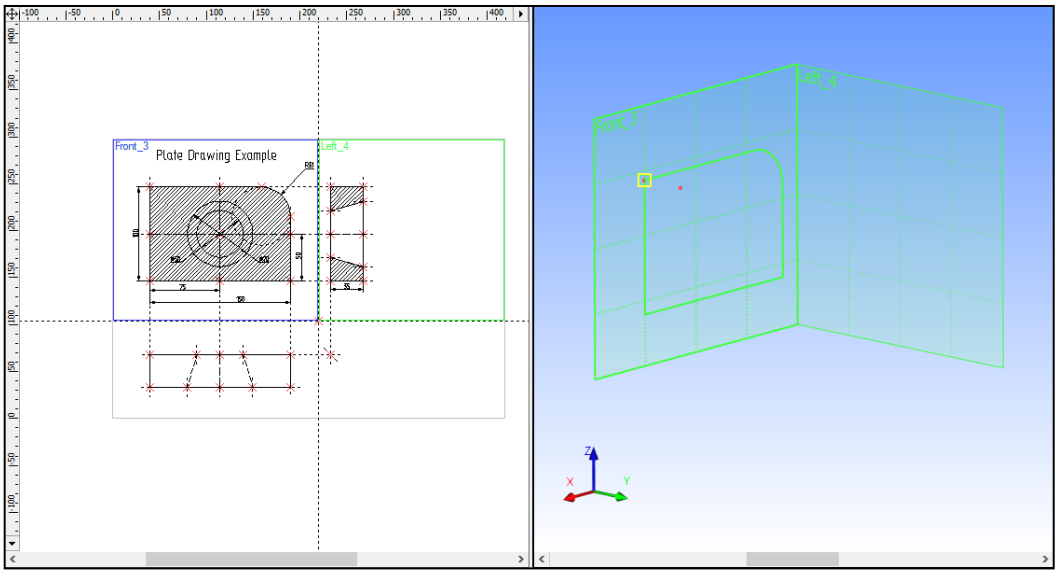
The system waits for further user action. Select the hatch **A** in the 2D window. The contour of the selected hatch and the workplane will get highlighted, and the respective profile will appear in the 3D window.



Once the hatch is selected, the system enters the mode for constructing a profile based on a hatch, and the following option activates in the automenu:

	<M>	Select 3D point for fixing Profile plane

The 3D node mark appears next to the cursor. Use the cursor to select one of the 3D nodes in the 3D view window. The profile will relocate to pass through the selected 3D node. The node itself and all its 2D and 3D construction references will be highlighted.



Confirm the 3D profile creation by pressing the icon in the automenu. All elements in either window will then be de-highlighted. We have just created a 3D profile. Quit the command to continue.

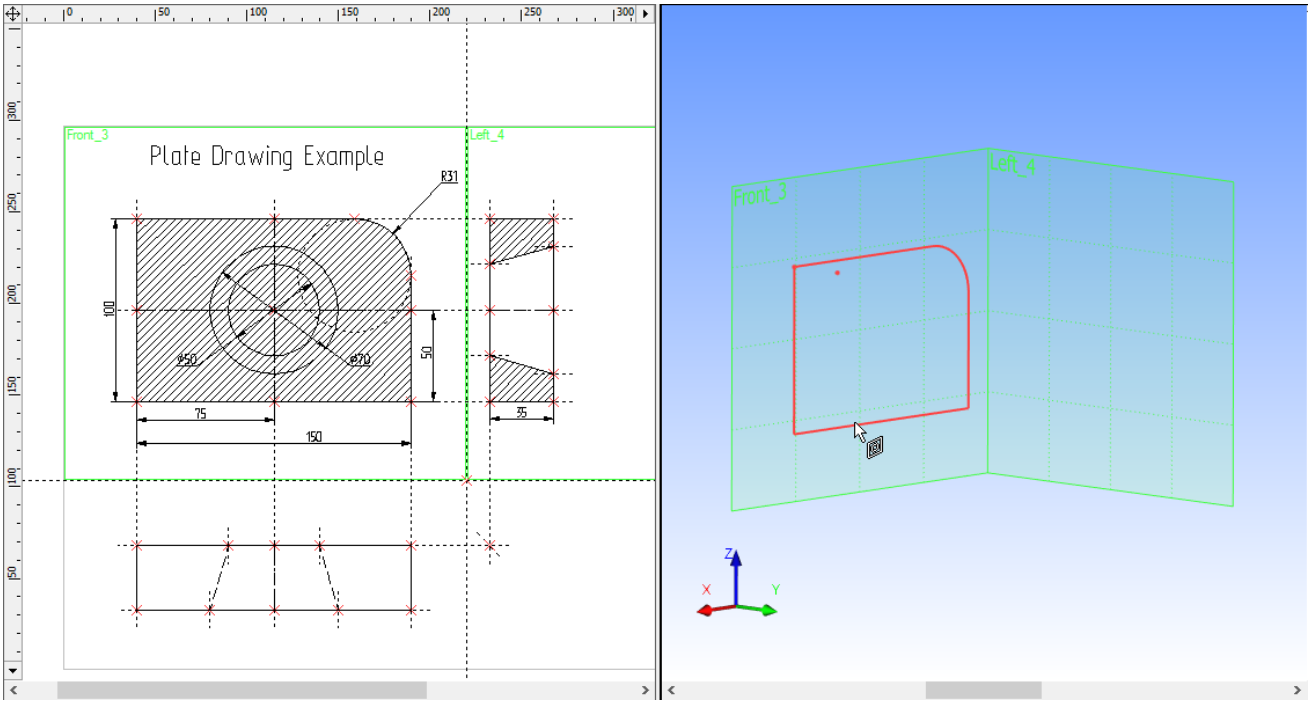
All auxiliary 3D elements have been constructed that are required for a three-dimensional body creation by extrusion.

Let's create an extrusion operation . The following option will be set in the automenu by default,

	<R>	Select Contour

As we are to use the just created profile as the contour, let's set up the filters for 3D profile selection. This can be done on the system toolbar by on the desired filter icon as well by clicking while holding the <Ctrl> key. All the rest of the filters will get turned off.

Move the cursor to the 3D window, place it by the profile and point at an edge of the profile. The cursor will gain a mark . Click to select the profile. The profile will be highlighted in the 3D window.



Keep in mind that selection of 3D elements required by 3D commands can be performed either in the 3D, or in the 2D view window. This is especially relevant to selection of 3D nodes and 3D profiles (contours) that can be selected in the 2D view as nodes and hatches respectively.

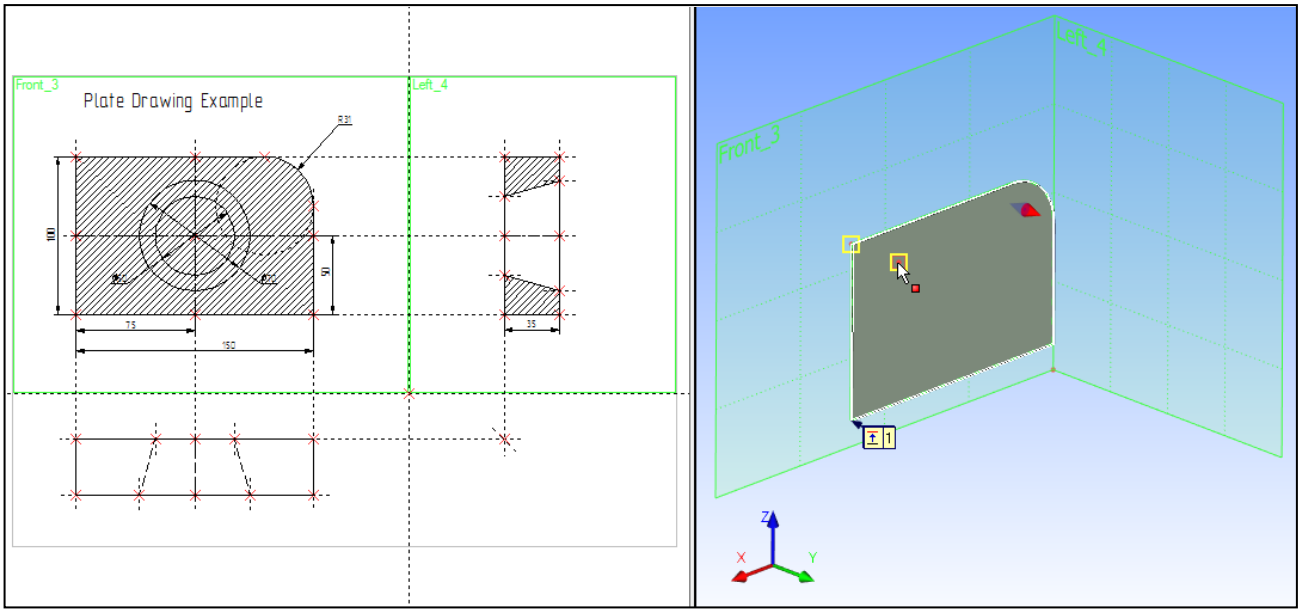
Next, we need to specify direction and the depth of the extrusion. In this example, the depth of the extrusion is equal to the thickness of the plate. This dependency can be maintained if the extrusion vector is defined based on the existing 3D nodes.

Once the 3D profile is selected, turn on the following option in the automenu,

	<F>	Select starting Point of Extrusion

Several views of the object can be used for specifying a 3D point. Make sure of the correct selection filter settings for 3D node selection on the system toolbar or in the pull-down list of active options.

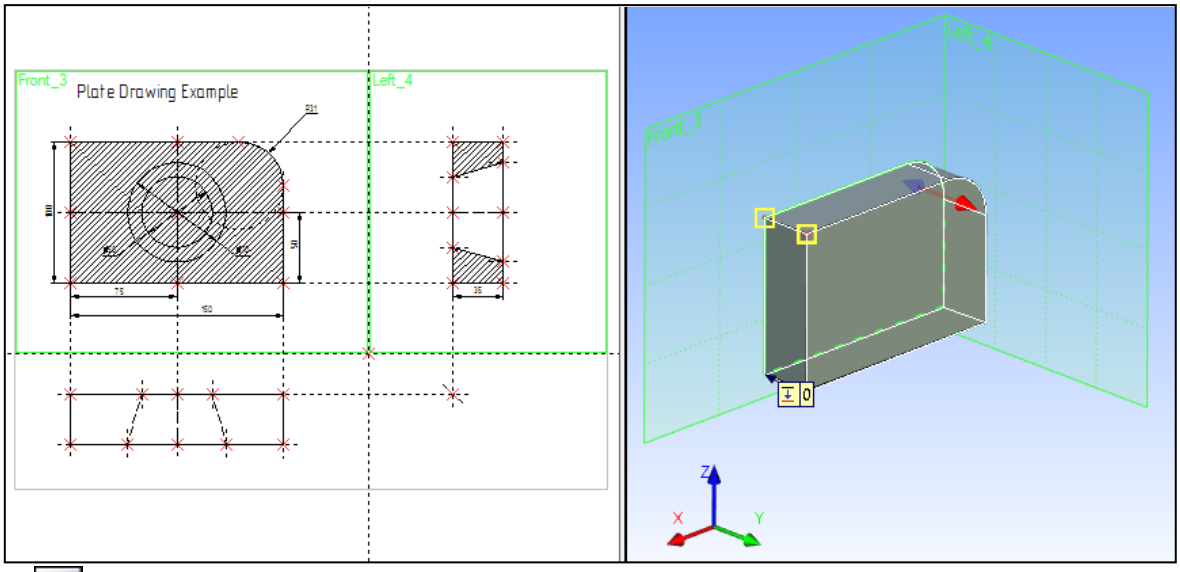
As the cursor is pointed at a 3D node, it gets the node mark . Select the 3D node, which the 3D profile plane is passing through.



After selecting the first 3D node, another option will be automatically turned on in the automenu,

	<S>	Select ending Point of Extrusion

Select the second 3D node with the cursor (point and click as usual). The preview of the current extrusion operation will appear on screen.

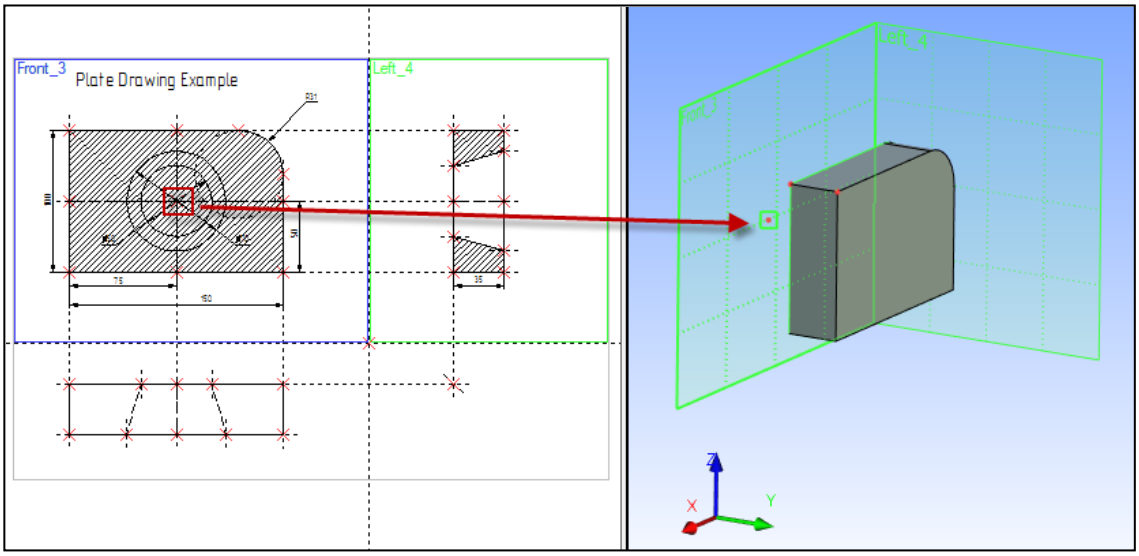


Press the icon, thus completing the extrusion operation. Quit the command.

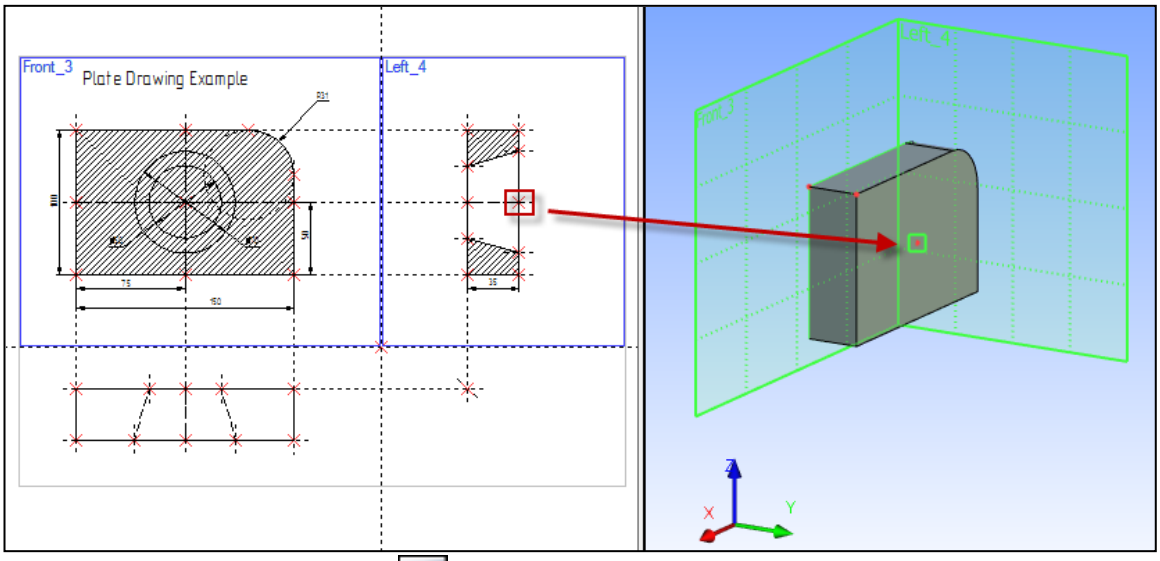
To create a hole in the three-dimensional model of the plate, we need to create the second 3D body. The latter will be used in the next step as the tool body for Boolean subtraction from the original extrusion. The 3D model of the hole is easiest to create by rotation operation.

To create a three-dimensional model by rotation, we need to define auxiliary 3D elements: the 3D profile and the axis of revolution for rotating the profile. To create the axis, we will need to construct 3D nodes.

Begin construction by calling the command **3N: Construct 3D Node.** Enter the command, and select a 2D node as shown on the diagram below, by pointing and clicking. A highlighted 3D node will appear in the 3D window.

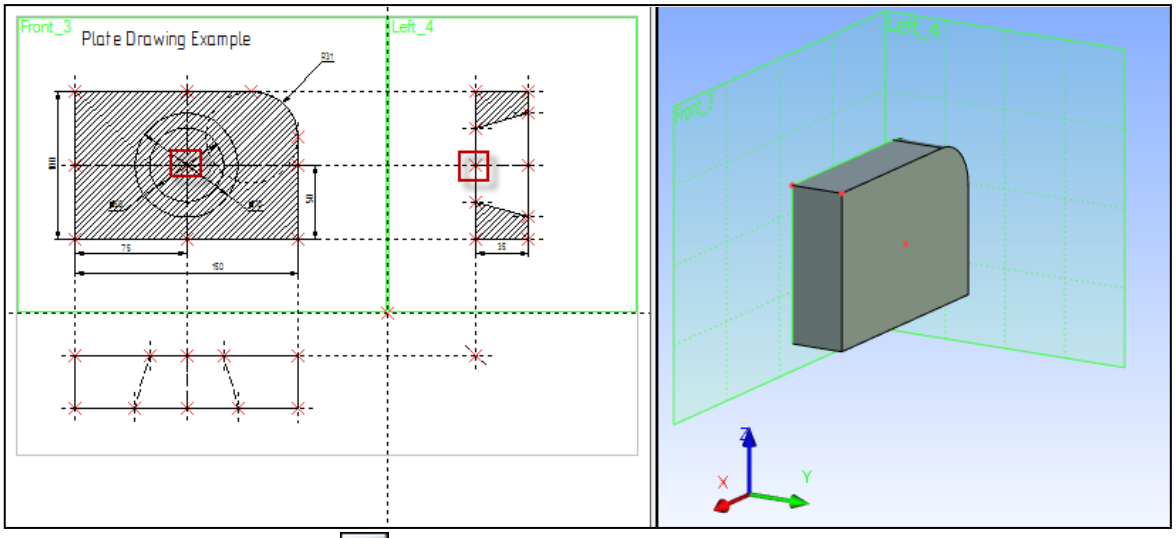


Next, move the cursor to the second 2D node, defining the second projection of the 3D node, and click. The highlighted 3D node in the 3D window will move along the Y-axis.



Confirm 3D node creation by pressing icon in the automenu. Highlighting turns off in both windows.

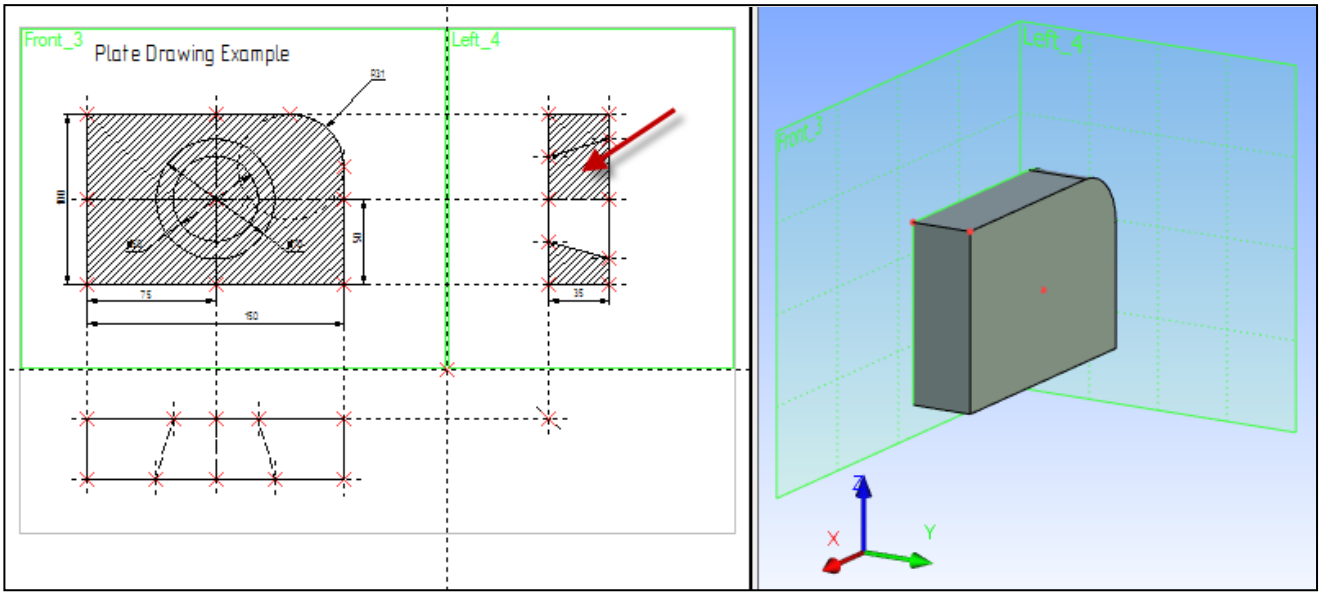
Create the second 3D node. It is necessary for defining the axis of revolution. The system is still within the command **3N: Construct 3D Node.** Go ahead with the second 3D node creation. Do this by selecting two 2D nodes as shown on the following diagram.



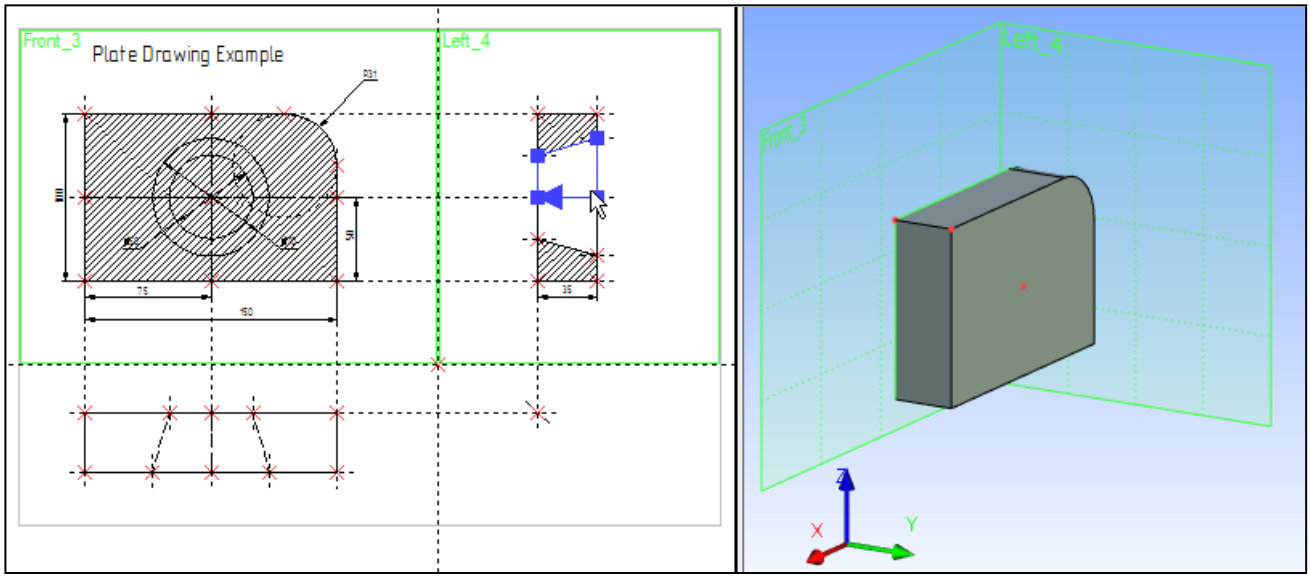
After the final selection press the icon in the automenu, and quit the command.

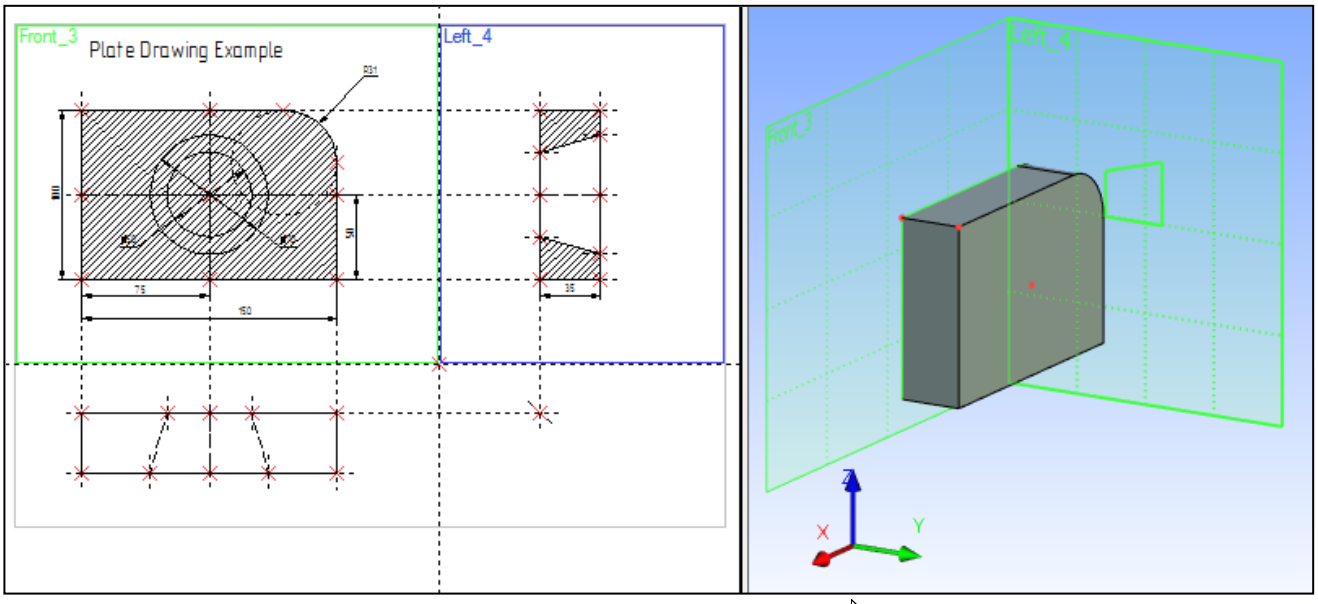
The next step is creation of the 3D profile.

Create a hatch B on the side view of the 2D drawing using the **H: Create Hatch** command. Make the hatch invisible to keep the actual drawing legible.



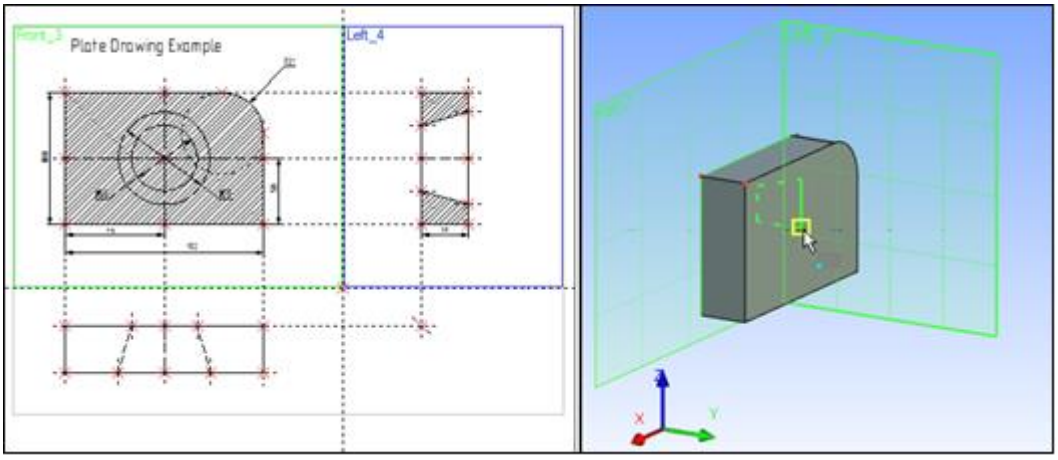
Call the command **3PR: Construct 3D Profile**. Use the cursor to select the hatch B. The selected hatch contour and the workplane will get highlighted, and a 3D profile appear in the 3D window.





Once the hatch is selected, the cursor will gain the 3D node mark prompting the user for selecting a 3D node for locating the contour plane in space. Use cursor to select one of the 3D nodes in the 3D

window. The profile will snap to the selected 3D node.



Confirm 3D profile construction by pressing the icon in the automenu.

The profile is now created. To continue with the three-dimensional model creation, quit the command.

We have created all required auxiliary 3D elements for creating the three-dimensional body by rotation operation.

Let's create the rotation operation. Call the command:

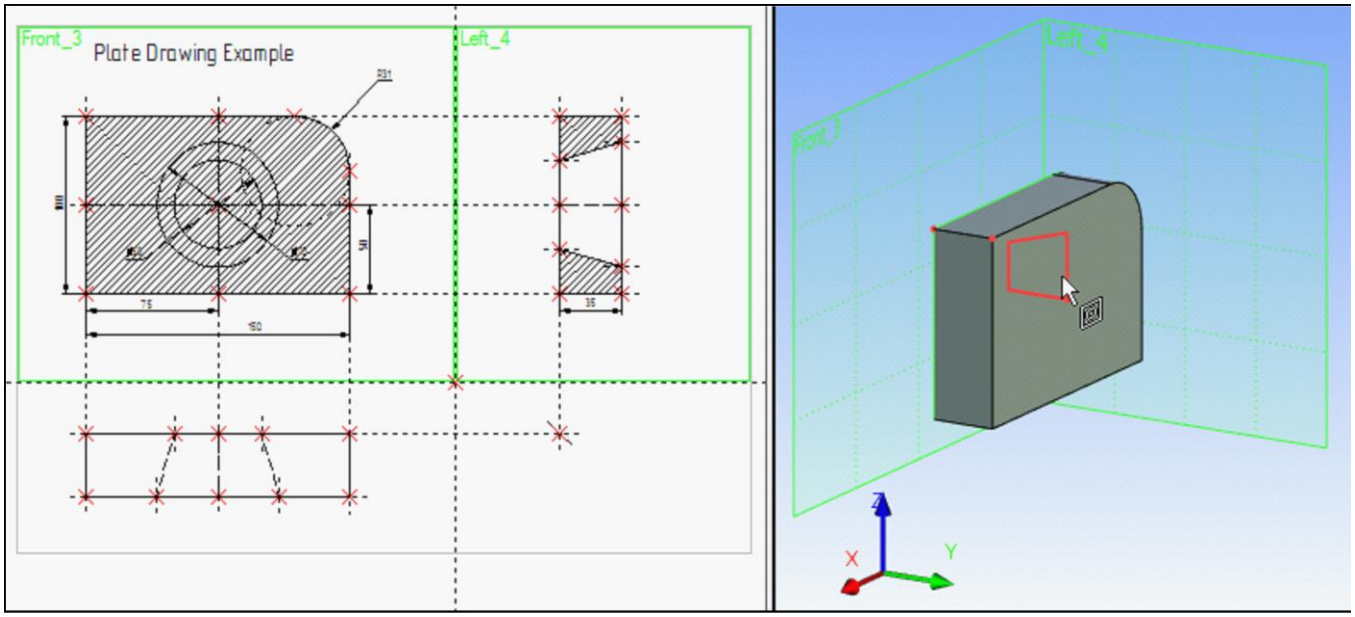
Icon	Ribbon
	3D Model → Create → Rotation
Keyboard	Textual Menu
<3RO>	Operation > Rotation

Once within the command, the following default option is set in the automenu:

	<R>	Select Contour

Various elements can be used as a contour for rotation operation. We will use a 3D profile created specifically for this purpose. Before selecting a profile, check on the system toolbar or in the pull-down list that the selector filters are set correctly.

As the cursor approaches the 3D profile, it gains the profile mark . To select the profile, click . The profile will be highlighted in the 3D window.

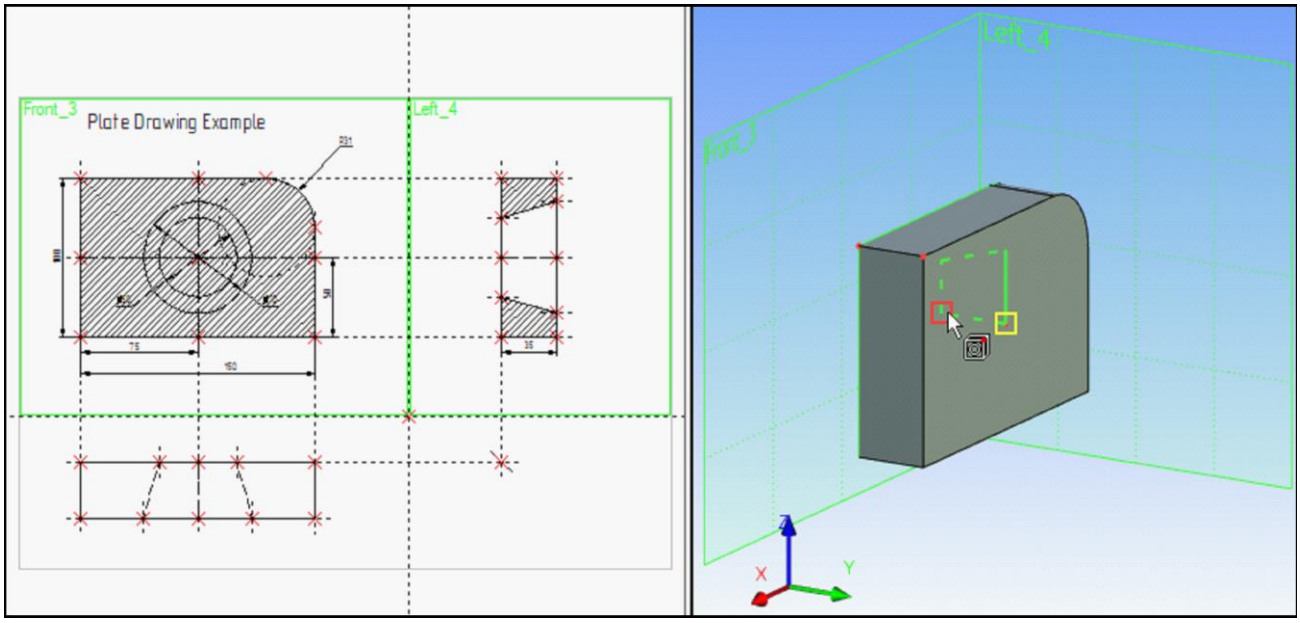


Next, define the axis of revolution by selecting two 3D nodes.

Set the following option in the automenu,

	<F>	Select 1st Point of axis

Move the cursor to a 3D node as shown below, and click .

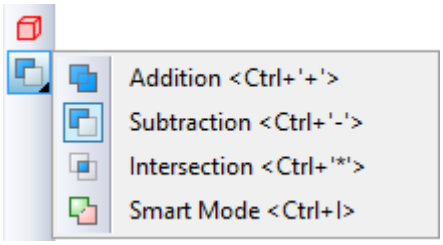


The option will activate in the automenu thereafter:

	<S>	Select 2nd Point of axis

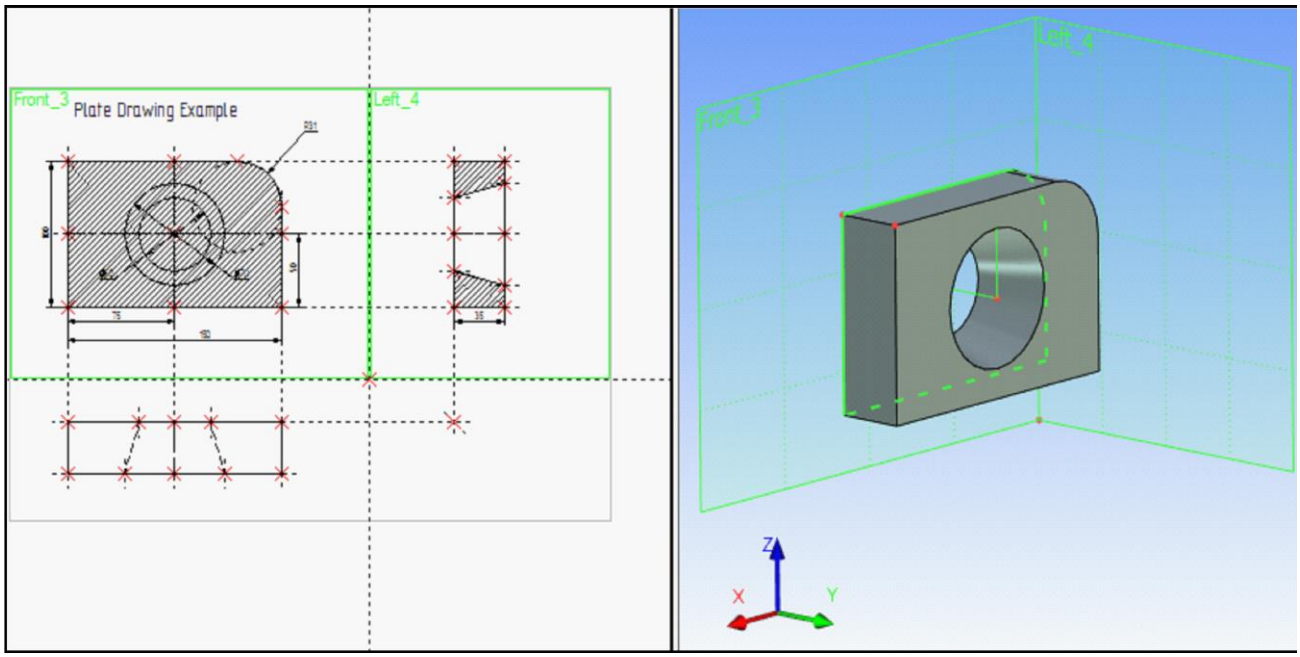
Move the cursor to the second 3D node defining the axis of revolution, and click . The body to be created will appear as a wireframe preview in the 3D window. The icon becomes accessible in the automenu, meaning that the rotation operation can now be completed. Before finishing operation, you need to choose «Subtraction» from Boolean operation pull-down list.

	<Ctrl><->	Substraction



Confirm rotation by pressing icon in the automenu.

The following three-dimensional image will appear in the 3D view window.

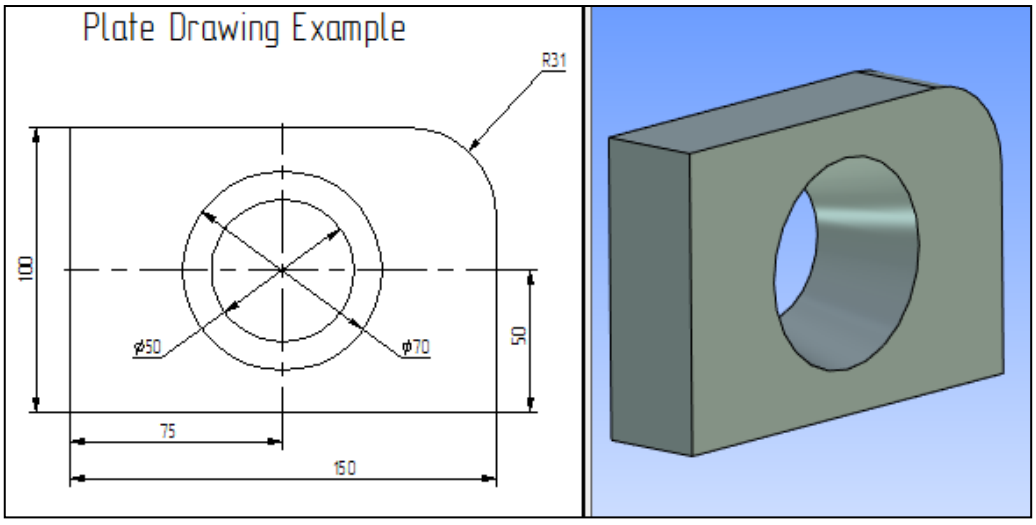


We have just created two simplest 3D bodies, an extrusion and a body of revolution. Tool body of revolution was subtracted from the original extruded plate.

You can find more information about Boolean operations in the chapter **Boolean operation**

In the three-dimensional model creation, we were using two-dimensional parametric elements of the T-FLEX CAD system as references. Therefore, parametric modifications of the two-dimensional drawing will drive parametric modifications of the three-dimensional model.

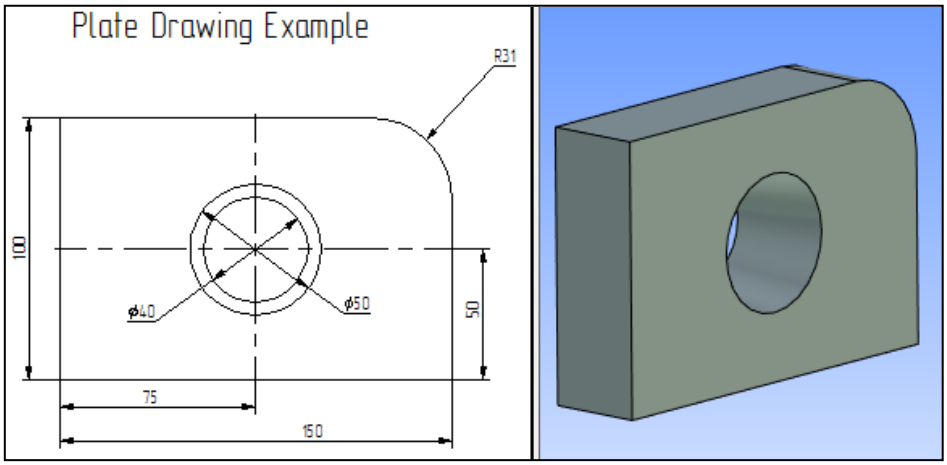
Modify one or more dimensions on the 2D drawing using the command **EC: Edit Construction**, or **V: Edit Variables**.



Then call the command:

Icon	Ribbon
	View toolbar →
Keyboard	Textual Menu
<3G>	Tools > Regenerate

This will cause recalculation of the 3D model geometry, and the model will adjust to the latest modifications made on the 2D drawing.



Working with the 3D View Window

The 3D view window (the 3D window) is one of the main system tools for working with the 3D model. It is used for the 3D scene rendering. The 3D window serves for viewing the model being created, as well as for selection of 3D objects for performing various manipulations with them (getting information about the selected object, editing, using as a base reference when creating operations and 3D constructions).

What is "3D Scene"?

3D scene is the entirety of all 3D objects (3D bodies and 3D constructions) that are present in the model. The size of the 3D scene is determined by the minimal cube fitting all 3D elements. When determining dimensions of the cube, the invisible 3D objects hidden from display by various methods are also included in the account.

Objects in the 3D scene are viewed using cameras. The camera that feeds the image in the 3D window, is called the active camera. By default, only the system camera is present in the model, which is located beyond the limits of the 3D scene. This is what is normally used to view the 3D scene. In most cases, just the system camera is enough for working with the 3D model. The position and orientation of the system camera can be controlled. The system camera cannot be deleted, neither can it be moved inside the 3D scene (the distance from the camera to the center of the 3D scene cannot be made less than half the 3D scene size).

In certain cases (for example, when surveilling inner parts of complex 3D models or when creating animation clips) the capabilities of just the system camera may not suffice. In such cases, the user can create and use one's own cameras for setting up a view in the 3D window. Custom cameras are attached to local coordinate systems (LCS). An LCS position determines a camera position. The respective section of this chapter is devoted to working with custom cameras.

The Methods of the 3D Scene Rendering

Visualization of 3D objects in the 3D view window can be done by various means (as a wireframe model, as solids, etc.). The 3D objects themselves do not change in this case, rather, their rendering method is changing.

There are the following visualization methods:

Wireframe - all three-dimensional bodies are displayed as a framework of their member edges. The method is convenient in that the background elements are not obstructed by the foreground ones. One can also see the objects located inside the body. This property is used when selecting construction elements for performing operations.

Shading - all three-dimensional bodies are rendered in a specific color defined for each body.

Rendering - all three-dimensional bodies are rendered with an account for the specifics of the selected material, whether for the entire body or assigned to a particular face.

Wireframe with hidden lines removal - the 3D window displays the 3D model wireframe without invisible lines. The fast algorithm of finding line visibility is used.

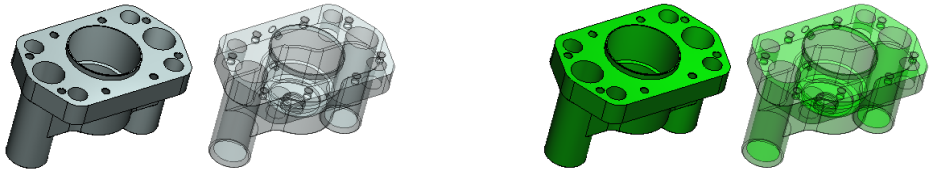
Wireframe with precise hidden lines removal - the wireframe model is displayed in the current position, without the invisible lines. This mode can be used only with the orthographic (parallel) projecting method.

The selected visualization method is applied by default to all elements of the 3D scene.

> Note that a visualization method is a characteristics of the 3D window, and does not depend on the camera used (whether the system or a custom one). That means, the visualization method assigned to a 3D window is maintained even when switching the active camera (that is, the one used at the moment for viewing the 3D scene).

The wireframe mode can be also assigned to individual 3D Bodies or operations. Such 3D object will be rendered in the wireframe mode always, regardless of the visualization method set for the 3D window. The sticky wireframe mode can be turned on by adjusting parameters of the respective 3D Body or 3D operation (see the chapter "General parameters of 3D elements").

For **shading** and **shading with materials** it is possible to additionally enable the *mode of semitransparent viewing*. When using the semi-transparency, the 3D model is displayed according to the rules of the main method of visualization but all faces of the bodies in the 3D scene become semitransparent.



Shading with materials | Shading with materials and semitransparency | Shading | Shading and semitransparency

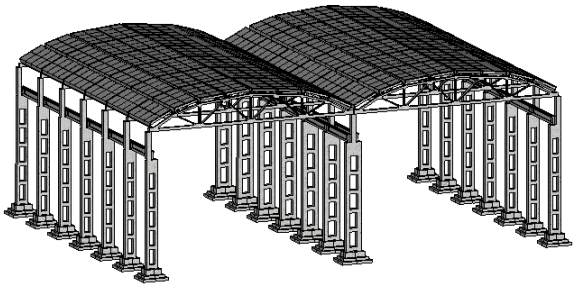
Transparent view can be enabled individually for any Body of the model. But such a Body will be displayed as transparent only when the mode of shading with materials is used. The transparent view can be enabled in the dialog of parameters of the Body or in the dialogs of parameters of operations included into the history of creation of the given Body (see the “General Parameters of 3D Elements” chapter).

Moreover, there is a possibility to make transparent the individual faces of Bodies. This can be done with the help of the **3AM: Apply material** command. Transparency of faces is also visible only in the mode of shading with materials.

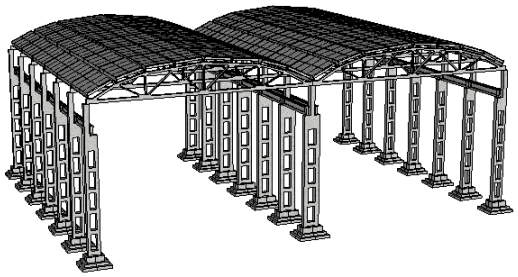
Working with the System Camera for Surveilling the 3D Scene

Projecting Method

The appearance of the 3D scene in the 3D view window depends, besides the specified visualization method, on the *projecting method* used by the system camera. The projecting method determines, how the camera will be showing objects in the 3D scene: without respect to the distance from the view point and the field of view angle - by the *orthographic (parallel) projection*, or by accounting for those parameters - the *perspective projection*.



Orthographic (parallel) projection



Perspective projection

A projecting method, unlike a visualization method, is a characteristic of the particular used camera, rather than the 3D window. When switching the active camera (for example, switching from the system camera to a custom one), the image generation in the 3D window will account for the new active camera projecting method.

Controlling the System Camera. Rotating (Spinning) and Moving a Camera

Initially after opening the 3D window, the system camera is located at the distance providing visibility of all elements of the 3D scene (that is, all 3D constructions and 3D Bodies present in the model). To view the 3D scene, you can rotate (spin) and/or move it arbitrarily, zoom in and out, change the orientation and view direction, as well as scale the image in the 3D window.

As the user watches the 3D window in the process of spinning (rotating) and moving the system camera, one feels as if the 3D scene itself is moving, rather than the camera. Therefore, when describing 3D model handling in the future, we will often refer to the process of rotating (spinning) and moving the system camera as the 3D scene rotation.

Camera's center of rotation

Rotating the image in the 3D window with the default system camera active is achieved by moving the camera about the scene center. If necessary, the center of rotation can be changed. The following can be selected as the new rotation point:

An operation's center of mass;

An arbitrary point in the 3D scene. The center of rotation will be the intersection of the ray passing through the selected point perpendicular to the screen plane with the face or edge (in the case of the wireframe display) of some 3D Body or any 3D construction element nearest to the camera. If the ray does not cross any of the 3D objects, then the center of rotation stays unchanged;

The 3D scene center (whenever necessary to return to the original center of rotation).

System camera rotation modes

There are the following system camera rotation modes:

Free rotation. This mode is preset by default. In this mode, a 3D camera can be rotated in an arbitrary direction about the rotation center;

Rotation about one of the axes of the coordinate system located at the center of the camera's rotation. the direction of the rotation coordinate axes can be:

- Coincident with the direction of the global coordinate system axes (*rotation with respect to the global coordinate system*),
- Defined by the coordinate system of the 3D window itself (*rotation with respect to the window coordinate system*) - one axis being directed perpendicular to the screen plane, the two others lying in that plane (one - vertically, the other - horizontally).

Panoramic rotation. In this mode the plane of the Z-axis is fixed. This mode creates the effect of viewing the room-scene with a video-camera which cannot be inclined (the Z-axis in this case plays a role of the vertical axis of the room). The mode of the panoramic rotation can be useful when working with the models of buildings, furniture, interiors etc.

Automatic Rotation

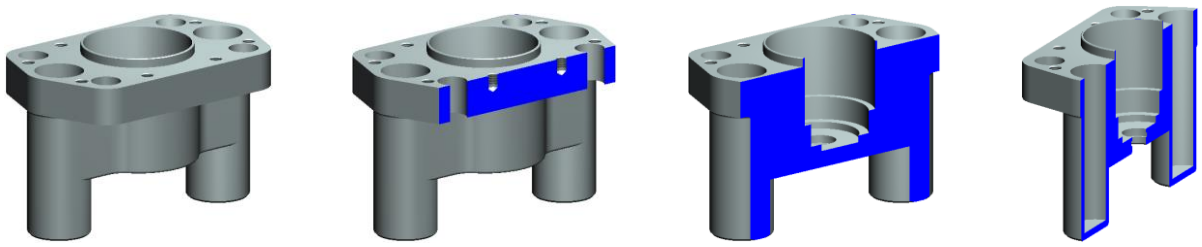
Automatic Rotation can be performed in any of the above-listed modes of the system camera rotation. To activate the automatic rotation mode, specify the direction and speed of rotation. Before terminating the automatic rotation mode, the system camera rotation command with such parameters will be repeatedly sent to the 3D window, which add the inertia effect to the 3D scene rotation.

Auto-Resize

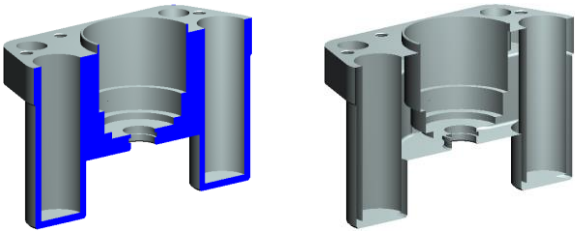
Auto-Resize is a 3D window mode, when the system camera's parameters are automatically updated upon each 3D model regeneration, so that the entire 3D scene is visible.

CLIP PLANE

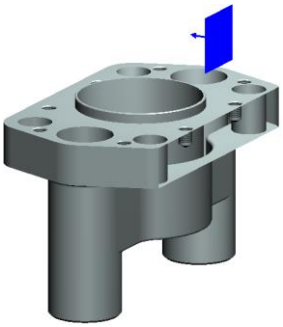
Use a *clip plane* as a handy visual examination tool for inner body parts. It is a plane, which can move along the specified direction and “slash” objects in the scene. The cut-off part of bodies is not displayed. No new bodies are created in this case. As the camera rotates, the orientation of the clip plane changes together with the changes in the scene position.



The resulting body section can be transparent (this would allow looking inside the body) or can be filled with blue color (if you need to examine the shape of the section).



The clip plane is not displayed by default. If necessary, the clip plane can be made invisible. It will appear as a small rectangle indicating the position and orientation of the clip plane.



Light Sources

Illumination of bodies in the 3D scene is provided by light sources. Light sources can be divided into two main types:

Stationary sources created by the user in the command **3L: Create Light Source**. Such light sources are independent of the used camera and its position. How to create stationary light sources will be described in details in the "Light sources" chapter of this manual;

Dynamic light sources, "mounted" on the active camera. The position of such light sources is defined relative to the active camera. When moving/rotating the active camera, the light sources will be also changing their position.

> Dynamic light sources, as well as the visualization method, are characteristics of the 3D window, that does not depend on the used camera (whether the system or a custom one).

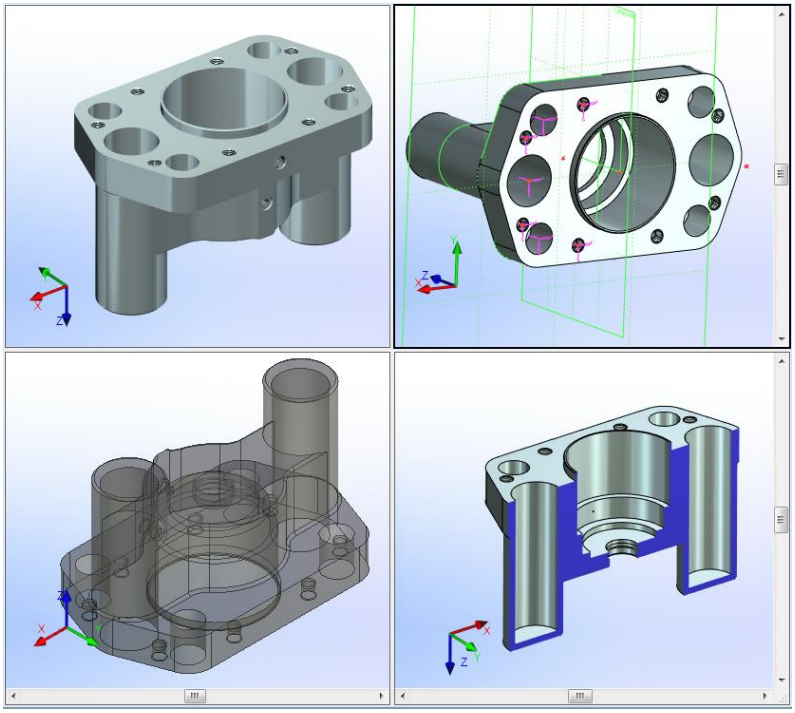
By default, there are two light sources in the 3D window attached to the active camera, an ambient light source (which uniformly illuminates the entire space of the 3D scene) and a directional source of white

light, located on the camera. If necessary, the user can change their parameters (color, intensity, and, for a directional light - the position with respect to the camera), as well as manually create additional dynamic light sources.

Creation and editing of light sources attached to the active camera is done in the special 3D view parameters dialog. You can also specify there the background color of the 3D window (the background color does not depend on the light sources present in the 3D window). The user can select any color as the background according to one's preferences. You can use a transitional background color (with a smooth change from one color to another).

WORKING WITH MULTIPLE 3D WINDOWS

When working with a 3D model, the user can open several 3D windows simultaneously. Individual parameters can be assigned to each of the open 3D windows: the visualization method, the orientation and view direction of the active camera, the background color and number of light sources on the active camera, the image scale, etc.



Division of the window of the document into parts can be fixed for subsequent sessions of work with the document. To do so, it is sufficient to enable the "Fixed set of windows" parameter in the dialog of the **ST: Specify parameters of document** command (the "Miscellaneous" tab) and save the document. When the document is opened next time, the window will be partitioned in the same way as it was at the moment of saving the document after enabling the aforementioned parameter. It will be impossible to modify the view of the window of this document.

USING 3D VIEWS

You can use the system preset *3D views* for quick setup of a 3D window. 3D View is a set of information about a 3D window, including information about the parameters of the used camera, about the light sources attached to it, the current rendering method, etc.

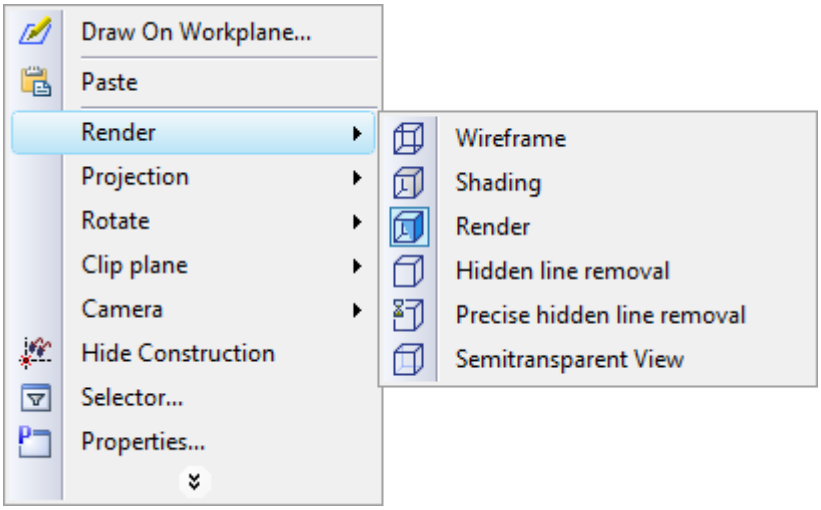
When loading one of the 3D views stored in the system, into the current 3D window, the 3D window is automatically set up according to the data of this 3D view.

The user can edit 3D views present in the system, adding to those unique sets of data about the state of the 3D window. 3D views are handled in the special 3D view parameters dialog.

MANAGING 3D WINDOW

Use the commands located in the "View" textual menu to manage the 3D window and the 3D scene display in it. Commands are grouped according to their purpose. Some of the commands are also available on the same-name "View" toolbar.

There is yet another way to call these commands. Point the mouse to a free space of the 3D window and right-click. The coming up context menu will contain all commands for managing the 3D window. The state of the context menu icons informs you about which mode of a particular command is currently active.



Defining a Visualization Method

To define a visualization method, use the commands:

Keyboard	Textual Menu	Icon
<3VW>	View > Render > Wireframe	
<3VS>	View > Render > Shading	

<3VD>	View > Render > Render	
<3VH>	View > Render > Hidden Line Removal	
<3VZ>	View > Render > Precise Hidden Line Removal	
<3VM>	View > Render > Semitransparent View	

Every method uses additional parameters. You can define those parameters in the 3D view parameters dialog described in the respective section of this chapter.

Rotating and Panning the 3D Scene

When working with a 3D model, the most commonly used is the *free rotation mode*; therefore it is preset in the 3D window by default. The camera can be moved around the rotation center by simply dragging the mouse with the depressed within the 3D window.

You can also use the keyboard for rotating the 3D scene. The <Left>, <Right>, <Up>, <Down> keys make the 3D scene rotate in the respective directions. You can rotate the camera about its own axis using the <PageUp>, <PageDown> keys. When the <Ctrl> key is pressed, all these keys work in the fast rotation mode.

> The speed of rotation for the normal and fast modes is set up in the **Customize > Settings** command on the "3D" tab.

By using while holding down the <Ctrl> key, you can move the camera in the plane parallel to the screen. In this way, the center of rotation moves together with the camera.

To change the image scale ratio, use with <Shift>. By moving the pointer up the screen, you zoom out of the image, by moving down - zoom in.

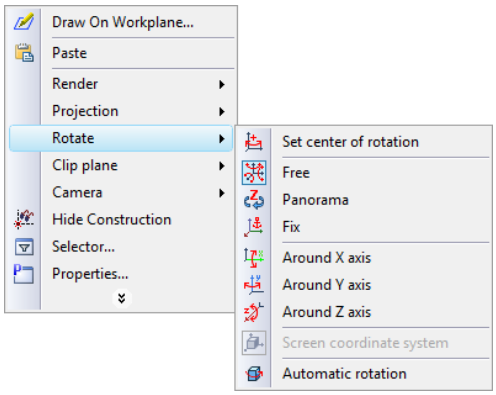
> This process is similar to the function of a real camera. The camera, however, stays still. Only the focal length of its objective lens changes.

If your mouse has a wheel, then you can control the degree of zooming on the image by spinning the wheel. Dragging the mouse with the wheel depressed moves the camera in the plane parallel to the screen plane.

Rotation about one axis can be activated by using one of the following commands in the **View > Rotate** textual menu or the same-name item of the context menu in the 3D window:

Keyboard	Textual Menu	Icon
<3RX>	View > Rotate > Around X axis	
<3RY>	View > Rotate > Around Y axis	
<3RZ>	View > Rotate > Around Z axis	

Calling any of those commands results in assuming the mode of rotating about the selected axis. Switching the axis, about which rotation is performed, is done by the same commands. You can judge which axis is currently used by the state of the respective icons in the textual or context menu.



Orientation of the rotation coordinate axes is determined by the state of the command **3RS: Toggle between 3D Scene Rotation Modes**:

Keyboard	Textual Menu	Icon
<3RS>	**View > Rotate > Screen coordinate system**	

When this command is active, rotation occurs in the window coordinates, when off - the orientation of the rotation coordinates coincides with the global coordinate system. This command activates automatically when switching to rotation about one axis. Therefore, by default rotation occurs in the window coordinates. You can judge the state of the command by its icon in the textual or context menu.

Rotating the 3D scene about one axis is done in the same way as in the free rotation -- using or the keyboard. The difference will be only in that rotation will occur only about one selected axis of the camera's rotation coordinate system upon any mouse movement or control key strikes.

To return to the mode of the free camera rotation, use the command:

Keyboard	Textual Menu	Icon
<3RF>	View > Rotate > Free	

The mode of fixed direction of the sight is enabled with the help of the **3RL: Fix direction** command. It can be invoked from the text menu “View > Rotate” or with the help of the context menu in the 3D window:

Keyboard	Textual Menu	Icon
<3RL>	View > Rotate > Fix	

When this mode is enabled, the direction of the sight gets fixed, i.e., rotation of the 3D scene is impossible. But it is possible to scale and translate the contents of the 3D scene. This setting is saved in the current document and active during subsequent sessions of work with it.

To turn off the mode of the fixed direction, it is required to select any other mode of rotation of the 3D scene.

The command **3RP: Rotation panoramic**:

Keyboard	Textual Menu	Icon
<3RP>	View > Rotate > Panorama	

In this mode the plane of the Z-axis is fixed. This mode creates the effect of viewing the room-scene with a video-camera which cannot be inclined (the Z-axis in this case plays a role of the vertical axis of the room). The mode of the panoramic rotation can be useful when working with the models of buildings, furniture, interiors etc.

Use the following additional commands for quick manipulations with the orientation and scale of the 3D scene display:

Keyboard	Textual Menu	Icon
<F3>	View > Scale > Zoom Area	
<Ctrl><Shift><PgUp>	View > Scale > Zoom In	
<Ctrl><Shift><PgDn>	View > Scale > Zoom Out	
<Ctrl><Shift><Left>	View > Scale > Pan Left	
<Ctrl><Shift><Right>	View > Scale > Pan Right	
<Ctrl><Shift><Up>	View > Scale > Pan Up	
<Ctrl><Shift><Down>	View > Scale > Pan Down	
<Ctrl><Shift><Home>	View > Scale > All Objects	
<Ctrl><Shift><End>	View > Scale > Maximize	
	View > Scale > Actual Size	
	View > Scale > Previous	

A detailed description of working with these commands is available in the “Main Concepts of System Operation” chapter of the “Fundamentals. Two-Dimensional Design” guide.

Defining the Center of the 3D Scene Rotation

To define the center of camera rotation, use the command **3RC: Select Center Of Rotation**:

Keyboard	Textual Menu	Icon
<3RC>	View > Rotate > Set center of rotation	

After the call, the current camera rotation center appears on the screen as a crosshairs. The following options become available in the automenu:

	<P>	Select any point as rotation center
	<O>	Select center of solid as rotation center
	<D>	Select center of scene as rotation center

To select the center of mass of a 3D operation as the center of rotation, simply activate the option , and then select the desired operation. The crosshairs marking the rotation center will move to the center of mass of the selected body.

The option allows specifying an arbitrary point as the center of rotation. For this, after selecting the option, simply pick the desired point (using) in the 3D window. If the ray passing through the given point perpendicular to the screen plane crosses some 3D object, then the center of rotation will be carried over to that point.

To return the camera rotation center to the 3D scene center, simply click .

It is possible to define arbitrary center of rotation without calling **3RC: Select Center of Rotation** command. This can be done by pressing without releasing for some time at any object within 3D scene. The special marker-cross will appear at the specified point. This marker signals about setting the position of the new temporary center of rotation. After finishing the rotation (when mouse button is released) center of rotation returns to the original position.

Activating Automatic Rotation (Spinning) of the 3D Scene

To activate automatic rotation, call the command **3RA: Automatic Rotation**:

Keyboard	Textual Menu	Icon
<3RA>	View > Rotate > Automatic Rotation	

To define rotation parameters, depress and, while holding it, drag the mouse in the rotation direction. After that, release . In that way, you define the direction and angular rotation velocity.

The automatic rotation mode is switched off by a repeated call of the same command.

Defining the Method of Projecting the System Camera

Use the following commands to select the method of projecting the system camera:

Keyboard	Textual Menu	Icon
<3VO>	View > Projection > Parallel	
<3VE>	View > Projection > Perspective	

Parameters of each projecting method are defined in the dialog of the current 3D view parameters. Working with this dialog is described in the "3D view parameters" section of this chapter.

Auto-Resize

To enable or disable the auto-fitting mode of the 3D scene, use the command **3VA: Auto-Resize 3D Scene**:

Keyboard	Textual Menu	Icon
<3VA>	View > Auto-Resize	

The auto resize mode will be active until the second call of this command.

Working with a Clip Plane

To activate a clip plane, use the command **3CL: Enable Clip Plane**:

Icon	Ribbon
	View → Clip plane → Active
Keyboard	**Textual Menu**
<3CL>	View > Clip plane > Active

A repeated command call turns off the clip plane.

To define the clip plane position, use the command **3CS: Set Clip Plane Position:**

Icon	Ribbon
	View → Clip plane → Set
Keyboard	**Textual Menu**
<3CS>	View > Clip plane > Set

Calling this command sets the plane parallel to the screen and activates it.

The position of the active clip plane (the distance from it to the view point) can be modified without calling additional commands, using the mouse together with the simultaneously pressed <Ctrl> and <Shift> keys. By moving the pointer up the screen, you move the plane "deeper" into the scene, and vise versa.

To reveal the clip plane in the 3D scene, call the command **3CD: Show Clip Plane Position**:

Icon	Ribbon
	View → Clip plane → Draw
Keyboard	Textual Menu
<3CD>	View > Clip plane > Draw

Repeatedly calling the command will remove the clip plane image from display.

Defining All 3D View Parameters in One Dialog

Use the convenient dialog of the 3D view parameters for quick setup of the 3D window, when you need to change its several characteristics simultaneously (for example, the visualization method, the orientation, the view direction and the method of projecting from the system camera, etc.). Some of the 3D view parameters (for example, the background color, the parameters of light sources on the camera) can be defined only in this style dialog.

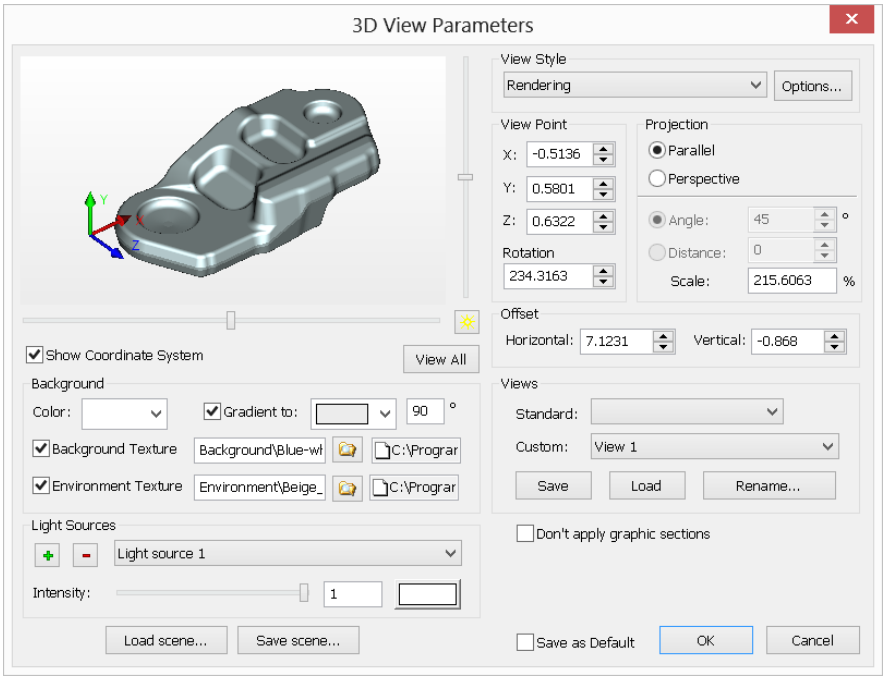
To call the dialog of the 3D view parameters, use the command **3VP: Set View Parameters**:

Icon	Ribbon
	View → Properties
Keyboard	Textual Menu
<PW>	View > Clip plane > Properties

After calling the command, a dialog window appears, which contains all current 3D view parameters.

The 3D window contents preview pane is on the left-hand side of the dialog window. It displays all the same as in the master 3D window, but smaller. You can work with the image in the preview pane in the same way as with the image in the 3D view window: spin, pan, zoom. To quickly view the contents of the 3D scene, you can use the **[View All]** button - clicking it scales the image in such a way that all contents of the 3D scene are visible. All manipulations made in the preview pane will be also applied to the image in the 3D view window (after closing its parameters dialog). The size of the 3D view properties dialog can be increased by moving the window boundaries with the cursor of the mouse when is being pressed.

The "**Show Coordinate System**" flag can be used for disabling the coordinate system display in the 3D view window. The displayed size of the global coordinate system in the 3D window is defined in the command **SO: Set System Options** (the "3D" tab).



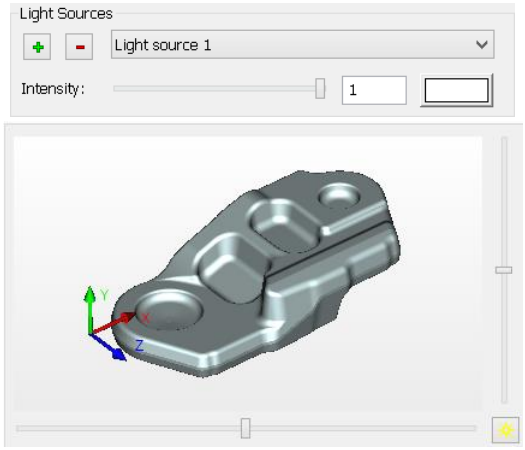
The **Background** group of parameters defines the background color of the 3D view window. To achieve a smooth transition of colors, set the **Gradient to** flag and specify two colors by selecting from drop-down lists. Clear the flag in order to make the 3D view window background monochrome. In such a case, specify just one color (the **Background color**).

The **Use background texture** flag is used to enable/disable the mode of using the texture as a background of the 3D view window. When the flag is disabled the 3D view window has the color that is specified in the background parameters.

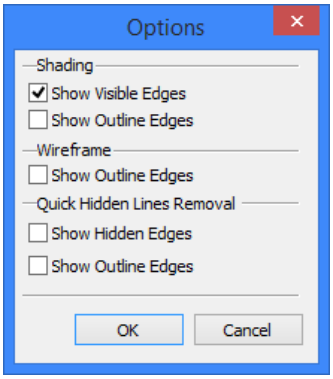
The **Use environment texture** flag is used to enable/disable the mode of using the texture as environment for 3D model. When the flag is disabled the specified texture will be suppressed.

The group of parameters in the lower-left corner of the dialog window is provided to manage light sources attached to the active camera in the 3D scene. This group includes the list of dynamic light sources existing in the current 3D window, buttons for adding/deleting light sources attached to the camera, as well as control elements for defining parameters of each of the light sources.

To edit parameters of an existing light source, select this source in the drop-down list. Control elements located below the list allow changing the brightness of the selected light source (using the slider or numerically), as well as its color. The selected light source can be deleted by clicking the button . Clicking the button creates another directional light source attached to the active camera.



The position of the selected light source with respect to the camera is defined using sliders located at the right and below the viewing pane of the 3D window contents. By default, dynamic light sources are placed directly on the camera (the sliders being positioned in the middle of their respective scales). By moving the sliders away from the center of the scale, one can move light sources right/left and up/down with respect to the camera. The button brings the light source back to the camera.



The method of the 3D scene rendering in this 3D window is defined using the **View Style** group of parameters. The drop-down list lets you select the desired method, while the Options... button serves for defining additional parameters of each mode. Clicking this button brings up the "Options" dialog, which specifies the requirements on drawing various lines of the 3D model - edges, outline (silhouette), hidden lines - for each visualization method.

The **View point**, **Projection** and **Offset** groups of parameters define the view direction and the method of projecting from the system camera, as well as its distance from the center of the 3D scene. In the parallel projecting method, the camera distance is defined by the **Scale** parameter. This parameter defines the degree of zooming on the image in the 3D scene.

In the perspective projection, the camera distance can be defined either by the field of view **angle** of the 3D scene, or directly by the **distance** between the camera and the center of the 3D scene. All three parameters are defined numerically. Additionally, the scale and the field of view angle can be controlled using the mouse, directly in the preview pane of this dialog, similar to the way it is done in the 3D window itself when moving/rotating the camera. The **Offset** parameter defines the horizontal and vertical shift of the 3D scene center with respect to the camera.

The **Views** group of parameters, located in the lower-right corner of the dialog, serves for quick setup of the 3D window state. The **Standard** drop-down list lets you changing the view direction of the system camera according to one of the engineering drafting standards (for example, Front (elevation) View, Back

View, Left (side) View, etc.). For this, simply select the desired scene orientation in the list. The **Custom** drop-down list serves for handling 3D views preset in the system.

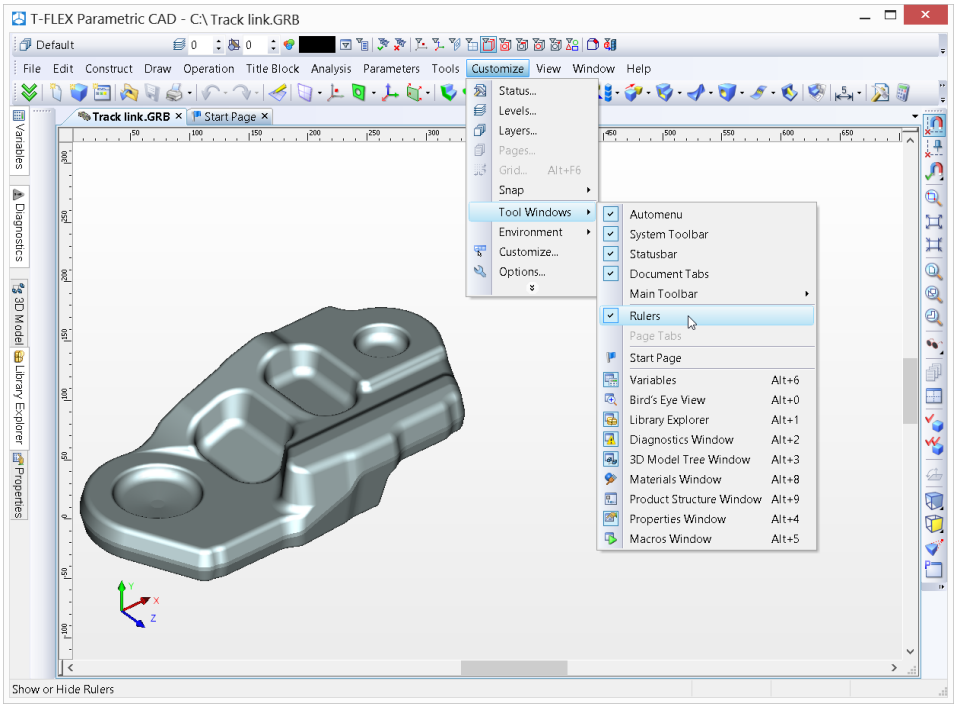
A 3D view can be loaded (that is, the 3D window set up according to the information stored in the 3D view), saved (by writing 3D View information about the current state of the 3D window) and renamed (by simply changing the name, under which the given set of information is stored in the system). To load a 3D view, select this 3D view in the list and click the **[Load]** button. Clicking the **[Save]** button writes information about the current state of the 3D window in the 3D View selected in the "Custom" list. Clicking the **[Rename...]** button calls an additional dialog for defining a new name for the selected 3D view.

RULERS IN 3D WINDOW

When working in 3D window the **Show** > **Hide rulers** command becomes available. This command can be invoked from the text menu «Customize > Windows». It is also available in the context menu that can be invoked upon pressing and indicating one of the toolbars.

Rulers allow us to orient ourselves in 3D space, estimate the size of a model and distances between the elements of a model. Coordinates displayed on the rulers constitute the coordinates on the imagined plane parallel to the plane of the screen.

Also with the help of rulers of 3D window it is possible to create, in transparent mode, the new workplanes and edit location of already existing workplanes. This will be described in more detail in the "Workplanes" chapter of this user's manual.



WORKING WITH CUSTOM CAMERAS

Creating and using custom cameras will be described in details in the "Cameras" chapter. Here we will just describe general techniques of working with them.

Custom cameras are used in the cases, when the system camera is not sufficient. In such a case, the user can create one's own cameras in order to get the desired effect - for example, for surveilling inner parts of complex 3D models or creating animation clips.

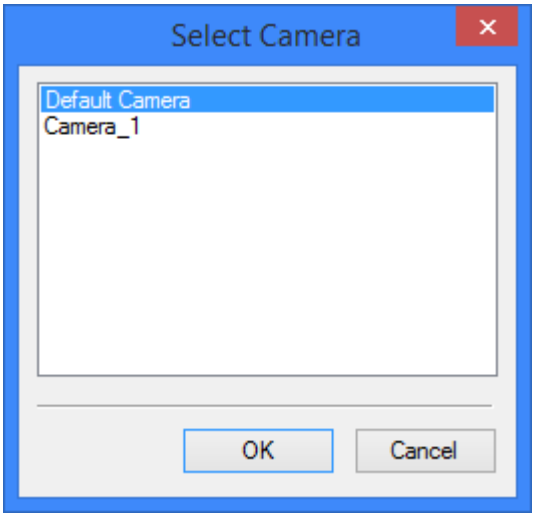
The position of a custom camera is defined using an LCS. This same LCS defines the viewing direction for a custom camera. A number of parameters are defined for a camera, which define the image it yields (for example, the projecting method).

To view the 3D window from the custom, rather than the system camera, select the custom camera as the active camera of the given 3D window. This can be done in the command **3VC: Select Camera**.

Icon	Ribbon
	View → 3D Model → Select Camera
Keyboard	Textual Menu
<3VC>	View > Camera

After calling the commands, the "Select Camera" dialog window appears, which contains the list of all cameras present in the current 3D model. In it, select the camera that will be used for viewing (surveilling) the 3D scene.

When working with multiple 3D windows, an individual active camera can be specified for each window.



While working with a custom camera, you can use auto resizing of the 3D scene image, using clip plane, modify existing and create new light sources on the camera. How to move and spin the active custom camera is described in details in the "Cameras" chapter.

Working with the 3D view parameters dialog is similar to the described case of using the system camera, with one exception: the parameters pertaining to the active camera will not be available for editing in the dialog. In the case of custom cameras, these parameters are defined in the parameters dialog of the camera itself.

> You cannot modify parameters of the active custom camera. If necessary, first select another camera as the active one (system or custom).

General Parameters of 3D Elements

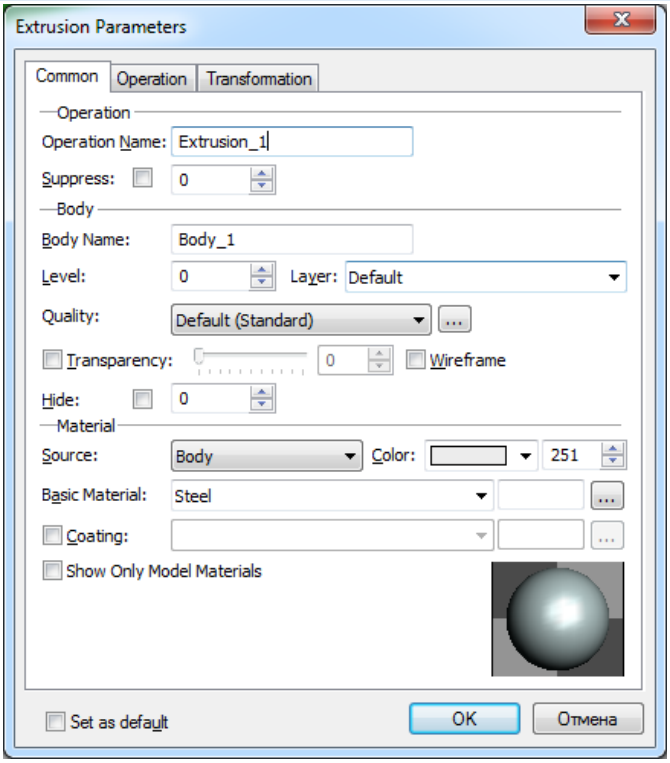
Similar to the 2D elements, each of the 3D elements has a certain set of parameters. Some of these parameters are common to all objects in the system, regardless of their dimension; others are specific to the 3D elements, while yet other parameters are only inherent to specific types of objects. To skip the description of the general parameters in each chapter, we will do it once, while reviewing only the specific parameters in the respective chapters.

«General» Tab in Dialogs of Parameters of 3D operations

The dialogs of parameters of all 3D operations contain the identical tab called "General". The set of parameters of this tab includes nonspecific parameters of the given operation and parameters of visualization of the Body in the generation of which this given operation is involved.

Parameters of visualization of a 3D image (level, layer, material, image quality) are specified for the entire Body. Therefore, the values of the parameters of visualization on the "General" tab will be identical for all operations that are included into the history of creation of the given Body.

When viewing the properties of the Body itself (the "Properties" command in the context menu of the tree of the 3D model), the dialog of parameters of the last, in the history of the given Body, operation is.



The "**Operation**" group:

- **Operation name** – the name of the 3D operation for which the dialog of parameters is invoked;

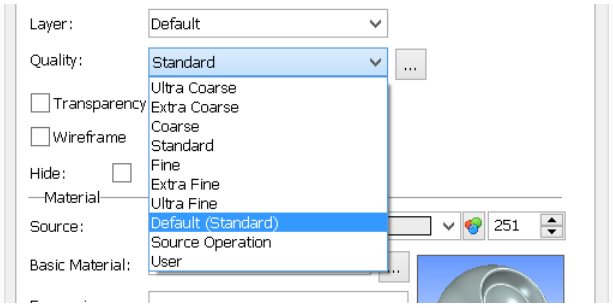
  This name is used for selection of the desired element from the list. The user can specify the name independently. In this case the system only keeps track of the uniqueness of the entered name. If the user does not specify the name, the system generates its own name consisting of the name of the element and the element's identification number. The default names for each type of elements are described in the file `Opernames.ini`.

- **Suppress** – this option is responsible for exclusion of the given operation from 3D model recalculation. The suppressed operation is not shown in the 3D scene. It is advantageous to use the given parameter during the stage of forming the individual parts of complex assemblies. The user can temporarily suppress the elements that have already been worked on and work with those elements which are required for further constructions. A real variable can be used as the value of the given parameter. The operation will be suppressed if the value of this variable is not equal to zero;

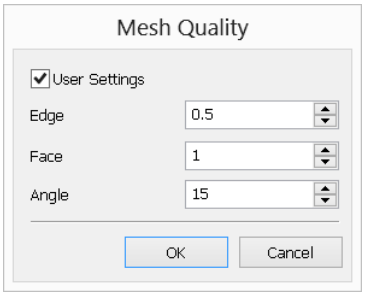
The "**Body**" group:

- **Body name**. The name of the Body in generation of which the given operation takes part.
- **Level, Layer** – common to the system parameters of the Body.

Quality. Parameter that defines the quality of 3D images of the Body. It defines the degree of discretization of the model into the grid of planar triangular faces when showing the image in 3D window. From the drop down list it is possible to choose the predefined sets of parameters that define the quality of the mesh:



- Ultra coarse;
- Extra coarse;
- Coarse;
- Standard
- Fine;
- Extra Fine;
- Ultra Fine;
- Dedault (Standard) – the values for model discretization parameters are taken from the **Customize > Document Parameters** command (the "Image quality" parameter on the "3D" tab);
- Source operation – the values for model discretization parameters are taken from the parameters of the operation that is a parent with respect to the operation indicated in the given dialog of parameters



– User-defined – the values for model discretization parameters are specified by the user. Additional button **[...]** invokes the dialog window for specifying the user-defined values of model discretization parameters (in units of a model). In the mesh parameters dialog it is possible to specify your own values of: allowance by edge, face precision and angular allowance.

The higher quality of model image increases the number of planar faces and slows down the work for large models or insufficiently productive video-cards. It is recommended to make the model image quality as low as possible.

When reading the models of the previous versions, the system automatically selects new parameters of the quality. In certain situations the automatically assigned quality can turn out to be lower or higher than the expected one. In these cases it is recommended to change it manually by selecting the corresponding value of the quality in the parameters of the document (the "3D" tab in the **Customize** > **Document Parameters** command) or in the parameters of the Body. The number of triangles that a specific model is discretized into, when a particular value of the mesh quality parameter is specified, can be determined with the help of the **Display graphics performance data** option in the the **3D** > **Graphics Settings** tab of the **Customize** > **Options** command.

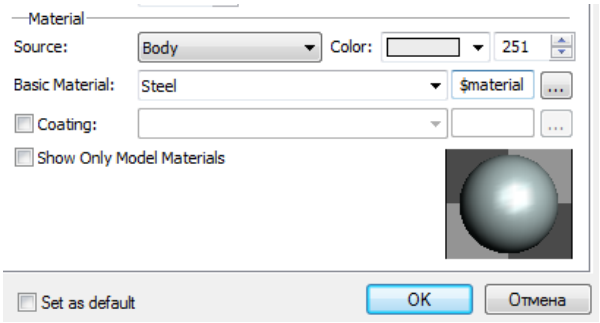
- **Transparency**. The flag, scroll bar and the field for editing the transparency value.
- **View as wireframe**. When using this option, the body will be displayed in the wire-frame view mode.
- **Hide** – this option allows us to hide the Body in the 3D scene without excluding it from the recalculation of the model.

All parameters in the "Body" group are accessible for editing only in the case when the current operation is uniquely linked with the Body. If the given operation generates several different Bodies, parameters of the given group are not accessible.

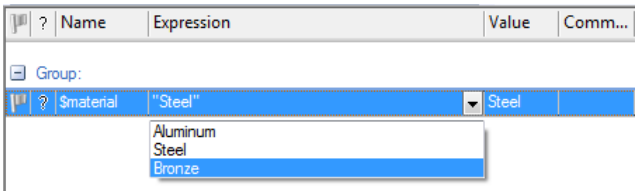
Parameters of the **Material** group allow us to make virtual models appear like a real product when the "Shading with Materials" visualization mode is used:

- **Source** — parameter which determines from where the material is taken. The possible variants are:
  - **Body**. Parameters are specified in the current Body;
  - **Source operation**. Parameters are inherited from the source operation;
  - **Current operation**. Parameters are specified in the current operation. This option is introduced to provide compatibility with the previous versions;
- **Color** — color of the Body in the modes "Wireframe view", "Shading", "Viewing with removal of invisible lines", "Viewing with precise removal of invisible lines".

- **Main material** - material of the Body used for evaluation of the mass inertial characteristics and engineering analysis. It is also used for determining parameters of the model view in the 3D window if an additional material of the "outer cover" is not specified. It can also be specified as a text variable.



The value of such a variable constitutes the name of an existing material. When the variable is created directly in the window of the operation's parameters, the list of possible values of the given variable is created simultaneously. This list contains all materials used in the given model.

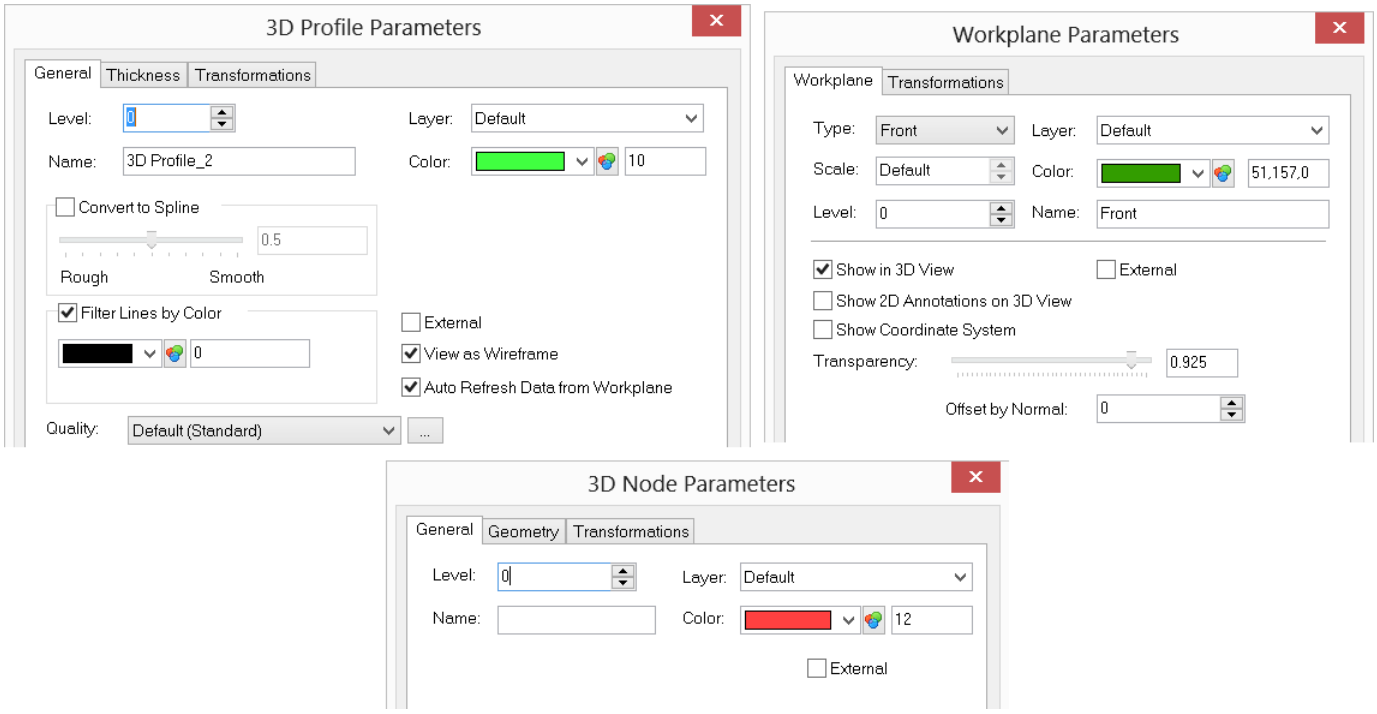


If the model is used as a 3D fragment, the material of the fragment can also be controlled with the help of such a variable.

- **Coating**. Additional parameter for specifying parameters of the model view in the 3D window. It allows us to specify the material that is different from the main material to use it for the purpose of visualization. In this case, the main material will still be used for engineering calculations. To specify the material of the outer cover, the corresponding flag must be enabled. The material can also be specified as a text variable.
- **Show only materials of a model** – auxiliary parameter that controls the structure of the materials selection list when working with the dialog of operation's parameters. If this flag is enabled, the "Name" list will show only those materials which were used in the given model. Material – is also an element of the 3D system. It contains the list of characteristics of the real material which we encounter in real life. The "material" element has parameters such as density, reflexivity, transmittivity and so on. All this is described in more detail in the special chapter devoted to this issue.

"General" Tab in Dialogs of Parameters of 3D Construction Elements

In the dialogs of parameters of 3D construction elements on the "General" tab there are the following common parameters:



Level. The value of the visibility level. It allows controlling the visibility of various elements.

Color. Element color.

Layer. The name of the layer to which the element belongs.

Name. If the user skips assigning a name, then the system generates its own name consisting of the element type and a through ID number. The default names for each element type are described in the file `Opernames.ini`. For example, the generated workplanes names are: Workplane_0, Workplane_27, etc.

"Transformation" Tab

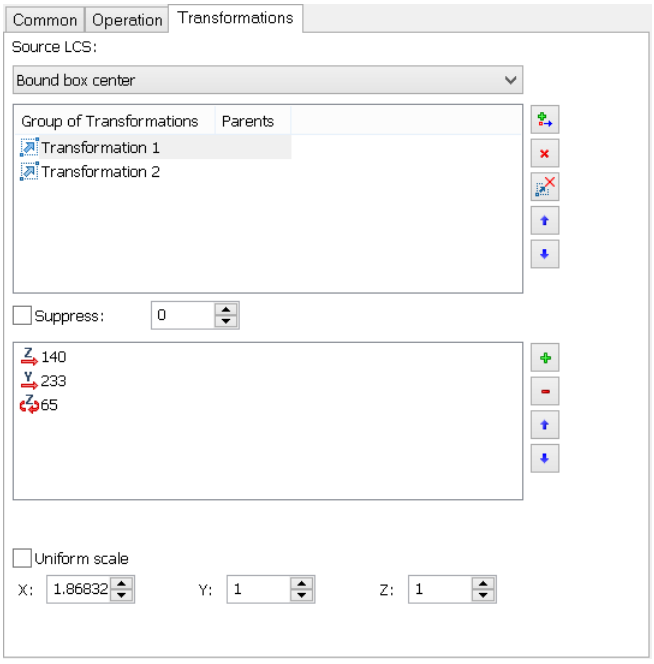
This tab contains information about the current object transformations.

A set of various transformations can be defined for any 3D object that will modify its original position in the space (the detailed description of handling transformations is available in the chapter "Assigning "Transformations" to 3D Entities").

"Offset/Rotation"-type transformation is defined by default for any 3D element with zero parameter values. It is initiated automatically at the element creation. A number of additional transformations are automatically defined for 3D fragments and Parts.

Additional transformations can be defined by two ways: by using the command **"3EG: Transform Element"** or directly on the discussed tab of the object parameters dialog box. The two ways are equivalent. The transformations defined by the command **"3EG: Transform Element"** will be displayed and accessible for editing in the parameters dialog box of the current object. Vice versa, any changes done to the set of a 3D object transformations in the tab "Transformation" of the parameters dialog box will be accessible for future editing in the command **"3EG: Transform Element"**.

The main difference between using the specialized command and the parameters dialog box is as follows. When creating a new transformation via the parameters dialog box, the reference elements of the transformation (axis, vector, point, coordinate system referenced by the transformation) are defined by the system automatically. When working in the command **"3EG: Transform Element"**, you can define arbitrary reference elements manually.



The list of the existing transformations is displayed in the upper-left corner of the tab. The rest of the fields display the parameter values of the current (selected in the list) transformation. The buttons next to the transformations list allow defining new transformations ( ), delete the existing ones ( ), move the transformations in the list up and down .. The order of transformations in the list defines the sequence in which these transformations are applied to the object.

The entry beneath the list displays the name of the selected transformation. It can be modified by entering a new value here.

Besides the geometrical parameters (offsets, rotations, scale), the additional parameters can be defined for the selected transformation:

Suppress. This parameter allows temporary excluding of the transformation from the object position calculation. A numerical variable can be used as the value of this parameter. The transformation will be suppressed, if the variable value is not zero.

Only for exploded view. With this flag set, the transformation will be taken into the account in the exploded state only (the command **3VX: Exploded View**).

The general parameter **Apply exploded view transformation of base elements** also affects the behavior of the current 3D object in the exploded state. With the flag set, all exploding transformations defined for the parent elements of the 3D object will be applied to this object as well. Consider, for example, a nut and bolt subassembly. The nut will first be subjected to the bolt exploding transformation, and then to its own. With the flag cleared, the object's position in the exploded state is defined by its own transformations only.

3D Construction Entities

WORKPLANES

A workplane is an auxiliary 3D construction element visualizing the abstract concept of the geometrical plane. The main purpose of a workplane is bringing two-dimensional drawing objects into the three-dimensional space. Such objects are used for creating flat 3D construction elements on their base, such as 3D nodes, 3D profiles, 3D paths, etc.). Each workplane is related with a certain page of a 2D drawing. One can say that workplanes provide a link between 2D drawing pages and the three-dimensional space.

Besides, workplanes are used as auxiliary elements in various 3D operations. For example, they help creating the section plane within a body in the operation "Cut by Section", or position the section plane when creating a section. The normal to a workplane can be used for defining an extrusion direction or a revolution axis, etc.

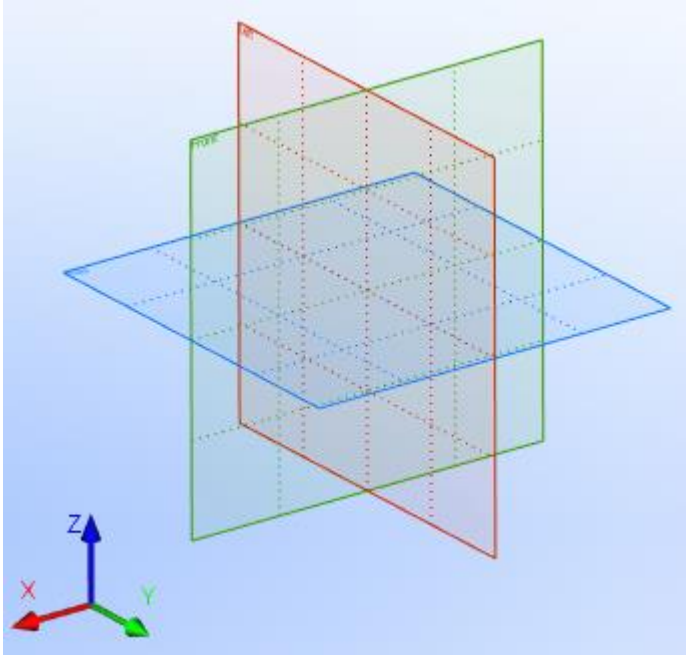
All workplanes in T-FLEX CAD are divided into two groups: standard and auxiliary. *Standard* are the T-FLEX CAD workplanes that coincide with, or parallel to, one of the principal coordinate planes of the world coordinate system (XY, YZ, ZX). Such workplanes are commonly used for creating base 3D constructions. However, to create a complete 3D model, using just standard workplanes may not be enough, thus prompting the use of *auxiliary* workplanes. The latter are created based on other 2D and 3D elements: already existing workplanes, vertices, edges and faces of 3D bodies, 2D projections, etc.

T-FLEX CAD provides a dedicated command for creating workplanes: **3W: Construct Workplane**, that allows creating various types of workplanes. Besides that, the system provides mechanisms that speed up creation of workplanes and drawing on those: creation of workplanes in "transparent mode" and drawing directly in the 3D scene in the active workplane mode, creation of workplanes parallel to already existing workplanes with the help of the special manipulators.

PRINCIPLES OF WORKING WITH WORKPLANES

Let's start getting acquainted with workplanes with the most simple and often used techniques of working with workplanes. Those include creation of standard planes as part of creating a new document (using the standard prototype) and creation of planes based on flat faces of a body. Those do not require use of a special command, allowing creating workplanes in "transparent mode".

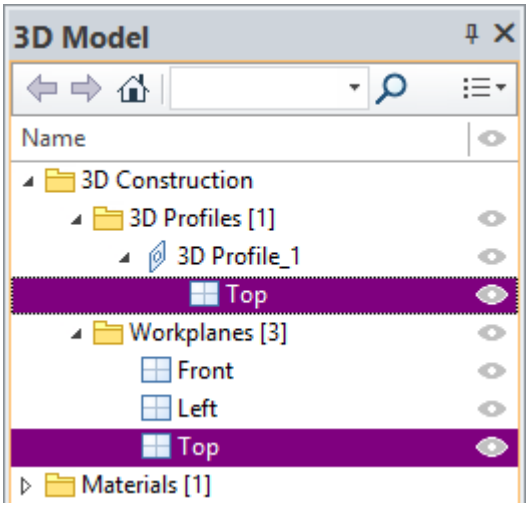
When creating a new 3D model based on a standard prototype, the three standard workplanes are created in the model from the beginning – "Front", "Left" and "Top". The image of the three standard planes is displayed in the 3D scene as three intersecting semitransparent rectangles defining the visual borders of the planes.



Note that the workplane borders are shown in the 3D scene for display purpose only, as they are actually infinite.

The outer borders of a workplane in the 3D window are drawn in a solid line, while the inner witness lines – in thin dashed lines.

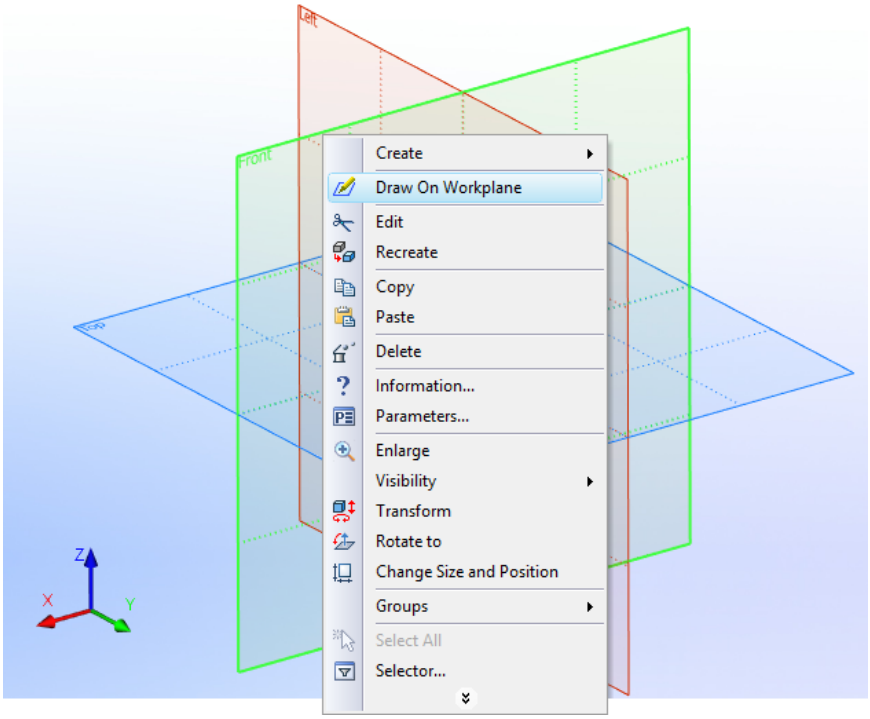
Moving the pointer to a line of a workplane highlights the workplane. A workplane can be selected for further manipulations by clicking, while right clicking calls the context menu for the workplane.



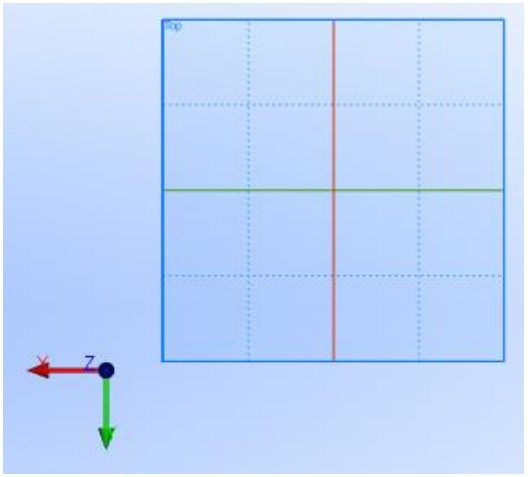
A workplane can also be selected in the 3D model tree, in the folder "Workplanes" or in a subtree representing the history of creation of some 3D element based on the workplane.

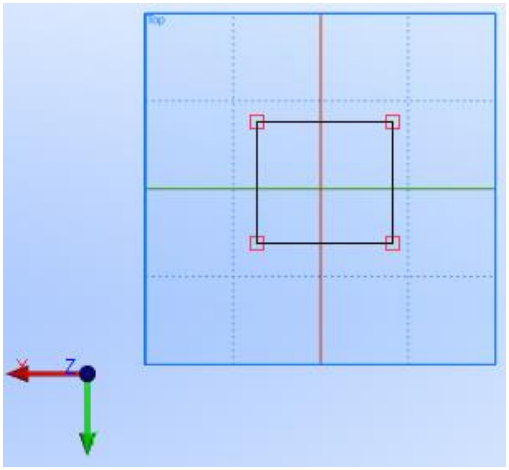
Let's review the main techniques of handling workplanes on a simple example. Suppose, we need to create a 3D profile on the workplane "Top", to use as a base for creating a 3D body.

Select the workplane "Top" in the 3D window or in the model tree, and right click. In the coming up context menu, select the item **Draw On Workplane**. The selected workplane activates.



The active workplane mode is intended for applying 2D elements on a plane in the three-dimensional space. When starting the mode of drawing on the active workplane, the 3D scene rotates in such a way, that the selected workplane becomes orthogonal to the viewing direction. From this moment, any relocation/rotation of the 3D scene is blocked. The main toolbar of the T-FLEX CAD (if it is not locked) will be switched to the mode “Workplane”. The 2D drawing commands become accessible in the textual menu.





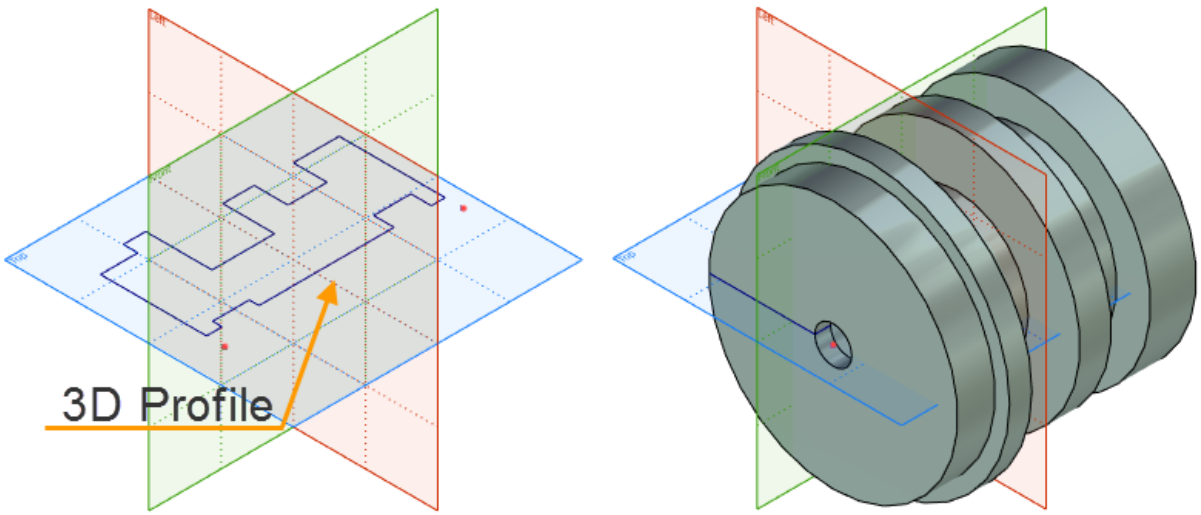
Now, you can draw on the workplane as if on a common 2D drawing page. Let's draw the contour of the future 3D profile directly on the workplane in the 3D window. (To do this, use the command **G: Create**

Graphic Line or **SK: Create Sketch**.) After that, finish working on the active workplane by simply pressing the option on the main toolbar (or in the context menu upon selection of any element):

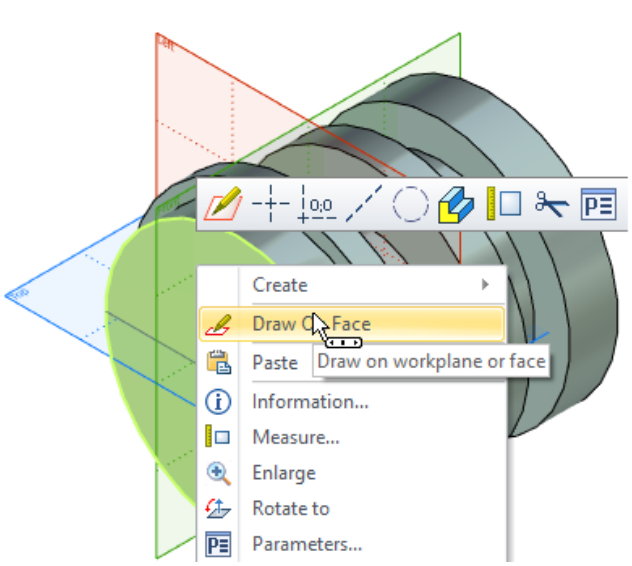
Finish	<Shift><End>	End Drawing On Active Workplane

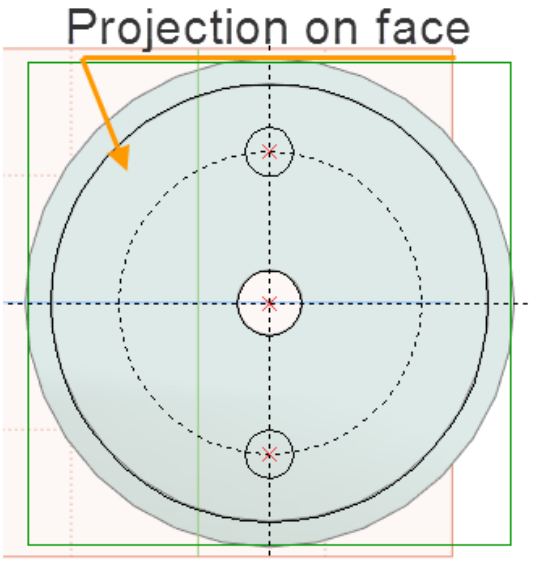
The 3D scene rotate back to its original orientation. The 2D commands will become inaccessible again. Meanwhile, a 3D profile appears on the workplane "Top", repeating the shape of the 2D constructions made by us on the workplane.

> The mode of automatic creation of 3D profiles on a workplane supports use of not only graphic lines, but also text and hatch. To create a parametric 3D profile, you should use lines and hatches in the same ways as you do when creating a parametric drawing (by snapping to construction lines).
>
> To create several 3D contours simultaneously on a single plane, it is possible to use graphic lines of different colors or hatches (this is described in more detail in the "3D Profiles" chapter).

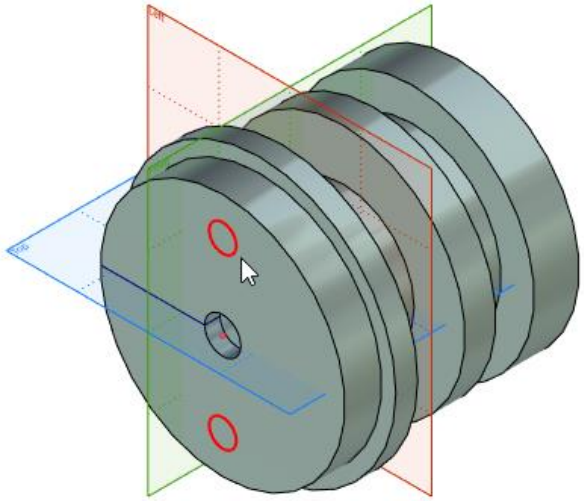


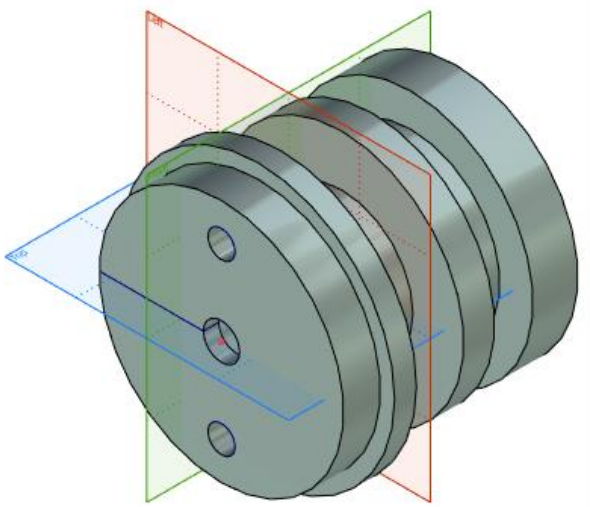
Further, suppose that the created profile is used in the rotation operation, resulting in a 3D body (see the chapter "Rotation"). Additionally, a hole or slot needs to be created on one of its slanted faces. To do this, you need to create a workplane lying on a face of the body. On this workplane, you will create a 3D profile – the contour of the future hole. The profile can then be extruded to the specified depth and, after that, subtracted by the Boolean operation from the original part. How can you do that? Simply move the pointer to the desired face and call the context menu. If the face is flat (as in our case), then the command **Draw On Face** will be provided in the context menu. Calling this command results in the following: a workplane will be created on the selected face, and activate. Besides, this source face is automatically projected on the created plane (see the chapter "2D projections"). The projection lines and their characteristic points can be used for creating the necessary 2D constructions on the workplane, for example, for defining hatches or construction lines to snap to.



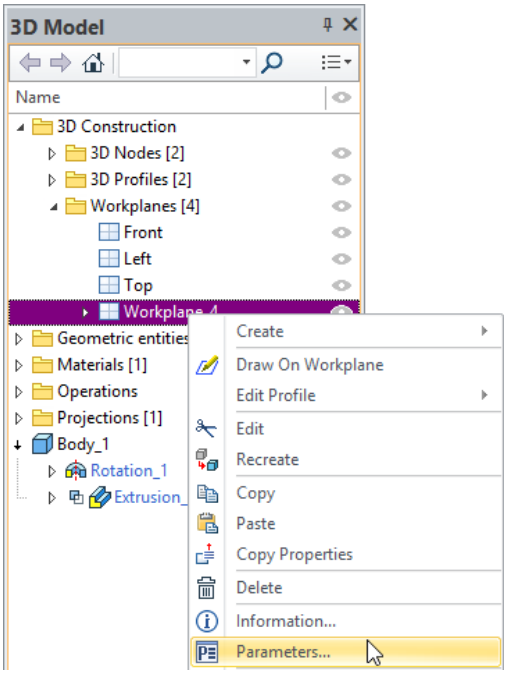


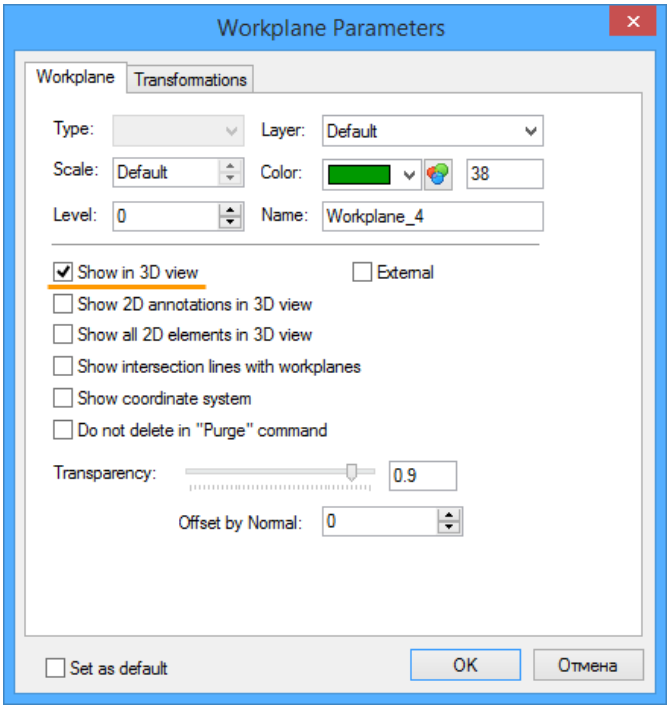
On the newly created plane, let's draw the contour of the hole, and then finish the drawing in the active workplane mode. The 3D contour will then be displayed in the 3D scene on the face we selected, that can be used for constructing the hole.

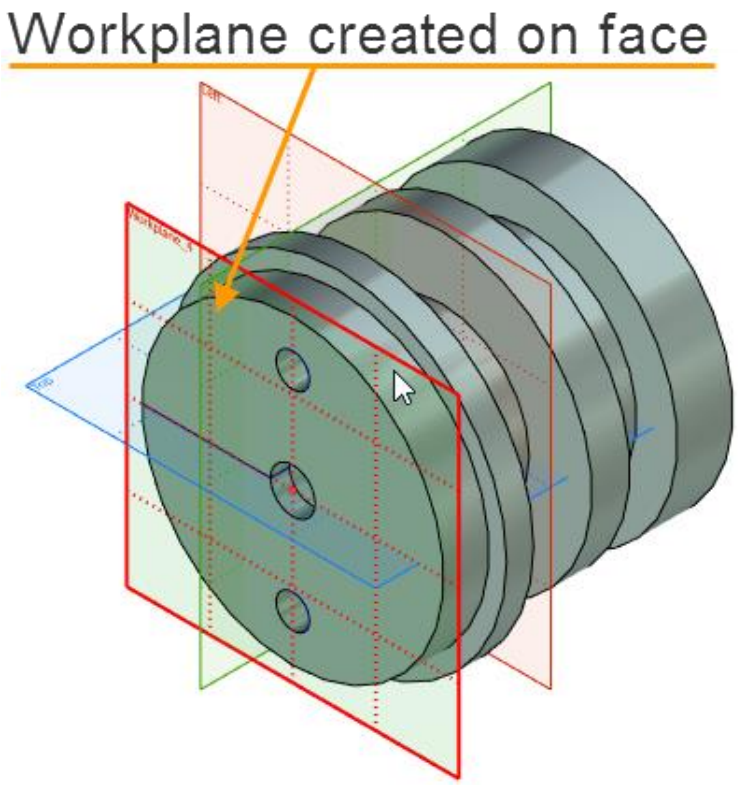




Note that the workplane created on a face is not displayed in the 3D scene. However, it appears in the 3D model tree. To make such workplane visible in the 3D scene as well, open its parameters dialog box (for example, by calling the context menu of that plane in the 3D model tree) and set the flag **Show in 3D View**.

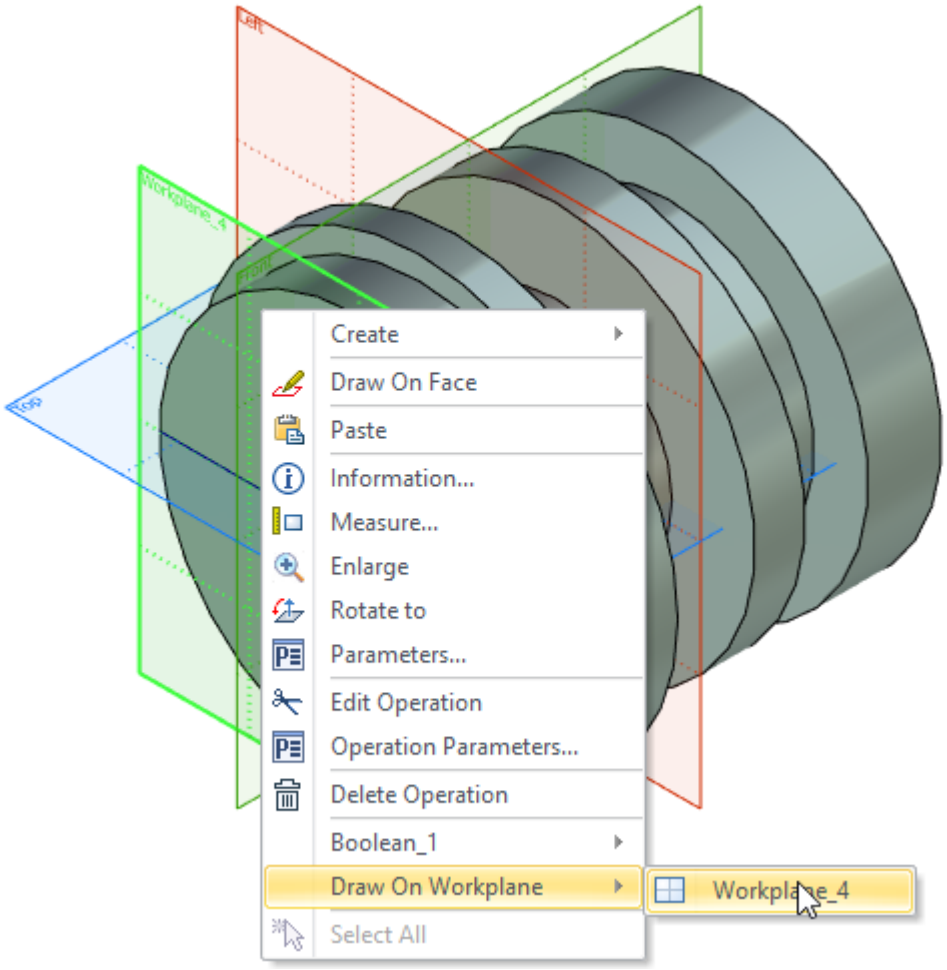






If you select the same face of the body again and call the context menu, you will find in it two similar commands: the already familiar **Draw On Face** and **Draw On Workplane**. We already know the purpose

of the first one – it creates another workplane based on the selected face. The second one allows activating the already existing workplane, created on the selected face (if you need, for instance, to edit the 2D constructions made on that plane). If several such workplanes are created, the desired one can be selected from the pulldown list.



Thus, you have learned the simplest and most commonly used techniques of creating workplanes. Other workplane creation techniques are used in the cases, when the standard planes and the planes created on the body faces are not enough for solving the posed tasks. Those techniques imply use of the dedicated command. Just before learning those techniques, let us clarify the relation between workplanes and the 2D drawing and review in details working in the active workplane mode.

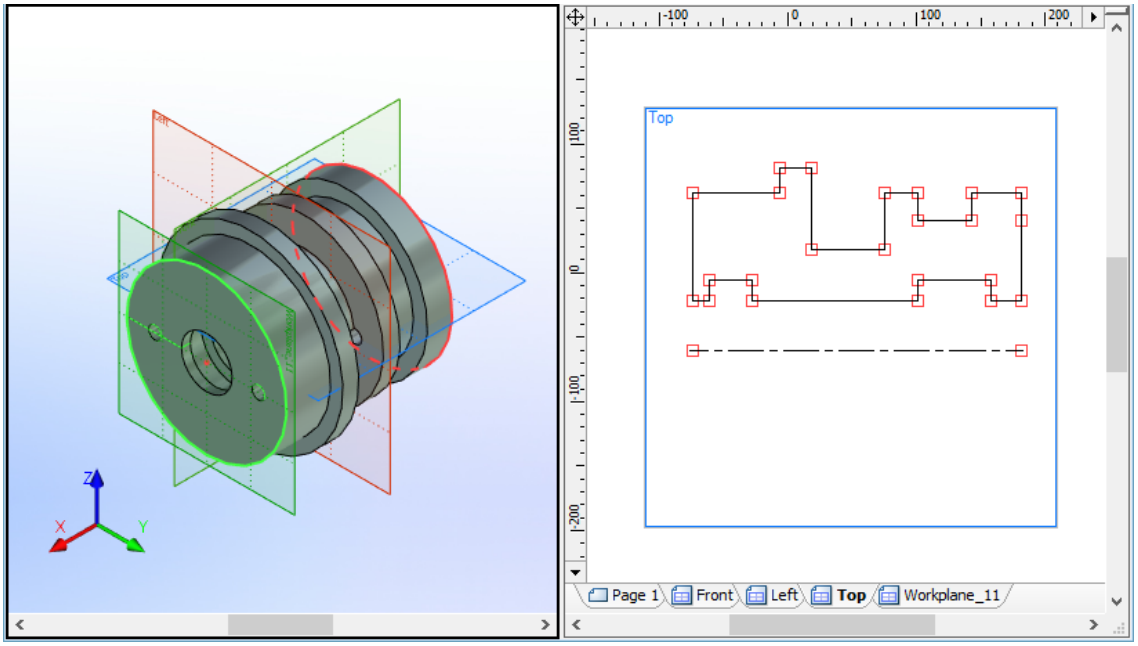
Workplanes and 2D Drawing

Any workplane in the document is linked to a page of the 2D drawing. Any time you create a workplane, a special 2D page of the type "Workplane" is automatically created in the document. The workplane will

be displayed on this page as a rectangle defining the visual borders of the plane in the 3D window. The name of the workplane is displayed in the upper left corner of the rectangle.

All 2D constructions made on this page are assumed by the system as lying on the workplane linked to this page. Vice versa, anything created directly on the workplane in the mode of drawing on the active workplane, appears on the linked 2D page of the document. In fact, one can think of a workplane as a 2D page in the 3D space. This rule applies to most workplanes in T-FLEX CAD. The exception is the workplanes created on the base of an existing 2D drawing page and its 2D constructions. No separate page is created for such workplanes. Instead, the link is established with the 2D drawing page that was used for creating the workplanes. Several workplanes of this type can be linked to such page in a 2D drawing. The 2D constructions belonging to this page can be linked to any of such workplanes. Usually, when selecting 2D elements intended for creating 3D elements on a workplane, the system suggests the workplane nearest to those elements, judging by the rectangular borders of the planes.

Therefore, 2D images can be created or edited on a workplane not only in the 3D window *in the active workplane mode*, but also in the 2D window. To do this, you need to open a drawing page in the 2D window, linked to this workplane. This approach is particularly useful when working with workplanes created in the 2D window, that is, based on an already existing drawing page. Workplanes can be activated only when they're located on a separate page of the 2D drawing. If several workplanes of the described type are linked to one page of a 2D drawing, they cannot be activated.



When working with the 2D pages of the workplanes created in the 3D window, first make sure that the display of the auxiliary pages is turned on in the command **PG: Pages**, and that the respective pages are displayed in the 2D window.

WORKING WITH ACTIVE WORKPLANE

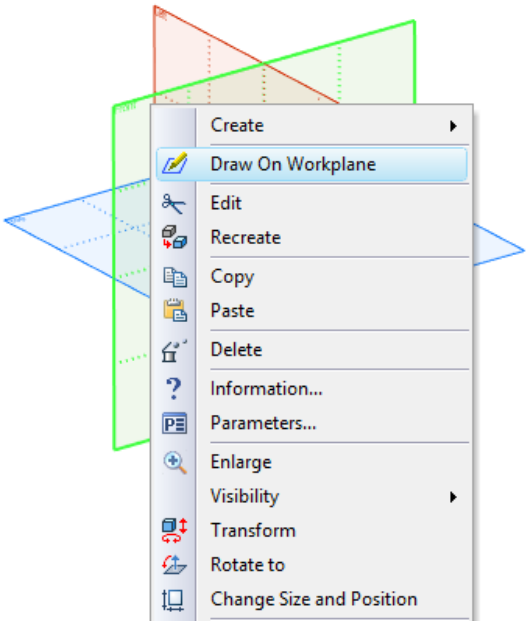
The mode of drawing on the active workplane is intended for placing 2D elements on a plane in the three-dimensional space and their further use in creation of the 3D model. Let's review this working mode in details.

Workplane Activation

A workplane can be activated in several ways. All possible ways of activating a workplane are listed here:

Drawing on the selected plane. Point the mouse to a workplane and click . The selected workplane gets highlighted. At this moment, you can start creating elements, using 2D drafting commands. Activate a 2D command. This will activate the workplane. On the main toolbar (if it is not locked) the button set "Workplane" will be shown.

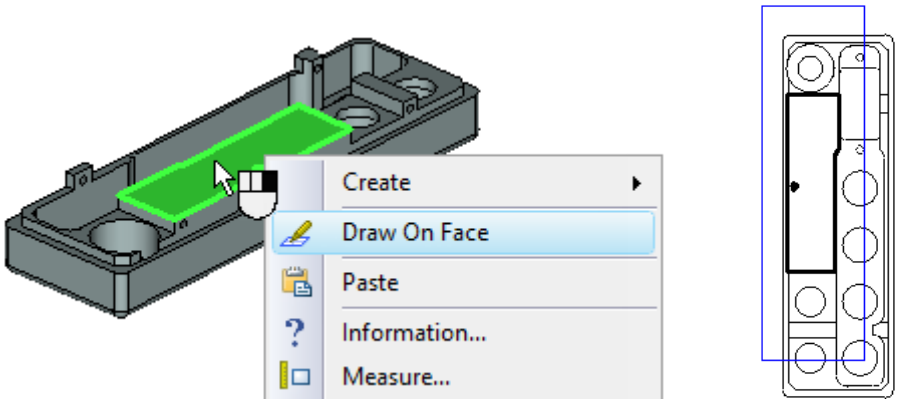
Activation from the context menu. Select a workplane in the 3D window or in the model tree and right click . In the coming up context menu, select the item "**Draw On Workplane**".



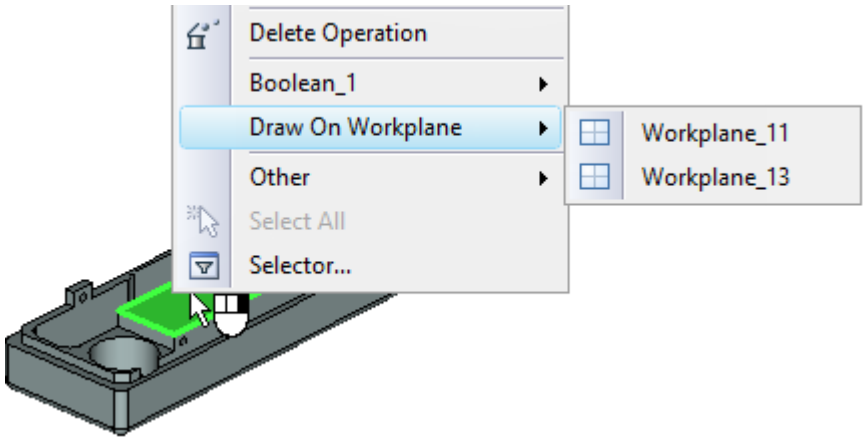
If such item is not present in the menu, that means, the plane cannot be activated. All necessary constructions should be done in this case in the 2D view (on the page where this workplane is located), and then carried over to the 3D scene by using the appropriate 3D commands.

Activation from the list. While the system is in the command waiting mode, point the mouse to an empty space in the 3D scene and right click . In the coming up menu, select the item "**Draw On Workplane...**". The list of workplanes available for activation in this document will be displayed on the screen. Select the desired workplane in the list and press **[OK]**.

Activation after creation. When creating a workplane on a face of a solid body, this workplane can be activated automatically. For this, point the mouse to the face to create the workplane on, and right click. In the coming up context menu, select the item **"Draw On Face"**. A workplane will be created based on the selected face, along with the page to contain all elements belonging to the workplane. The created workplane will be activated.



Activation from the context menu with plane selection. This is activation of an existing workplane created based on a body face. Point the mouse to a face of the body and right click. If there are workplanes created on this face, then the coming up context menu will have the item **Draw On Workplane**. Selecting this command displays the list of workplanes created on the current face. The workplane selected in this list will be activated.



Active Workplane Control

Upon activating a workplane, on the main toolbar (if it is not locked) the button set "Workplane" will be shown. The options on this toolbar allow performing the following actions:

		Rotate Active Workplane Orthogonal To View Direction

If the whole 3D scene needs to be rotated while working with the active workplane, use the option:

		Rotate 3D Seen/Select 3D Elements

You can also rotate the 3D scene by the mouse while holding down the <Alt> key. While in the 3D scene rotating mode, you can select 3D elements (as for projecting on the workplane, etc.). To continue drawing on the active workplane, turn off this mode.

If you prefer working in the 2D window, you can open the 2D drawing page corresponding to the active workplane. This is done by the option:

		Open 2D Window With Active Workplane

The page is displayed in the full-screen mode. It also contains the projection of the selected face. All constructions made on this page are automatically carried over the active workplane. To return to the 3D window, turn off this option.

If you need to project some model elements on the workplane, use the option:

		Project Elements On Active Workplane

This option is available only when working in the 3D window. Upon selecting this option, the icons appear in the automenu for selecting 3D model elements (operations, faces, edges) that you need to project on the active workplane. Upon finishing the selection, confirm the projection creation with the icon .

A special option is provided for projecting a 3D node or a body vertex on the active workplane

		Create Projection of 3D Node or Vertex

Upon activating this option, simply select the nodes or vertices to project on this workplane.

If the projected elements fall beyond the displayed bounds of the active workplane, their projection on the active workplane will not be displayed in the 3D window. In such a case, it may be convenient to use the 2D window of the active workplane. Alternatively, you can change the size of the workplane.

To align the cutting plane with the active workplane (see the chapter "Working in the 3D view window"), set the option:

		Move Cutting Plane To Active Workplane

Additional control over the cutting plane is provided when using the option .

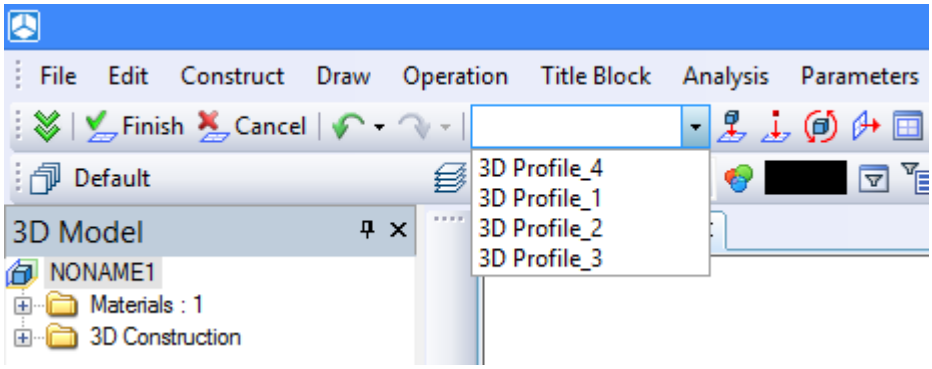
To finish drawing on the active workplane, press the icon:

	<Shift><End>	End Drawing On Active Workplane

If the 3D window is active, you can also point the mouse to an empty area in the 3D scene and right click . The coming up context menu will have the item **End Drawing On Active Workplane**.

While working in the active workplane mode, you can use any construction and 2D element creation commands. If the workplane was created on a body face, then the face projection on this workplane will be automatically created on the drawing page linked to this workplane. While working with 2D drafting commands, you can snap construction and graphic elements to the lines of the projection and their joint points.

In the "Workplane" and "Workplane (draft)" sets of the main toolbar of T-FLEX CAD, the element of control with the list of 3D profiles constructed by graphic lines of the given plane is available. If one of the profiles is selected from this list, the graphic lines that form this profile are automatically displayed on the screen in large size and newly created graphic lines are assigned the color of the selected profile.



Upon finishing working on the active workplane, the system automatically creates a 3D profile, using the following elements: hatch contours, graphic lines and TrueType-font text. If several hatches or text were created on the 2D page of the workplane, then multiple 3D profiles are automatically created (by the number of identified hatches and/or text). If graphic lines of different colors exist on the workplane, the system, upon exiting the mode of drawing on the plane, will prompt the user to create several 3D profiles – according to the number of used colors. Each of the created profiles will contain only the lines of one specific color.

To finish working on active workplane without creating a 3D profile, use the option:

Cancel	<Alt><End>	Cancel Drawing On Active Workplane

While working in the active workplane mode, you can call 3D operation creation commands. In such a case, the system finishes drawing on the workplane, and automatically highlights the created profile if the operation requires profile selection.

SETTING VIEW OF WORKPLANES IN 3D SCENE

Appearance of workplanes in the 3D scene is controlled by collection of parameters found in the settings of the system, of the individual document and the workplane itself.

First of all, it is possible to control the transparency of the workplanes (from totally transparent up to totally opaque). Parameter that controls the degree of transparency of an individual workplane is available in the dialog of parameters of the given plane (see the "Parameters of workplanes" section).

At the corner of the rectangle that constitutes the image of the workplane, the name or the type of the given plane is usually indicated. It is possible to hide/show them with the help of the "Workplanes > Draw name" parameter in the dialog of the **ST: Specify Parameters of Document** command (the "3D" tab). The font that is used for displaying the name of the workplane is specified in the **SO: Set System Options** command, on the "3D" tab.

The font size of the workplane is specified by the **Workplane font** parameter on the **3D** tab in the dialog of the **ST: Specify Parameters of Document** command. The larger the value of this parameter, the larger the font size will be.

Inside the rectangle of the workplane, the coordinate system of the given plane can be displayed. The visibility of the coordinate system is controlled with the **Show coordinate systems** parameter in the dialog of parameters of the given plane (see the "Parameters of workplanes" section). By default the coordinate system of the workplane is not displayed.

In the **SO: Set System Options command**, the tab **3D**, it is also possible to specify the required number of internal lines on the image of the workplane (the **Workplanes\Number of lines** parameter). There, you can also define the mode of selecting workplanes – by the bounding rectangle lines only or by all displayed lines (the **Workplanes\Selection by all lines** parameter).

WORKPLANE CREATION

In the beginning of this chapter, we described the most convenient ways of making workplanes – creation of standard planes together with the new document creation (using the standard prototype) and, later, creation of workplanes based on flat body faces. The dedicated command **3W: Construct Workplane** provides wider opportunities for workplane creation:

Icon	Ribbon
	3D Model → Construct → Workplane
Keyboard	Textual Menu
<3W>	Construct > Workplane

The command allows creating workplanes of the following types:

- Standard workplane.
- Workplane parallel to a geometrical plane defined by various 3D elements (face, other workplane, flat edge, LCS, etc.).
- Workplane passing through a 3D point (3D node, body vertex, path or 3D profile, edge midpoint, etc.).
- Workplane passing through a 3D line.
- Workplane orthogonal to a 3D curve.

- Workplane tangent to a surface.
- Workplane based on a local coordinate system.
- A copy of an existing workplane (transformed).
- Workplane based on a 2D projection.
- Placed proportionally between two planes or faces;
- Workplane based on a local 2D view.

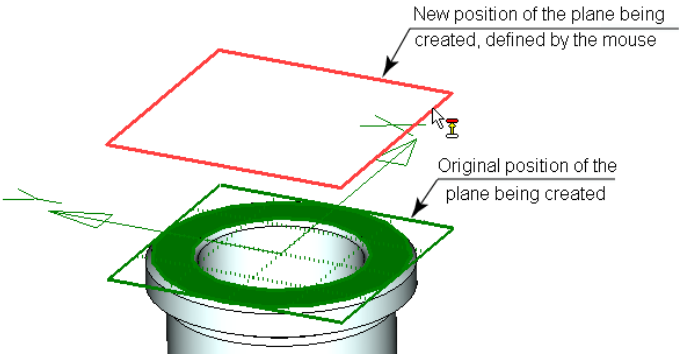
Creating Workplane Parallel to a Geometrical Plane

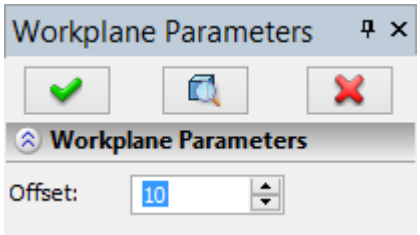
To create a workplane parallel to an arbitrary geometrical plane, use the following option of the command automenu:

	<1>	Select Plane that defines Workplane position

The set of filters for this option allows selecting a 3D element in the 3D scene, defining the desired geometrical plane. This can be another workplane, a flat face (the face underlying plane is actually used), a flat elliptical edge, LCS (one of its principal planes), etc.

Upon selecting the 3D element in the 3D scene, the new workplane is displayed. The original position of the created plane coincides with the position of the geometrical plane defined by the parent 3D element. Exception is the case of creating a workplane based on another workplane – in such a case, a small initial offset is used by default. The final position of the plane being created can be defined by entering a numerical value of the offset from the original 3D element in the command property window. Alternatively, the offset can be defined directly in the 3D scene, using the mouse. For this, you need to move the pointer to the displayed image of the plane being created (the pointer will change to), depress and, while holding down the left mouse button, move the plane to the desired position.





One can also specify an additional 3D element to define the workplane position. To define the position of the plane being created by using additional 3D elements, the following options are used:

	<3>	Select first Point of Workplane position
	<5>	Select Surface to which Workplane is tangent
	<7>	Select Edge to which Workplane is tangent

The workplane will be oriented parallel to the original 3D element (or, rather, the geometrical plane defined by this element) and pass either through the specified 3D point or tangent to the face (or, rather, to its underlying surface) or to the edge. If two tangency configurations are found for the specified face or edge, the point will be selected, that was nearest to the pointer at the time of selecting the tangency element. The workplane will assume its new position instantly after selecting one of the elements.

If the original 3D element is a face or edge, an additional option will be provided in the automenu:

	<8>	Project Selected Face to Workplane Being Created

This option allows projecting the selected face on the page of the workplane being created. The option is turned on by default.

Upon the confirmation by the option , the workplane being created assumes its new position according to the specified offset.

There is another way of creating a workplane on a 3D element. While the system is in the command waiting mode, move the pointer to the 3D element (a flat face or edge) and right click. In the coming up context menu, select the command **Create** > **Construct Workplane**. As a result, the workplane creation command will be launched, with automatic selection of this 3D element.

Alternatively, when selecting a face, in the same context menu you can select the command "Draw On Face". As a result, a workplane will be created based on the specified face, and the system will enter the mode of working on the active workplane.

Creating Workplane through a 3D Point

To create a workplane based on a 3D point, you need to select the following option in the command's main automenu:

	<3>	Select first 3D Point of Workplane position

Using this option, you can select a 3D element in the 3D scene, that defines the necessary point. This could be a 3D node, a body vertex, a center of an elliptical edge or a midpoint of a straight one, an LCS origin, etc. The coordinate origin of the workplane being created will be positioned in the selected point.

The following additional options we will appear in the automenu after picking the option :

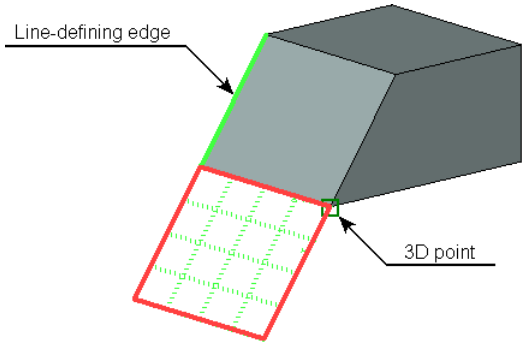
	<3>	Select first 3D Point of Workplane position
	<4>	Select a line through which the workplane goes
	<5>	Select second 3D Point of Workplane position
	<7>	Select third 3D Point of Workplane position
	<8>	Select Plane from which Workplane angle is measured

	<9>	Select Direction
	<T>	Select Surface to which the Workplane is tangent

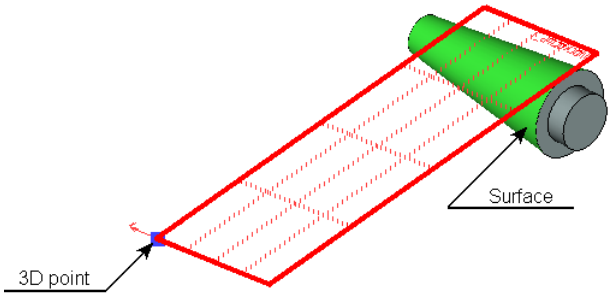
Using these options, you can construct the following types of workplanes based on the selected 3D point:

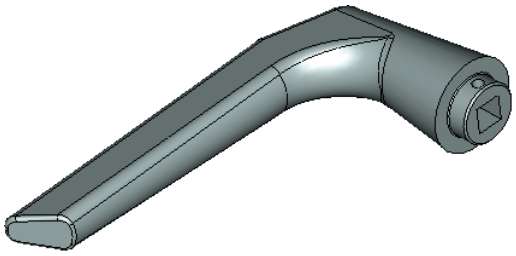
- Workplane going through a 3D point and a line;
- Workplane going through a 3D point and tangent to a cylindrical or conical surface.
- Workplane going through two 3D points;
- Workplane going through two 3D points and in the specified direction (or through two points at an angle to the specified direction);
- Workplane going through two 3D points at an angle to a plane;
- Workplane going through two 3D points and tangent to a conical, cylindrical or spherical surface.
- Workplane going through three 3D points.

To construct *Workplane going through a 3D point and a line*, after selecting the base 3D point you need to specify a line, through which the plane should go. The line is selected with the option . The selected line should not go through the coordinate system origin point. In this case, the X-axis of the coordinate system of the workplane being created will be directed along the selected line, while the Y-axis will be going through the coordinate system origin point perpendicular to the line.

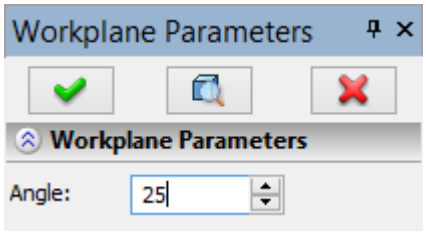


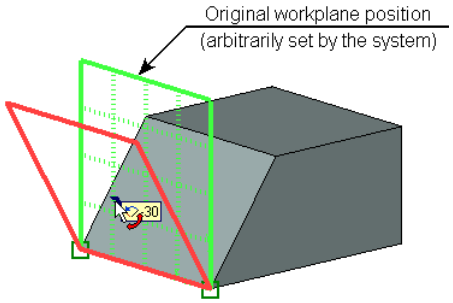
To construct *Workplane going through a 3D point and tangent to a conical or cylindrical surface*, after defining the 3D point you need to select the desired surface with the option . From the two tangency possibilities, the one will be selected that is closer to the cursor position at the instant of selecting the surface.





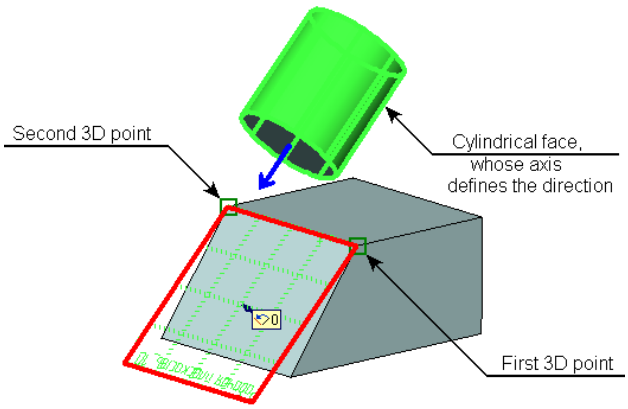
To construct *Workplane going through two 3D points*, you need to select an additional 3D point after selecting the first 3D point, using the option . The workplane is constructed as going through the axis defined by the selected 3D points. The workplane position in the space is set arbitrarily by the system. If necessary, the workplane position can be modified by rotating it at the specified angle relative to the original position. The angle is defined in the command's properties window or directly in the 3D window using a dragger.

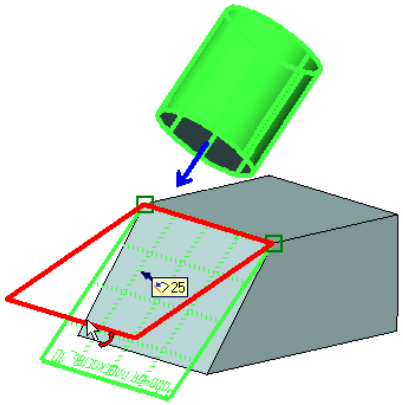




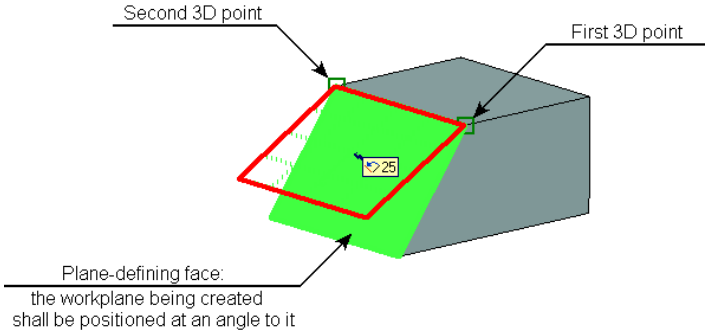
To construct *Workplane going through two 3D points in the specified direction*, after defining the first 3D point you need to select the second 3D point (the option) and a 3D element defining a direction (the option). The direction can be defined with a planar edge or 3D path, a normal to a surface, an axis of a cylindrical or conical surface, etc.

If necessary, it is possible to define an additional rotation angle of the workplane relative to a specified direction (in the properties window or in the 3D window using a dragger).



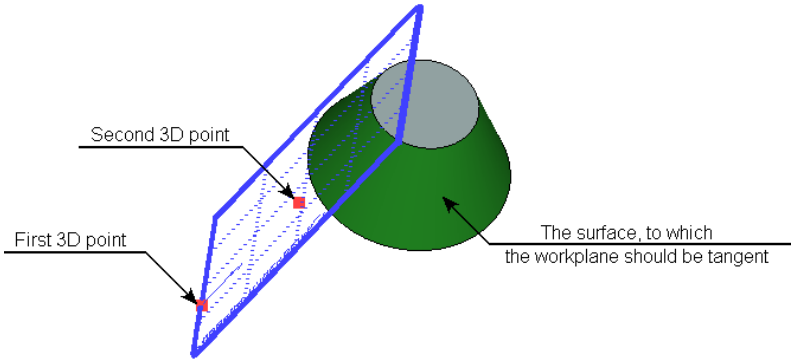


To construct *Workplane going through two 3D points at an angle to a plane*, after defining the first 3D point you need to select the second 3D point (the option) and a plane (the option). To define a plane, you can select another workplane, a flat (planar) face, a planar edge, an LCS, etc. As in the previous cases, the angle is defined in the properties window or in the 3D window using a dragger.

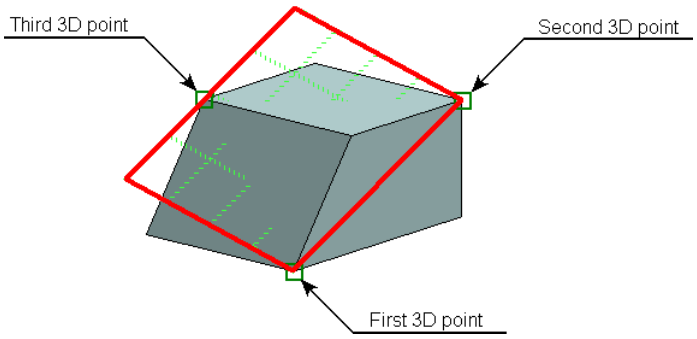


To construct *Workplane going through two 3D points tangent to a surface* (conical, cylindrical or spherical), you need to specify the first 3D point, and then the second 3D point (the option) and a surface (the option). The X-axis of the created workplane coordinate system will be directed from the first selected point to the second one.

When selecting a conical or cylindrical tangency surface, the construction result will be defined similarly to that of a workplane by a single 3D point with a tangency to a surface. The second point definition could merely help define the tangency configuration.



To construct *Workplane going through three 3D points*, you need to define the first 3D point, then select the second 3D point (the option) and the third 3D point (the option). The X-axis of the created workplane coordinate system will be directed from the first selected point to the second one, and the Y-axis will be perpendicular to it.



The workplane creation must be confirmed by the option .

Creating Workplane through a 3D Line

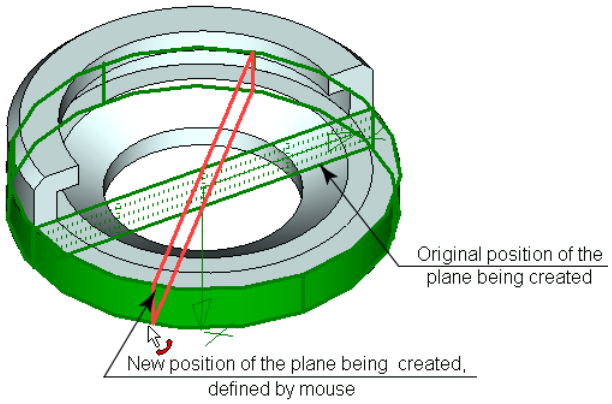
A workplane can be defined as passing through a flat edge or a 3D line defined by a straight edge, an axis of revolution, an LCS axis or by other 3D elements. To define a workplane of such type, the following options of the main command automenu are used:

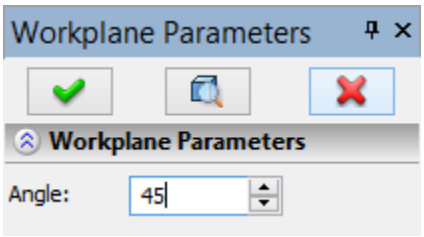
	<5>	Select flat Edge through which Workplane passes
	<6>	Select Axis for the Workplane

To select a flat edge (the option), simply move the pointer to the edge and click . To select a 3D straight line (the option), select a 3D element suitable for defining a straight line (an axis). The list of 3D elements allowed for selection is defined by the state of the selection filters in the pulldown list of the option.

The origin of the coordinate system of the workplane being created is defined by the start point of the edge (the closing point in the case of periodical edges) or by the midpoint of the specified line. The X-axis is directed along the edge (tangent to the edge in the case of closed edges) or along the line.

Initially, the workplane being created can be rotated above the selected edge or 3D line by defining the rotation angle by a numerical value in the command property window. Alternatively, the plane rotation angle can be defined directly in the 3D scene by using the mouse. To do this, move the pointer to the image of the plane being created (the pointer will change to), depress and, while holding down the left mouse button, move the plane to the desired position.





Plane rotation can also be done with respect to a plane that can be selected by using the option:

	<3>	Select Plane from which Workplane angle is measured

This option allows defining a geometrical plane by specifying a 3D element lying on it: a flat curved edge, a path or profile, a flat face, a workplane, etc.

The workplane being created will be rotated about the edge or 3D line by the specified angle with respect to the selected element (originally, this angle is considered equal to 0°). The rotation angle can be defined by a numerical value in the property window or input directly in the 3D scene, using the dragger.

The workplane rotation angle can also be specified by selecting an additional 3D point for the plane to pass through. To define the point, use the following automenu option:

	<5>	Select first Point of Workplane position

When selecting a 3D line or straight flat edge, there is yet another way of rotating the plane – by specifying a face to which the workplane should be made tangent. The face can be of cylindrical, conical or spherical shape. The face selection is done by the option:

	<7>	Select Surface to which Workplane is tangent

The face won't get selected if its shape is different from the specified above, or if it cannot be made tangent to the workplane being created. Upon selecting the face, the workplane being created will rotate about the edge or the 3D line to become tangent to the face. The workplane creation must be confirmed by the option .

Creating Workplane Orthogonal to a 3D Curve

To set the mode of creating this type of workplanes, in the main command automenu select the option:

	<7>	Create Workplane Orthogonal to Path

Upon selecting this option, the following actions become accessible:

	<2>	Select Path to which Workplane is orthogonal

This option allows selecting a 3D element defining the source curve to which the workplane should be made orthogonal. This can be a 3D path, 3D profile or its boundary, an edge or a loop of edges.

Upon selecting the source curve, the image of the workplane is displayed orthogonal to the curve, at its start point. The position of the workplane can be adjusted by selecting a 3D point for the workplane to pass through:

	<3>	Select Point that sets Workplane position

Upon selecting the 3D point, the workplane will reposition in the 3D scene to pass through the selected point while staying orthogonal to the source curve. The workplane origin will be placed at the intersection point. If the selected point is not lying on the path or edge, then the X-axis will be directed to this point. Otherwise, the direction of the X-axis is defined by the system automatically.

The workplane creation must be confirmed by the option .

Creating Workplane Tangent to a Surface

The mode of creating this type of workplane is set by the option:

	<8>	Select Surface to which Workplane is tangent

After that, you need to select the surface to become tangent to the workplane being created. This is done by the option:

	<1>	Select Surface to which Workplane is tangent

This option allows selecting a 3D element suitable for defining a surface. This can be a face, a 3D profile or a simple body, a worksurface, workplane, or a flat edge.

Upon selecting the face, you need to select a 3D point, whose projection on the selected surface is to become the tangency point between the workplane being created and the surface (the projection is done by the shortest distance to the surface):

	<2>	Select first Point of Workplane position

Workplane Origin Selection

This option is available only when creating workplanes based on 3D elements in the ways described above. To modify the origin point resulting from the workplane construction, you can use the option:

	<O>	Define Workplane coordinate system

After that, the options will appear in the automenu for defining 3D points, whose projections on the workplane being created, will define the position and orientation of its local coordinate system axes:

	<1>	Select Point that defines Workplane Origin

	<3>	Select Point of X-axis direction
	<F>	Eliminate Workplane coordinate system definition

The option allows selecting a 3D point, whose projection on the workplane will be considered the new origin. The option allows selecting a 3D point, whose projection on the workplane will be defining the direction of the X-axis of the plane coordinates. The point selection can be undone using the option .

To exit the mode of defining the plane coordinate system, use the option:

	<Alt+Down Arrow>	Return to previous mode

Creating Copy of Workplane

To construct a workplane of this type, select the following option in the main menu:

	<A>	Create Workplane Copy

As a result, the following options become available:

	<W>	Select source Workplane
	<1>	Select source LCS
	<2>	Select target LCS
	<F>	Cancel LCS selection

To create a copy, do the following steps:

- Select source Workplane.

The selection can be done either in the 3D window or in the 2D window. You can set the desired offset and rotation angle values in the tab “Transformation” of the workplane parameters. Rotations in this case are done with respect to the world coordinate system. This is sufficient for creating a copy.

- Select source LCS.

Defining the transformation parameters at this step results in rotations with respect to the selected coordinate system.

- Select target LCS.

The workplane position is defined by repositioning of the source workplane according to the transition from the source to the target coordinate system. Additionally, you can define the transformation parameters. Rotations will be done with respect to the target coordinates.

The workplane is created on the current drawing page. All parameters of the copy plane that are relevant to the 2D view (origin, displayed borders) are the same as the source workplane parameters.

Creating Workplane Based on Local Coordinate System

A local coordinate system (LCS) is an auxiliary 3D construction element used for defining 3D object positions in the space. A workplane based on an LCS coincides with one of its principal planes – XY, XZ, YZ. Since LCS axes can be directed arbitrarily, use of this construction technique allows positioning the workplane in any desired point in the space and in any orientation.

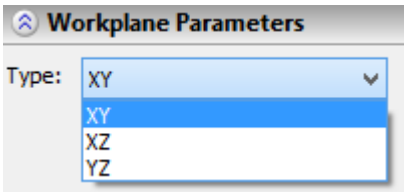
To create such workplane, in the main command automenu select the option:

	<L>	Construct Workplane based on LCS

As a result, the following actions are accessible:

	<N>	Set attachment point
	<L>	Select target LCS
	<B>	Define Workplane borders

To create the workplane, just select a local coordinate system (in the 3D window or in the 3D model tree) using the option . The image of the workplane being created will be instantly displayed in the 3D window, coinciding (by default) with the XY plane of the selected LCS. One of the corners of the displayed workplane border rectangle will be at the LCS origin. At this step, you can already complete the plane creation by pressing . In this case, a separate "Workplane"-type page will be created in the 2D drawing for the new workplane.



You can change the default positioning of the plane with respect to the source LCS. Firstly, you can set a different principal plane of this coordinate system. To do this, in the property window select the desired option. As you do, the workplane image in the 3D window will instantly adjust its position.

Secondly, you can select a 2D node in the 2D window, using the option , to be assigned as the origin of the coordinate system. In this case, the workplane is created on the page of the 2D drawing to which the selected 2D node belongs. The image of the plane in the 3D window will move to make the LCS coincide with the point on the plane defined by this node.

Finally, you can additionally change the size and position of the displayed borders of the workplane, using the option . The borders are changed in the same way as when creating a standard plane.

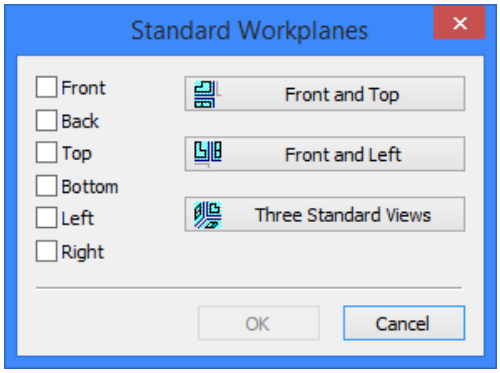
Creating Standard Workplane in 3D Window

In the cases when the new document was not based on the standard prototype, you may need to manually create one or several standard planes. To do this, activate the 3D window, and in the workplane creation command select the option:

	<S>	Construct standard Workplane

A standard workplane can be created either in the 2D or in the 3D window. The way of creating the plane and the obtained result depend on what window was active at the time of selecting this option.

The dialog box appears in the screen for selecting standard views. The workplanes will be made parallel to the respective principal coordinate planes, as defined by the selected views. One can select several standard views simultaneously in the list on the left-hand side. Confirming the selection by pressing the **[OK]** button leads to automatic creation of the workplanes for the specified views. Separate "Workplane"-type 2D pages will be created for each workplane. Such pages are not displayed in the 2D window by default.



The displaying of the workplane pages in the 2D window can be turned on using the command **Customize** > **Pages**, by setting the flag **Show Pages/Workplanes**. Such workplanes are displayed in the 3D view by default. The position in the three-dimensional space is determined by the world coordinates that provide the common origin for all workplanes created in this way.

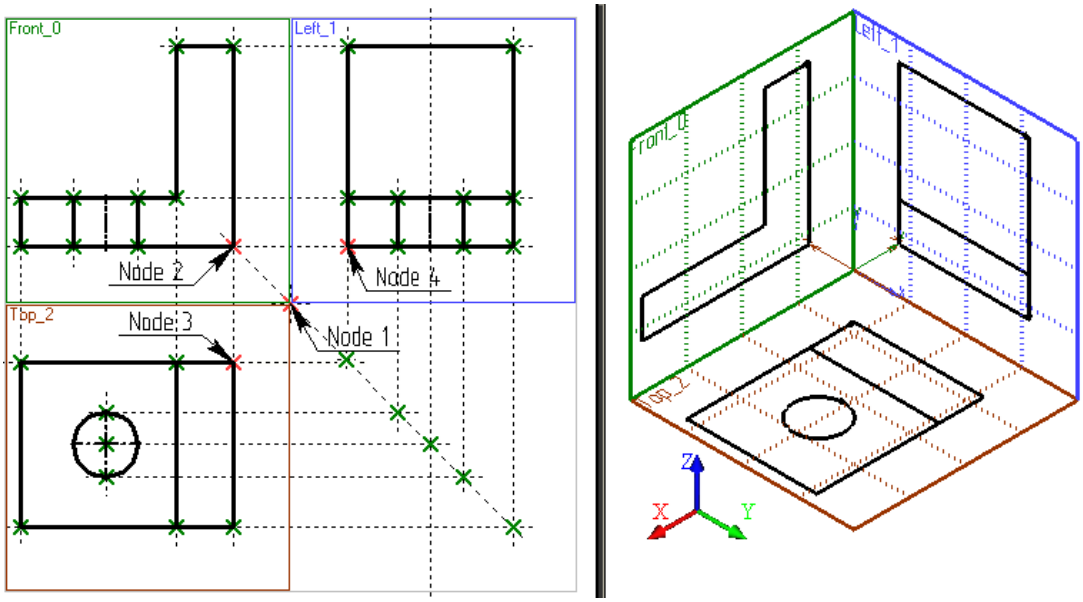
The origin of each workplane is located in the center of the rectangle that defines the workplane display in the 3D window. A standard workplane origin can be modified only in the editing mode. In this case, the origin selection is done in the same way as when creating workplanes based on a 2D view (the section "Creation of standard workplane in 2D window").

Simultaneous creation of several standard workplanes

The dialog box launched by the option has another three additional options represented by the graphic buttons: [**Front and Top**], [**Front and Left**], [**Three Standard Views**]. You can simultaneously create several standard workplanes by using these options.

Creating Standard Workplane in 2D Window

Creation of workplanes in the 2D window, that is, based on an already existing page of a 2D drawing, containing 2D constructions, is mostly used with the technique "From 2D to 3D". Usually, standard planes are created for each of the main drawing views. The 3D constructions on the planes are built based on the drawings on the respective drawing views. This allows carrying the elements of a complete 2D drawing over the three-dimensional space.



To create a standard workplane in the 2D window, use the option . The sequence of steps for creating the plane in this case is as follows:

- Determine, to which of the principal coordinate planes the workplane being created should be made parallel. This is done by selecting the view for the workplane. For example, by selecting the front or back view, we assume that the plane being created is positioned parallel to the XZ plane of the world coordinate system.
- Select a 2D node on the current page of the drawing (*optional in some cases*). The node will define the position of the workplane borders rectangle on the page, as well as the origin of the plane local coordinates. If no workplane is yet linked to this drawing page, then you do not need to specify the node. In this case, one of the corners of the drawing format frame will be assumed as the local coordinate system origin.
- The default workplane borders are set as follows. The specified 2D node is considered the first corner of the border rectangle. The second corner is placed at the nearest corner of the drawing format frame. Standard planes are assigned a specific corner of the format frame, the front view being the upper left one, the bottom view – the lower left, the left view – the upper right.
- The correct definition of the displayed borders of the workplane helps simplify the future creation of 3D constructions on the plane from the 2D drawing. When selecting 2D elements to use for 3D

constructions, you need to define, in which workplane those elements belong. The system can do it automatically by taking the plane, whose border rectangle covers the selected 2D elements. If the borders of the planes on the 2D page do not align with the respective drawing views, the system choice will be incorrect, and you will have to select the workplane manually. Therefore, the attachment node should be selected in such a way, that the workplane border rectangle agreed with the drawing view intended for the plane. If this cannot be achieved by selecting the attachment node, the border size and position of the workplane being created can be additionally adjusted, as described in the next paragraph.

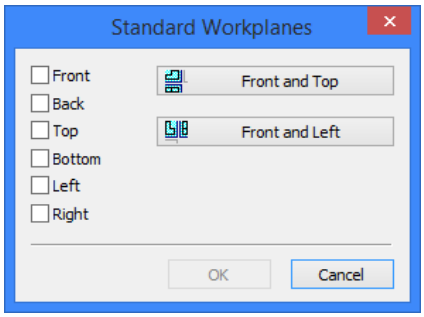
- Change the default settings of the plane displayed borders in the 2D window (*optional*). The borders can be modified in two ways. The first way is selecting another corner of the drawing format frame as the second point of the border. The second way is to define the borders of the plane anew by specifying the new positions for both corners (by selecting two 2D nodes).
- Select a 3D node defining the position of the workplane in the space (*optional*). By default, the standard workplane coincides with one of the principal coordinate planes of the world coordinate system. The origin of its local coordinate system is positioned in this case at the world coordinates origin. Instead, if defining the 3D node, the plane being created is carried over to the position defined by the selected node. That means, the resulting plane will be parallel to the principal coordinate plane while passing through the 3D node. The plane local coordinates origin is also placed at the selected 3D node.
- Define the displayed borders of the plane in the 2D window (*optional*). The default borders of a workplane are defined as follows. The first point is considered the point selected at the step 2. The second point is considered the nearest corner of the drawing borders. In the case of standard planes, a certain corner point is used (for the front view – the upper left one, for the bottom view – the lower left, for the left view – the upper right). The user can modify these borders (by selecting, for instance, two 2D nodes).
- Upon performing the necessary steps, confirm the workplane creation by the option:

	<Y>	Finish input

The workplane created in this way is not displayed by default in the 3D window. To have it displayed in the 3D scene, set the flag “Show in 3D View” in its parameters dialog box.

Let's review each step in details.

Activate the 2D window and select the option "Construct standard Workplane". The dialog for selecting the standard view will be displayed on the screen. Unlike its counterpart for creating the standard plane in the 3D window, this dialog allows selecting only one standard view from the list on the left-hand side.



If no workplanes exist on the current drawing page, then the rectangle will be drawn the size of the drawing format, marking the workplane being created.

Workplane creation can be completed at the step, using the option . The workplane local coordinate system origin will be set automatically.

If, however, there is at least one workplane on the current drawing page, then the workplane image will be displayed on the drawing only after performing the further steps. The following additional options are provided in the command automenu:

	<N>	Set attachment point
	<M>	Set relation with 3D Node
	<B>	Define Workplane borders

Suppose that the "Left" view was selected at the previous step. This means, the workplane being created will coincide (or stay parallel) with the YZ plane of the world coordinate system. After that, you can select a 2D node that will define the local coordinate system origin on this workplane. The node is selected using the option . The workplane creation can be finished at this step. In this case, it will coincide with the YZ plane of the world coordinates.

If necessary, the workplane borders set by default can be modified (the option). When calling this option, the following additional options become available in the automenu:

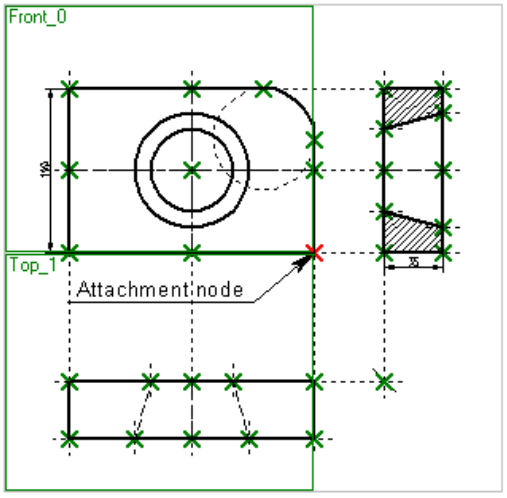
	<B>	Set border's 1st/2nd corner
	<Tab>	Define Workplane borders
	<K>	Eliminate borders
	<W>	Return to previous mode

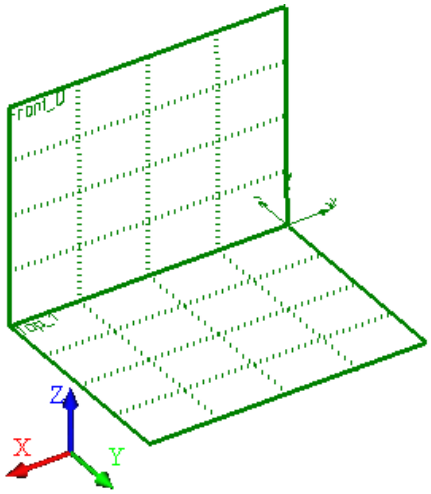
The borders of the workplane can be redefined in two ways. You can simply select another corner of the drawing format frame as the second point, using the option : pressing the icon repeatedly makes the

system toggle through all corners of the format frame. Alternatively, you can define the plane borders anew, attaching them to two nodes by the option . Attachments to the nodes can be canceled by the option .

The final step can be exact positioning of the plane being created in the space by specifying a 3D node (the option).

As in the case of creating a standard plane in the 3D window, multiple planes can be simultaneously created in the 2D window. However, in this case only two additional buttons are present in the dialog lunched by the option : [**Front and Top]** and [**Front and Left]**. Using these options, you can create two standard workplanes at once. In this case, simply specify the common coordinate system origin as a 2D node.





Creating Workplane Based on 2D Projection

The workplane created by a 2D projection is positioned in the same way as the projecting plane of the selected 2D projection. For example, if the projecting direction is defined by a body face, then the workplane will lie on this face. If the projecting direction was defined by a section, then the workplane will lie in the plane of the section, etc. Such construction technique allows creating, for example, workplanes parallel to the plane of an arbitrary section, whose orientation in the space is different from any other body faces or any existing workplanes.

To create a workplane by a 2D projection, just select the desired projection while in the workplane creation command. The created workplane is placed on the same page of the 2D drawing as the selected projection. The origin of the coordinate system of such plane is not defined explicitly. It is assigned by the system automatically as coinciding with the origin of the drawing page containing the 2D projection.

To create such workplane, select the option in the main command automenu. The following actions become accessible as a result:

	<J>	Construct Workplane based on 2D Projection
	<B>	Define Workplane borders

The 2D projection to use for the workplane creation is selected by clicking in the 2D window while the option is active. After selecting the 2D projection, a rectangle is displayed on the current page, defining the borders of the workplane being created.

The display of such workplane in the 3D view is controlled by the respective parameter in the operation settings.

The workplane borders are set by the system. If necessary, those can be redefined using the option.

The workplane creation is finalized by pressing.

Workplane between two planes or faces

A workplane in between two planes is created using the command's main automenu option:

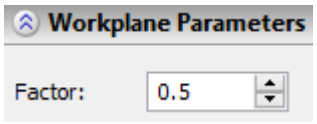
	<E>	Create Workplane between other Workplanes

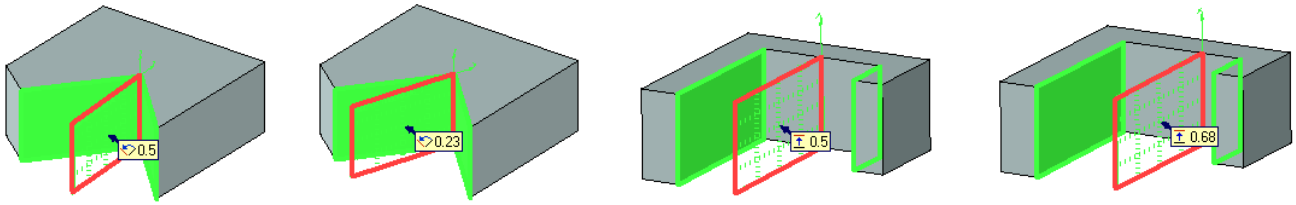
After selecting this option, you need to select two planes using the options:

	<1>	Select First Workplane
	<2>	Select Second Workplane

If the source planes are parallel, then the resulting plane divides the distance between them in the specified proportion (this is defined by a numerical parameter). If the source planes are not parallel – then it is the angle that is divided proportionally.

The parameter value that defines the position of the workplane being created relative to the source planes is entered in the command's properties window or in the 3D window using a dragger. The default preset value is 0.5, so that the created workplane is the symmetry plane for the selected planes.





Creating Workplane for Auxiliary 2D View

The workplane for an auxiliary 2D view is created using the option:

	<D>	Construct auxiliary Workplane

Workplane creation for an auxiliary 2D view is most often used in the technique "From 2D to 3D". As a reminder, let us mention that the 3D model creation is based in this case on a completed 2D drawing. First, workplanes are created according to the views of the drawing. Next, auxiliary 3D elements, such as nodes and profiles, are created based on 2D constructions. The 3D model is finally built based on the created auxiliary elements.

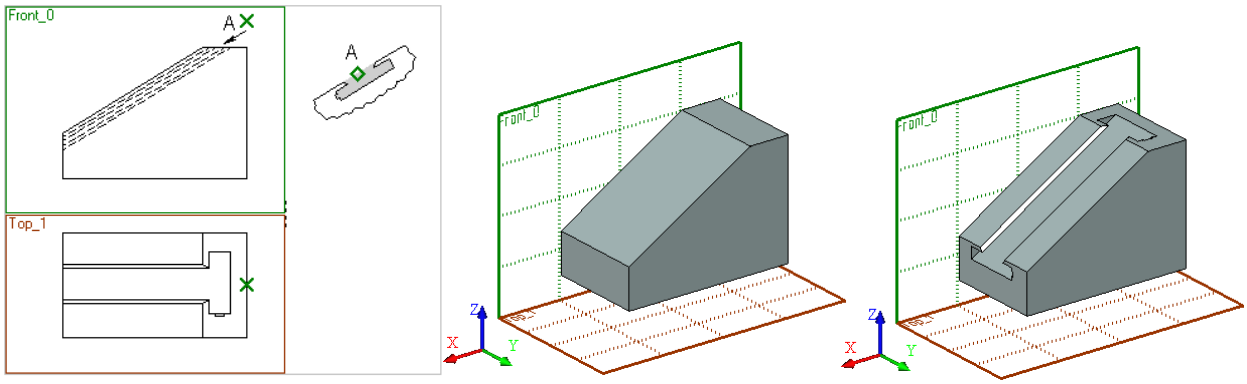
If a 2D drawing contains only main part views, creating just standard workplanes is enough. However, if additional views are present on the drawing (auxiliary or local section views, or simple slanted sections), whose projection plane is not parallel to any of the standard workplanes, you may need to use a *workplane based on an auxiliary 2D view*. Such a plane is oriented orthogonal to the selected auxiliary view direction. Therefore, creating a workplane based on an auxiliary 2D view amounts to "re-creating" the imaginary underlying plane of the projection defined by this auxiliary view.

> Note: the terms "projecting", "section" and "views" used in this chapter refer to general concepts of drafting and design, rather than to the T-FLEX CAD elements "2D Projection", "Section" or the 2D detailing element "Drawing View". The use of those terms is unrelated to the means of design which can be, for instance, manual drafting of a part's 2D representation.

Plane creation procedure

The preconditions to creating an auxiliary view-based workplane are as follows. The 2D drawing of the part must be fit with all necessary main views and sections of the part, including the source auxiliary view. The source workplane must be created in the *2D window*. This workplane must correspond with the part view that defines the view direction or the section plane of the auxiliary view in question.

For clarification, let's review the following example. Suppose, a 3D model is built based on the 2D drawing shown below. The general shape of the part was "re-created" by the front view and the top view. Now, we need to create a slot on the slanted face of the part based on an existing view "A".



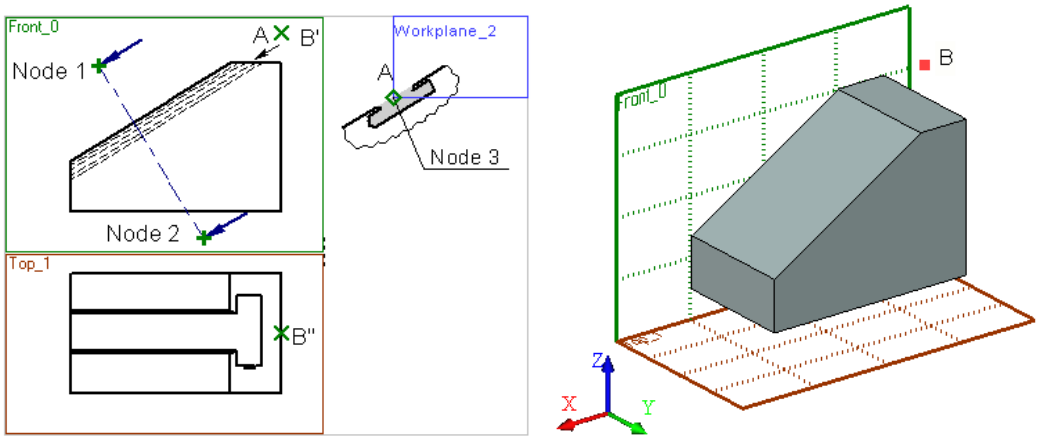
To create the slot, we need to create a workplane orthogonal to the view direction of the local view "A". This plane will help creating a 3D profile based on the view image. The resulting profile will then be used for creating the slot using an extrusion and a Boolean subtraction.

The workplane of the view "A" will be orthogonal to the "Front" plane. Their intersection line must be orthogonal to the arrow marking the view direction.

The workplane creation starts with specifying two 2D nodes ("Node 1" and "Node 2") defining the intersection line between the workplane being created and the existing plane. Upon specifying the nodes, the system automatically identifies the original workplane and the view direction for "projecting" on the auxiliary view plane being created. The workplane viewing direction is marked by two arrows that appear next to the selected nodes. This defines the preliminary positioning in the space of the workplane being created.

> The choice of the original plane and the view direction can be modified, if necessary. Note also, that the view direction initially defined by the system depends on the order of the nodes selection.

The next step is selection of a 2D node for the attachment point of the workplane being created on the 2D page. Normally, you select one of the 2D nodes belonging to the auxiliary view image. The same node defines the workplane borders and its local coordinate system origin. The first corner of the plane border rectangle will be at the selected node, while the second, the diagonal one – at the nearest corner of the drawing format frame on the current document page. In the example, let's select "Node 3" as the attachment node for the plane being created. The newly created workplane border rectangle is then displayed on the current page of the drawing.



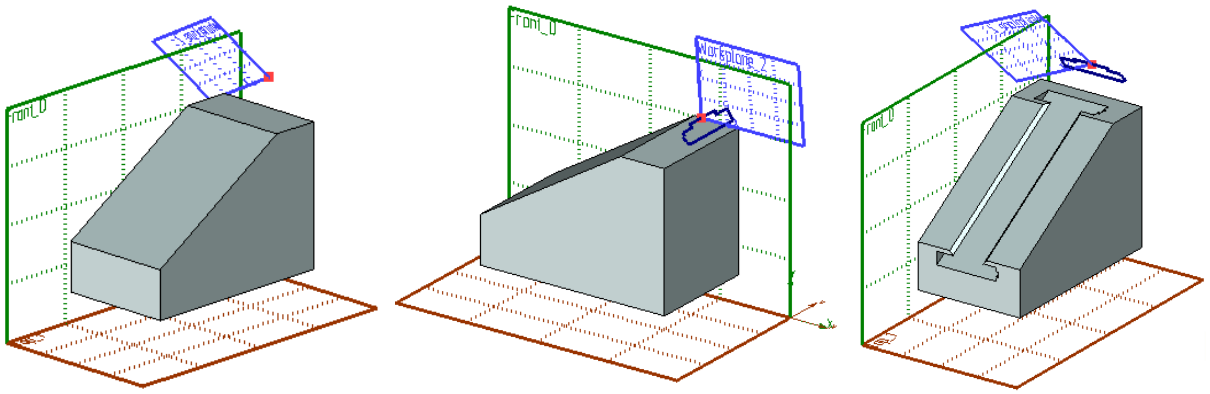
The local coordinate axes of the plane being created are automatically rotated, together with the plane itself, by the angle equal to that between the planes' intersection line and the vertical. This ensures the correct positioning in the space of the 3D elements that will be created on this plane from the lines of the auxiliary view.

Additionally, one can define a 3D node for the workplane being created to pass through. This step is optional. If no 3D node is specified, then the plane will be constructed through the world coordinates origin. The plane is positioned in the space in such a way, that the 2D attachment node defined at the

previous step becomes the projection of this 3D node on the plane being created. The attachment 3D point defines the exact positioning of the plane being created in the space.

In the described example, the new plane positioning is adjusted by using the 3D node "B" (constructed from its projections B' and B''). As a result, the plane will assume the position shown on the following diagram.

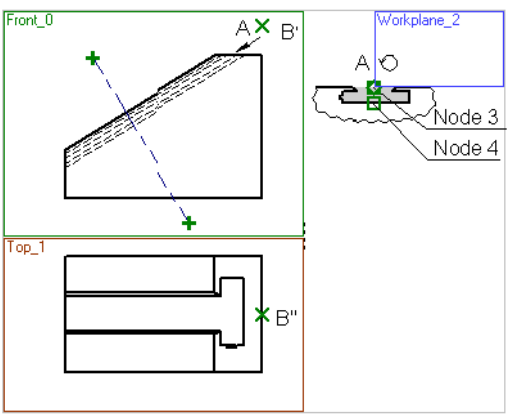
> To view the new plane position in the 3D space, set the flag in its parameters that controls the plane display in the 3D scene.

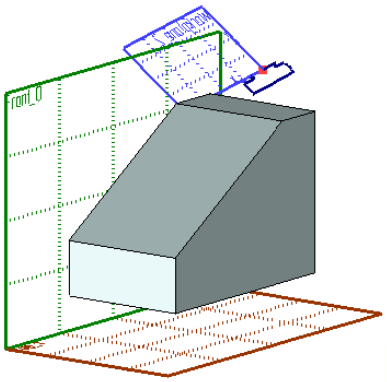


When creating a workplane of an auxiliary view, you can redefine the plane borders in the 2D window. To do this, specify 2D nodes on the current page of the document as the corners of the displayed plane border rectangle. This can be skipped in the described example.

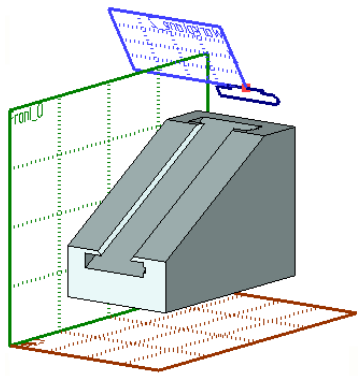
Note the hatch shown as gray filling on the view "A". If you create a 3D profile on the new plane based on this hatch, then the profile will be positioned just as a part element supposed to be in order to yield the projection on the created plane per the view "A".

If the image on the view "A" were rotated as shown on the diagram below, then we would need to additionally define the rotation angle for the plane being created. In such a case, two additional 2D nodes are to be specified ("Node 3" and "Node 4") to define the rotation axis. As a result, the local coordinate system of the plane being created is rotated additionally by the angle equal to that between this axis and the intersection line between the planes.





The plane and the respective profile orientation without the additional rotation angle



The plane and the respective profile orientation with the defined additional rotation angle

Plane creation

Creation of a workplane based on an auxiliary 2D view is done by using the following option in the command automenu:

	<D>	Construct auxiliary Workplane

Upon selecting this mode, the additional options appear in the automenu:

	<D>	Select 1st/2nd Node of view direction
	<W>	Select Other Workplane
	<A>	Set view rotation
	<Tab>	Switch view direction
	<N>	Set attachment point
	<M>	Set relation with Node
	<B>	Define Workplane borders

The default option is activated upon turning on the mode of creating the plane by an auxiliary 2D view. This option allows selecting subsequently two nodes in the 2D window (using), which define the planes intersection line. Upon selecting the first node, the system automatically determines and highlights the original workplane candidate. If a wrong plane was selected by the system, it can be changed by the option . Upon specifying the second node, the view direction of projecting on this plane is determined. The initially determined view direction can be flipped by the option .

Upon specifying both 2D nodes and, therefore, the intersection line between the planes, the option activates automatically in the automenu. It allows selecting the plane attachment node on the drawing page in the 2D window. The plane creation can be finished at this time by pressing in the command automenu or in the property window. Alternatively, you can continue with the plane creation by selecting a 3D node with the option , for the plane to pass through. This option activates automatically in the automenu after defining the attachment 2D node.

The default workplane borders can be modified using the option . Modifying borders of a workplane by an auxiliary 2D view is similar to modifying the borders of other types of workplanes described above.

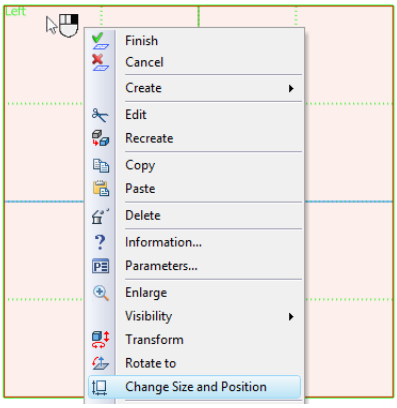
Additional rotation of the plane coordinate system (when dealing with a rotated image view) is defined by the option . Upon activating the option, specify two additional 2D nodes. The rotation direction depends on the order of selecting the nodes.

WORKPLANE SIZE MODIFICATION

To modify the displayed borders of any workplane that are not attached to 2D constructions, you can use the command **PZ: Set Paper Size**:

Icon	Ribbon
	Workplane → Modes → Page size
Keyboard	Textual Menu
<PZ >	Customize > Page Size

Modifying the displayed borders of a workplane changes the size of the 2D page created for this workplane.

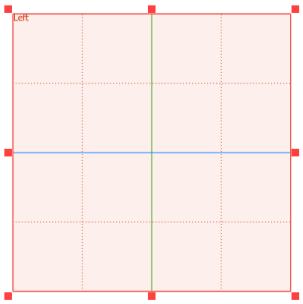


The command is accessible only when the page is active in the 2D window, whose size needs to be changed, or when the workplane is activated.

There is yet another way of calling this command. Move the pointer to the workplane and right click. This can be done either in the 3D window or in the 2D window. In the coming up context menu, select the item **Change Size and Position**. (This item does not appear in the context menu for workplanes created from existing 2D pages filled with constructions.)

As result of the described actions, the workplane and its respective page are displayed as shown:

To modify the size of the workplane, move the pointer to one of the square handles on the workplane border. The pointer will change the appearance, reflecting the current resizing action. Depress and move the mouse in the desired direction. The exact values can be entered in the command parameters.



To move the workplane, point the mouse within the border of the workplane. As the pointer changes appropriately, depress and move the workplane to the desired position.

The new size and position of the workplane is confirmed by .

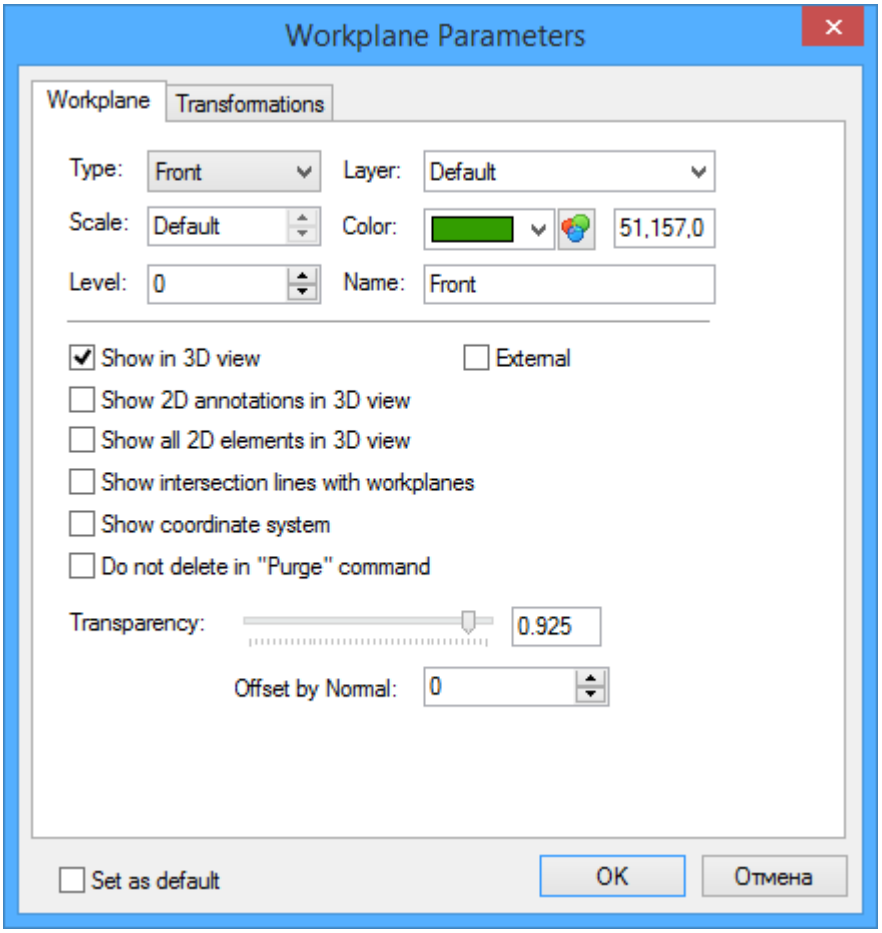
> The boundaries of the workplane can also be modified with the help of the special manipulators (see the "Manipulators of Workplanes" section).

WORKPLANE PARAMETERS

As any system elements, the workplanes are assigned a set of parameters. Their values can be defined in the dialog box called by the option:

	<P>	Set entity Parameters

The workplane parameters dialog box has two tabs: **Workplane** and **Transformation**.



The tab “Transformation” is described in details in the chapter “General parameters of 3D elements”. The tab “Workplane” contains general system parameters (**Layer**, **Color**, **Level**), the parameter **Name** (default names are “Workplane_0”, “Workplane_1”, etc.), which is common across all 3D elements, and several workplane-specific parameters. Those are:

Type. At plane creation, you do not edit the plane type in this dialog box, since the plane type is always predefined. When editing, you can modify the types of standard workplanes only, as, for instance, replacing “Front” by “Left”.

Scale. Defines scaling of the 3D construction elements created with the help of this workplane. The default value is “Default”, that is, the scale of the 2D drawing page, on which the created workplane is based.

Sometimes, 3D constructions are based on 2D elements drawn to a scale different from the page scale. In this case, the workplane scale needs to be defined equal to the scale of the 2D elements. Suppose, for example, that the scale of a 2D drawing page is equal to 1:25. The main drawing of the part on this page is done to the scale of the page. Suppose, however, that a local view is magnified to the scale 1:4. A workplane is created on this drawing page. To capture the

geometrical data of the local view, this workplane should use the same scale as the view, that is, 1:4.

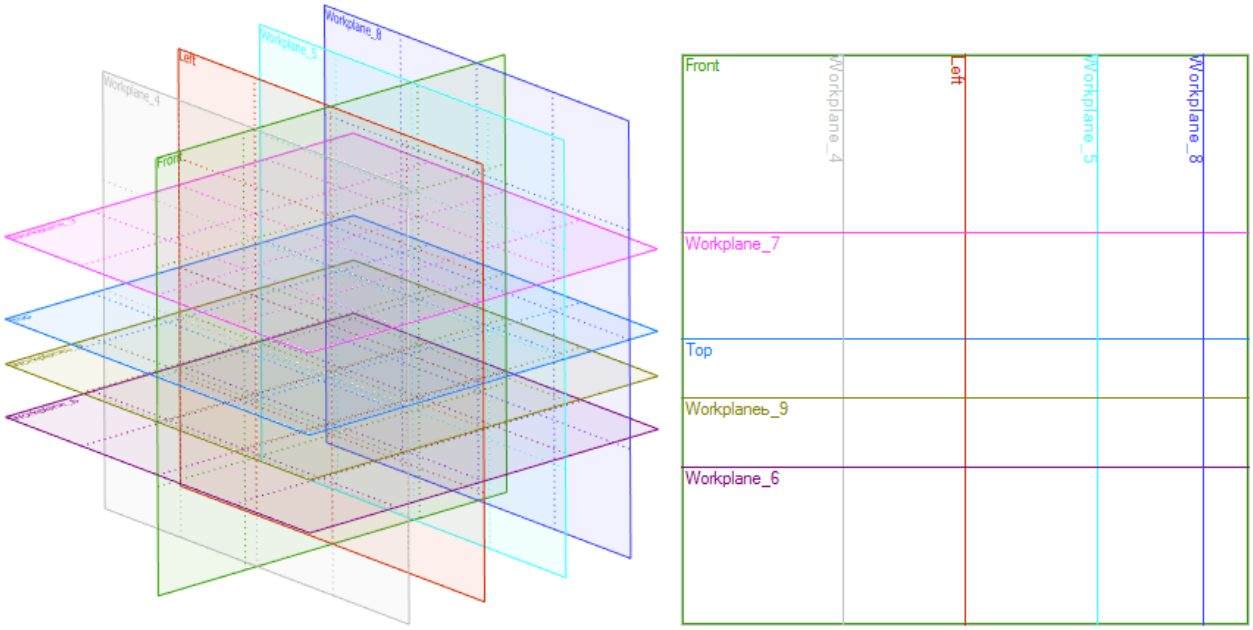
External. Setting this flag allows referencing the given workplane in assemblies when inserting the current file as a fragment.

Show in 3D View. This flag controls the display of the workplane in the 3D view.

Show 2D Annotations on 3D View. This parameter allows carrying the annotation elements from a Workplane over to the 3D space. When the flag is set, the annotation elements (dimensions, leader notes, roughness symbols) that were created on that Workplane become displayed in the 3D window, and you can work with them as with the usual 3D annotation elements.

Show all 2D elements in 3D view. When the flag is set, all hatches, graphic lines and other elements of the 2D drawing are displayed in the 3D scene.

Show intersection lines with workplanes. When the flag is set, not only workplane boundaries but also intersection lines with the other model workplanes will be shown.



Intersection lines can be selected to create new construction lines.

Show Coordinate System. This parameter controls the view of the workplane's own coordinate system on its image in the 3D scene. By default, this parameter is turned off, i.e., the coordinate system is not displayed.

Do not delete in Purge command. When the flag is set the workplane will not be deleted, even if it is not used for the operations creation. Option is useful when workplane contains sketch, which removal from the document is undesirable.

Transparency. This parameter allows a user to control the transparency level for the rectangular box of the workplane in the 3D window.

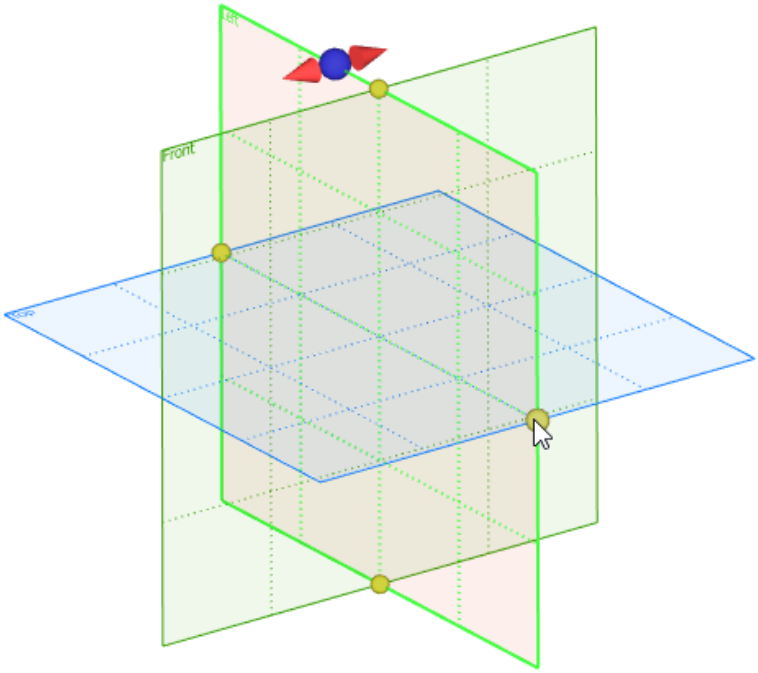
Offset by Normal. This parameter defines the offset along the normal to the workplane with respect to the parent element. For example, for the workplanes created parallel to a geometric plane, the offset is measured from the initial geometric plane. For the workplanes constructed independently of other 3D objects (for example, for standard workplanes), the offset is specified with respect to the zero point of the global coordinate system.

MANIPULATORS OF WORKPLANES

For convenience of work with workplanes in T-FLEX CAD, special manipulators can be used.

When selecting a workplane in the 3D scene outside the command, on the workplane's image the special manipulators will appear in the form of a rectangle.

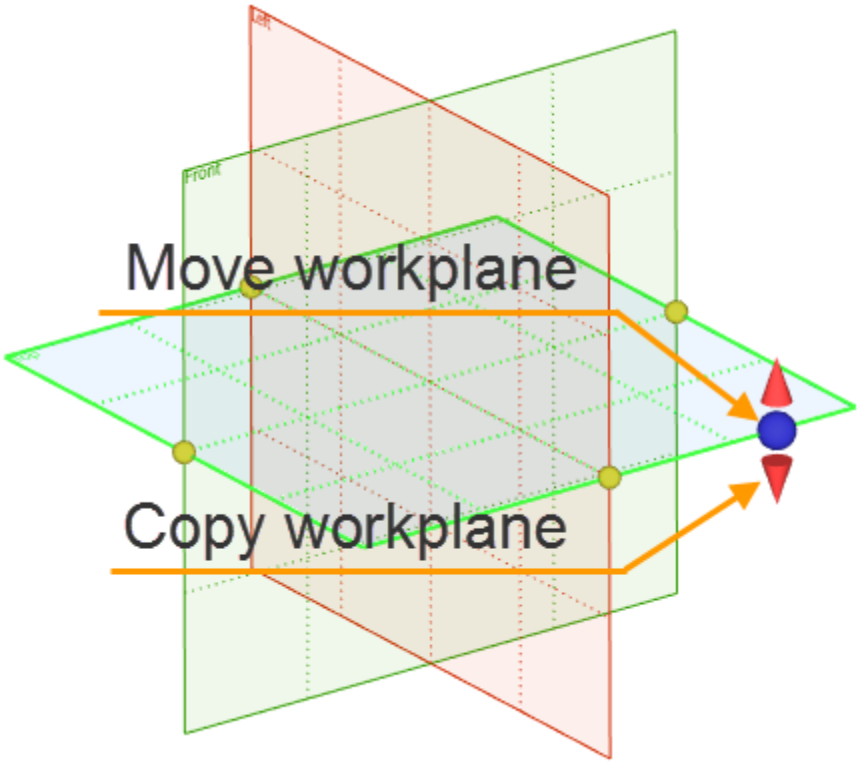
Small manipulators in the form of spheres located at the mid-sides of the workplane's rectangle allow us to quickly modify the visible boundaries of the workplane. .



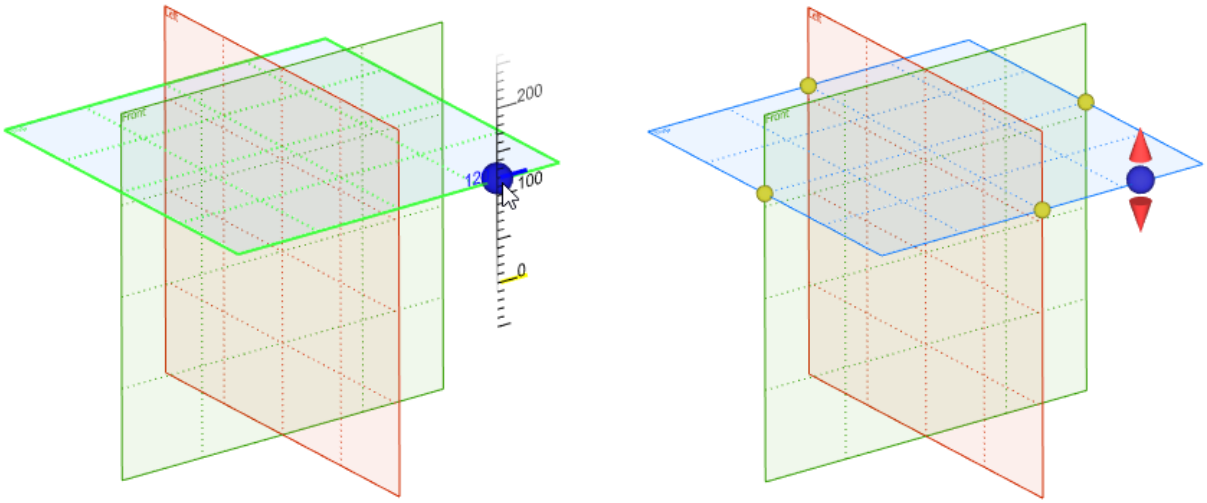
When the cursor is moved to such a manipulator, the size of the manipulator is slightly increased. Working with the manipulator can be carried out in two ways. First, it is possible to point at the manipulator and press . The selected boundary of the plane will start moving after the cursor. Pressing for the second time will fix the new location of the boundary of the workplane. The second way – by pointing at the manipulator, press and without releasing the pressed mouse button, move the selected boundary of the plane to the desired location. In this case, to fix the new size of the workplane, it is sufficient to release the pressed mouse button.

In both cases, when the manipulator is activated, in the system toolbar appear the additional elements for exact specification of the new location of the boundary of the plane being edited.

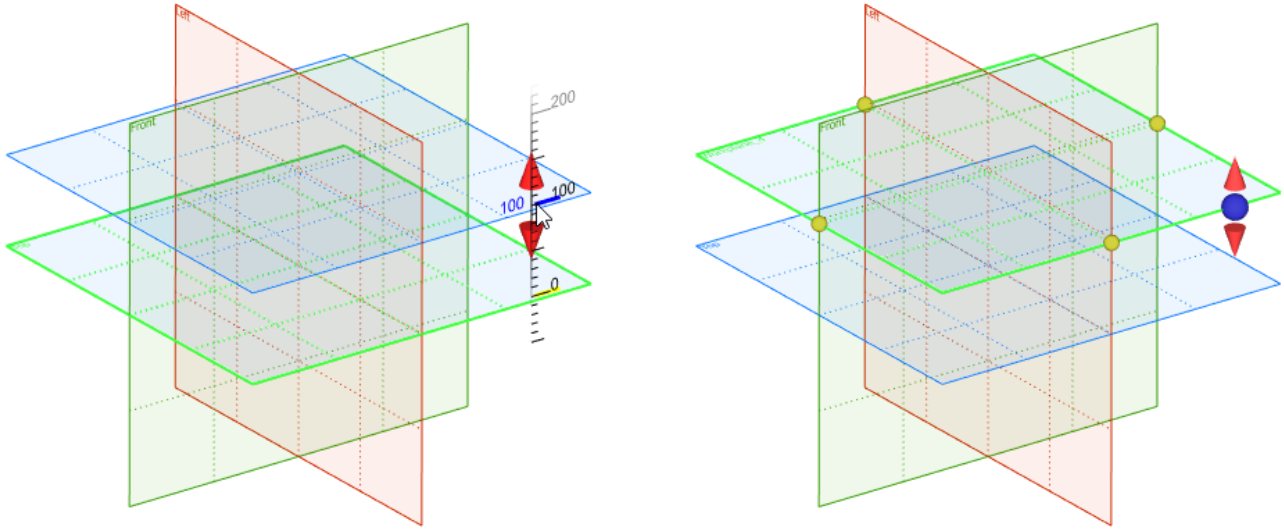
The large manipulator, which appears at that place where the cursor was at the moment of workplane selection, allows us to quickly move the plane and also create new workworkplanes. The given manipulator practically consists of two manipulators: sphere at the center – *manipulator of translation* – serves for the translation of the selected plane parallel to its original location; and the cones – *manipulators of copying* – allow us to create new planes parallel to the selected one.



After the plane translation manipulator is activated, the image of the plane being edited is "duplicated". One copy remains at the initial position, the second copy of the plane starts dynamically moving after the cursor. Between them a dimension line that measures the magnitude of the distance between the planes is drawn. After pressing for the second time (or after releasing the pressed mouse button) the workplane is fixed at the specified location. As in the case of modifying the boundaries of the plane, the exact value of the translation can be specified in the system.



Working with the manipulator of copying is carried out in a similar way. The only difference is that as a result of this operation not the original plane but its copy is moved to the specified location. In this way any number of parallel planes can be created.



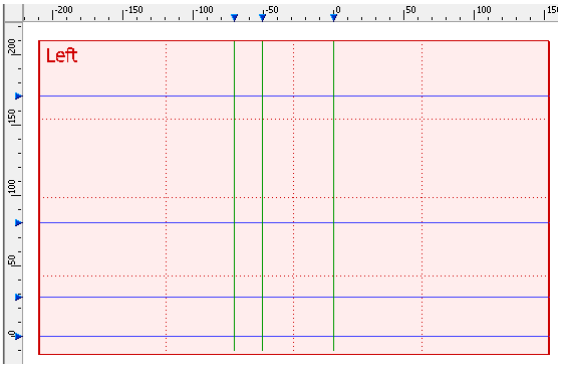
Working with Workplanes with the Help of Rulers of 3D window

When working in the 3D window with the enabled rulers, there are additional capabilities for quick translation and copying of workplanes.

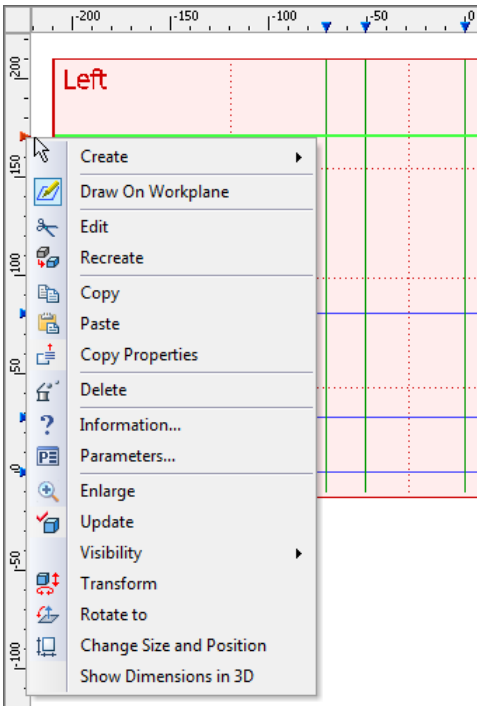
Visibility of rulers in the 3D window can be controlled with the help of the **Customize** > **Windows** > **Ruler** command of the text menu. Also, the identically-named command can be used in the context menu that appears upon pressing and selecting one of the toolbars.

The rulers in the 3D window allow us to quickly estimate the sizes and distances in the 3D scene. Coordinates shown on the rulers constitute the coordinates on the imaginary plane parallel to the plane of the screen.

In case when the workplane is orthogonal to the plane of the current 3D view and occupies in it the vertical or horizontal location, the special marker on the ruler (or) will show the location of such a workplane.



The marker of the workplane is closely linked with the workplane itself. When the cursor is moved to the sign of the marker, the color of the marker is changed to red. Moreover, the workplane itself is highlighted. Pressing on the sign of the marker invokes the context menu for the plane that corresponds to the given marker.



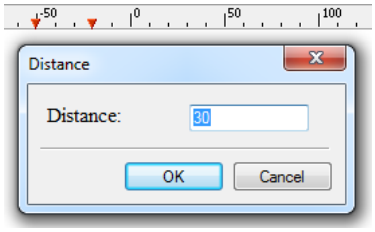
When the marker is selected with the help of , the mode of translation/copying of the workplane is activated. On the screen the popup dynamical toolbar appears that allows us to specify the exact value of the translation of the plane.

The translation of the selected plane is carried out if, after selection of the marker with the help of , the user moves the cursor without releasing the pressed mouse button. To fix the new location of the workplane, it is sufficient to release the mouse button.

Creation of the new workplane – copying – takes place if, after selection of the marker with the help of , the pressed mouse button is released. Pressing for the second time in this case will fix the location of already the new workplane – copy of the original plane.

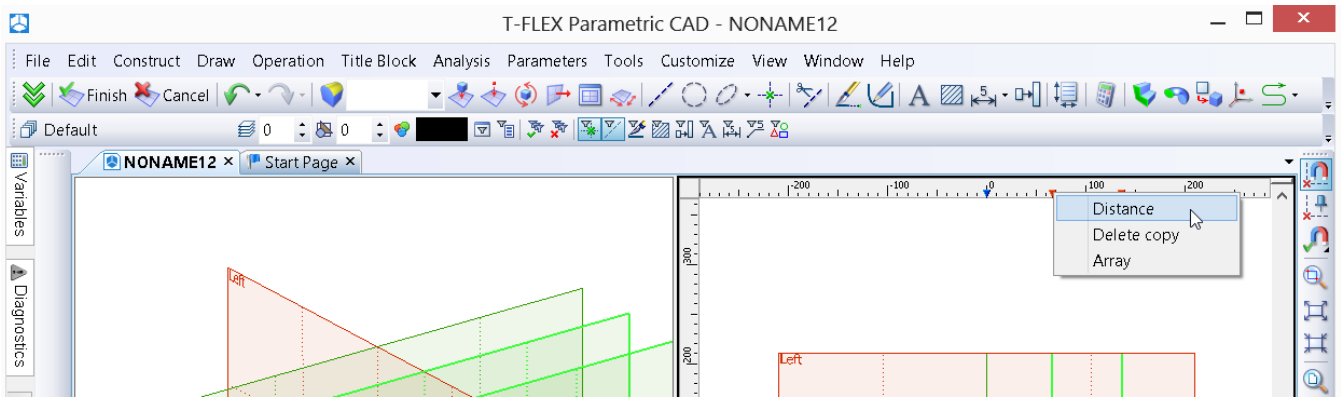
By using the <Shift> and <Ctrl> keys, it is possible to select several markers at once. Translation and copying in the case of multiple selection are carried out similarly to the selection of one marker. In the context menu the additional commands will be available:

- **Distance** (*available only upon selection of two markers*) – this command is used for quick modification of the distance between the workplanes. When invoking the command, the "Distance" window appears in which it is necessary to indicate the required value of the distance. After pressing**[OK]**, the system will displace the corresponding workplanes in such a way that the distance between them becomes equal to the specified value;

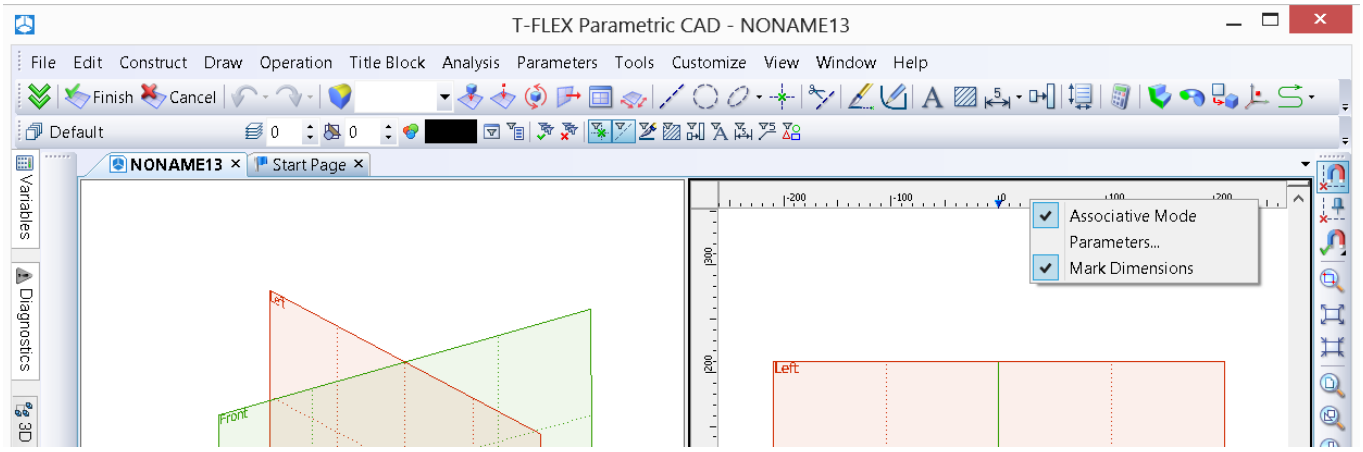


- **Uniformly** (*available only upon selection of three or more markers*) – this command displaces the planes, which are connected with the selected markers, in such a way that the distance between all adjacent planes becomes the same;

- **Array...** – create array of planes (*available only upon selection of two markers of planes*).

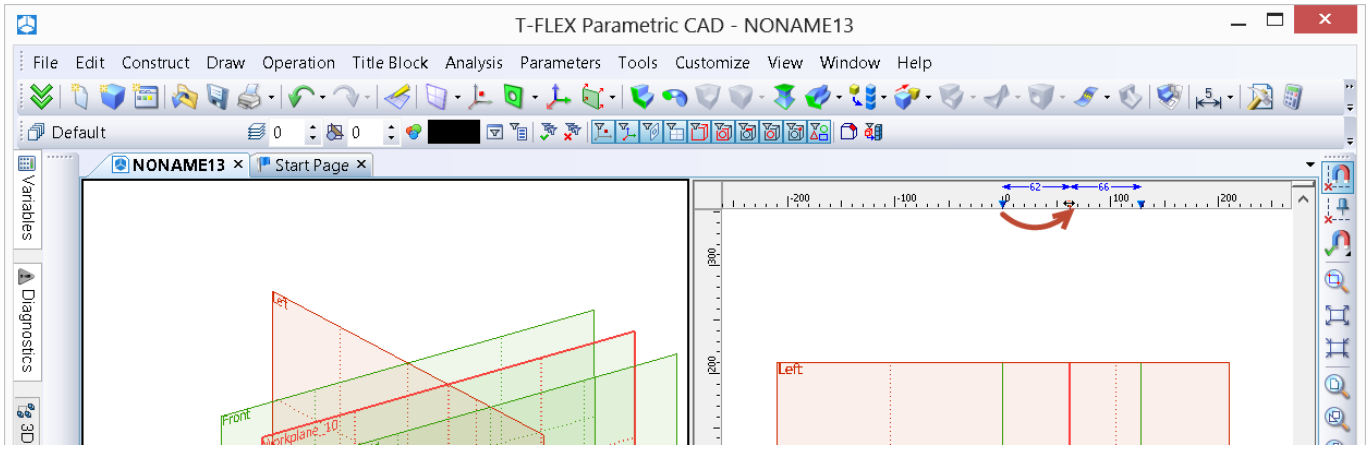


In the context menu of the ruler of the 3D window two additional commands are available – **Associated mode** and **Show dimensions**.



The "Associated mode" flag controls the mode of creation of workplanes with the help of tick marks on the rulers. When this flag is enabled, the new workplane is created with the shift with respect to the original workplane. When the flag is disabled, location of the new plane will be fixed in absolute coordinates without snapping to the original workplane.

The "Show dimensions" mode controls the display of distances between the planes. When this flag is enabled, the distances of the selected workplanes between each other (if several of them were selected) and the distances to the first and last workplanes along the given axis are shown on the ruler as dimensions.

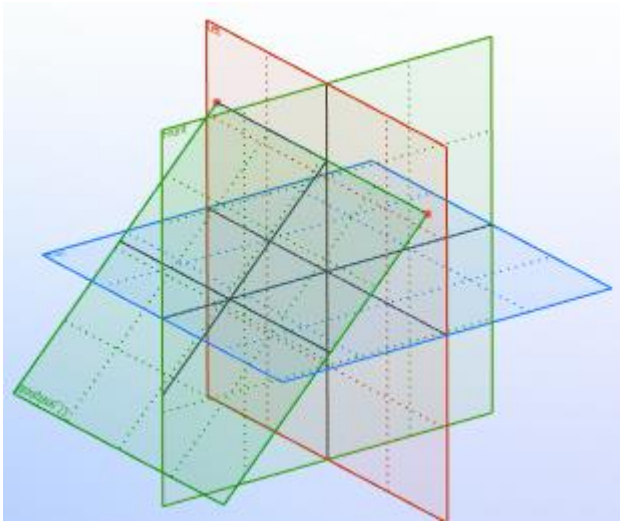


- Grid of Workplanes

In T-FLEX CAD there is one more auxiliary tool for working with the workplanes – *grid of workplanes*.

The grid of workplanes – is a set of lines of intersections of all workplanes of the model. Intersections are shown with the gray lines. Visibility of the grid is controlled with the **Show/hide grid of workplanes** command of the "View" toolbar. By default, the grid is not displayed.

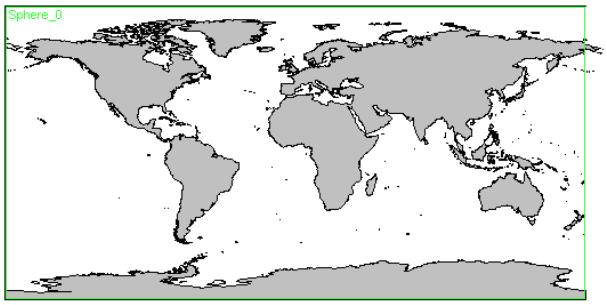
The nodes of the grid of workplanes can be used for snapping, once they are selected as geometric points in different 3D operations. In this case 3D nodes will automatically be created at the nodes of the grid.

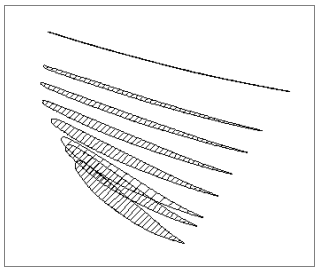


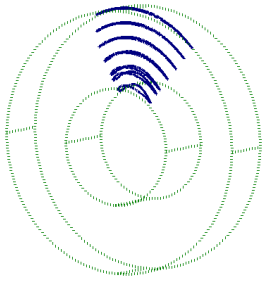
WORKSURFACES

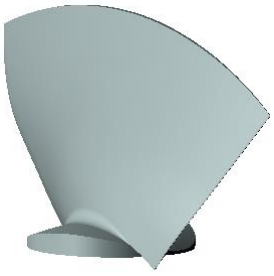
Worksurfaces are auxiliary 3D elements that represent geometrical surfaces of certain type. A cylinder, sphere or torus can be used as such surfaces. The functional purpose of worksurfaces is similar to that of workplanes. Generally, those are intended for carrying 2D objects over to the 3D space. Each worksurface is always related with a parametric region on a page of a 2D drawing. Auxiliary 3D elements, such as 3D nodes, 3D profiles and 3D paths, can be created based on a flat 2D image located on this page. Those elements will be placed on the respective worksurface – a cylinder, sphere or torus.

The main purpose of worksurfaces is creation of auxiliary objects (3D profiles, 3D paths, 3D nodes) for creating parts whose surfaces are curved in two directions: turbine blades, boat propellers, ship hulls, aircraft fuselages.









MAIN CONCEPTS

Worksurfaces are defined parametrically in the special coordinate systems:

- *cylindrical*;
- *spherical*;
- *toroidal (ring-shaped)*.

The parametric coordinates are always counted with respect to an orthogonal (Cartesian) coordinate system. (The meaning and the range of the coordinates depend on the surface type.) The particular Cartesian system can be either the world coordinate system, or a specifically selected local coordinate system (LCS).

The position of a point in the **cylindrical coordinates** is defined by three parameters: the radius of the cylinder, the shift along the cylinder axis (Z-axis) and the angle (or the arc length) between X-axis and the radius-vector of the point projection onto the X-Y plane.

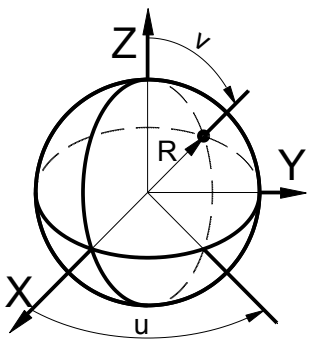
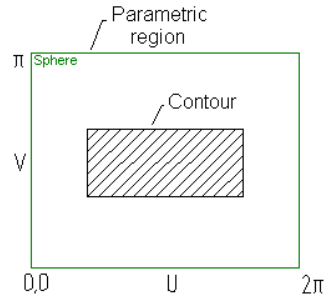
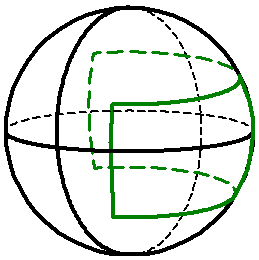
The position of a point in the **spherical coordinates** is defined by three parameters: the radius of the sphere, the angle between X-axis and the radius-vector of the point projection onto the X-Y plane, and the angle between the point radius vector and Z-axis.

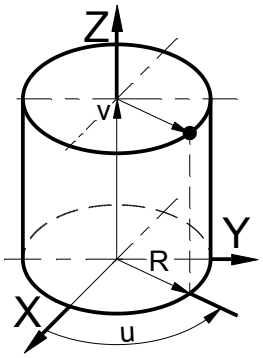
The position of a point in the **toroidal coordinates** is defined by four parameters: the two defining radii of the torus, the angle between X-axis and the point projection on the X-Y plane and the angle between the point radius vector and Z-axis direction.

The parameters defining the radius of the cylinder, that of the sphere or the two torus radii are called *fixed*, as those are the same for all points belonging to one surface. This is a constant characteristic of the given surface. The cylinder and the sphere have one fixed parameter – the radius, while the torus – two (the two torus radii).

Therefore, the position of a point belonging to a surface of any type is actually defined in a special coordinate system by two orthogonal parameters – the coordinates (*U* and *V*). Due to this fact, a mapping can be established between any of the special coordinate systems and a rectangular region on a plane (*a parametric 2D region*). This 2D region represents an "unfolding" of the surface onto the plane (according to the rules of mapping the UV-coordinates into the Cartesian coordinates for the selected type of surface), playing the same role as the image of the workplane on a page of a 2D drawing. Besides, the 2D region defines the range of the UV-coordinates.

Introducing a parametric 2D region allows working with a surface as if with a common workplane.

Graphic Illustration	**Parametric 2D region**	**Profile in 3D**
Spherical coordinates		
	 U – angle from X-axis (0-2π) V – angle from Z-axis (0-π) R – radius of the sphere	

<table>
<tr><td colspan="3">Cylindrical coordinates</td></tr>
<tr>
<td> 
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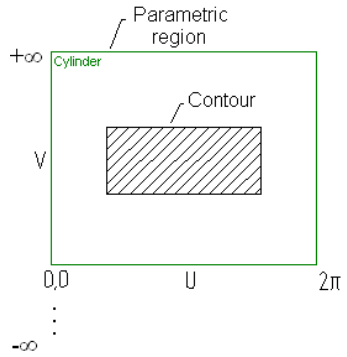
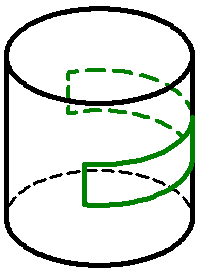
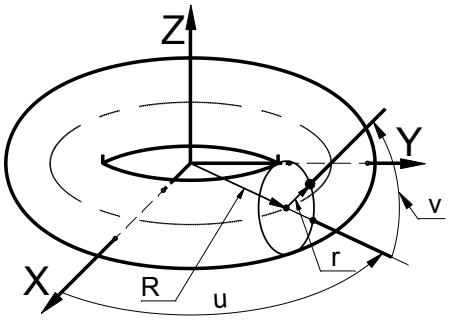
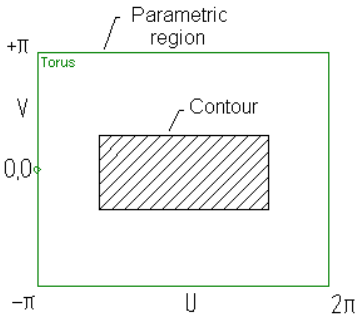
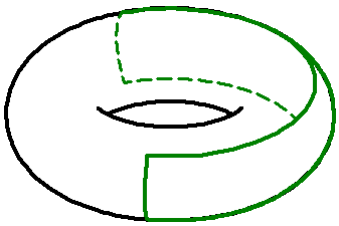
<td> 

U – angle from X-axis (0-2π) or arc length counted from X-axis (0-2πR)
V – Z-coordinate (-∞,+∞)
R – radius of the cylinder</td>
<td>  </td>
</tr>
<tr><td colspan="3">Toroidal coordinates</td></tr>
<tr>
<td> 
</td>
<td> 

U – angle from X-axis (0-2π)
V – angle from the vector R in the plane ZOR (-π,+π)
R – torus major radius,
r – torus minor radius</td>
<td>  </td>
</tr>
</table>

Data Required for Creating a Worksurface

When creating a worksurface of any type, you need to define:

The parametric 2D region (a rectangular region in the 2D drawing) – the unfolding onto a plane of the surface being created;

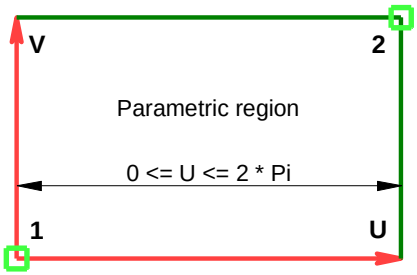
The fixed parameter value for this surface (the radius of the sphere or cylinder, or the two torus radii);

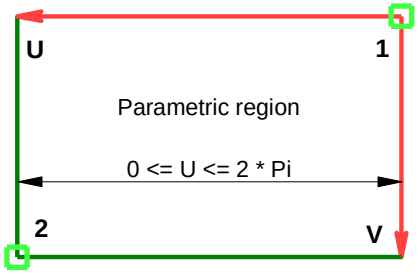
The original Cartesian coordinate system (the world coordinate system or an arbitrary LCS), with respect to which the parametric surface coordinates are defined.

Parametric Region

The parametric 2D region of a worksurface is defined on one of 2D pages of the current document. The size and position of the parametric region in the drawing units is defined either by the system automatically or by the user (by specifying two 2D nodes as the diagonal corners of the region). The size of this region in the parametric coordinates is always constant, depending on the type of the surface being created (entered as the "ranges" in the table above). The region extents along the *X*-axis are mapped onto the allowed range of the parameter *U*, along the *Y*-axis – onto *V* (the reverse configuration is also supported).

In the cases when the position and size of the parametric region are defined by the user (by selecting two 2D nodes), the direction of the worksurface axes is determined by the nodes selection order and position. The diagrams below show the configurations of the UV-axes directions with respect to the order of node selection.

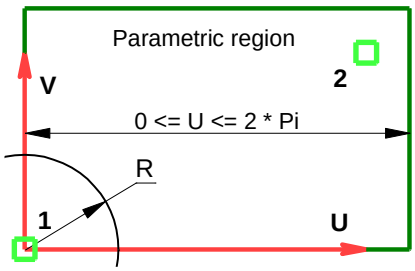




When the parametric 2D region of the worksurface is defined automatically, the axes are always directed as shown on the right-hand-side figure.

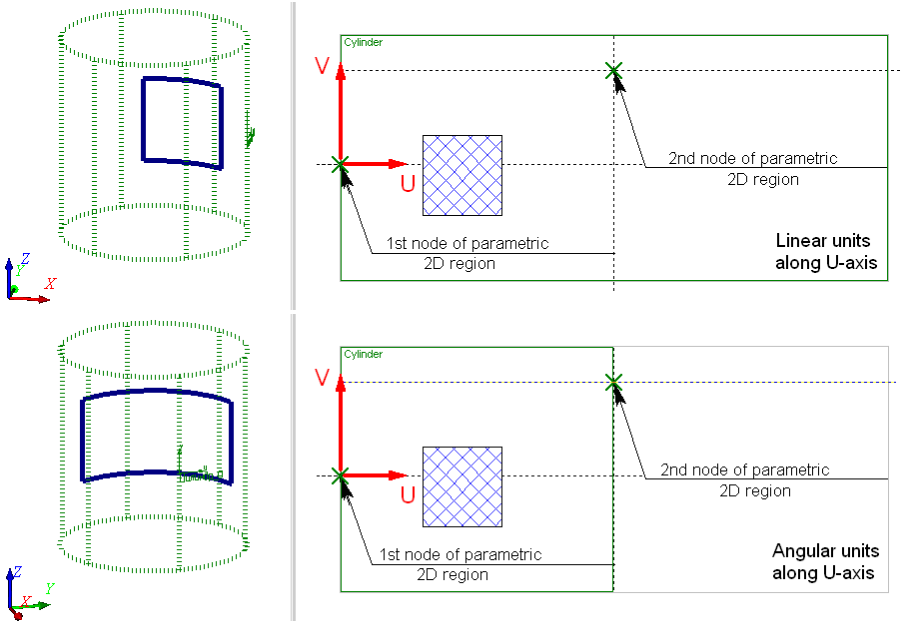
In spherical and toroidal coordinate systems, both parameter axes represent angular coordinate values. The exception is the cylindrical coordinate system. Since a cylinder can be "unwrapped" onto a plane without distortions, linear units can be used as cylindrical coordinates. The V-axis units are always linear, while the units of the U-axis can be either angular (the angle between the radius vector and X-axis) or linear (U being the linear distance as measured along the cylinder unfolding).

When using linear units along the U-axis of the cylindrical coordinates, the second 2D node defines only the axes orientation. The size of the parametric region along the U-axis in this case is equal to $2\pi R$, where R is the cylinder radius. The diagram below shows the way of defining the parametric region for a cylindrical surface, when linear units are used in both axes.

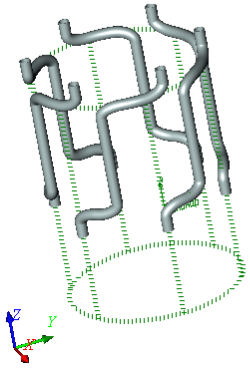


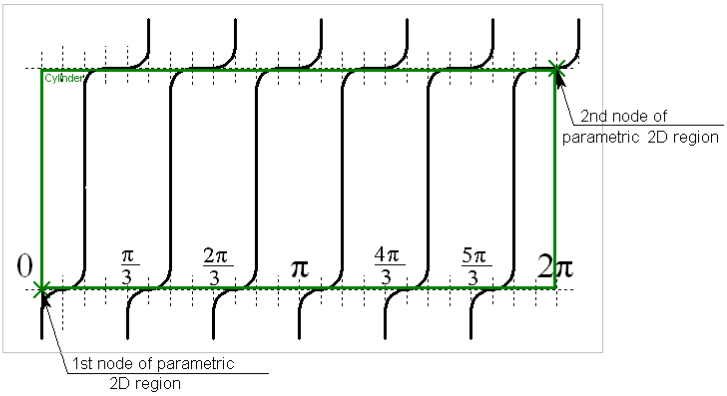
When using angular units along the U-axis in cylindrical coordinates, both nodes are used. The linear distance between the two nodes along the U-axis is considered equal to the angular value of 2π. The linear coordinates in this direction will be converted into the angular ones based on this constant.

The choice of units along the U-axis – linear or angular – depends on a particular task. If the 2D construction to be carried over on a cylindrical worksurface is described in linear units, then it may be more convenient to use linear units along the both axes of the cylindrical coordinates. In this case, the 2D construction will be carried over to the 3D space preserving all relations between the construction elements. Use of angular units in such a situation will cause distortions in the transition from 2D to 3D. As in illustration, consider a simple example of creating a 3D profile on a cylindrical worksurface using a 2D hatch of square shape. One can easily see that the profile shape is far from square when using the angular units along the U-axis. Meanwhile, when using the linear units, the 3D profile matches the shape of the hatch.



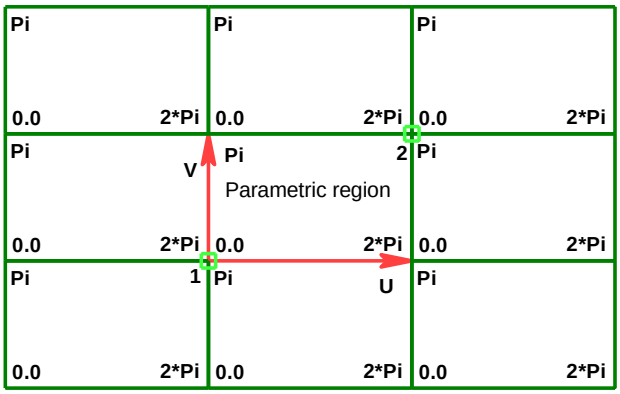
Use of the angular units along the U-axis of the cylindrical coordinate system makes sense only in the cases when the original 2D constructions are described using angular values. For instance, in the following example, the original 2D data for creating a model of a heat exchanger are defined in the angular values on the surface of a cylinder. Use of angular units helps easily getting the desired result (provided that the boundaries of the parametric region of the worksurface were defined correctly).



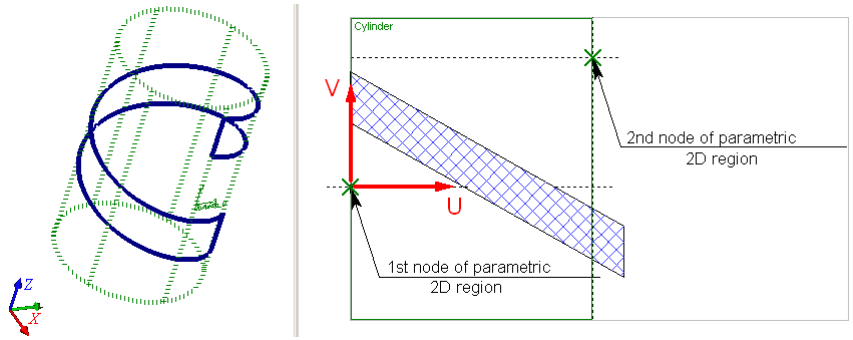


To be comfortable when working with angular units, use the following rule. The distance between the 2D nodes along the U-axis must be set so that the total width of the parametric region along the U-axis (that is, the respective distance between the nodes defining its boundary) is a multiple of 360 (in degrees), or 2π (when using radians).

When working with worksurfaces of any type, keep in mind the following. Any of the worksurfaces are periodic in one or both UV-parameters. Therefore, a parametric surface has cycles in the direction in which it is periodic. The diagram below shows periodicity of a parametric 2D region for spherical and toroidal surfaces.



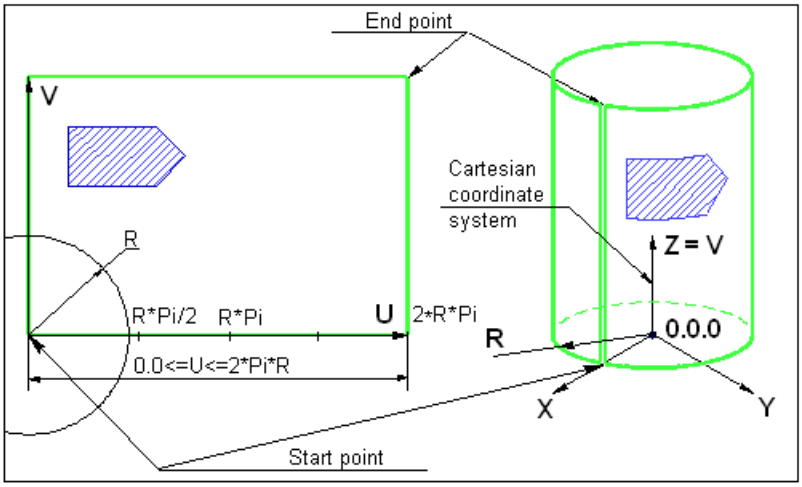
In the case of a cylindrical surface, the parametric 2D region cycles only along the U-axis.



The Fixed Parameter and the Cartesian Coordinate System of the Worksurface

A fixed parameter value (two parameters – in the torus case) is defined with a numerical value. When creating a worksurface by an existing geometrical surface, the fixed parameter value is determined by the specified offset along the radius of the source geometrical surface.

The position of the worksurface in space is defined by selecting a Cartesian coordinate system. The parametric coordinates will be counted with respect to the system. An arbitrary LCS can be selected as such a system. If no local coordinate system is specified, the world coordinates are used.



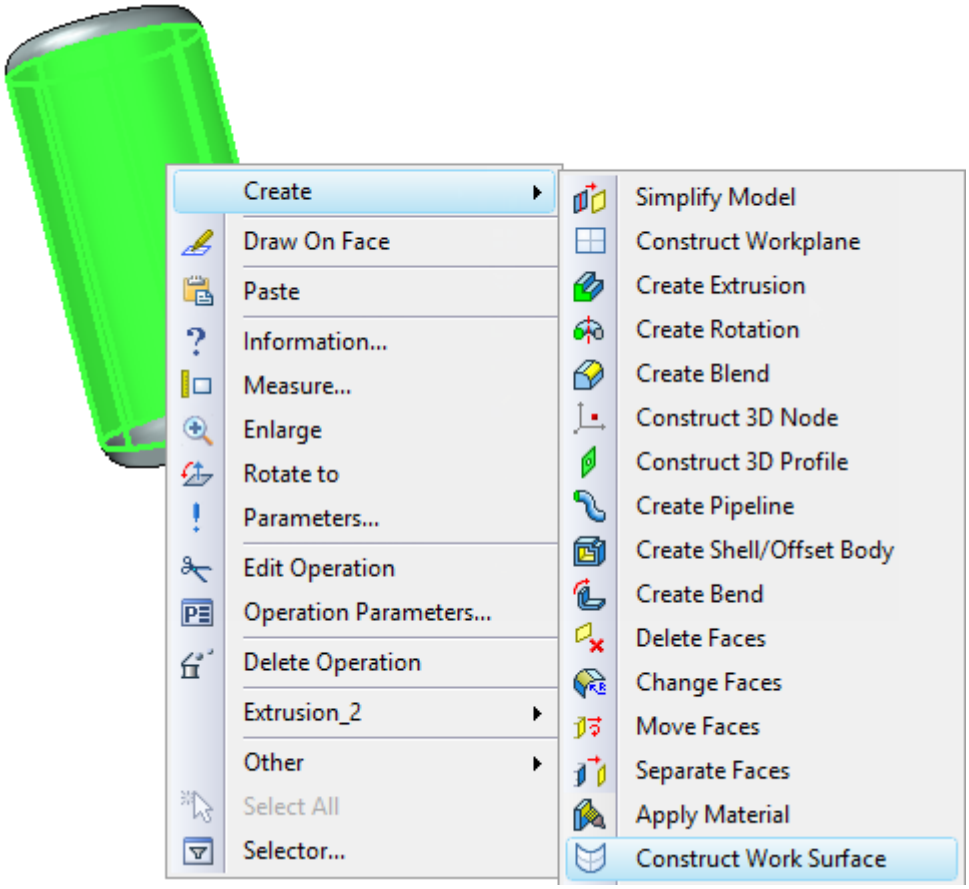
When constructing a worksurface by an existing geometrical surface, the coordinate system of the source geometrical surface is used as the Cartesian coordinate system.

Introduction to Worksurfaces

To understand how worksurfaces are created and used, let's review a brief example.

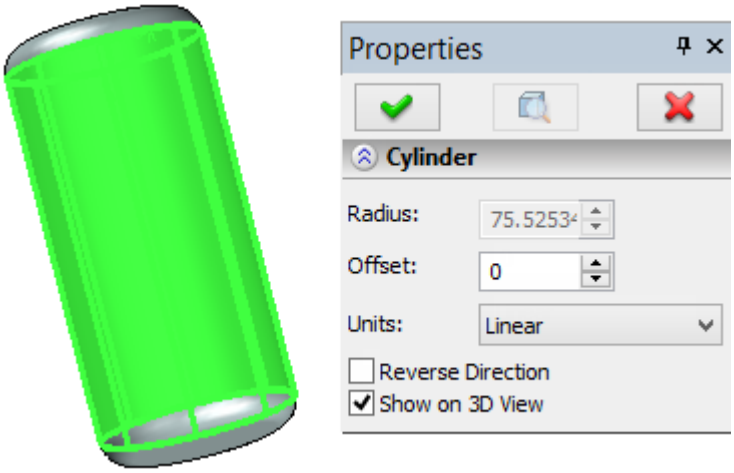
Suppose, we need to wrap a text on a cylindrical body face. To do this, let's create a cylindrical worksurface coinciding with the surface of that cylindrical body face. Then it is possible to create a 2D element-text on that surface, and use it to create a 3D profile and a text-like looking body.

To create a worksurface, select a cylindrical face of the source body in the 3D window and call its context menu. In the context menu, select the command **Create/Construct Worksurface**.



As a result, the command will be launched, **3SU: Construct Worksurface**.

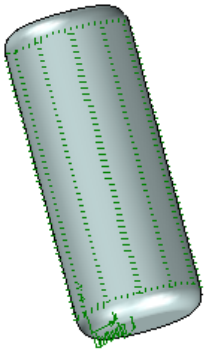
Upon entering the command, the cylindrical face will be automatically selected. To create the worksurface, just click . As a result, a cylindrical worksurface will be created, which coincides in the radius and position in the space with the source face, or, rather, with its geometrical surface.



The command's properties window displays the parameters of the surface being created. Those do not need to be changed.

The "Offset" parameter defines the difference between the source face radius and that of the surface being created. The required "0" value is already set by default.

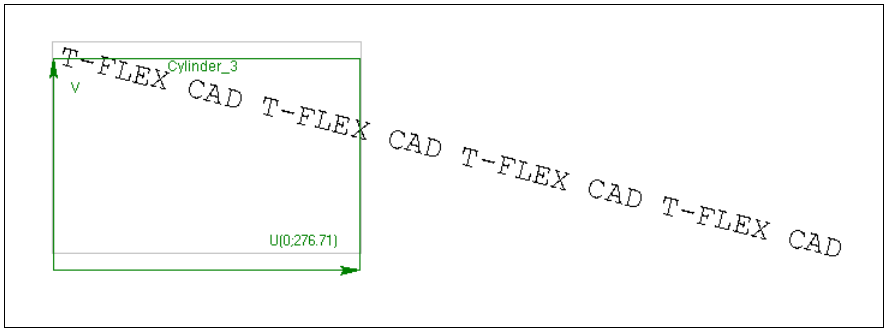
The "Units" parameter defines, what units will be applied along the U-axis. The default setting "Linear" should also be left unchanged.

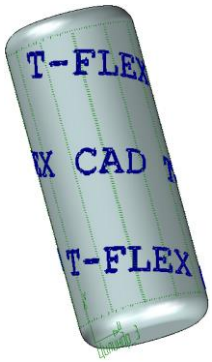


By now the worksurface is created. Now we need to create the 3D profile-text on it. To do that, select the created surface and call its context menu by right-clicking. In the menu select the command **Draw On Workplane**. A separate window will open with the 2D drawing page, corresponding to the selected worksurface. A box will be marking the parametric 2D region of a worksurface on the page.

Let's create the required text on the 2D page. Finish drawing on the active worksurface by clicking Finish on the control panel of the active workplane (actually, a surface in this case).

Upon leaving the mode of drawing on the active worksurface, a 3D profile will be automatically created based on the 2D text.





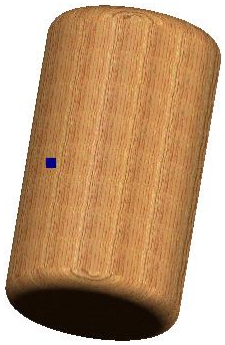
What is left is to extrude the 3D profile to the required height and perform a Boolean addition of the resulting text and the source cylinder.

Let's see another example. In it, we need to apply a text on a cylindrical-shape plate while maintaining a specified offset distance from the plate borders.

In this example, two worksurfaces will be used. We will construct the first one by the cylindrical face of the source body. On it, let's create the 3D profile of the plate contour. We will construct the second worksurface by the outer face of the plate. The 3D profile-text will be created on the latter worksurface.

The difference of this example from the previous one is that in this case we need to know how precisely the worksurfaces and their source faces are positioned relative to each other. Otherwise, it will be impossible to precisely position the plate on the cylindrical body, and the text – on the plate.

To precisely position the worksurface, one can use an additional 3D point. The surface coordinate origin will be placed at that point.

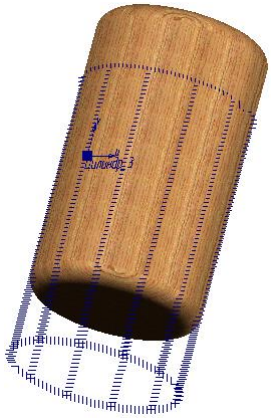


Let's create a 3D node on the face of the source body where the lower-left corner of the plate shall be positioned. Next, as in the previous example, we need to select a cylindrical face of the source body, call its context menu and launch the command **Create/Construct Worksurface**.

We again will leave unchanged the default parameters of the worksurface being created. But, before confirming the surface creation, we will enable the automenu option:

	<1>	Select starting point

Using this option, we will select the 3D node created earlier. The surface coordinate origin will be positioned where the node projects (maps) on the surface.



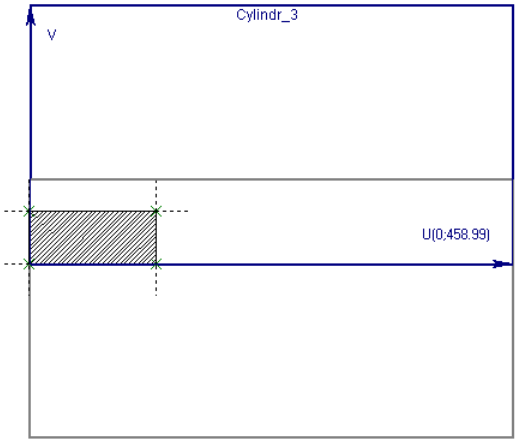
After selecting the 3D node, the worksurface creation can be finilized by clicking . After that we will select the worksurface and call from its context menu the command **Draw On Workplane**. As in the previous example, after calling the command, a separate window will open with the 2D drawing page corresponding to the selected worksurface.

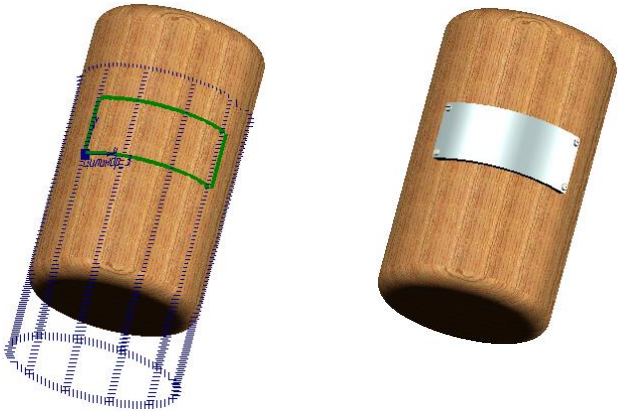
A box on the page will be marking the parametric 2D region of the worksurface. The worksurface coordinate origin (the lower-left corner of the box) corresponds to the 3D node that defines the lower-left corner position of the would-be plate.

By accepting the lower-left corner of the box as the base point, let's create the necessary 2D constructions to create the plate contour. To create the 3D profile, we need to apply graphic lines and/or hatch. Finish working in the mode of drawing on the active worksurface by clicking

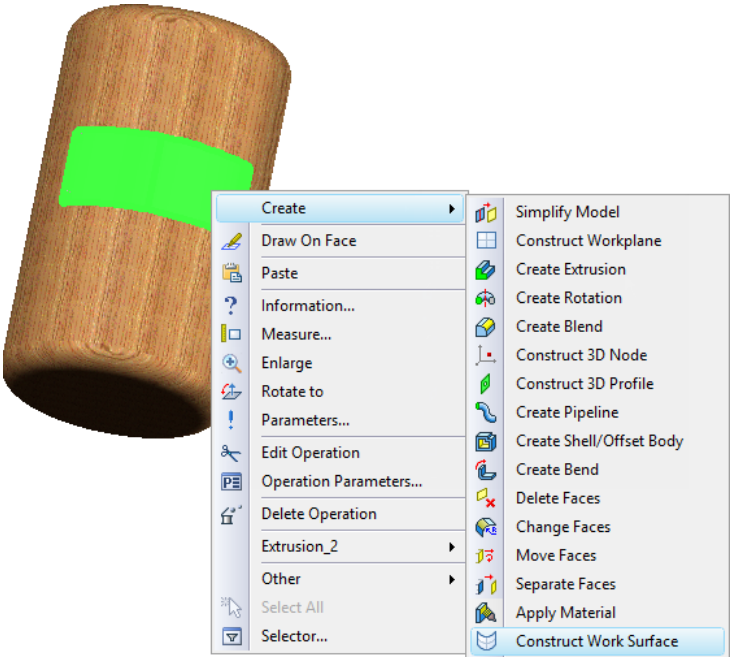
on the

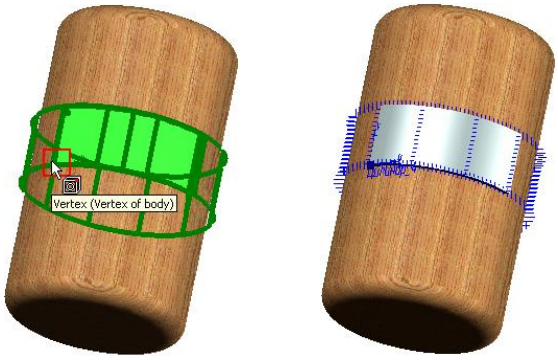
control panel of the active workplane. Upon exiting the mode of drawing on the worksurface, the plate's 3D profile will be automatically created. The plate body is created by extruding the resulting 3D profile.



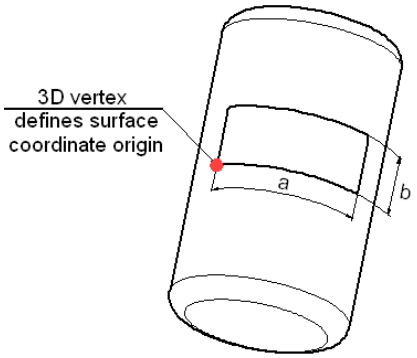


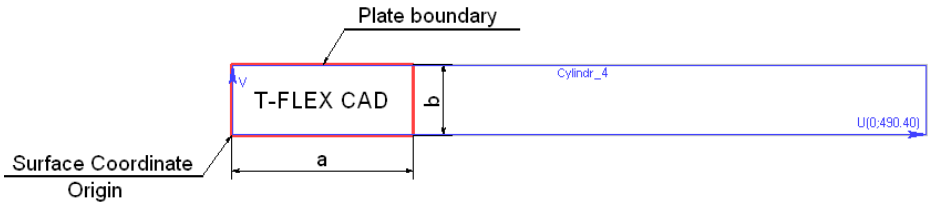
To create text on the plate, we will construct a second worksurface. The construction will be similar to the previous cases. To define the surface coordinate origin, select one of the vertices (the lower-left one) of the plate's upper face.





After creating the second worksurface, we will go into the mode of drawing on it (the command **Draw On Face** in the surface's context menu). On the worksurface page, mark the plate boundaries. After that, apply 2D text at the desired position on the plate.





Next, the 3D profile on the worksurface is created based on the 2D text. The profile is extruded, and a Boolean addition with the plate body is performed.

In the described example, we relied on a 3D point selection as the worksurface origin to precisely position the 3D profile contour on the worksurface. The same goal could be achieved using a different method.

Working in the mode of drawing on the active worksurface (the command **Draw On Workplane** in the context menu) is mostly analogous to working in the respective mode on a workplane. The only difference is that one can draw on a workplane both in 2D window and directly in the 3D window. For a worksurface, only the mode of drawing in the 2D window is available. Besides that, all other functions of this mode are supported for a worksurface, including the capability of projecting (mapping) a 3D point on the active plane/surface.

In our example, one could have skipped selecting additional 3D nodes as the origin for worksurfaces, but rather have them mapped in the drawing-on-surface mode. After that, it is possible to create all underlying construction for the 3D profile contours based on the created 2D nodes-the projections.

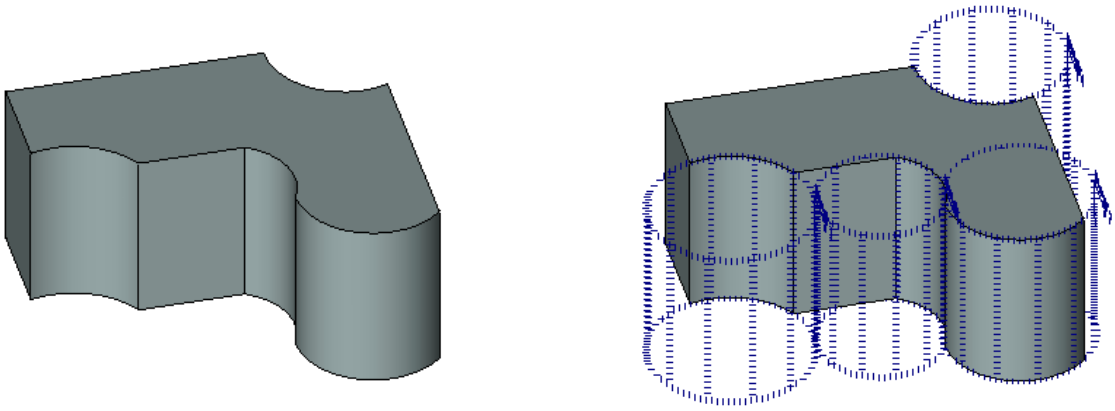
Methods of Creating Worksurfaces

There are three methods to create a worksurface:

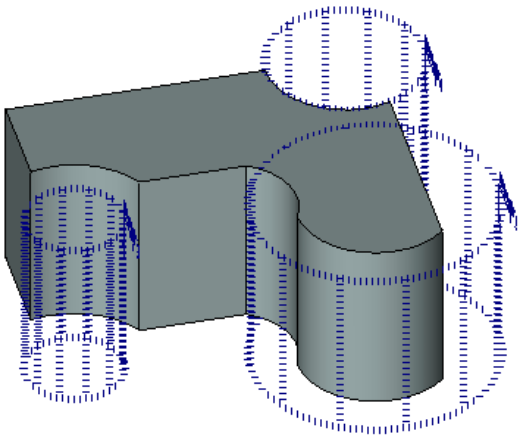
- Creating a worksurface by an existing geometrical surface of the respective type (a surface underlying a face, 3D profile, another worksurface). This method was used in the aboveDescribed examples.
- Creating an arbitrary worksurface (that is, without being based on existing geometrical surfaces in the 3D model) in the 3D window.
- Creating an arbitrary worksurface based on 2D constructions (in the 2D window).

Worksurface Based on Existing Geometrical Surface

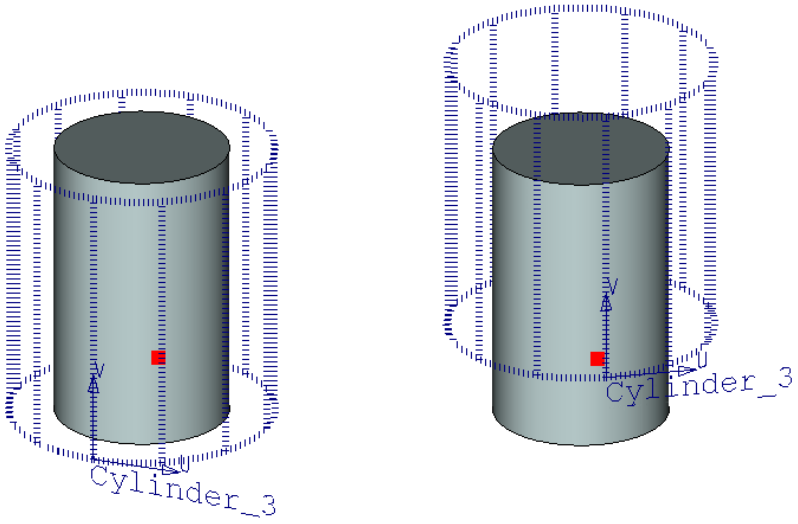
Worksurface creation by an existing geometrical surface is performed in the 3D window. To define the source data, select a 3D object related to the intended geometrical surface. As a result, a worksurface is created that coincides with the source geometrical surface.



Additionally, you can specify an offset amount (two offsets for a torus), which defines the difference between the source and target surface radii. In this case, a worksurface will be created as an equidistant (offset) to the source geometrical surface.



The source geometrical surface coordinate system is used as the Cartesian coordinate system for the created worksurface. The coordinate system position for the worksurface being created can be adjusted by selecting an additional 3D point. The point will determine the worksurface coordinate system origin (that is, the X-axis direction of the Cartesian coordinate system). Two 3D points are defined for a spherical worksurface – first determining the X-axis direction, and the second adjusting the Z direction (and, therefore, the Y-axis).



Without selecting an additional 3D point

With an additional 3D point selected (defined with a 3D node on a face)

A separate 2D page of the "Workplane" type is automatically created for a constructed worksurface. The page's format frame size is determined automatically by the size of the 3D object that was used to define the source geometrical surface. The parametric 2D region of the worksurface is also created automatically. Its size is equal to the size of the page format frame.

The created worksurface is by default displayed in the 3D window.

Arbitrary Worksurface

An arbitrary worksurface is also created in the 3D window. To create a worksurface by this method, you need to specify the type and the radius (two radii for a torus) of the surface being created.

Additionally, you can specify an LCS, relative to which the parametric coordinates of the worksurface will be counted.

As in the previous method, a separate 2D page of the "Workplane" type is automatically created for such worksurface. The dimensions of the page format frame and the parametric 2D region worksurface are automatically determined by the system.
Worksurfaces of this type are also displayed in the 3D window by default.

Worksurface Based on 2D Constructions (in 2D Window)

Creation of a worksurface of this type is done in the 2D window, on an already existing drawing page of any type. To create the surface, you need to define the boundaries (two diagonal corners) of its parametric 2D region. The region boundaries are defined using two 2D nodes. The position and the order of selecting the 2D nodes determine the direction of parametric axes of the worksurface. Besides that, the radius is to be defined (two radii for a torus) of the surface being created.

Additionally, you can specify an LCS, relative to which the parametric coordinates of the worksurface will be counted.

A worksurface of this type can be created on a 2D page that is already related with other worksurfaces or planes. In this case, the boundaries of the parametric 2D region are used by the system to automatically match the 2D elements found on this page, with the respective worksurface/plane (when creating 3D elements – 3D profiles, 3D paths).

If several worksurfaces/planes are related to a 2D drawing page, those cannot be activated.

RULES OF CREATING WORKSURFACES

To create a worksurface, use the command **3SU: Construct Worksurface**:

Icon	Ribbon
	3D Model → Construct → Worksurface
Keyboard	Textual Menu
<3SU>	Construct > Worksurface

Upon calling the command, the following options are available in the automenu:

	<S>	Select Face
	<1>	Spherical coordinate system
	<2>	Cylindrical coordinate system
	<3>	Ring-shaped coordinate system

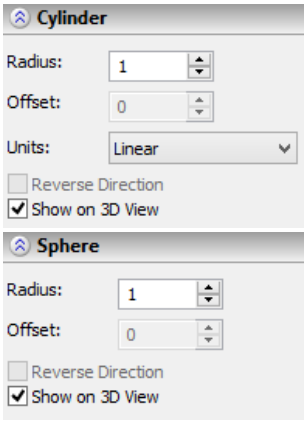
The first option – – serves to construct a worksurface by an existing geometrical surface.

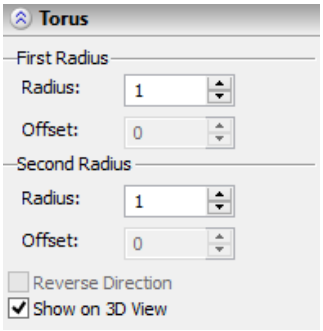
The other three options – , and – are used to construct either an arbitrary worksurface in the 3D window or a worksurface based on 2D constructions (in the 2D window). How a worksurface will be created depends on where the **3SU: Construct Worksurface** command call originated – from the 3D window or from the 2D window.

Creating Worksurface by an Existing Geometrical Surface

To create a worksurface by an existing geometrical surface, use the option . It is turned On by default. With the option enabled, select a 3D object in the 3D window or in the model tree, suitable to define a surface of the cylindrical, spherical or toroidal type. This could be a face of the respective type, a 3D profile, or an existing worksurface.

After selecting the source 3D object, the system will automatically determine what-type worksurface will be created. The command's properties window will have the dialog to define parameters of the given worksurface.





Depending on the type of the created worksurface, you can define the values of one or two offsets in the properties window. The offsets determine the difference between the radius of the source geometrical surface and the radius of the worksurface being created.

Original direction of worksurface coordinate axes corresponds to direction of the parent surface. **Reverse Direction** flag in the Properties dialog box will reverse direction of both axes.

Additionally, the option is available in the automenu to select a 3D point defining the worksurface coordinate origin:

	<1>	Select starting point

When creating a spherical worksurface, yet another option will be available in the automenu – to define the second 3D point to adjust the Z- and Y-axis direction of the Cartesian coordinate system::

	<2>	Select Pole point

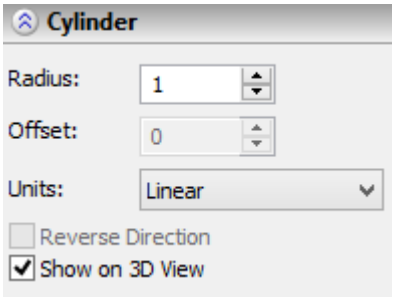
A 3D point (3D points) selection can be canceled with the option:

	<D>	Cancel selection of points setting surface orientation

Creating an Arbitrary Worksurface (in the 3D Window)

To create a worksurface of this type, you must call the command **3SU: Construct Worksurface** from the 3D window. After entering the command, you need to specify the type of the surface being created, by selecting the desired automenu option (, or).

After selecting the surface type, the command will go into the mode of creating the worksurface of the selected type. Define the radius of the surface being created in the command's properties window (two radii for a torus). The surface creation can be completed right after defining the radius (radii), by clicking (in the automenu or in the command's properties window).



To define an LCS, relative to which the parametric coordinates of the worksurface will be counted, use the option:

	<L>	Select target LCS

The option to cancel a local coordinate system selection is:

	<D>	Eliminate target LCS

The system will automatically define the parametric 2D region of the worksurface being created (on a separate drawing page of the "Workplane" type). If necessary, you can manually define the boundaries of the parametric 2D region of the worksurface being created, using the nodes on the same or another 2D page. To do that, use the same options as when creating a worksurface based on 2D constructions.

Creating an Arbitrary Worksurface Based on 2D Constructions (in the 2D Window)

To create a worksurface of this type, you need to call the **3SU: Construct Worksurface** command from the 2D window. Upon entering the command, just like in the previous case, you need to specify the type of the surface being created (the option , or).

After that, you need to define the parametric 2D region of the worksurface being created. If there are no worksurfaces or planes on the current 2D page, then the system will select the region boundaries automatically according to the page format frame size. If, on the other hand, the current drawing page is already related with worksurfaces or planes, or if you are not satisfied with the system choice, define the boundaries of the parametric 2D region by selecting two 2D nodes with the options:

	<F>	Select Node as origin of parametric region
	<S>	Select Node as second border of parametric region

The start and end points should not coincide. Otherwise, it will be impossible to determine the orientation of the U and V axes.

By default, the range along the **X**-axis of the parametric region is mapped onto the range of **U** parameter, the **Y**-axis – onto **V**. This can be changed by the option:

	<3>	Shift coordinate axes

The fixed parameter value of the worksurface being created (the surface radius) is defined in the command's properties window.

If the worksurface is to be created relative to a local coordinate system, then you need to use the option:

	<L>	Select target LCS

To cancel a local coordinate system selection, use the option:

	<D>	Eliminate target LCS

The visible boundaries of a worksurface on the drawing are by default aligned with the defined boundaries of the parametric 2D region. The visible boundaries can be modified with the options:

	<B>	Define Workplane borders
	<Tab>	Switch Workplane borders
	<K>	Eliminate borders

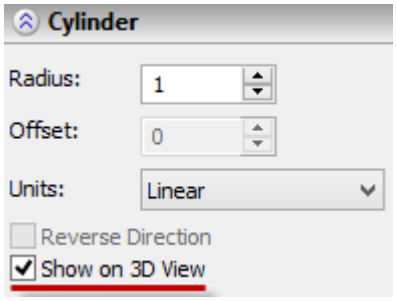
These options work in the same way as those provided for common workplanes.

By default, a created worksurface of this type will not be displayed in the 3D scene. To make it appear in the 3D window, enable the **Show on 3D View** flag in the command's properties window.

You can finalize worksurface creation by clicking (in the automenu or in the command's properties window).

WORKSURFACE REPRESENTATION IN 3D VIEW

It is possible to control the visibility of the working surface by the special flag **Show in 3D View** available in parameters dialog box when creating/editing a worksurface. By default this flag is turned on, that is to display the worksurface in 3D view. When flag is turned off the worksurface will be not shown in 3D view.



3D Nodes

A 3D node is a T-FLEX CAD element, representing a point in the three-dimensional space. Main use of 3D nodes is as a reference for snapping other three-dimensional objects: additional workplanes, profiles, local coordinate systems, etc. 3D nodes help defining objects necessary for performing operations, such as an extrusion vector, an axis of revolution.

There are several types of 3D nodes. These can be divided into three main groups: nodes based on existing 3D elements, nodes defined in absolute coordinates, and nodes based on 2D nodes.

The display size of 3D nodes on the 3D scene is defined in the command **ST: Set Document Parameters** on the tab **3D** > **View**.

The Ways of Node Creation

To create a 3D node, use the command **3N: Construct 3D Node**:

Icon	Ribbon
	Draw → Construct → 3D Node
Keyboard	Textual Menu
<3N>	Construct > 3D Node

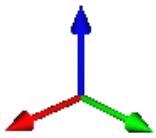
This command helps creating the following types of 3D nodes:

1) Based on existing 3D elements:
   - 3D nodes;
   - Vertices;
   - Local coordinate systems;
   - 3D profiles/paths;
   - Edges;
   - Workplanes;
   - Faces;
   - Operations;
   - At the center of elliptical edge;
   - At the center of a sphere or torus;
   - At an intersection of elements;
   - By shortest distance between elements;
   - By faces and offsets;
   - At the center of gravity;
   - At an intersection of three planes;

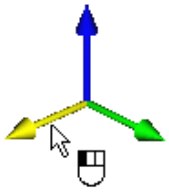
2) In absolute coordinates;
3) By two projections.

Using Dragger for Creating 3D Nodes

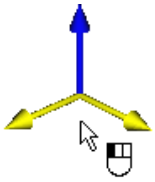
The draggers can be used for creating 3D nodes in the following situations: in absolute coordinates, relative to another 3D node, vertex or a local coordinate system. Draggers can also be used for defining a node relative to the center of an elliptical edge or a surface, which is a sphere or torus. A dragger appears as a coordinate system. It allows moving a 3D node along one axis or simultaneously in two dimensions.



A dragger for defining 3D node position



Selecting a dragger axis for moving the node along one coordinate axis



Selecting two dragger axes for moving the node in two dimensions

To move a 3D node along one axis, simply move the cursor to one of the node dragger axes. The respective axis will be highlighted in **yellow color** (by default). Upon clicking, the selected axis becomes **pink** (by default), and the dragger starts moving along the selected axis following the cursor. The subsequent clicking fixes the dragger (and the respective 3D node) in the position of the click.

Moving the node simultaneously in two dimensions is done similarly. To do this, simply move the cursor to the dragger and place it between two axes. The axes will highlight. Upon clicking the dragger starts moving after the cursor in the plane defined by two axes. The subsequent fixes it in the new position.

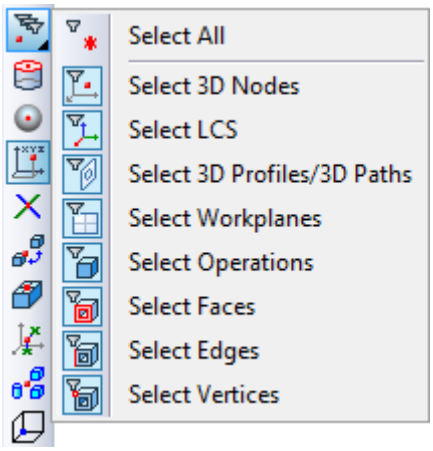
Moving the dragger of the 3D node being created causes update of the node coordinates (offsets) in the property window. Vice versa, modifying the coordinates of a 3D node in the dialog box moves the node dragger.

Main Node Creation Techniques on a 3D Element or Relative to a 3D Element

The following automenu option provides for most common ways of creating 3D nodes based on existing 3D elements:

	<S>	Create 3D Node on element or relative to element

The option becomes active right after entering the command. To create a 3D node based on a 3D element, simply select the desired 3D element in the 3D scene or model tree. Such a 3D element can be an existing 3D node, a local coordinate system, 3D profile or path, a workplane, operation, face, edge or body vertex. When selecting 3D elements, pay attention to the state of element selection filters in the pull-down list under the option or in the system toolbar.



Depending on the selected element, the appearance of the automenu or the property dialog box will adjust accordingly.

Creating Node Based on Existing 3D Node, 3D Vertex or Local Coordinate System

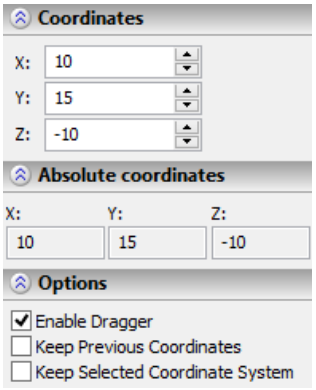
Selecting an existing 3D node, 3D vertex or a local coordinate system brings up the node coordinate definition dialog box in the property window, and a coordinate system-shaped dragger in the 3D scene. The dragger coordinate system origin coincides with the selected 3D element (the origin in the case of a local coordinate system). The dragger axis directions coincide with: the world coordinates axis directions - when selecting a 3D node or vertex, and with the axes of the selected coordinate system – when selecting a local coordinate system.

The position of the node being created is defined in local coordinates either directly on the 3D scene, or using a dragger, or else by entering exact numerical values in the section **Coordinates** of the property window. The absolute node coordinates (in the world coordinate system) are displayed in the section **Absolute coordinates** of the same dialog box.

Too quickly reset coordinates of a node to 0, use the following additional automenu option:

	<Z>	Move Node to coordinate system origin

Additional options, collected in the section **Options**, help tuning the command for the following modes:



Enable dragger. Toggles the display of the dragger of a 3D node in the 3D scene. When unchecked, the dragger is not displayed.

Keep previous coordinates. When set, the relative coordinates, defined for the node being created, are remember and will be preset by default in the dialog box when creating the next node.

Keep selected coordinate system. When set, the selected 3D element (a vertex or a local coordinate system) will be re-used as the origin for new nodes to be created.

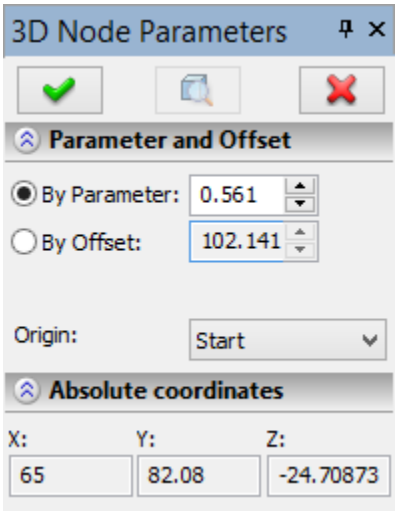
The created 3D node is placed in the specified position with respect to the parent element. Upon any modifications to the parent position, the created 3D node will also change its position according to its relative coordinates defined at creation.

Creating Node Based on Single Connected 3D Profile, 3D Path, or Edge

Selection of a 3D profile (single connected only), 3D path or edge activates the mode of creating a node on the selected element. The selected element is highlighted. To create a node, upon selecting the parent element do the following:

- Select the origin point for defining the node position on the parent element;
- Specify the position of the node being created on the parent element by the offset distance from the reference point (along the parent element).

When the mode starts, the property window displays a dialog box for defining node parameters. The dialog has two sections. The section **Parameter and Offset** serves for defining the reference point and the node position on the parent element. The section **Absolute coordinates** displays the current absolute coordinates of the node being created.

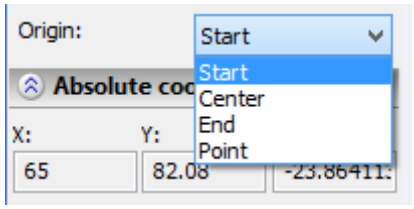


The origin point for positioning the node can be specified as: *start*, *end*, *center* of the parent element or an arbitrary *3D point*.

> The start point of a 3D profile, path or edge can be distinguished from the end point by the kind of element highlighting: an arrow at one of the ends of the highlighted element indicates the end point. In the case of closed 3D elements, the positions of the start and end points coincide.

In the case of using an arbitrary 3D point as the origin, two possibilities exist. If the selected point belongs to the parent element, then it will be used as the origin point. Otherwise, the reference point will be the nearest point on the parent element to this specified 3D point.

Selection of the origin point is done in the property window from the list under the **Origin** item. The pull down list of values contains the following options: **Start**, **End**, **Center**, **Point**. Before selecting the latter option, specify a 3D point to be used as the origin for defining the node position on the parent element. An additional option is provided in the automenu for selecting the point:



	<G>	Set point for offset origin

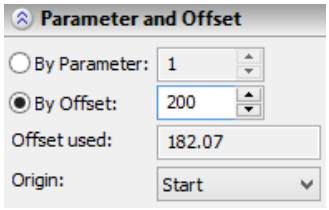
Upon activating the option, one can select the desired 3D point right in the 3D scene. The 3D node used as the origin can also be selected in the 3D model tree.

To reject the selected point, use another additional option:

	<X>	Cancel selection of offset origin

The node position on the parent element can be defined either in the property window or directly in the 3D scene. When using the property window, first choose the way of defining the distance by setting the switch **By Parameter/By Offset** as desired:

- *By Parameter*. The distance from the origin point is set in the relative units (0 – the position of the origin, 1 – the distance equal of the length of the whole parent element). For example, if the midpoint of the profile/path is used as the origin, then parameter values can vary in the range [-0,5;0,5].
- *By Offset*. The distance from the origin is set in the model units.



Should the user enter the value of the offset greater than the length of the parent element, the node is constructed at its end. An additional field appears in the property window, **Offset used**, that displays the actual distance from the origin point to the created 3D node.

As the parameter or the offset is modified in the dialog box, the node in the 3D scene moves accordingly.

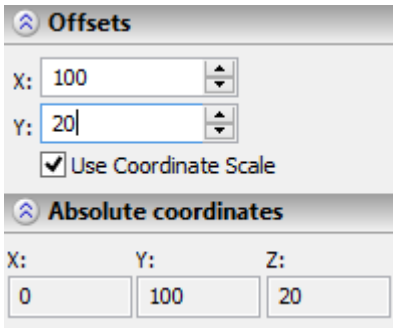
In the 3D scene, simply point the mouse cursor to the desired position of the node on the object and click . On subsequent clicks in the 3D scene, the node will move to the new position.

> At the beginning of node creation, its position coincides with the position of the mouse click selecting the parent 3D element.

Creating Node Based on Workplane

Selecting a workplane creates a 3D node on the workplane. The node position is defined in X-Y or U-V coordinates in the workplane coordinate system. The coordinates can be defined in the 3D scene using the mouse or by numerical values in the property window.

Upon selecting the plane, the preview image of the node appears at the position of the mouse click selecting the plane (or at the coordinate system origin, if the plane was selected in the model tree). On subsequent clicks in the 3D scene, the node will relocate to the new position.

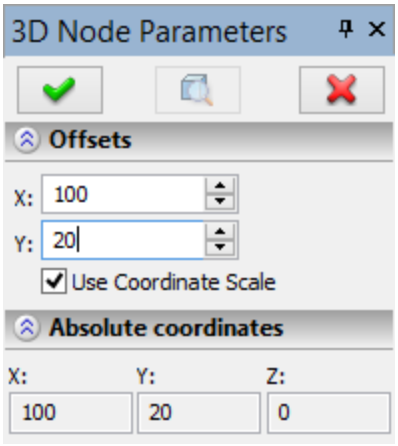


In this mode, the property window contains a dialog box for specifying the exact values of the node coordinates (the section "Offsets"). Meanwhile, the additional section **Absolute coordinates** tracks the position of the node being created in the world coordinates.

The flag **Scale** defines the way of specifying the node coordinates: with the flag set, those are the X-Y coordinates, cleared – the U-V coordinates of the workplane.

Creating Node Based on Underlying Surface

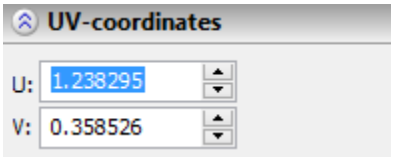
Selecting face as a parent element allows creating a 3D node on its underlying surface. The node position on the surface is generally defined in U-V coordinates.



In the special case of selecting a flat face, one can define X-Y coordinates of the node as the local coordinates on the face plane. The origin position and the direction of the X and Y-axes in this case are defined by the system arbitrarily.

Upon selecting a face (in the 3D scene or in the 3D model tree), it is highlighted (by default – *green*). The node will be displayed at the point of the mouse click selecting the face. The desired position can be defined by another click in the 3D scene or via the property window.

The kind of the dialog box in the property window depends on the type of the selected face. If the face is flat, the dialog box is similar to that used for node creation on a workplane. It allows defining the exact values of local coordinates of the node being created (X-Y or U-V, depending on the state of the flag **"Use Coordinate Scale"**). When selecting a non-flat plane, the dialog box allows defining U-V node coordinates only.



Creating Node Based on Operation

When selecting an operation as the parent element, the command switches to the mode of node creation at the center of gravity of the body or a group of bodies, as described below. The selected operation becomes the first body in the group that will be used for constructing the 3D node.

Special Ways of Creating Nodes Based on Existing 3D Elements

Special ways of creating 3D nodes based on existing 3D elements are separated into a special group of options in the automenu, inactive by default.

Creating Node at Center of Elliptical Edge or Center of Sphere or Torus

A 3D node can be constructed at the center, or, rather, relative to the center of an elliptical edge or a surface of a sphere or torus. To do this, use the following automenu options of the node creation command.

To create a node relative to the center of an elliptical edge:

	<C>	In center of elliptical edge

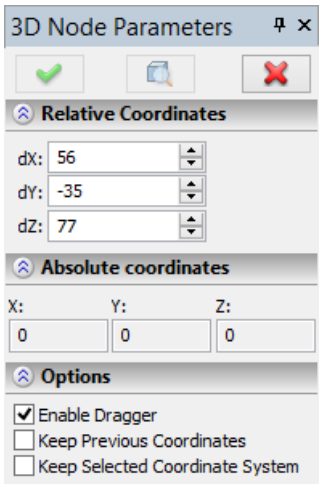
Upon calling the option, specify an elliptical or circular edge (note that a circle is a special case of an ellipse). The node will be created relative to the center of the ellipse to which the selected edge belongs.

To create a node relative to the center of spherical or torus surface, use the option:

	<U>	In center of spherical or torus face

Upon calling the option, specify a spherical or torus face. The node will be created relative to the center of the surface (sphere or torus) to which the selected face belongs.

In either case, node creation is similar to creating a node relative to another 3D node or vertex. Selection of the parent element (edge or face) brings a dialog box for defining the coordinates of the node in the property window, and a coordinate system-shaped dragger in the 3D scene. The origin of the dragger coordinate system coincides with the center of the edge or surface (depending on the option used). The directions of the dragger axes coincide with the directions of the axes of the world coordinate system.



The position of the node being created is defined in the relative coordinates or directly in the 3D scene by using the dragger or by specifying the exact numerical values of the offsets in the section **Coordinates** of the property window.

The absolute coordinates of the node (the world coordinates) are displayed in the section **Absolute coordinates** of the same dialog box.

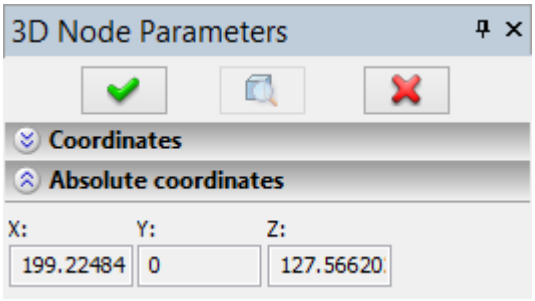
To quickly reset the specified offsets, use an additional automenu option:

	<Z>	Move Node to coordinate system origin

Creating Node at Intersection of Elements

A 3D node can be constructed at an intersection of two 3D elements: the **first** and the **second**. To construct a node, virtually any elements can be used: a path and a body, two paths, an edge and a body, etc. The pairs of 3D elements admissible for selection are defined solely by the possibility of getting an intersection between those resulting in a point. To exclude situations when the intersection of the specified elements is not a point, only a curve or a line are allowed as the first element, while the second element to intersect can be a curve, a line, a sheet body or a solid body.

To create a node, specify the intersecting elements. The order of selecting the elements is not important. The second element to intersect can be selected ahead of the first one. Upon specifying both elements, the system defines the point of intersection for creating a node there. Meanwhile, the property window displays its absolute coordinates. If several intersection points exist, then the one is selected by default that was nearest to the element selecting clicks. A different intersection can be selected if necessary.



This mode is turned on by the option:

	<Q>	Construct 3D Node on intersection of elements

Upon calling this option, the command switches into the mode of defining the first element. The automenu gets additional options:

	<W>	Select curve as first element
	<L>	Select axis as first element

These options allow selecting a curve or straight line to use as the first intersecting element. When selecting 3D elements for the line, pay attention to the state of the element selection filters of these options.

To reject the first selected element, use the option:

	<X>	Cancel selection of first element

To proceed with defining the second element, use the option:

	<T>	Select second element

Upon defining the first element, the command automatically switches to the mode of defining the second element.

The following options are used in the mode of defining the second element:

	<W>	Select curve as second element
	<L>	Select axis as second element
	<F>	Select surface as second element
	<E>	Select sheet body as second element
	<B>	Select body as second element

As appears from option names, those allow selecting a curve, straight line, plane, surface, solid body, or sheet body as the second intersecting element. When selecting 3D elements defining a line, surface, or body, the element selection filters are used.

To reject selection of the second element, use the option:

	<X>	Cancel selection of second element

To return to the mode of specifying the first element, use the option:

	<T>	Select first element

If selected elements have multiple intersections, use the following option to select the desired one:

	<N>	Select another intersection

Creating Node by Shortest Distance between Elements

A 3D node can be created by the shortest distance between two 3D elements:

	<R>	Construct 3D Node using shortest distance between two elements

Upon calling this option, specify two 3D elements. Either one can be an edge, loop, face, 3D path or body. The second element can also be a 3D point. The elements can be specified in any order, however, note that a 3D node is always created on the first element at the location nearest to the second element.

Upon selecting the mode of 3D node creation by the shortest distance between elements, the command goes into the mode of defining the first element. The following additional options appear in the automenu:

	<E>	Select Edge
	<L>	Select Loop
	<F>	Select Face
	<T>	Select 3D Path
	<B>	Select Solid

These options allow selecting a 3D object of the required type as the first element.

Proceed with defining the second element using the option:

	<N>	Select second element

Upon defining the first element, the command goes into the mode of defining the second element automatically.

In the mode of defining the second element, similar options are used. The following options appear in the automenu for selecting the second 3D element:

	<E>	Select Edge
	<L>	Select Loop
	<F>	Select Face
	<T>	Select 3D Path
	<B>	Select Operation
	<V>	Select 3D point

To return to the mode of specifying the first element, use the option:

	<N>	Select first element

Creating Node by Faces and Offsets

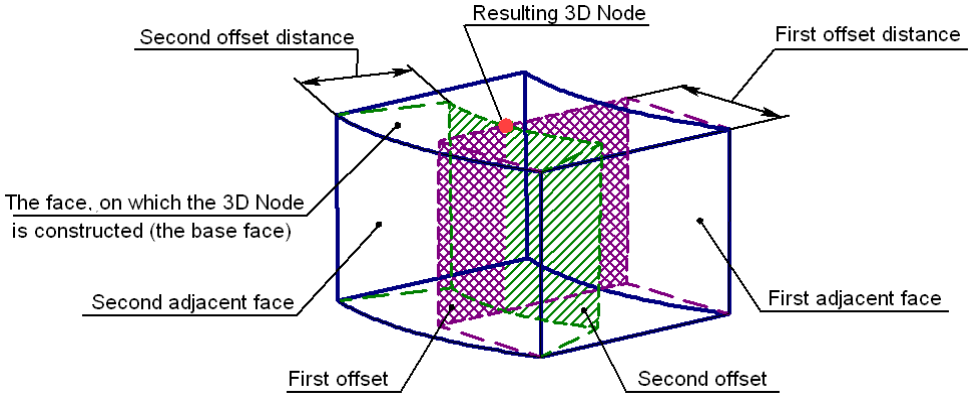
Creating a 3D node by faces and offsets from a solid is a special case of creating a node on a face. In this mode, first specify the base face for the node to be placed on. The required node position on this face is defined using two additional faces of the same body, adjacent to the base face (having a common edge), and their offsets. To create the node, offsets are constructed from the additional faces at the specified distances. The node is defined as the intersection point of the base face and the offsets.

All three faces can have the underlying surface of any type (a plane, torus, sphere, cone, spline surface). Alternatively, the additional faces can be selected by selecting the edges that join the faces with the base face.

In this mode, 3D node construction is done as follows:

- The offsets from the additional faces are constructed. The offset distances are defined by the values entered for those faces. In the case of a positive distance, the offset is constructed inside the body, negative - outside.
- An intersection of the base face and the offsets of the additional faces are calculated. The resulting intersection will be the point of the 3D node creation.

In simple cases, the face surfaces can be extended beyond the bounds (both the base and the offsets). For a plane and cone, this extension is unlimited, while for a torus or a sphere it is closed. This allows, for example, creating a node beyond the bounds of the base face, yet lying on its underlying surface (when the offset distance is negative). This also allows constructing 3D nodes in the situations when the offsets from the additional side faces do not intersect with the base face.



The mode of creating a 3D node by faces and offsets is invoked via the following automenu option provided in the command:

	<O>	Construct 3D Node using faces and offsets

Upon starting this mode, additional options for selecting faces appear in the automenu. Select the base face using the following option:

	<F>	Select face containing 3D node

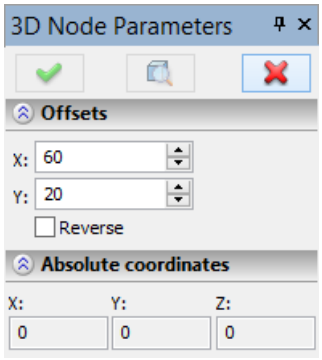
Was selected face is highlighted in *green* (by default). A set of options for selecting additional faces or their defining edges becomes available in the automenu:

	<E>	Select Edge that sets the first offset
	<T>	Select Face that sets the first offset
	<D>	Select Edge that sets the second offset
	<G>	Select Face that sets the second offset

The values of the offset distances from the side faces that define the position on the 3D node being created, are entered in the property window, in the section **Offsets**.

Setting the additional flag **Reverse** flips the direction of counting the offset distances. The node position on the base face would correspond to the negative values of the offsets. When the flag is cleared, the positive offsets are inside the body, meaning that the node is created within the face.

The desired node position on the base face can be directly specified in the 3D scene by clicking . Meanwhile, the property window will display the corresponding values of the offsets.



Creating Node at Center of Gravity

Creation of a 3D node at the center of gravity of one or several bodies is performed using the following option:

	<M>	Create 3D node in center of mass

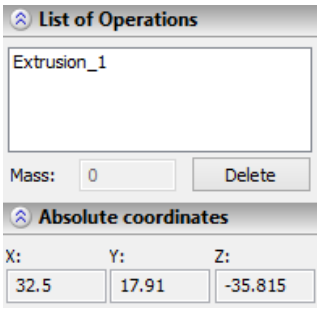
To enter this mode, you can also use the option , selecting a body as the defining 3D element.

In the mode of creating a 3D node at the center of gravity, one or several bodies need to be specified to define the node. The center of gravity is calculated for the specified set of bodies, defining the position of the 3D node being created. The actual material properties are accounted for when computing the center of gravity.

Upon calling this mode, the following additional options appear in the automenu for selecting the necessary set of bodies:

	<B>	Select operation to create 3D node in center of mass
	<L>	Select all operations

Operations can be selected in the 3D scene by clicking or in the 3D model tree. Selected operations are included in the list of the defining operations in the property window (the section **List of Operations**).



The property window, besides keeping the track of the selected operations, supports other actions. For example, one can find the mass by clicking one or several operations in the list of selected, and reading the result in the **Mass** box. The list can be edited. Selected operations in the list can be removed by pressing the pushbutton **[Delete]**.

An additional section in the dialog box, **Absolute coordinates**, helps keeping track of the position of the node being created in the world coordinates.

One or several selected operations can be excluded from the list not only via the property window, but also by direct selection in the 3D scene. To do this, use the following options:

	<F>	Cancel selection of operation
	<D>	Cancel selection of all operations

These options become accessible only after at least one operation has been selected.

Creating Node at Intersection of Three Planes

Creation of a 3D node at a point of intersection of three planes is carried out with the help of the option:

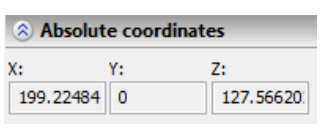
	<K>	Point is at intersection of three planes

To create a node in this mode, three planes are specified. To select the planes, use the options of the automenu:

	<0>	Select first plane
	<1>	Select second plane
	<2>	Select third plane

The given options allow us to select any 3D object capable of defining a plane in the space – planar edge or face, workplane, LCS, etc. When selecting 3D objects, the user should pay attention to the state of filters of selection of the elements for the given options.

After specifying all three planes, the coordinates of a point of intersection of the given planes will appear in the properties window. After pressing the 3D node is created at the given point.

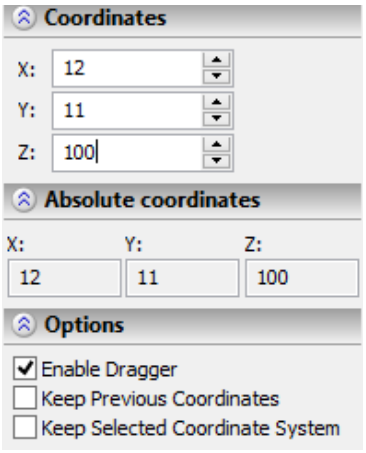


Creating Node in Absolute Coordinates

To create a 3D node in absolute coordinates, use of the option:

	<A>	Construct 3D node in absolute coordinates

Upon selecting this mode, the property window displays a dialog box for defining absolute coordinates of the node. Simultaneously, a coordinate system-shaped dragger appears in the 3D scene. The dragger coordinate system origin is placed at the point (0,0,0) in the world coordinates. The dragger axis directions coincide with the directions of axes of the world coordinate system.

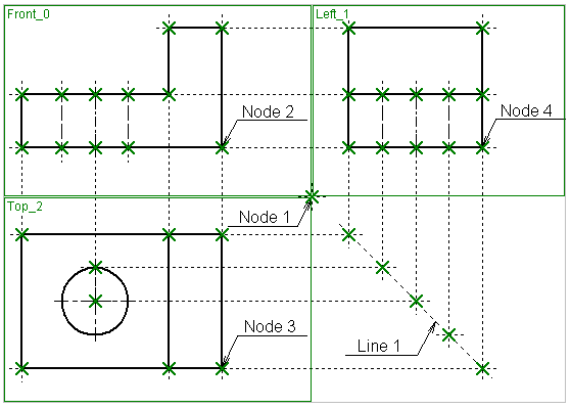


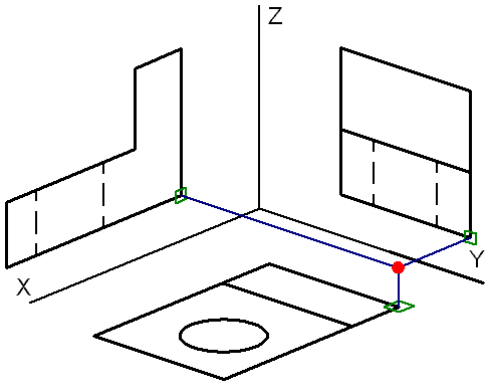
The position of the node being created is defined either directly in the 3D scene by the dragger, or by entering the exact coordinate values in the section **"Coordinates"** of the property window. To quickly reject the specified position of the node, use the following additional automenu option:

	<Z>	Move Node to coordinate system origin

Creating Node by Two Projections

A 3D node can also be created based on its two projections on different workplanes. 2D nodes are used as projections.





To create a 3D node, specify two 2D nodes lying on different workplanes. To create a 3D node directly on a workplane, specify just one 2D node, that is, just one node projection. The workplane and the two node coordinates will define the position of the 3D node in space in this case. To define the height of the node

being created above the workplane, the second projection needs to be specified, that is, the second 2D node. The two projections define a point in space unambiguously.

> When defining a 3D node by one projection, one can use a 2D node lying on a worksurface. As a result, the 3D node is created lying on the given worksurface.

When defining a node by two projections, erroneous situations might occur. The projection is the point at the intersection of the workplane and the normal. The second projection is the second intersection. To define a point in space, the two normals must intersect. If two projections do not define a point in space, a relevant message is display in the diagnostics window. As a rule, errors occur when defining projections on two views that do not have an explicit projective relation.

For instance, in the case shown on the diagram, one can select Node_2 is the first projection that belongs to the workplane "Front". For the second projection, one can select, for instance, Node_3. Two projections unambiguously define a point in space.

To create a 3D node in reviewed example, one can also use Node_3 on the plan view and Node_4 on the side view. To avoid erroneous situations, the drawings on these views have a projective relation, as defined by the projecting line Line_1.

The mode of creating a 3D node by two projections is launched via the automenu option:

	<J>	Create 3D node by two projections

Upon calling the option, the command automenu subsequently brings the options for defining 2D node projections:

	<F>	Set 3D node 1st projection
	<G>	Set 3D node 2nd projection

One can select a 2D node by clicking in the 2D window. The selected node is highlighted. Also highlighted is the workplane that contains the node by default (a workplane contains an element by default when the element appearance on the screen is within the workplane bounds). To select a different workplane, use the option:

	<W>	Select other workplane

To cancel use of the second projection (in the case when it is already defined), push the following automenu option:

	<D>	Delete 2nd projection of 3D Node

The absolute coordinates of the node being created are tracked in the property window.

CREATING 3D NODES BY GRID OF WORKPLANES

Upon creating/editing 3D nodes, in situations, when selection of a 3D point is required, there is a capability of transparent construction of a 3D node by the grid of workplanes (without invoking the **3N:**

Construct 3D node command). When the grid is displayed in all operations, in which selection of a geometric point is required, it is possible to select the points at the nodes of the grid of workplanes, and also the points of intersection of paths and planes. In the latter case, at the nodes of the grid a 3D node is automatically created.

3D PROFILES

A 3D profile is one of the most important elements in T-FLEX CAD. 3D profile is an oriented contour in the three-dimensional space. A 3D profile is oriented based on the orientation of the elements used for defining the profile (a workplane, worksurface, face or a set of edges).

A 3D profile can be created based on a parametric 2D contour located on a workplane page. This allows propagating parametric properties of two-dimensional drawings into the realm of solid modeling.

Sweeping a profile by any means in the space is the base operation for a solid body creation.

MAIN CONCEPTS. TYPES OF PROFILES

Profile Geometry

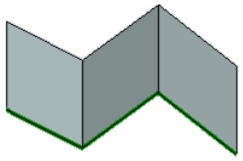
In a general case, a 3D profile is a sheet geometry element. For working convenience, profiles in the 3D scene are usually rendered as a wireframe, that this is a set of bounding lines (contours). However, one can set the 3D profile display mode is a sheet body.

3D profile in wireframe mode

3D Profile in sheet body mode

The contour of a 3D profile can be open. Such a profile will appear and behave as a wireframe geometry body in either display mode. For example, extruding such a profile will result in a sheet body, rather then a solid body as usual.

3D profile with open contour



Result of extruding 3D profile with open contour

A 3D profile might contain several contours (bounding lines).

Types of Profiles

All 3D profiles can be divided into five groups by the ways of their creation:

1. Profiles created based on 2D elements – hatches.
2. Profiles created based on 2D elements – text.
3. Profiles created based on 2D elements – graphic lines.

The above listed types of profiles can be created by the user explicitly, using a dedicated command, or created automatically by a system while managing the active workplane. These profiles are used at the

initial stage of a solid body creation. As the body is being built by various operations, new surfaces, faces and edges are created, that can be used for subsequent operations. These elements can be used as a base for creating profiles of other types:

4. Profiles created based on solid body elements – loops (a closed chain of edges) or faces.
5. Profiles created by using existing profiles. For example, one can copy profiles or create offset profiles. One can also project or lay an existing profile on a three-dimensional solid body or on some of its faces.
6. Profiles created by unfolding a surface or a set of surfaces.

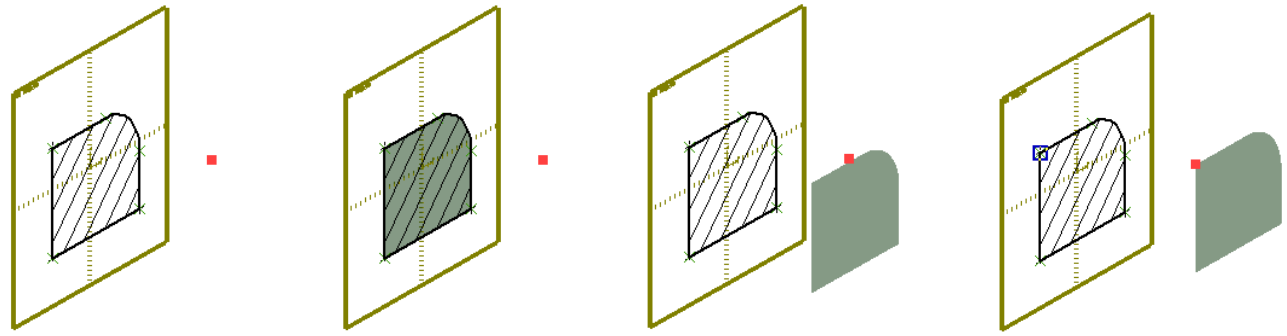
PROFILES BASED ON 2D ELEMENTS

3D profiles created from the elements of 2D drawings are widely used in creation of 3D models. To create a profile, one can use hatches, text and graphic entities located on workplane and worksurface pages. When using a 2D element lying on a workplane page, the resulting 3D profile will be flat. A 3D profile created from a 2D element on a workplane will repeat the shape of the underlying surface.

Profiles Based on Hatches

When creating a hatch-based 3D profile, its plane (surface) coincides with the workplane or a surface, whose page contains the original hatch.

A flat 3D profile (based on a hatch on a workplane) can be repositioned in space. This can be done by using a 3D node through which the plane of the profile being created should pass. If that is not sufficient, one can refine the position of the 3D profile in space by specifying a 2D node (on the workplane containing the hatch), to which the selected 3D node should be snapped.

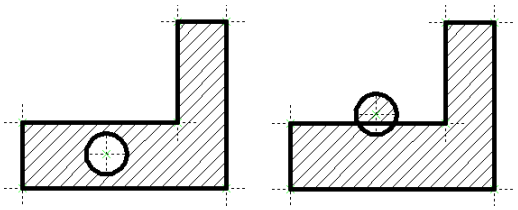


To create a 3D profile, one can use any hatches, including the ones created for decorating a 2D drawing. Those can be of any type: hatches, fills, outlining or hidden. However, the hatches created for a 2D drawing, are not always suitable for creating a profile of the required shape. In this case, one has to create a special hatch for the 3D profile. If such hatch is not necessary on the 2D drawing, we recommend using the "Invisible" type or put such elements on a special layer.

Note that one workplane (or, rather, its page) might contain several hatches, any of which can be used for creating 3D profiles.

Self-intersecting contours

Use of self-intersecting hatch contours is not allowed for creating a 3D profile. For example, if a hatch consists of two contours, then the situation shown on the left-hand side diagram is correct. The right hand side diagram shows a situation that leads to creating an erroneous contour.



Initially, a 3D profile could be created based on a correct hatch contour. However, after a later editing of 2D construction elements, an erroneous situation may occur, which is the self-intersecting contour. In this case, an error message is output in the diagnostics window, and the 3D profile regeneration suspends until after the problem is fixed.

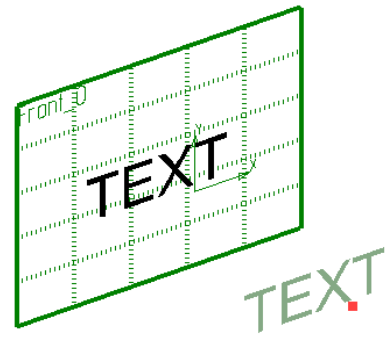
This rule (about prohibiting self-intersection of elements of a 2D contour used for creating the 3D profile) is applied not only to the profiles created from hatches. Self-intersections should be avoided also when using text (some TrueType fonts create self-intersections in certain letter combinations), and graphic lines.

Profiles Based on Text

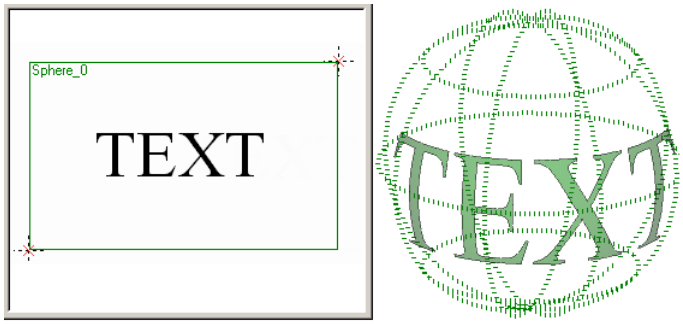
3D profiles based on 2D text are created identical to those based on hatches. Additional nodes may be specified for positioning a flat profile in the space. However, there is a limitation when creating text-based 3D profiles. Only TrueType fonts are supported.

The text in this case is considered a union of closed contours, where each contour is a separate character. Thus, the resulting 3D profile is a multi-contour one.

If elements are encountered in a text that cannot be transformed into the profile contour lines, these are ignored. For example, if a roughness symbol was inserted in a text, then only letters and numbers in the roughness notation will be used for generating the contour lines, while the strokes of the roughness symbol itself will be ignored.



Creating 3D Profile from text on Workplane

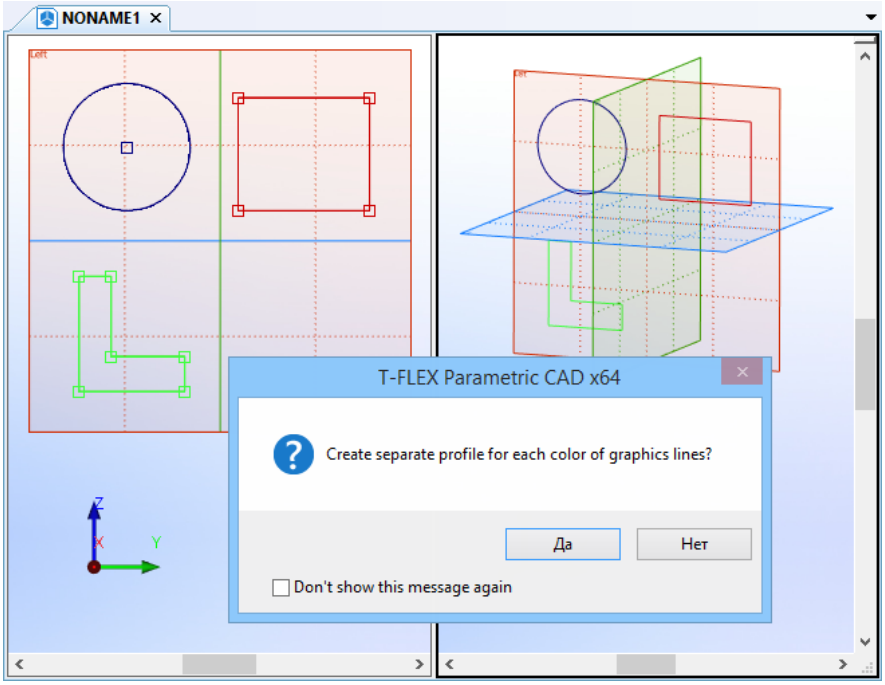


Creating 3D Profile from text on Work Surface

Profiles Based on Graphic Lines on Workplane

A 3D profile can be created based on graphic lines lying on the specified workplane. The involved lines may constitute any image. This can be a connected sequence making a closed or open contour, or simply a set of separate lines. The graphic lines are even allowed to intersect. However, in this case one will have to use the *thickening option* for the profile being created (see the section “Thickening flat profile”).

If all graphic lines on the workplane have the same color, then only one 3D profile is always created. To create several profiles, the graphic lines of different colors can be used. When creating a profile, the system will prompt the user to create a separate 3D profile for each color of the graphic lines. Upon approval, the number of the 3D profiles that will be created will be the same as the number of the colors that were used in creating the image on the workplane. By default, each 3D profile will be assigned the same color as the original graphic line.



In parameters of each 3D profile that was created by graphic lines, the lines whose color served as a basis for the creation of the given profile will be marked. If desired, the user can modify the color of the original lines later or altogether refuse the filtering of lines by color. In the latter case, the profile will be reconstructed by all graphic lines of the given plane.

If the image on the workplane is later modified (for example, new graphic lines added or existing ones deleted), the 3D profiles created based on this image will adjust accordingly.

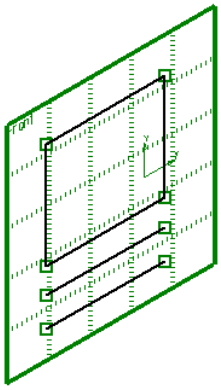
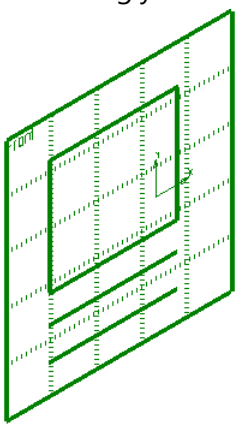


Image on Workplane



3D Profile

Automatic Creation of Profiles from 2D Elements (on Active Workplane)

A 3D profile can be created by the system automatically at the end of a session on the active workplane. In this case, the workplane must contain at least one 2D element of any of the following types:

Hatch;

A graphic line created by the command **SK: Create Sketch** or **G: Create Graphic Line** (the lines of 2D projections are not considered in this case);

Text (using a TrueType font).

The 3D profile creation algorithm depends on the type of 2D elements present in the workplane. In the case when only graphic lines are present on the workplane, the system automatically creates a single new profile (the single-contour or multi-contour one). When later adding new graphic lines, the system will automatically add contours to the existing profile. If the identified graphic lines intersect, then use the *thickening option* for creating a 3D profile.

When several different types of elements are present in the active workplane, all suitable for creating a profile, then the system works differently. The graphic lines are ignored in this case. The profiles based on hatches and text only will be created at the end of the active workplane session. A separate profile will be created for each hatch and each text. Only if no 3D profile could be created from any hatches or text in the workplane, the graphic line-based profile will be created.

PROFILES BASED ON 3D ELEMENTS – LOOPS, EDGES, 3D PROFILES

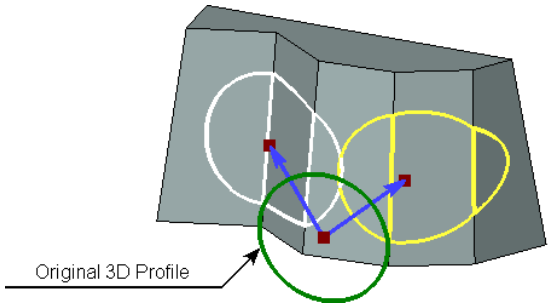
Profiles Based on a Loop or a Face

This type of a 3D profile is created based on a loop (a closed chain of edges) or face (set of faces) of the 3D solid body. The geometry of this kind of profile is fully defined by the selected elements of the 3D body and adjusts to the body geometry modifications. In a general case, this type of profile may not be flat.

Projecting Profile on a Face or a Body

A 3D profile can be created as a projection of another existing 3D profile on all faces of a 3D body or on its single face. In the case of projecting on all faces, the body itself is selected, when projecting on a face – a specific face. In this case, remember the following restriction: the projection of the selected profile should fully fit within the selected surfaces.

The projection vector of the original profile is defined either by one 3D object, capable of defining the vector (straight path or edge, LCS), or with the help of two 3D points (3D nodes, vertices of bodies, etc.). If the projection vector is not defined, then the projection is constructed along the normal to the selected face. Selecting the body is equivalent to selecting a set of faces. Therefore, in this case the projection vector will be defined as the normal to the nearest face.



Two 3D Profiles are projections of the same original profile on the body faces. Blue arrows (not shown in the 3D scene) indicate the direction of the projection vectors defined by 3D Nodes.

Copying 3D Profile

To create a 3D profile as a copy of an existing profile, simply select the original 3D profile. The thus created profile will coincide in space with the original profile. Various transformations defined among the profile parameters will be applied with respect to the world coordinate system.

If necessary, one can additionally specify the original and the target coordinate systems. In this case, profile copying translates the profile from the original to the target coordinate system. The transformation parameters defined for the profile will be applied with respect to the original coordinate system in this case.

The capability of copying 3D profiles relieves the user from creating additional workplanes.

3D Offset Profile

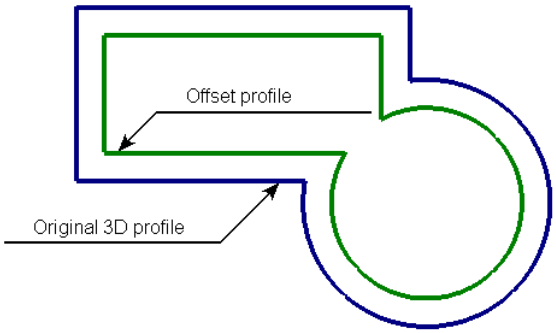
3D offset profiles can be constructed from *flat* 3D profiles only. The original profile can be a multi-contour one.

To create an offset profile, select the original profile and specify the offset with respect to the original profile. Offsets can be defined in two ways:

By defining a 3D node for the offset to pass through. The node should lie in the plane of the original profile.

By a numerical amount of the offset distance for the target profile from the original profile.

The diagram shows an example of creating an offset profile at a negative distance.



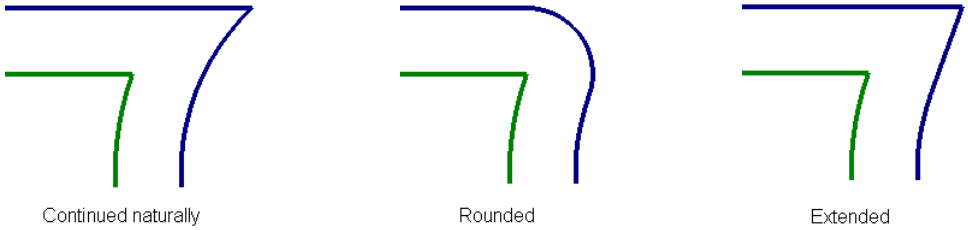
This drawing is located in the folder " *Examples\3D Modeling\ Profile\Offset Curve*".

When creating an offset, three ways of handling gaps between the offset strokes are possible:

Natural – the curves that make the original contour are continued naturally. Straight lines are continued straight, arcs - round.

Round – rounds are constructed between the ends of the neighboring strokes of the contour being created.

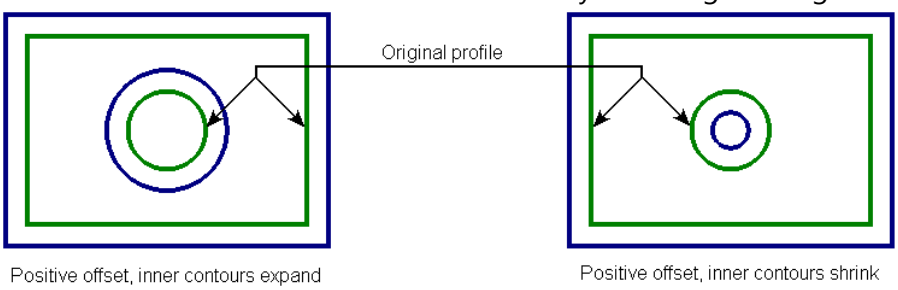
Extend – the gaps are closed by straight lines tangent to the strokes at their ends.



If the original profile to offset from contains inner loops, their handling will be defined separately, regardless of the offset distance sign. One of the two options can be chosen:

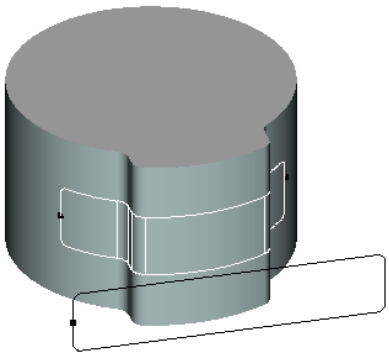
Enlarge – offsets to inner contours are constructed by expanding the original contours, regardless of the offset distance sign.

Decrease – offsets to inner contours are constructed by shrinking the original contours.



Laying Profile on a Face or a Body

This functionality allows creating a new profile by laying the original *flat* profile on a face or set of faces in the body.



How does it work? A point is selected in the profile plane to be tied to a point in the face. The point selected on the profile is projected on the selected point on the face. After that, the profile is wrapped around the face, as if a paper label on the bottle. One can use additional elements for precise profile positioning.

The reference point in the profile plane used for projecting on the surface is defined by a 3D node lying on the original profile.

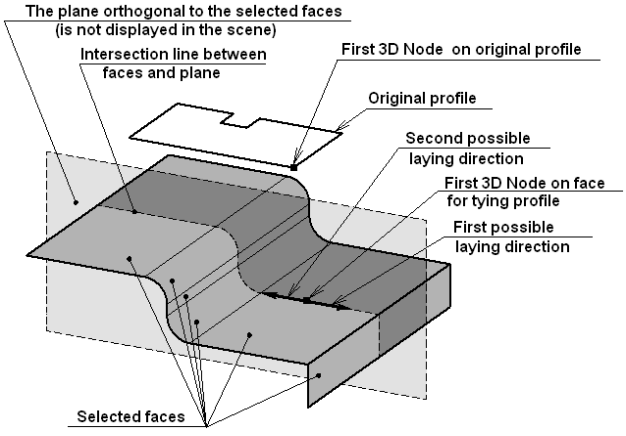
The area for the profile to be laid on can be represented either by a separate body face or by a set of adjoining faces.

The profile transformation is performed without rips or shrinkages of its surface –the area of the original and the resulting profiles is the same. This condition poses restrictions on the used faces or sets of faces.

> The faces allowed for laying profile are those that can be created by simple extruding. However, those don't have to be produced by the extrusion operation per se. For example, a cylinder is allowed, since it can be created by simple extruding, even if it was actually created by the rotation. On the other hand, a cone, sphere or torus is not allowed, since it cannot be created by the simple extrusion.

The point on the face used for tying the original profile is defined by a 3D node. The profile is tied to the first selected point (the first point selected on the profile is snapped to the point on the face).

Then, a plane is constructed through the selected 3D node on the face orthogonal to all used faces. If the selected faces were constructed by an extrusion, then this plane would have been orthogonal to the extrusion direction. The profile will be laid along the intersection line between this plane and the selected faces. At this point, two possibilities exist for defining the direction of the profile laying along the intersection line. The profile can be laid on either side of the reference point (see the diagram).

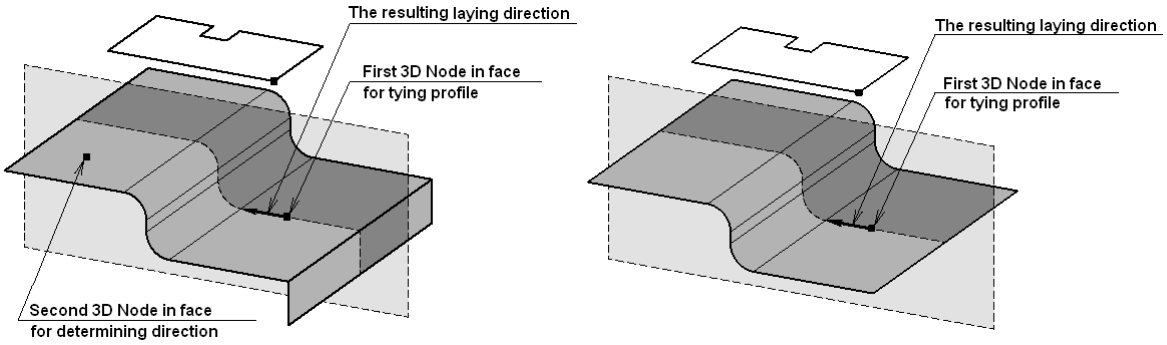


> The plane orthogonal to the selected faces, the intersection line between the plane and the faces and the direction vector of laying the profile are not displayed in the scene. Those are shown in the diagrams for illustration purpose only.

To define the direction of laying the profile uniquely, one can select the second point on the used faces. The selected node will define which out of the two possibilities should be used as the direction of laying the profile.

> If the set of faces is defined in such a way, that the first point belongs to a side face, then the laying direction is determined automatically – towards the adjoining faces. This is the case when you do not have to select the second point on the face.

Both cases described above are illustrated by the following diagrams:



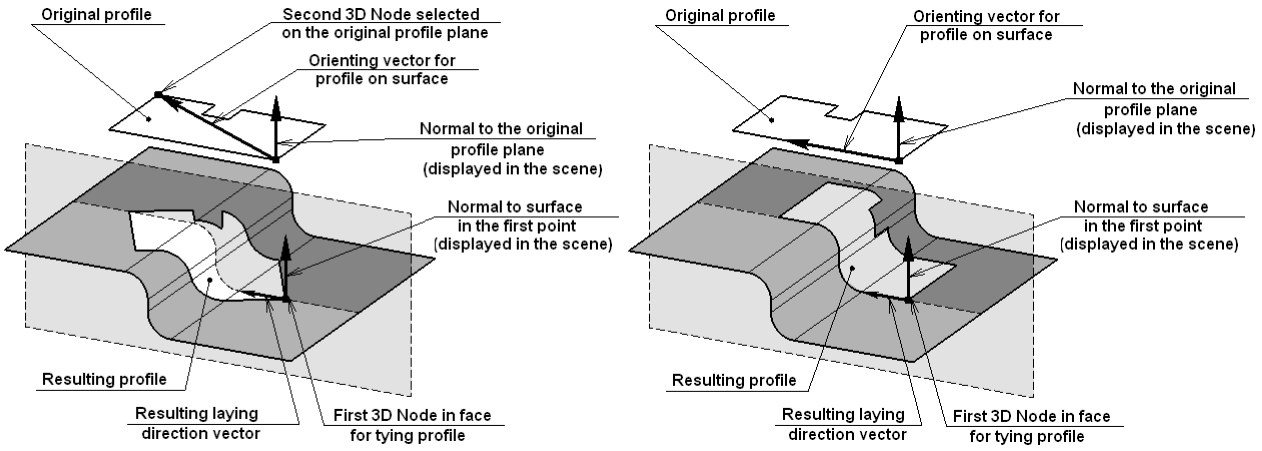
The diagram on the right hand side shows the case, when the first selected 3D node (on the surface) belongs to a side face. In this case, the laying direction is determined automatically.

Next, **the profile orientation on the face** needs to be resolved.

The vector that defines the profile orientation on the face (set of faces) can be defined in two ways: by using the second point on the profile plane, or without it. This vector is not displayed in the scene, and is shown in the diagrams for clarity.

When using the second point, the start and the end of the new vector will be defined by the first and second points on the profile plane. In the cases, when the second point on the profile plane is not selected, the direction is determined by the following algorithm:

- The laying direction vector beginning at the first point on the surface is projected on the profile plane. This gives the direction of the new vector for profile orientation.
- The start of the profile-orienting vector is placed in the first point on the profile plane.



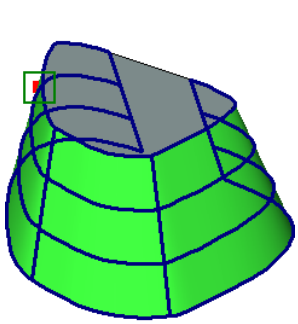
The arrows that are drawn from the first point on the profile and on the face show the normal directions to the profile plane and to the surface. When laying the profile, the profile plane is reoriented to have its normal coincide with the normal to the surface in the first selected point. The normal direction of the profile plane can be flipped to the opposite. This allows turning the laid profile 180° about the laying direction axis.

Profiles – Unfoldings of a Surface or a Set of Surfaces

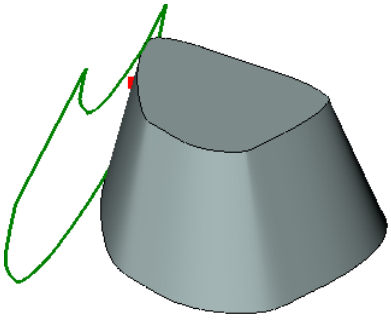
Constructing Unfolding of a Ruled Face

To construct a 3D profile as an unfolding of a ruled surface, select a face of the respective type. The face can be as either closed or open, with or without holes of various shapes.

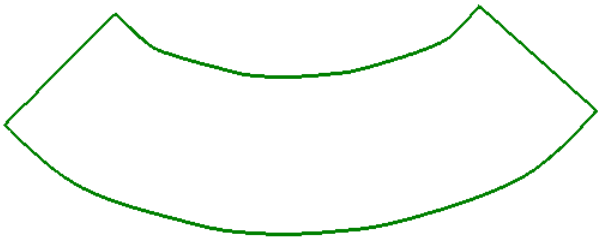
The profile is constructed in the plane tangent to the surface of the original face. The tangency point of the profile or face can be specified by any 3D object which is capable of defining a point (for example, 3D node or vertex of the body). Specified point has to lie on the original face. If the selected face is closed (periodic), the same 3D point is used for defining the cutting line of the unfolding. The cutting line follows the generatrix line through the given node.



Selecting ruled face and 3D Node for creating unfolding profile



Resulting profile - unfolding of the ruled face



Plan view of the profile

Creating Unfolding of a Cylindrical Face

To create a 3D profile as an unfolding of a cylindrical face, select the face to unfold and the tangency point for the plane of the face from which the unfolding profile is being created.

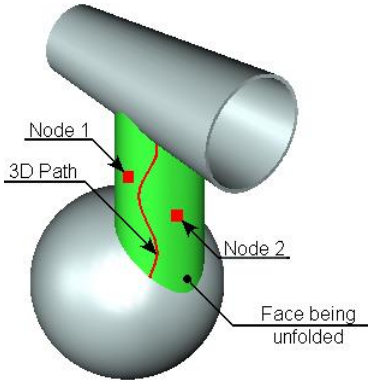
If the face is closed, the cutting line of the unfolding may pass along:

The generatrix passing through the tangency point between the profile plane and the selected face surface.

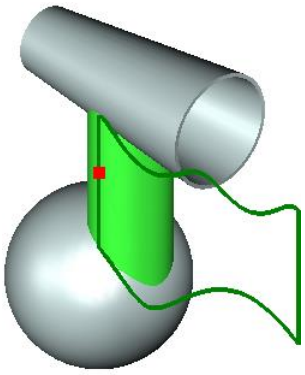
The generatrix passing through an additional 3D point.

A 3D path. The path must satisfy the following requirements:

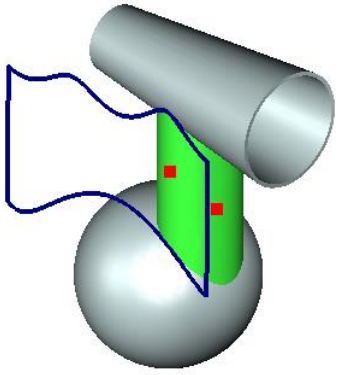
- Lie in the same geometrical surface as the face being unfolded. (To achieve that, one can create an arbitrary 3D path first, and then project it on the face. One can also create a path in cylindrical coordinates with the same position and radius as the face being unfolded.)
- The ends of the path must lie on the end boundaries of the face.



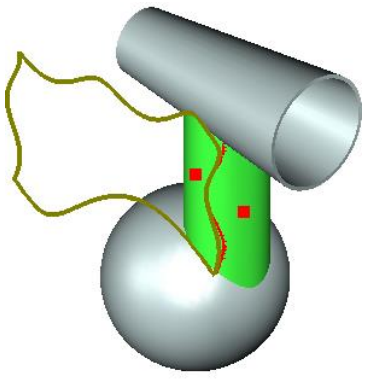
The diagrams show an example of constructing a 3D profile as an unfolding of a closed cylindrical face, using all three types of face cutting for unfolding. Node 1 was used for defining the tangency point between the profiles and the original face, as well as for defining the cutting line for one of the profiles. The additional Node 2 and 3D path were used for defining the unfolding cutting line in two other cases. The resulting profiles lie in the same plane tangent to the surface of the original face at the point defined by Node 1, but differ in the cutting line.



Profile with cutting line defined by the same node as the tangency point (Node 1)



Profile with cutting line defined by an additional 3D node (Node 2)

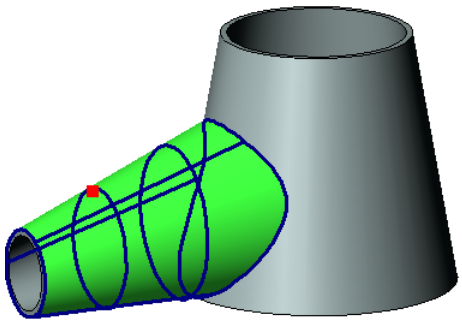


Profile with cutting line defined by an additional path

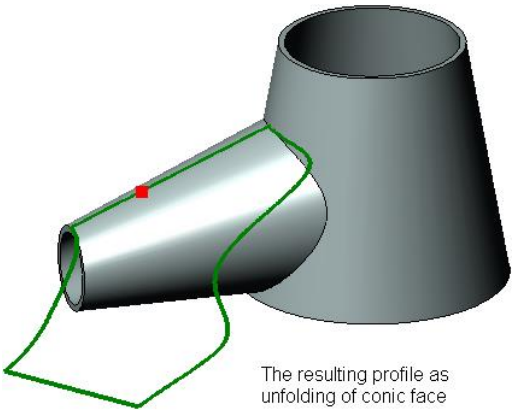
Creating Unfolding of a Conic Face

To create a profile as an unfolding of the conic face, select a conic face. Allowed for selection are faces whose extended surface makes an arbitrary round cone.

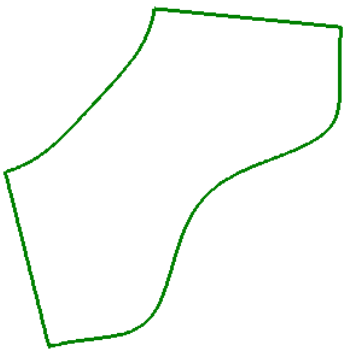
The profile being created is constructed in the plane tangent to the surface of the original face, just as in the case of creating other types of unfolding profiles. The tangency point can be specified by any 3D object, which is able to define a point (for example, 3D node or vertex of the body). Specified point has to lie on the original face. An additional 3D point can be defined for closed faces, to be used for defining the unfolding cutting line. The point defines the choice of the generatrix, along which to cut the unfolding of the face. If no additional point is specified, the unfolding is cut along the generatrix passing through the tangency point between the profile and the face.



Selecting face to unfold



The resulting profile as unfolding of conic face



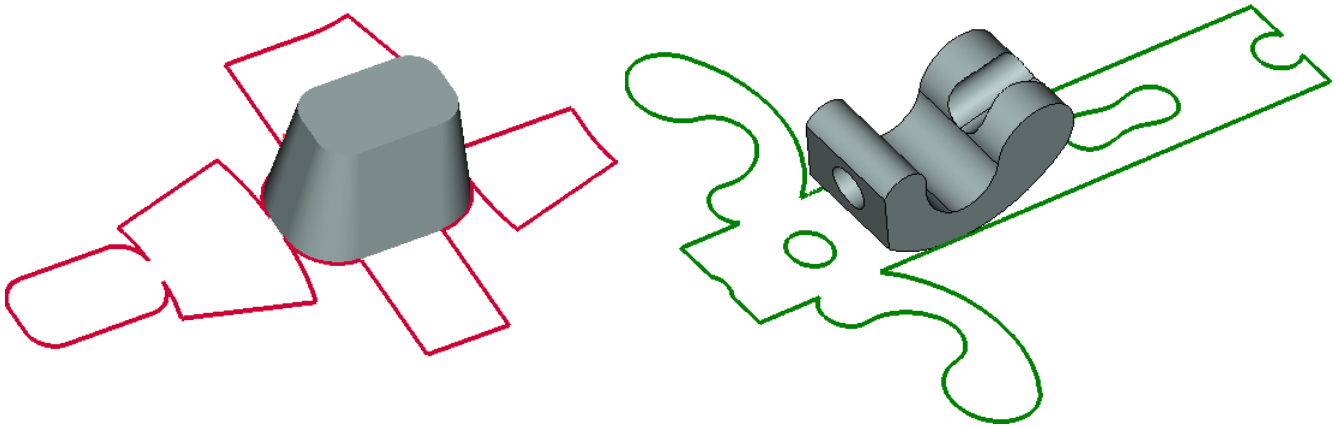
Plan view of created profile

Creating Unfolding of a Set of Faces

Creating a 3D profile as an unfolding of a set of faces is mostly similar to creating other unfolding profiles.

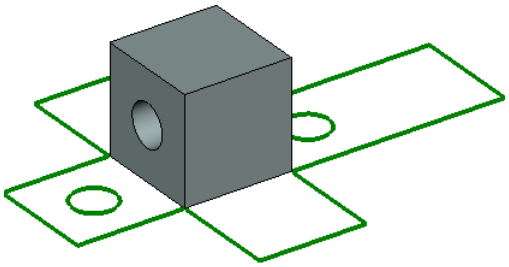
To create such a profile, select the body faces to unfold. The selected faces must have common edges. When creating the profile, each face's surface is unfolded (if flat and unfoldable), and the set of faces is unfolded about common edges.

The position of the profile in space is defined by a 3D point on one of the selected faces. The plane of the profile will be tangent to that face at the specified point.

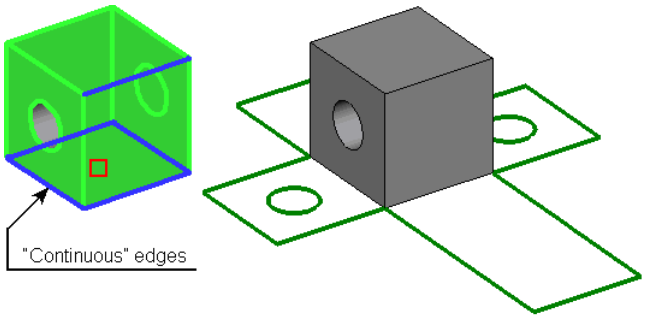


Several solutions could be possible when creating a profile. (An example is unfolding all faces of a cube.) By default, the system picks up a solution automatically. To make a specific choice of profile construction,

one can additionally select "inseparably linked" edges that will be used as the unfolding lines. When specifying edges, one should define the links between all necessary faces. Otherwise, the faces will be ignored in the profile creation, for which "inseparably linked" edges are not defined that would connect those with other faces in the set.

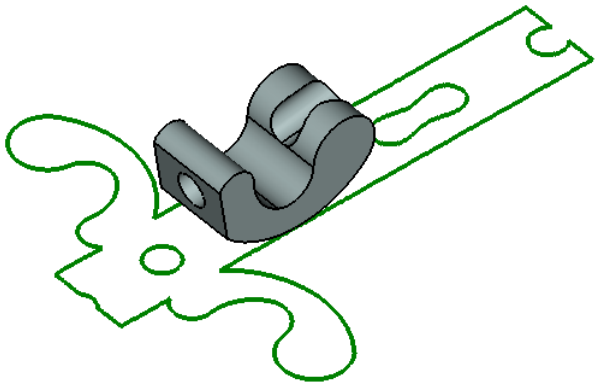


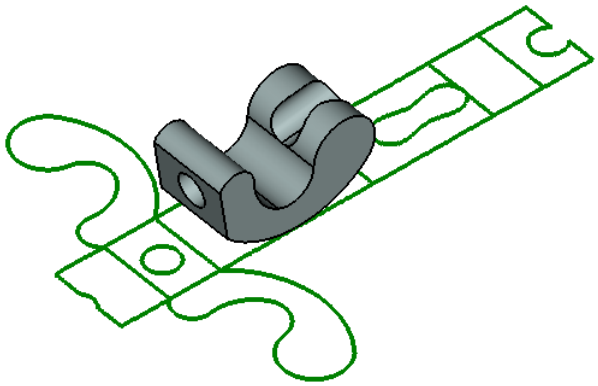
Unfolding choice picked automatically by the system



3D profile unfolding created with "inseparably linked" edges specified

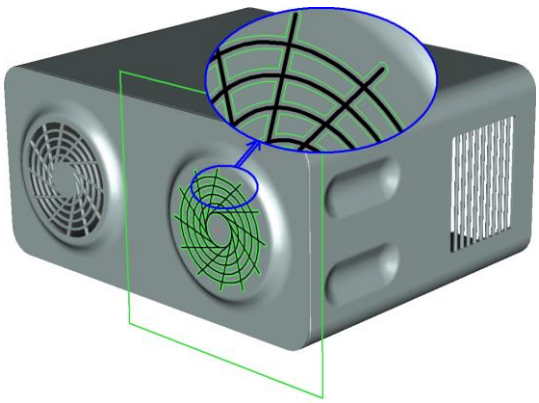
By default model edges will not be visible on the unfolded profile. However they can be created if necessary.





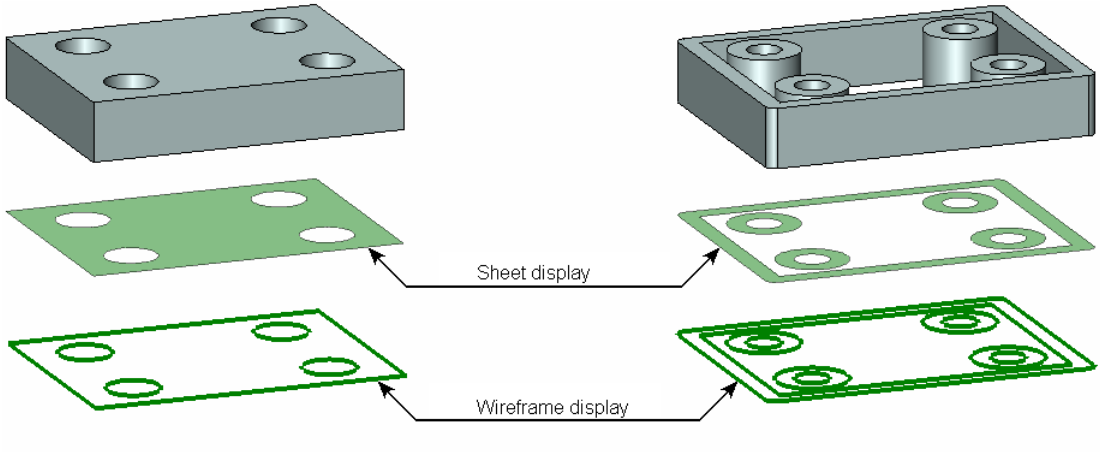
THICKENING FLAT PROFILES

Any flat profile representation can be changed using *Thickening mode*. In this mode, the contour lines of the profile are offset. The new, thickened, contour is constructed based on these lines. The contour geometry may change in this case, which can be readily seen in the display of the sheet profile or the solid body resulting from it.



When creating a 3D profile based on 2D elements, the use of this mode may become the only way of constructing the profile. This happens, for example, when the original graphic lines, hatch or text strokes used for constructing the profile have self-intersections or forks.

In this case, the system will not be able to create the profile, unless in this mode, and will automatically prompt the user for turning on the thickening mode.



Original profile in wireframe and sheet display modes and its extrusion

Thickened profile in wireframe and sheet display modes, and its extrusions

Thickening of a 3D profile depends on the following parameters that affect the resulting shape:

- *Thickness type* (the direction(s) of offsetting the original contour lines);
- *Offset distance* (distances) from the original 3D profile contour lines;
- *Ends handling* of the new 3D profile contour lines;
- *Gap handling* between the hanging ends of the offset 3D profile contour lines;
- *Closed area handling* (specific to 3D profiles created from graphic lines on a workplane).

Gap handling and *closed area handling* work the same way as the respective parameters in the offset profile creation. The rest of parameters are described here.

Note that different shapes of the original 3D profile contour lead to different behavior of this mode.

Thickening types and line offsetting

Offsets can be created at either or both sides of the original lines, at specified distances in each direction, equal or unequal. The offset distance is defined by a numerical value.

In theory, the way of constructing offsets is determined by the *thickening type* that allows the following choices:

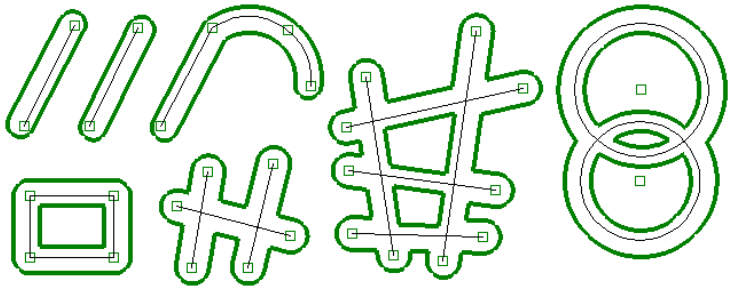
Outward, inward – the offset lines are constructed on one side of the original contour lines; for closed contours, the direction of offsetting with respect to the original lines is defined by the selected mode (that is, either by expanding or by shrinking the original contour); the offsetting direction for open contours is automatically picked by the system.

Symmetric – the offset lines are constructed on both sides of the original contour lines at equal distances;

Double sided – the offset lines are constructed on both sides of the original contour, at different distances.

Depending on the selected type of thickening, one or two *Offset distances* are required. One value is required for the first three choices of thickening ("outward", "inward", "Symmetric"), two – when the "Double sided" option is chosen.

In real life, however, the shape of offset lines is determined, besides the thickening type, by the shape of the original contour. It depends on the presence in the original contour of self-intersections or forks, as well as closed areas. For example, a contour with self-intersections or forks in most cases can't be thickened on two sides by different offset distances. Use of choices "Outward" or "Inward" is not allowed for such contours either. The result of subjecting it to the option "Double sided" will also depend on the presence of closed areas in the contour. This choice of thickening is also impossible if there aren't such areas in the original profile contour. To be precise, use of the option in this case will result in the same shape as the choice of "Symmetric", that is, the same offset distance will be maintained on either side of the original contour lines (the second offset distance being ignored). If the contour with self-intersections or forks contains closed areas, then the inward offsets (the ones created in closed loops) use one of the distances, while the outward ones (all the rest) use the other offset distance.



Handling ends of lines in thickened profiles

When thickening an open contour, the offsets from the hanging ends of lines can be joined by the following two ways:

Round – the ends of the offset lines are joined by circular arcs.

By line – the ends of the offset lines are joined by straight lines.

"Round"-type end handling

"By lines" end handling

CREATING 3D PROFILES

To create any 3D profile, use the command **"3PR: Construct 3D Profile"**. The command is called as:

Icon	Ribbon
	3D Model → Construct → 3D Profile
Keyboard	Textual Menu
<3PR>	Construction >3D Profile

To create a 3D profile, upon calling the command, select the required type of constructing the profile. The further steps depend on the selected type of the profile.

Creating 3D Profile Based on 2D Hatch or Text

To create a 3D profile based on a 2D hatch or text, use the following options of the command main automenu:

	<H>	Select Hatch
	<A>	Select Text

The normal flow of the 3D profile creation based on a hatch or text consists of the following steps:

1. Selecting a hatch or text (in the 2D window or on the active workplane). This simultaneously defines the workplane or worksurface to which the selected element belongs by default. (The hatch or text belongs to a workplane if it is located within the bounds of the workplane in the 2D window or at a position nearest to the boundaries of this plane.) The hatch and the workplane (surface) are marked as selected in the 2D window. Meanwhile, the preview image of the profile being created appears in the 3D view window.

   To alter the workplane selection, use the option:

	<W>	Select Other Workplane

   This will result in selecting another workplane or worksurface. Subsequent use of this option allows looping through all workplanes present in the active page. Meanwhile, one can watch how the profile being created adjusts dynamically in the 3D window.

   When using a 2D element lying on the workplane, profile creation completes at this point. In the case of a flat profile, one can additionally:

2. Select a 3D node to snap the contour plane to:

	<M>	Select 3D Node for fixing Profile plane

3. Select a 2D node on the contour to snap the specified 3D node to:

	<N>	Select Node to be moved to 3D Node

To complete the 3D profile creation, use the automenu option or the respective button in the property window.

To undo the snapping to the nodes, use the following automenu options:

	<K>	Break link with 3D Node
	<F>	Break link with Node

Those undo the respective actions of repositioning the contour.

Creating 3D Profiles from Graphic Lines on Workplane

To create a 3D profile from graphic lines on the workplane, use the following option of the command main automenu:

	<W>	Create profile by sketch

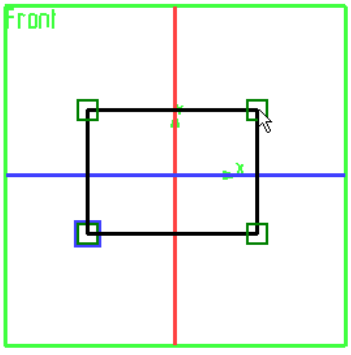
To create the profile, upon turning on this mode, simply select the desired workplane in the 3D window or in the 3D model tree and confirm the profile creation by .

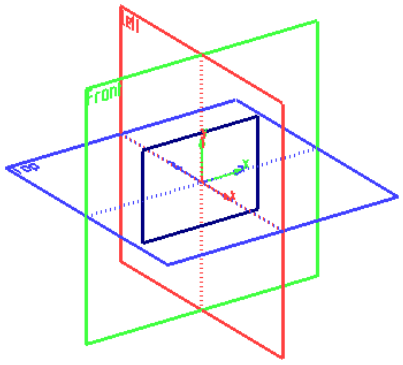
Automatic Profile Creation when Working on Active Workplane

To have 3D profiles be created automatically, select or activate a workplane (the command **"Draw On Workplane"** in the context menu). Once the necessary 2D elements are created in the workplane, such as graphic lines, hatches or text, the profile creation occurs upon completing the active workplane session by the option:

	<Shift><End>	End Drawing On Workplane

The 3D profile will automatically be created upon completing this mode.





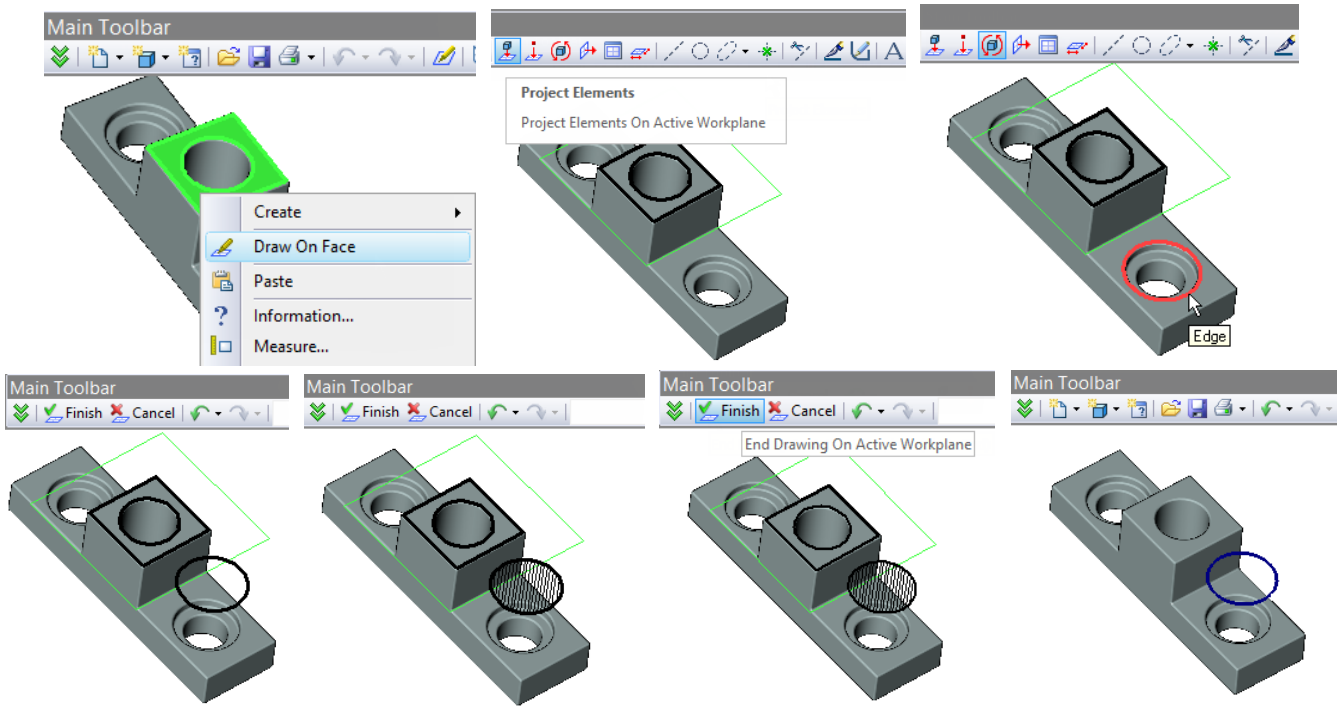
When on the active workplane the graphic lines of different colors exist, the system, before creating a profile, will issue the message: *"Create a separate profile for each color of graphic lines? Yes/No"*. In case of positive answer, several 3D profiles will be created – according to the number of used colors. If creation of several profiles is declined, one 3D profile is constructed by all graphic lines of the workplane.

The active workplane mechanism allows constructing profiles on flat faces of three-dimensional bodies. To do this, move the pointer over a flat face and right-click, and then select the item "Draw on Face" in the coming up menu. A new workplane will be created based on the selected face, along with its respective page. All newly created 2D elements will be placed there. The created workplane will become

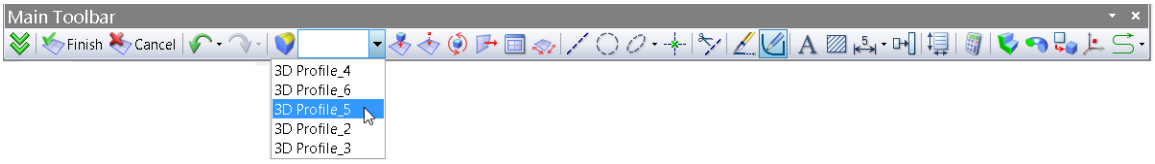
active (see the chapter "Workplanes"). The selected face is projected automatically on the workplane. Next, do the steps described above.

To project other model elements on the active workplane, use the options:

	Project Elements On Active Workplane
	Create Projection of 3D Point of Vertex



When drawing on a plane on which the 3D profiles by graphic lines have been already created, the control element with the list of available on the given plane profiles is present on the system toolbar. If one of the profiles is selected in this list, the graphic lines forming the profile are shown automatically on the screen in large scale, and the color of the selected profile is assigned to newly created lines.



In this mode the **"Color only for current page"** flag is enabled in the parameters of graphic lines. This means that the selected color will be active only upon creation of graphic lines on the page of a workplane.

Creating 3D Profile Based on a Loop or a Face

To create a 3D profile based on loop or face, use the following option of the main command automenu:

	<L>	Select Loop
	<F>	Select Face

To create the profile, upon turning on one of these modes, simply click on an appropriate 3D element in the 3D window: a loop (a closed set of body edges) or a body face. The selected elements are highlighted. When selecting a set of body faces, one can adjust the selection, if necessary: a single face – by clicking on the face again, all currently selected faces – by the automenu option:

	<K>	Cancel selection

To complete the profile creation, use .

Creating Profile - Projection of Existing Profile on a Face or a Body

A profile – projection of another 3D profile on a face or body – is created by using the following command option:

	<J>	Construct 3D Profile as projection of 3D Profile to face

Selection of the body or face on which the original 3D profile will be projected is done by one of the following automenu options provided in this mode:

	<B>	Select Operation
	<F>	Select face for projecting 3D Profile

A body or face can be selected in the 3D window by clicking . A body can also be selected in the 3D model tree. The selected body or face is highlighted in the 3D window.

The profile being projected is selected (also directly in the 3D window or in the 3D model tree) using the option:

	<R>	Select 3D Profile to project

The selection made is sufficient for the profile normal projection. In this case, the profile creation can be completed by pressing . In other cases, the projection vector needs to be defined for the original profile by specifying subsequently two 3D points or by selecting one 3D object which is able to define the vector.

To define the vector by two 3D points, subsequently use the options:

	<M>	Select first 3D points of projection direction vector
	<S>	Select second 3D points of projection direction vector

To specify the direction vector by one 3D object the option is provided in the automenu:

	<L>	Select projection direction

To cancel selection:

	<K>	Cancel selection of projection direction

When selecting elements in the described order, the system automatically switches the options for proceeding with the next step. You can do selection in arbitrary order; however, in this case you have to pick the appropriate options manually.

Creating 3D Profile – Copy of Existing 3D Profile

For creating a 3D profile as a copy of an existing 3D profile, use the option:

	<M>	Construct 3D Profile as a copy of another 3D Profile

To create a profile copy, first select the original 3D profile in the 3D scene or in the 3D model tree. The profile selection option activates automatically, as the mode is turned on:

	<T>	Select profile to copy

Once the profile is specified, one can complete the creation with the option . The created profile will coincide with the original one in this case (if no transformation is defined for it).

To define the coordinate systems that determine the translation of the profile being created off the original one, use the following automenu options:

	<1>	Select Source Coordinate System
	<2>	Select Target Coordinate System

If the required LCS are absent in the current model, the following options can be used:

	<K>	Select Point for Creating Source Coordinate System
	<W>	Select Point for Creating Target Coordinate System

These options allow a user to automatically create a desired LCS by specifying its origin point. The axes of the coordinate system will be directed according to the direction of the axes of the global coordinate system.

The selected local coordinate systems are highlighted in the 3D scene. The preview image of the profile being created appears in the position determined by the specified local coordinate systems.

A selection of a coordinate system can be instantly rejected by the option:

	<F>	Cancel Coordinate System selection

Creating 3D Profile - Offset

To create an offset, select the option

	<Q>	Construct 3D Profile as an offset to 3D Profile

The original profile can be selected in the 3D scene or in the 3D model tree, while under the activated automenu option:

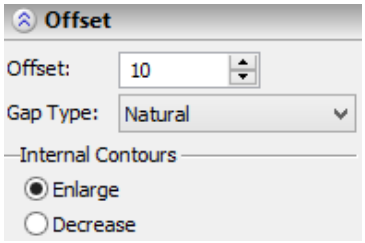
	<T>	Select 3D Profile

The 3D node determining the offset distance of the profile is defined by the automenu option:

	<N>	Select offset by 3D Node

The following automenu option is provided for rejecting the selection of the 3D node:

	<K>	Cancel link with 3D Node



To define the offset distance by a numerical value, one can use the command property window (the parameter **Offset** in the same-name section of the dialog box). Other offset parameters can also be defined there. The choice of handling gaps occurring in the offset contour is defined by the parameter **Gap Type**. The choice of handling the inner contours of the profile (if any) is defined by the selector **Internal Contours**.

Laying Profile on a Face or a Body

The mode of laying a profile on a face is called by the following option of the main command automenu:

	<D>	Lay profile on sequence of faces

To create such a profile, do the following steps:

1. Select the original 3D profile:

	<1>	Select planar profile

2. Select the first 3D node for defining the profile orientation on the surface:

	<2>	Select point on planar profile

   This option activates automatically right after selecting the profile.

3. Select the second 3D node for defining the profile orientation on the surface (*optional*):

	<3>	Select second point on planar profile

4. Select a face or a set of faces on which the profile will be laid:

	<4>	Select surface

An incorrect selection can be canceled using the option:

	<7>	Cancel surfaces selection

5. Select the first 3D node for tying their original profile to the selected set of faces:

	<5>	Select first point on source surface

6. Select the second 3D node for tying the original profile to the selected set of faces (*optional*):

	<6>	Select second point on source surface

7. Flip (if necessary) the normal direction of the profile plane to the opposite (*optional*):

	<Tab>	Change profile orientation

Creating 3D Profiles – Surface Unfoldings

Creating unfolding of a ruled face

To create a 3D profile as an unfolding of a ruled face, use the following option of the main command automenu:

	<B>	Unfold ruled surface

Selection of the face to unfold is done by the option:

	<A>	Select face to unfold

To specify defining the tangency point between the profile being created and the source face, use the option:

	<B>	Select 3D node as tangency point of profile and source face

Creating unfolding of a cylindrical face

Unfolding of a cylindrical face is constructed by the option:

	<C>	Unfold cylindrical surface

The face to unfold is selected with the help of the option:

	<A>	Select face to unfold

To specify the 3D point defining the tangency point between the profile being created and the source face, use the option:

	<B>	Select 3D node as tangency point of profile and source face

To define the split line for unfolding a closed face, use the following options:

	<C>	Select 3D path to define split line
	<D>	Select 3D point to define split line

Creating unfolding of a conic face

To create a profile as an unfolding of a conic face, use the option:

	<E>	Unfold conic surface

The face to unfold is selected with the help of the option:

	<A>	Select face to unfold

To specify the tangency point between the profile being created and the source face, use the option:

	<B>	Select tangency point of profile and source face

To define the split line for unfolding a closed face, use the following option:

	<C>	Select 3D point to define split line

Unfolding of a set of faces

Creating 3D profile as an unfolding of a set of faces is done by the following option of the main command automenu:

	<I>	Unfold group of connected faces

The source set of faces is specified by subsequently selecting faces by the option:

	<A>	Select surfaces

You can specify the inseparably linked edges of the source set (if necessary) by the option:

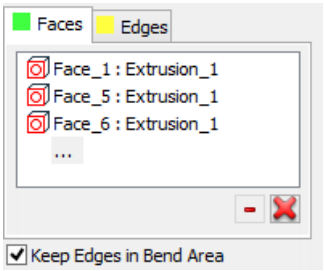
	<B>	Select inseparably linked edges

To specify the 3D point defining the spatial position of the profile being created (that is, the tangency point between the profile and a face from the source set containing this point), use the option:

	<C>	Select point on profile

To reject the selected set of inseparably linked edges, use the options:

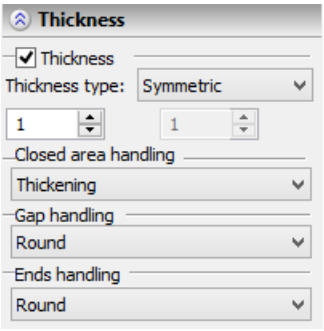
	<D>	Cancel faces selection
	<E>	Cancel selection of inseparably linked edges



Parameters dialog box can be used for cancelling the selection of faces or edges. Selected faces and edges are added to the appropriate lists ("**Faces**", "**Edges**"). button will delete the currently selected face or edge. button deletes all selected faces or edges.

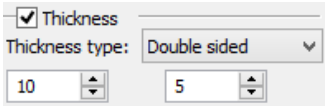
Option **"Keep Edges in Bend Area"** will create internal edges-bounds on the unfolded 3D profile corresponding to edges of the model faces.

Creating Thickened Profile



The mode of thickening a 3D profile of any type can be set in the command property window, in the section **Thickness**. To turn on the mode, one needs:

1. Check the box **Thickness**;
2. Select the profile thickening type from the **Thickness** combo box: **Outward**", **Inward**", **Symmetric**, **Double sided**;
3. Specify the offset distance from the original contour. If the option "Double sided" was selected in the previous step, specify two values for the distances;



4. Define the way of handling the gaps in the offset contours that occur due to thickening by selecting the desired option from the drop-down list of parameters **Gap handling**: **Round**, **By line**, **By curve**".
5. Select the choice of handling ends of the contours being created from the drop-down list of parameters **Ends handling**: **Round**, **By line**.
6. For 3D profiles created from the graphic lines on the workplane, one can additionally define the way of handling closed contours by selecting one of the following options from the drop-down list of the respective parameter: **Thickening** or **Closing area**.

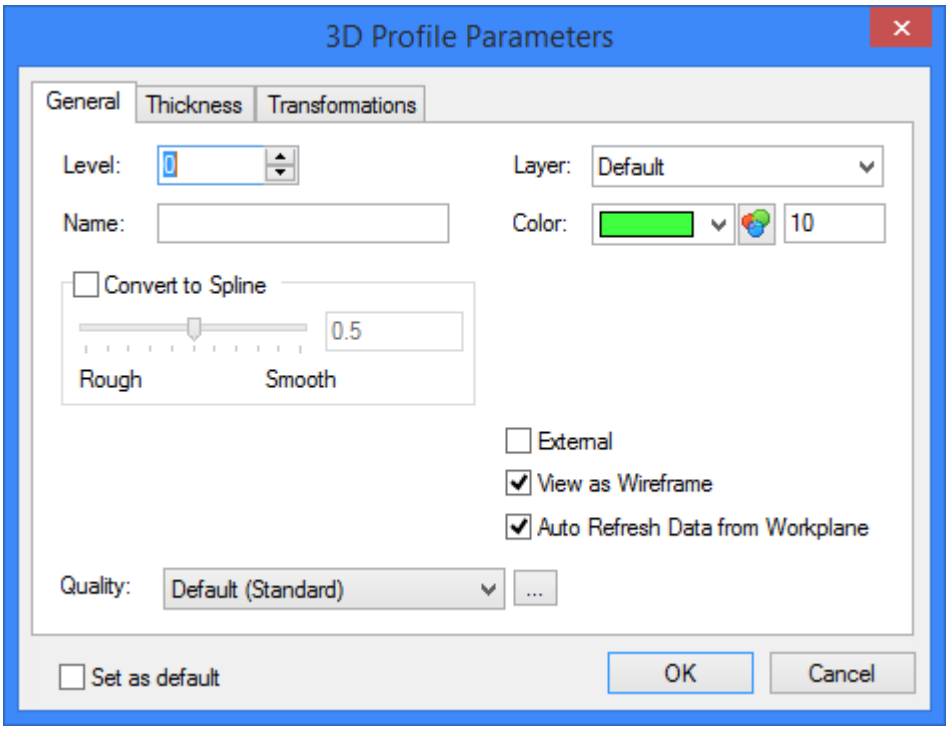
Parameters of 3D Profiles

As any other system element, each 3D profile has a set of parameters, which is the same for all types of profiles. The parameter values can be defined by calling the parameters dialog box via the option:

	<P>	Set entity parameters

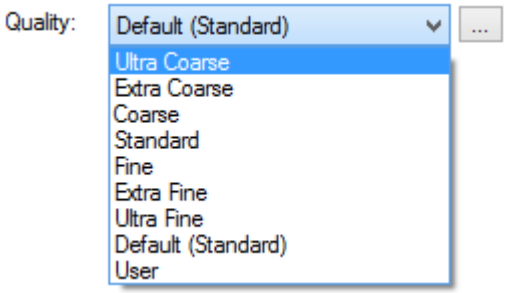
The parameters dialog box of a 3D profile has three tabs: **General**, **Thickness** and **Transformation**. The tab **Thickness** contains the parameters that duplicate the contents of the same-name section of the property window provided with the profile creation command, as described above. The tabs **General** and

Transformation are described in details in the chapter "General parameters of 3D elements". We will mention here just the special parameters provided in the tab "General" specifically for 3D profiles:

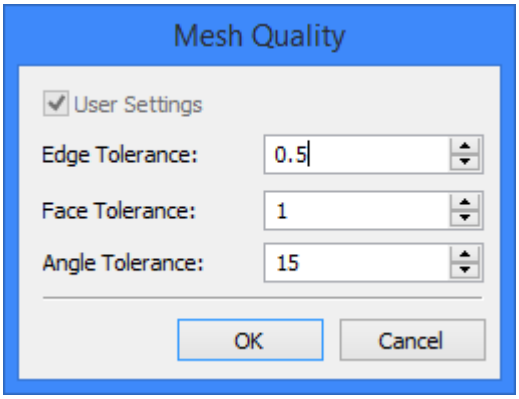


Quality. Parameter determining the quality of the image of a 3D profile. It defines the degree of model discretization into the grid of triangular planar faces for display of the image in 3D window. From the drop down list the predefined set of parameters specifying the quality of the grid can be selected:

- Ultra coarse;
- Extra coarse;
- Coarse;
- Standard;
- Fine;
- Extra fine;
- Ultra fine;



- Default (Standard) – the values of parameters of model discretization are taken from the **Customize > Document Parameters** command (the "Image quality" parameter on the "3D" tab);
- User– the values of parameters of model discretization are specified by the user. Additional button**[...]** invokes the dialog window for specifying user-defined values of parameters of model discretization (in units of a model). In the dialog of mesh parameters it is possible to specify the user's own values: allowance by edge, face precision and angular allowance.



External. If the model is used as a 3D fragment, then the profiles with this parameter set are made accessible in the assembly mode. Those can be used for creating operations. The position of such a profile is fully determined by the position of its respective model. Only parameters can be modified. The profiles derived from 3D fragments have special names (Reference Profile No). These profiles can also be assigned the attribute External, that will allow their use in the assemblies of higher level.

Auto Refresh Data from Workplane. With this flag cleared, some changes in the work plane page will not be reflected in the 3D profile even after a full regeneration (such as, for instance, adding new contours made of graphic lines).

View as Wireframe. This flag controls the way of rendering the profile in the 3D scene. If it is cleared, the profile is rendered as a sheet body. The flag is on by default, and the profile is displayed in the wireframe representation.

Convert to Spline. This parameter allows converting each profile contour into a single spline. Each contour of the thus converted path or profile consists of a single segment, with the intermediate vertices being removed. The approximation accuracy is controlled by the parameter from 0.0 (Rough) to 1.0 (Precise) by the slider or via the accuracy input box, similar to the mesh density. In certain cases, such 3D profiles are more suitable then those based on polylines (as, for instance, in the "Loft" operation).

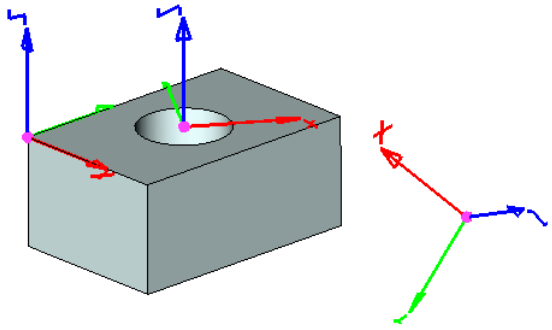
This parameter does not affect combination profiles, whose several faces have adjoining edges (for example, if the profile is created from a set of adjoining faces).

Select lines by color. The given parameter has the meaning only for 3D profiles that were created by graphic lines on the workplane. This flag controls the way of creation of the 3D profile. If it is disabled, the profile contour is constructed on the basis of all graphic lines of the workplane. When the flag is enabled, the profile contour is created only on the basis of the graphic lines of the specified color.

Local Coordinate Systems

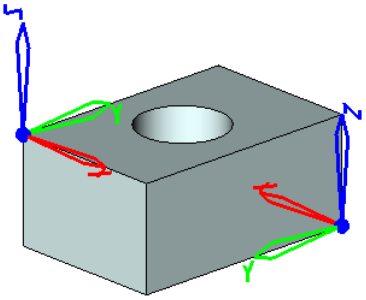
A local coordinate system (LCS) is an auxiliary 3D construction element, that allows positioning 3D objects in the space. LCS's are widely used in 3D modeling, for example, for building 3D assemblies, copying 3D bodies, defining transformations, etc.

An LCS is displayed in the 3D scene as three named arrows, indicating the directions of the axes of the given coordinate system. The point where the arrows drawing is the LCS's origin. The size of an LCS's image is controlled in the command **ST: Set Document Parameters**, on the tab **3D** > **View**, by the parameter **Size** > **LCS**.



On the basis of the LCS it is possible to create special 3D construction elements of the T-FLEX CAD — 3D connectors. 3D connectors are used for attaching fragments when creating 3D assemblies. They represent themselves the LCS of a special type which are able (besides performing the function of attaching the fragments) to pass the values of the variables of one 3D fragment into another fragment. The use of the 3D connectors significantly simplifies positioning of the parts and matching of the fragment parameters when designing the assemblies.

The image of the 3D connector in the 3D window is analogous to the image of the standard LCS, but the axes are marked with "volume" arrows.



Steps of Creating LCS

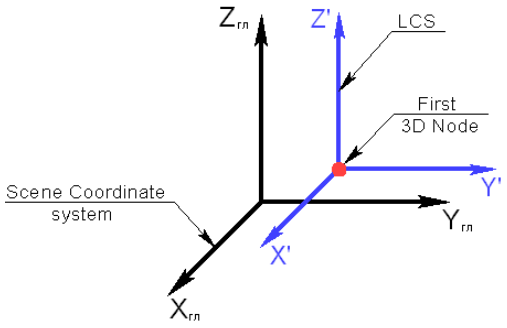
Generally, an LCS creation includes the following steps:

1. **Selecting LCS origin.** A local coordinate system is created in the selected origin point with the same axis orientation as the global coordinate system.
2. **Selecting X-axis direction**. X-axis of the LCS resulting from the previous step, is reoriented in the specified direction.
3. **Selecting Y-axis direction**. Y-axis of the LCS resulting from the previous step, is reoriented in the specified direction.
4. **Selecting surface for additional reorientation**. The LCS resulting from the previous step, is reoriented in such a way, that its X-axis becomes pointing at the nearest point on the specified surface.
5. **Selecting tangency surface**. The LCS resulting from the previous step, is reoriented to achieve tangency with the specified face/surface.
6. **Selecting required LCS transformations**. The LCS resulting from the previous step, is rotated about its axes.
7. **Specifying additional non-geometrical characteristics of the LCS** (associated elements, visibility in the assembly, etc.).

Some steps can be skipped. The only required action is selecting the origin of the LCS.

Defining the Origin of the LCS

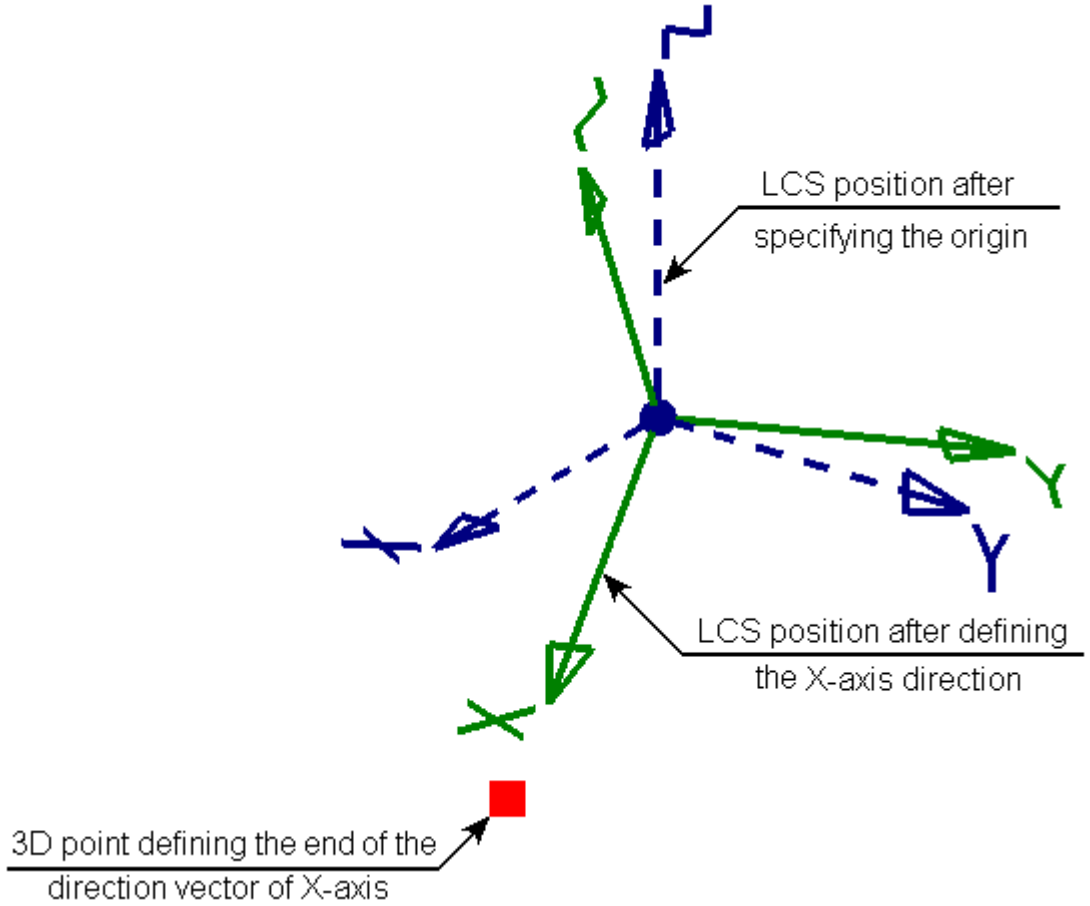
The origin of LCS is defined by a 3D point. An LCS is created in the specified 3D point with the axes orientation (X', Y', Z' on the diagram) coinciding with the orientation of the axes of the global coordinate system.



Defining X-Axis Direction of LCS

To define X-axis direction of the LCS being created, specify either the direction vector itself or its 3D end point (the first point of the vector is the 3D point defining the origin of the LCS). As a result, the coordinate system is reoriented in such a way, that the X-axis assumes the desired position.

> The rotation axis is defined as follows: a plane is constructed through the two positions of the X-axis –the old (resulting from the previous step) and the new (defined at the current step). The normal to this plane, passing through the origin of an LCS, is exactly the rotation axis.

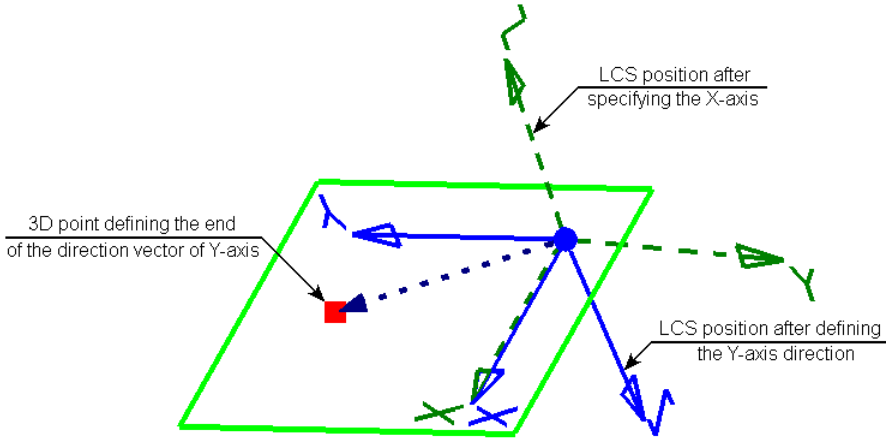


Defining Y-Axis Direction of LCS

The Y-axis direction is defined, as in the case of the X-axis, either the direction vector itself, or by its 3D end point (the first point of the vector is the 3D point defining the origin of the LCS).

> The step is performed only upon the condition of completing the previous one, which is the selection of the X-axis direction. You cannot define the Y-axis direction without specifying the X-axis direction.

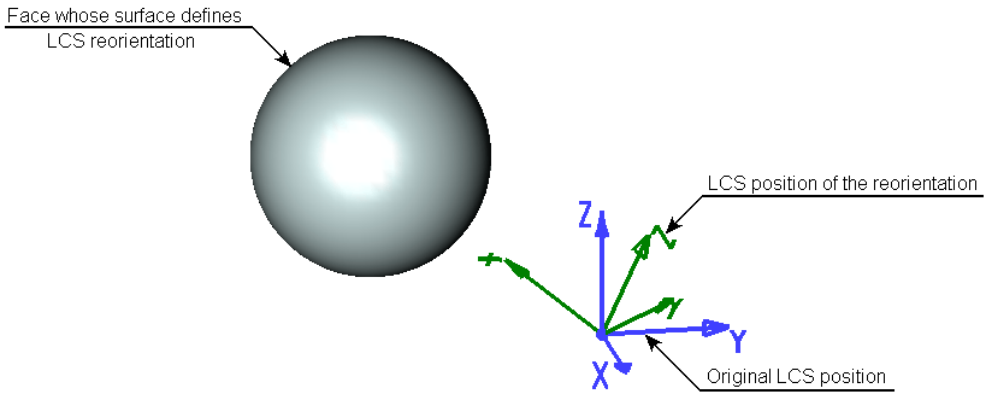
The vector of the Y-axis direction does not define the Y-axis direction, rather, the position of the plane XY, in which this axis must be located. The plane will pass through the X-axis of the LCS and the specified vector of the Y-axis direction. As a result, the X-axis of the LCS maintains its position, and the Y-axis is located in the specified plane at the angle of 90^0 to the X-axis. The direction of the Z-axis is defined automatically based on the X and Y axes.



Reorienting X-Axis of the LCS towards the Nearest Point on the Selected Surface

Reorienting the X-axis of the LCS towards the nearest point on surface is used in the case, when it is impossible to achieve the desired position of the LCS by selecting vector of the X-axis direction.

Upon selecting the reorientation surface, the LCS being created is reoriented in such a way, that its X-axis is directed towards the point of the selected surface nearest to the origin of the LCS.



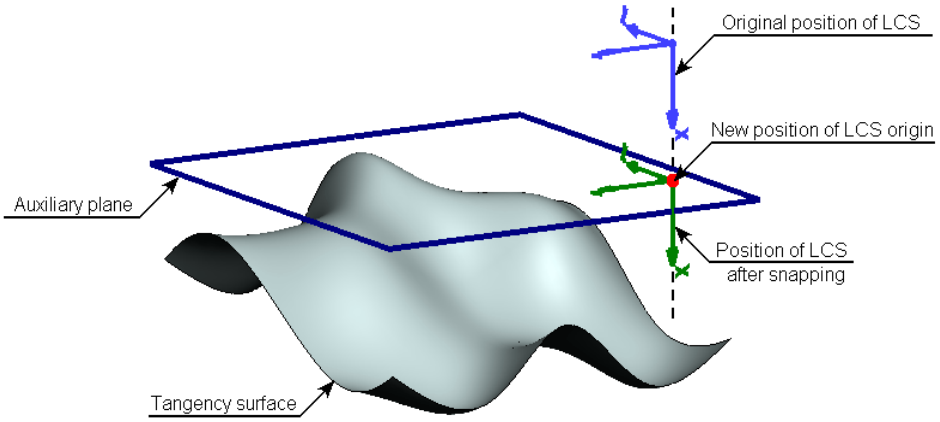
The reorientation surface can be represented by a worksurface or plane. You can also select a face, flat edge or a simple body (a body, all whose faces lie on one surface). In this case, the reorientation surface is represented by the geometrical surface underlying the selected element.

Reorienting an LCS is done similar to rotating an LCS at the time of defining the original X-axis direction.

Snapping LCS to Tangency Point on Surface

Snapping LCS to tangency point on a face allows adjusting the position of the origin of the LCS being created.

The new position of the origin of the LCS is defined as follows: an auxiliary plane is constructed parallel to the plane YZ LCS and tangent to the selected surface. If there are several such planes, the nearest to the origin of the LCS is selected. The intersection point of the auxiliary plane and X-axis will become the new origin of the LCS.



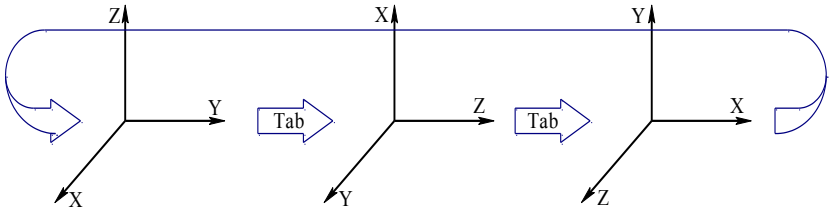
The tangency surface is defined in the same way as the reorientation surface. To do this, you can select a worksurface or plane, face, flat edge or a simple body. However, limitations are applied in this case:

If a plane is used as the tangency surface (a workplane, and underlying plane of a flat face or flat edge), it must be perpendicular to the X-axis of the LCS;

If the tangency surface is defined by a cylindrical face – the axis of the cylinder must be perpendicular to the X-axis of the LCS.

Modifying Axes Orientation of LCS

To modify axes orientation of an LCS, you need to use stepwise rotation of the LCS about any of its axes. In one pass, the LCS is reoriented by 90^0 about the selected axis. Additionally, a provision is made for cyclical rotation of the axes of an LCS. The result of the cyclical rotation is shown on the diagram.



If both ways of modifying axes orientation of an LCS are combined together (rotation about one axis and cyclical rotation), then the specified rotations about the specifics axes are performed first, and the cyclical rotation follows.

Additional Characteristics of LCS

In addition to geometrical parameters defining location and orientation of the axes of the local coordinate system, for the LCS it is possible to specify a series of additional characteristics which are necessary when using the LCS for attaching 3D fragment in the assembly.

Visibility of LCS in Assembly

Four visibility modes can be specified in the assembly drawing for the LCS being created in the 3D fragment document:

- *Invisible in assembly*. This mode is set on by default. The LCS created in the fragment document, will be not accessible in the assembly document. The LCS of the fragment will be visible neither in the 3D window, nor in the LCS list of the assembly model;
- *Visible in assembly on one level*. This LCS will be visible in the assembly model and available for snapping to it under condition that the fragment containing this LCS is a fragment of the first nesting level. The location of such coordinate system is determined by the location of the 3D fragment. This LCS can be used for attaching other 3D fragments, and also for creating any construction elements and operations of the assembly model;
- *Visible in assembly on all levels*. The LCS will be visible in the assembly document and accessible for snapping to it for any nesting level of the fragment in the assembly;
- *Externally invisible in assembly, but available for snapping to it*. This mode is used most frequently for 3D connectors operation. In this mode, by default, the LCS of the fragment is not displayed in the 3D window of the assembly model. However, upon bringing the cursor to the LCS (or to its associated elements) the LCS will be "shown" in the assembly model and it will be possible to select it for snapping. Such LCS is also displayed in the list of the local coordinate systems of the assembly (with gray color, as invisible).

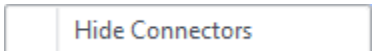
Visibility of Fragment's LCS in Assembly

The following types of LCS can be visible in the model tree of an assembly: "On one level", "On all levels", "Connector".

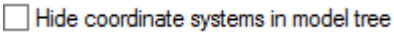
Limitations:

A) "Connector" and "On all levels" external LCS are invisible in the model tree until any other element is not snapped to it.

You can disable the option **Hide connectors** in the context menu of the model tree to show the LCS.



B) LCS will be hidden in the model tree if the option **Hide coordinate systems in model tree** is enabled in the fragment parameters.



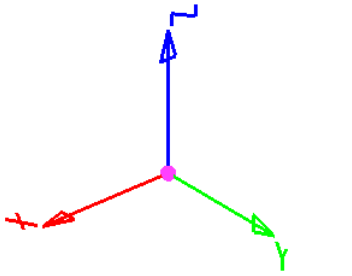
LCS for Attaching 3D Fragment

Location of the 3D fragment in the assembly gets fixed by combining two coordinate systems: source one – belonging to the fragment, and target one – located in the assembly model. The LCS created in the

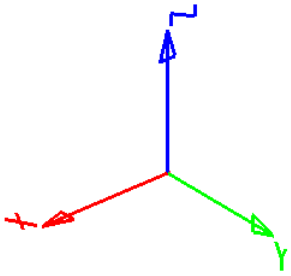
fragment document and assembly document are usually used as the source and target coordinate systems.

However, not all LCS created in the 3D fragment document can serve as a fragment source coordinate system. The only LCS that can be selected as the source coordinate system are the LCS marked as *used for fragment attachment*. This parameter is specified in the LCS properties when creating and editing the LCS.

The coordinate systems marked as used for fragment attachment will be displayed in the 3D window in a special way: a sphere is drawn at its origin. By this symbol, it can be distinguished from others.



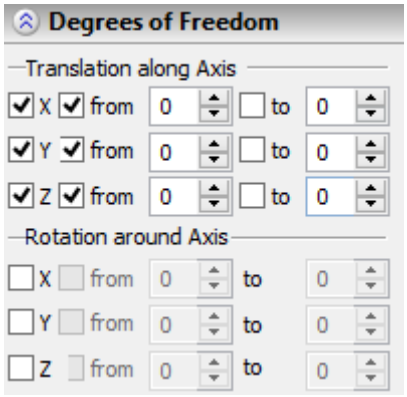
LCS with "Use for Fragment Insertion" flag On



LCS with "Use for Fragment Insertion" flag Off

Degrees of Freedom

For the LCS which will be used as the source coordinate system upon 3D fragment attachment, it is possible to specify additional conditions in the form of allowed degrees of freedom. They define behavior of the 3D fragment with respect to the fixing LCS. When inserting a 3D fragment, the conditions with degrees of freedom are copied from the LCS parameters to the 3D fragment parameters and subsequently are taken into account when calculating the set of mates of the designed 3D assembly model.

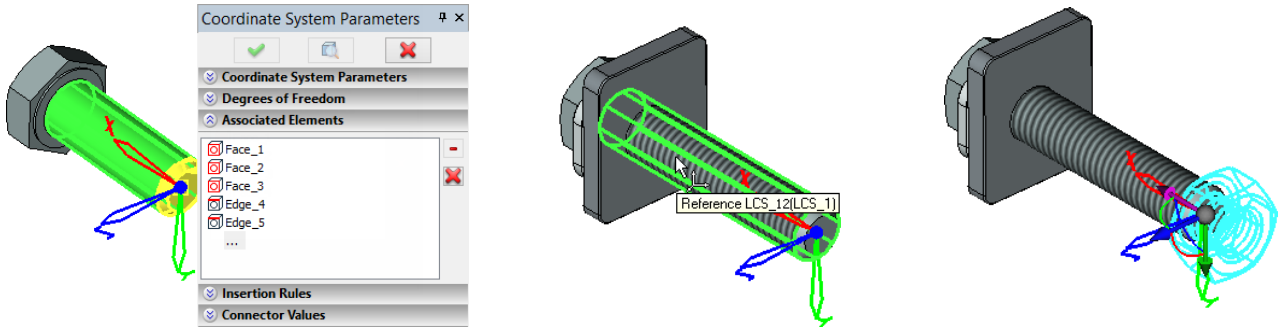


There are 6 degrees of freedom of the fragment – 3 translations and 3 rotations with respect to the LCS axes.

For each degree of freedom of the LCS, allowable bounds on the fragment displacement can be specified.

Associated Elements

For the LCS it is possible to specify associated, i.e., connected with the LCS elements (vertices, faces, edges, etc.). It will allow a user later on to perform snapping to the given LCS by indicating not the LCS itself, but the elements connected with the LCS. For example, if for the “external” LCS created in the fragment of the bolt, faces and edges of the bolt are specified as associated elements, then in the assembly when attaching a nut to this bolt, it will be sufficient to point at any face or edge of the bolt with the cursor – the LCS connected with these elements will be highlighted and selected as a result.



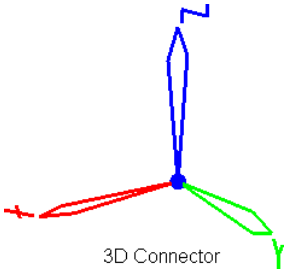
3D Fragment Insertion Rules upon Snapping to Given LCS

For LCS which is going to be used as the target LCS upon 3D fragment attachment, so-called "insertion rules" can be specified. They imply the necessity of specifying additional transformations upon insertion of the fragment with snapping to the given LCS.

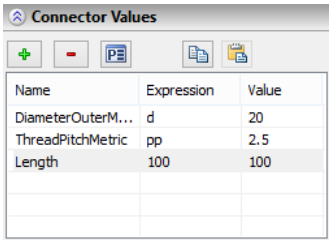
For example, when a nut is snapped to the bolt LCS, it is usually necessary to specify additional translation of the nut along the bolt axis. That is why when designing a bolt model, in the insertion rules for the bolt LCS, it is possible to indicate the necessity of translation along the X-axis.

3D Connector

3D connector represents itself the LCS for which *named values* are specified. The named values determine information (values of variables) which a connector must pass to the 3D fragments switched to this connector. For example, among the bolt fragments included in the library of the standard T-FLEX CAD elements, 3D fragments containing data about the current diameter of the bolt were created. Upon snapping the fragment-nut (from the same library) to such connector, the nut diameter will be automatically replaced with the value taken from the connector.



The image of the 3D connector differs from that of the standard LCS by the form of the axes arrows. Specifying position and direction of the 3D connectors axes is performed in the same way as upon creating the standard LCS.



The named connector values are specified in the dialog of the LCS parameters (3D connector). They can be constants (a constant is indicated as an expression for such named value) as well as variables (the name of the variable of the current document is indicated as an expression for such named value).

CREATING LCS

To create an LCS, use the command **3O: Create Local Coordinate System**:

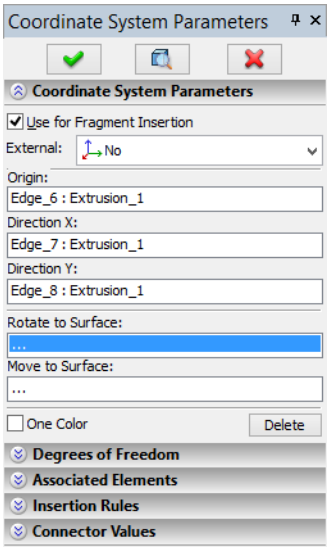
Icon	Ribbon
	3D Model→ Construct→ LCS
Keyboard	Textual Menu
<3O>	Construct > LCS

When creating an LCS, the properties window and the command automenu are used together.

To select a geometrical element defining some property of the LCS (the origin, X and Y axis direction, the reorientation surface, etc.) you can simply focus in the respective input box of the properties window. In this case, the option for defining the given parameter activates automatically in the automenu. The option combo box allows setting up filters for geometry elements selection. The opposite is also true: upon activating one of the automenu options, the input focus in the command's properties window automatically switches to the input box of the LCS parameter, corresponding to that option.

Specifying Geometric Characteristics of LCS

Note that geometry elements, defining various LCS parameters (the origin, X and Y axis direction, the reorientation surface, etc.) can be selected in arbitrary order. Those, however, will be applied in the order described earlier in this chapter.



To select a 3D point defining the origin of the LCS, focus in the input box **Origin** of the properties window (the section **LCS parameters**). The following automenu option will become active:

	<1>	Select 3D Node or vertex as LCS origin

Once the 3D point of the coordinate system origin is selected, the image of the LCS being created is displayed in the 3D window. As the rest of the characteristics are specified, the direction of the LCS axes can be modified.

To define the X-axis direction, use the entry **X axis direction** of the properties window and the automenu option:

	<2>	Select point that sets direction of X axis

The Y-axis direction of the LCS is defined in the **Y axis direction** entry of the properties window and the automenu option:

	<3>	Select point that sets direction of Y axis

The surface, whose nearest point defines the reorientation of the X-axis of the LCS, is specified in the input box **Rotate to Surface**. The respective automenu option is:

	<4>	Rotate LCS to be orthogonal to Face

To select the tangency surface, use the **Move to Surface** entry of the properties window and the automenu option:

	<5>	Offset LCS to be tangent to Face

The selected geometry elements (3D node, vertex, edge, etc.) are specified in the respective entries of the properties window. To reject an element, focus in this entry and press the button **[Delete]**. Moreover, by pressing or <Esc> it is possible to successively cancel selection of all elements.

To modify axes orientation of an LCS, use the following options:

	<A>	Rotate LCS around X axis by 90°
	<O>	Rotate LCS around Y axis by 90°
	<Z>	Rotate LCS around Z axis by 90°

The cyclical reassignment of the axes of an LCS is done by the option:

	<Tab>	Change LCS axes orientation

Additionally, in the command's properties window you can specify the parameters required when using this LCS for positioning fragments.

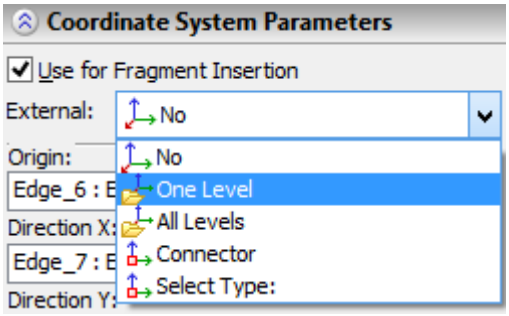
The detailed description of those parameters is provided in the section **LCS parameters** of this chapter.

Once the desired LCS is built, confirm its creation by (in the properties window or in the command automenu). To reject creation of the given LCS, use the automenu option:

	<F>	Reset target LCS

Before confirming the LCS creation with the help of in the command's properties window, the LCS characteristics (see below) necessary for snapping 3D fragments can be specified.

Choosing LCS Visibility Mode in Assembly

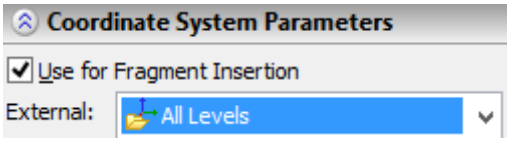


When inserting the current document into an assembly as a 3D fragment, the drop down list of the parameter **"External"** in the group **"Coordinate System Parameters"** of the command's properties window is used for selecting LCS visibility mode:

- **No**. LCS will be always invisible in assembly;
- **One level**. LCS will be visible in assembly on one level;
- **All levels**. LCS will be visible in assembly on all levels;
- **Connector**. LCS will be invisible in assembly but available for snapping to it (this option is usually used for 3D connectors).

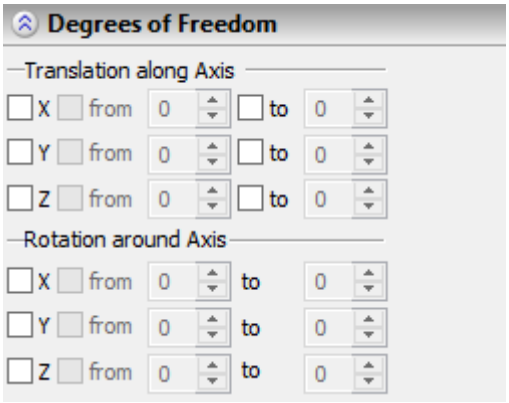
LCS as Source Coordinate System of 3D Fragment. Specifying Degrees of Freedom

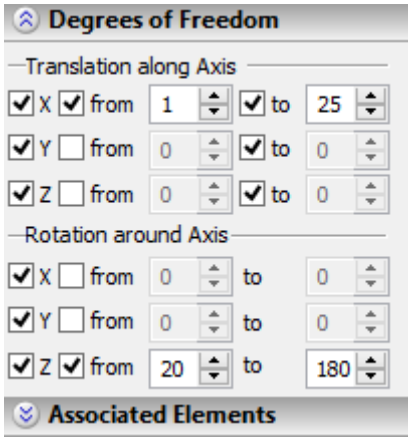
To utilize the LCS being created as the source LCS of the fragment (upon inserting the current document into an assembly as a 3D fragment), it is necessary to set on the flag **Use for Fragment Insertion** found in the command's properties window, in the group **Coordinate System Parameters**. By default, this flag is turned on.



When the flag "Use for Fragment Insertion" is on, in the group **Degrees of Freedom** it is possible to additionally indicate degrees of freedom of the 3D fragment when given LCS is used as the source fixing LCS.

There are 6 degrees of freedom for a fragment – 3 translations and 3 rotations with respect to LCS axes. By default, all degrees of freedom are forbidden – the switched off flags found next to the strings of all degrees of freedom manifest about that.





For enabling any of the degrees of freedom, it is necessary to set on the flag to the left of the corresponding string. In this case, the fragment can move along the given axis of the LCS (or rotate around the given axis of the LCS) with no restrictions.

For specifying start and/or end limits of permissible motion by allowed degrees of freedom, it is necessary to set on flags before the fields **from** and/or **to** and specify desired value in the opened entry field.

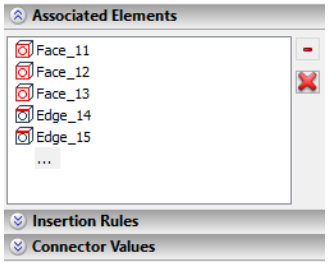
Note that for rotation around the LCS axes, two (start and end) limits of motion have to be specified at the same time. For translation along the axis, the limits can be specified independently (for example, only start, or only end limit).

Selection of Associated Elements

For specifying associated elements, the command's property window (group "**Associated Elements**") and automenu can be used.

To specify associated elements with the help of the automenu, it is necessary to turn on the option:

	<E>	Select Associated Elements



This option allows a user to select the associated elements for the LCS just by picking them in 3D window. The drop down list of the option contains filters for choosing the elements. The selected 3D elements are put into the list "Associated Elements".

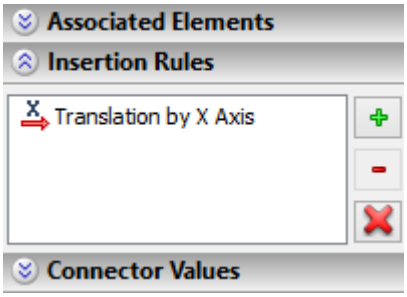
The option can be enabled just by putting the cursor over the string of the list containing "...".

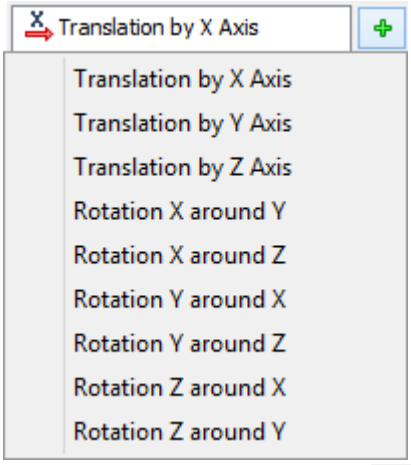
For removing selected elements, the buttons to the right of the list are used. The button allows a user to remove one element – selected in the list at the moment of pressing the button. When pressing the button , all selected associated elements of the LCS are removed at once.

Specifying 3D Fragment Insertion Rules upon Attaching to Given LCS

Additional transformations of translation/rotation, which the system automatically prompts a user to carry out when the fragment is attached to given LCS, can be specified in the group "**Insertion Rules**" found in the command's properties window.

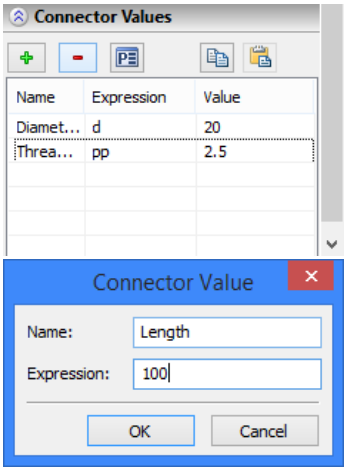
To create a new rule, press the button . Upon pressing this button, a drop down menu appears in which a desired type of transformation needs to be chosen. The selected transformation is added to the list.





For removing one of the specified transformations, choose it in the list and press the button . Pressing the button will lead to deletion of all transformations at once.

Specifying Named Values for 3D Connector



When creating 3D connector the named values are specified in the group **"Connector Values"** in the properties window for the LCS. In the list found in the given group of the properties window, the names of the named values of the connector, their expressions and current values are indicated. The buttons located above the list are used for filling in the list of the named values.

To add a new named value to the list, the button is used. After pressing this button, the dialog window "Connector Value" will appear, in which the name of the named value being created and expression assigned to it have to be specified. Both constant values as well as the variables of the current document can be used in the expression. After pressing **[OK]** the specified named value will appear in the list.

It is recommended to specify "meaningful" names for the named connector values sticking to the same name creation rules in all 3D models. This will simplify the setup of the connection between the connector values and external variables of the fragments snapped to it. For example, at first sight complicated names like "DiameterOuterMetricThread", "ThreadPitchMetric", "LengthInnerMetricThread", "LengthInnerInchThread" used in the standard T-FLEX CAD libraries, are actually made up according to the common to all libraries rules and exactly show what information is transferred by the given connector value.

Pressing the button removes the named value selected in the list at the moment of pressing the button.

By pressing the button , it is possible to rename already created named connector value or the expression specified for connector value.

The button allows a user to copy all named connector values and their expressions to the clipboard.

The button adds the lines located in the clipboard of the system to the list of the named values (if in the list there are no named values with the same names). Copying the named values via clipboard works in the framework of the same instance of T-FLEX CAD application. Thus, it is possible to copy the named values from one document to another.

LCS PARAMETERS

Parameters of an LCS are defined in the dialog box invoked by the option:

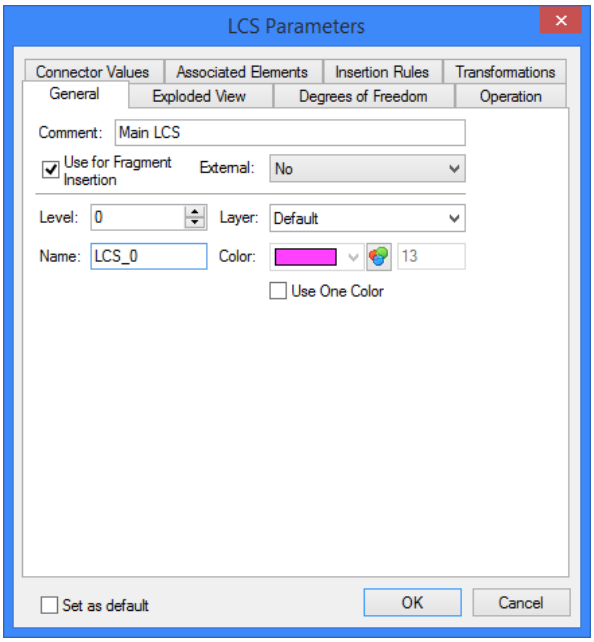
	<P>	Set entity parameters

Some parameters of this dialog are common across all 3D elements. Their detailed description is provided in the chapter **General parameters of 3D elements**.

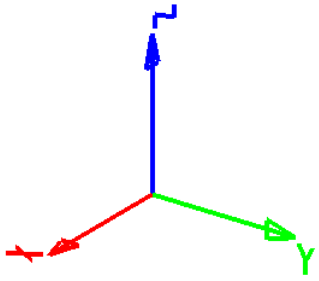
Other parameters of the given dialog duplicate parameters of the LCS properties window already described above. Thus, only those LCS parameters which were not mentioned in the previous section of this chapter will be described in this section.

You can define the following parameters of an LCS on the tab **General**:

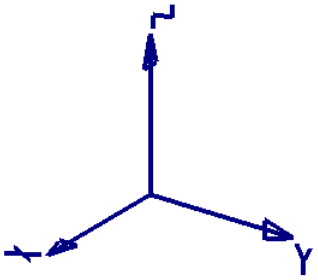
Comment. This parameter allows specifying a text string – the description of the given LCS. The comment helps selecting a coordinate system from the list of those available for positioning, when applying a 3D fragment.



Use One Color. This flag is responsible for the LCS viewing colors. When it is turned off, the LCS axes are drawn with different colors. Each axis color (red, green, or blue) corresponds to the color of the global axis. When the flag is turned on, the LCS is drawn with one color ("**Color**" parameter).

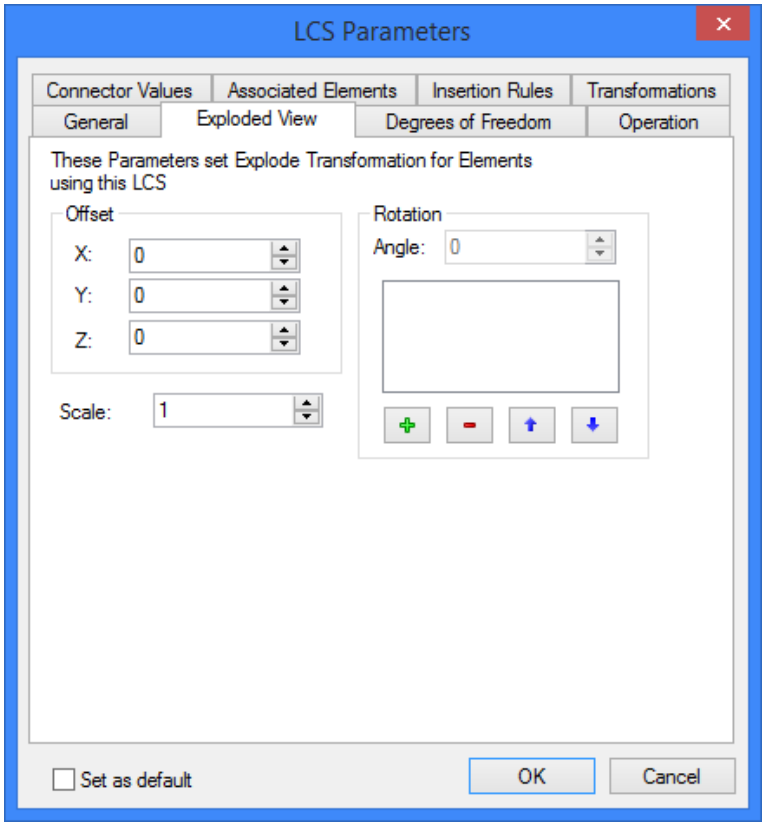


Display of LCS by different colors



Display of LCS by one color

On the tab **Exploded View** you can specify the transformations that will be applied to 3D fragments attached to the given LCS in the exploding mode (the command **Tools** > **Exploded View**).



3D PATHS

3D path is used as an auxiliary 3D element in many system 3D operations. It represents a spatial curve that may consist of segments of different types.

Any 3D path has direction, and, therefore, the start and end points. The direction of the 3D path will affect the result of using it in various operations.

A 3D path can be open or closed. The start and end points of a closed path coincide.

METHODS OF CREATING 3D PATHS

The command **3PA: Construct 3D Path** is used for creating 3D paths:

Icon	Ribbon
	3D Model → Construct → 3D Path
Keyboard	**Textual Menu**
<3PA>	Construction >3D Path

This command unites various methods of creating 3D paths. Those methods can be divided into two groups:

Creating 3D paths based on already existing 3D elements:

- 3D path based on a sequence of connected edges;
- 3D path as a 3D spline through 3D points;
- 3D path as a sequence of 3D paths;
- 3D path as a 3D path projection on a face;
- 3D path as a copy of an existing 3D path;
- 3D path as an outline edge;
- 3D path as an offset from a curve (of a 3D Path) lying on a surface;
- 3D path based on a section of a body by a plane;
- 3D path by a parametrically varying 3D node.

Creating 3D paths based on 2D elements (located on the workplanes or worksurfaces):

- 3D path by a hatch;
- 3D path by 2D paths;
- 3D path by two projections (two 2D paths).

To create an arbitrary 3D path, you need to accomplish the following steps:

1. Select the method of creating a 3D path.

Each method has its separate option within the command's automenu. Upon selecting one or another option, the automenu provides additional options corresponding to the given method of creating a 3D path.

Some methods of creating a 3D path can be automatically activated by the system if you select the source object of a certain type in the 3D scene or in the 3D model tree right after entering the command. For example, selecting a 3D node activates the mode of creating a 3D path by a 3D points; upon selecting a 3D path – the mode of creating a 3D path copy; selecting a 2D path on the active workplane sets the mode of creating a 3D path by 2D paths.

2. Select the source objects for creating a path and the required parameters (if necessary);
3. Confirm the creation by in the automenu or in the command's properties window.

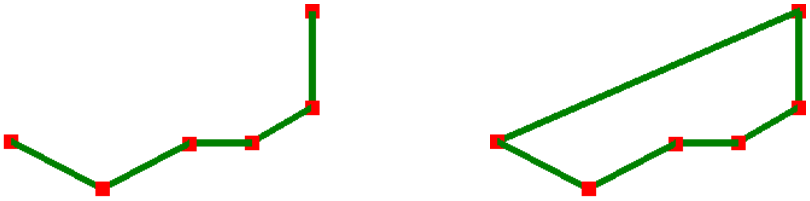
CREATING 3D PATHS BASED ON 3D ELEMENTS

Creating 3D Path as Spline through 3D Points

Splines are constructed based on a set of 3D points. The points of the set determined the spline control points. Modifying positions of the source 3D points will affect the shape of the 3D path built through these points.

There are three types of 3D splines:

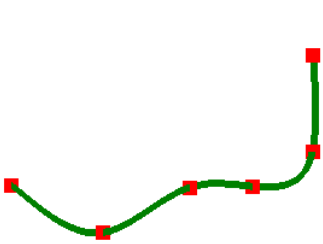
3D polyline is a spatial curve made of straight segments connecting the points of the source set. A polyline can be either open or closed (when a closed polyline is created, the first 3D point of the source set is used twice – for defining the start and the end of the path);



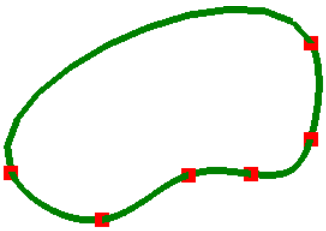
Open polyline

Closed polyline

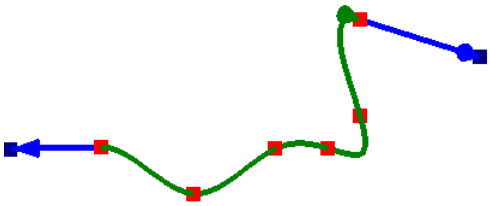
3D spline through nodes is a spline passing through all control points. Such spline can be periodic or nonperiodic. When a periodic spline is created, the first 3D point is used twice. When creating a nonperiodic 3D spline through nodes, you can additionally specify the spline direction vectors at the spline's boundaries (the start and the end);



Nonperiodic 3D spline through nodes



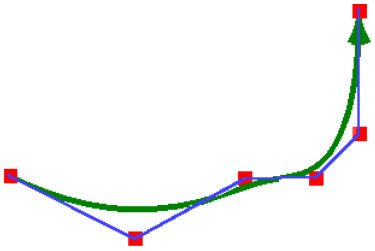
Periodic 3D spline through nodes



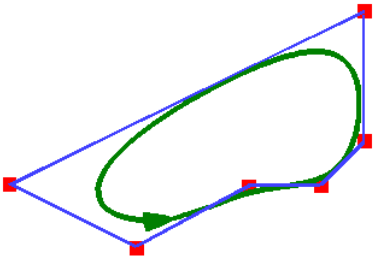
Nonperiodic 3D spline through nodes with specified directions at the bounding points

3D spline by polyline (a NURBS – non-uniform rational B-spline) – an arbitrary-degree spline by a control polyline. The control points are used for building the control polyline The spline itself does not pass through them (except the first and the last point in the case of an open spline). The spline's parameters allow specifying the degree, weights at control points, parameterization type and positions of the parameterization nodes.

Just like the previous case, the 3D spline by polyline can be periodic or nonperiodic.



Nonperiodic 3D spline by polyline



Periodic 3D spline by polyline

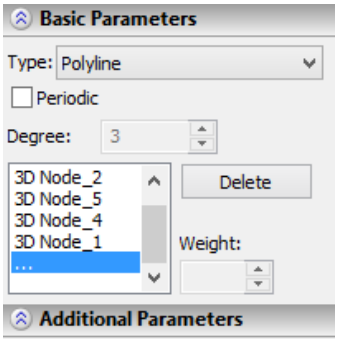
To create a 3D path as a 3D spline, in the command's automenu select the option:

	<S>	Construct 3D Path as spline based on 3D Nodes

A 3D spline creation usually begins with specifying a set of 3D points through which the spline will be constructed. The following option is used for selecting 3D points:

	<M>	Add 3D Point to Spline

The selected 3D points are added to the list of the spline control points in the properties window. To exclude one of the points, pick it in the list and click **[Delete]**.



The type of the spline being created – periodic or nonperiodic (closed or open in the case of a polyline) – is also determined by means of the properties window (the check box **Periodic**). When the check is cleared, a nonperiodic spline is created (or an open polyline). When the flag is set, the spline will be periodic (closed in the polyline case).

The further steps of the spline creation depend on its type. The spline type selection is also done in the command's properties window (the parameter **Type**). The appearance of the properties window and the automenu changes according to the selected type.

The 3D path as a polyline does not use any more parameters. Its creation is complete at this point.

When creating a 3D path as a spline through nodes, the following pairs of options are available in the automenu for defining spline directions at the bounding points:

	<L>	Set direction at Spline start
	<R>	Set direction at Spline end

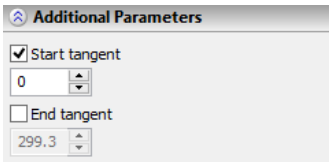
When selecting the option or , the automenu brings the options for selecting the second point of the direction vector or the vector itself at the corresponding bounding point:

	<N>	Set direction at Spline start by 3D Point
	<D>	Set direction at Spline start by Vector
	<N>	Set direction at Spline end by 3D Point
	<D>	Set direction at Spline end by Vector

To define the second point of the direction vector, select an additional 3D point. The direction vector will be pointing from the respective bounding point to the specified 3D point. To specify the direction vector explicitly, you need to select a 3D element suitable for defining a vector in the space. The direction vector is created by the parallel translation of the selected vector to the spline's bounding point.

The length of each directional vector affects the shape of the spline being created - the longer is the vector, the longer portion of the curve is affected by this vector. The vector length can be determined automatically (based on its defining 3D elements) or be manually set by the user. By default, the vector length is taken from its defining 3D elements. For example, when defining the direction vector by a 3D point, the vector length will be equal to the distance between the specified 3D point in the respective bounding point of the spline.

To manually specify the length of the direction vector at the start or at the end of the spline, you need:

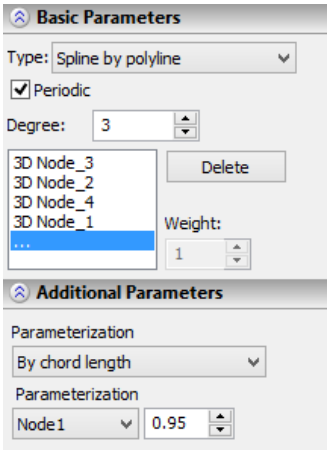


1. In the group **Additional parameters** of the command's properties window set the appropriate flag: **Start tangent** or **End tangent**".
2. In the input box becoming accessible below, enter the desired vector length.

To abandon the selected direction vector at the start or end of the spline, you can use the automenu options:

	<K>	Clear direction at Spline start
	<F>	Clear direction at Spline end

When creating a 3D spline by polyline, the shape of the curve being created is determined by the spline parameters in the command's properties window:



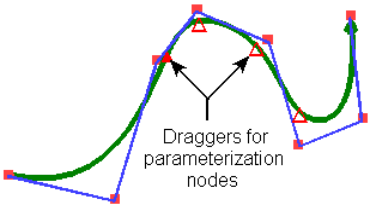
Weight coefficients of all spline's base 3D points. The coefficient must be greater than or equal to 0. The more is the weight of a point, the closer to it the spline will pass. The default value set for all control points of the spline is "1". To modify the value of the weight of the particular point, select it in the list and enter the desired value in the **Weight** parameter's input box.

Degree of polynomials of the piecewise-continuous function, describing the spline being created is the **Degree** parameter. The maximum allowed value of the degree is the number of the spline control points minus one. The default degree set for a spline with at least four control points is equal to 3.

The higher is the spline degree, the more "rigid" it becomes. That means, the spline barely reacts to moving a particular control point. The low-degree splines pass closer to the control points, and changes in their positions stronger affect the spline shape.

Parametrization type (**By Chord length**, **Uniform**, **Centripetal** and **Manually**). In the case of the uniform parameterization, the shape of the curve does not adapt to the length of the control polyline segments, which may cause cusps and self-intersections on small segments. The two other types of parameterization ("by Chord length" and "Centripetal") this does not happen, while different lengths of the polyline segments make different contributions. The effect of the parameterization type on the shape of the curve being created is nonlinear and may differ case-by-case.

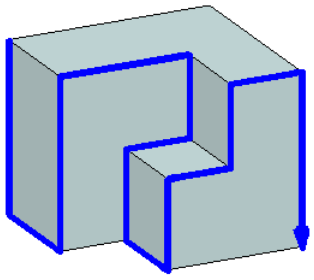
In the case of the manual parameterization, the user can change the shape of the curve by manually changing positions of the spline parameterization nodes. The most convenient way is to do this directly in the 3D scene by the draggers defining positions of each parameterization node.



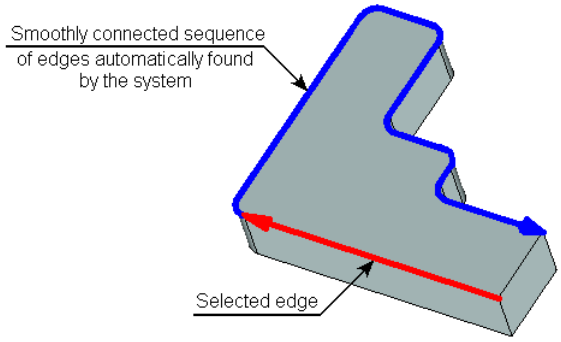
3D Path by Connected Edges

A 3D path is created by a set of edges of an operation body, 3D profile or another path. The set is defined by subsequently selecting its member edges. The selected edges must be connected with each other, forming a continuous chain. A set consisting of a single edge is also allowed.

A connected sequence of smoothly joined edges can be found by the system automatically upon selecting one of its member edges.



3D Path by connected edges



This method of creating a 3D path can be set by the following automenu option provided in the command:

	<E>	Construct 3D Path using edge sequence

Upon turning on this mode, the following auxiliary option activates automatically in the automenu:

	<E>	Select Edge

This option allows selecting all necessary edges by subsequently picking them in the 3D scene by .

For automatic selection of editing connected sequence of smoothly joined edges, you need to activate another option:

	<S>	Smooth edge chain selection mode

Upon activating this option, simply select any edge that belongs to the intended sequence. The rest of the edges will be selected automatically.

To exclude several consequent selected edges starting from the last one, use the option:

	<D>	Delete last Edge from 3D Path

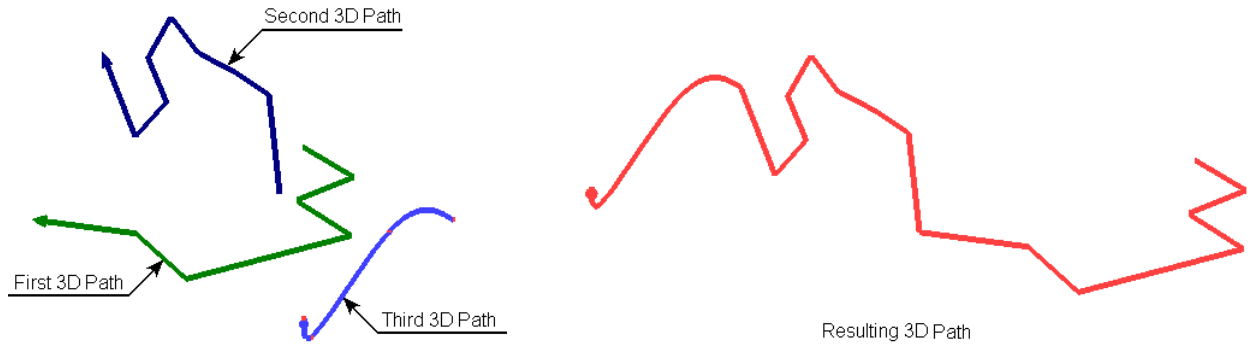
To cancel selection of all edges at once, use the option:

	<K>	Cancel selection

3D Path as a Sequence of 3D Paths

This method allows creating a 3D path by combining an arbitrary number of 3D paths. Any paths are allowed as the source, except closed ones.

The resulting 3D path is created as follows: the start of the second path is carried over to the end of the first selected 3D path by parallel translation, the start of the third one is carried over the end of the second, and so on. Each subsequent path is connected to the end of the previous one.



To create a 3D path in this way, select the option:

	<T>	Construct 3D Path using 3D Path sequence

Selection of 3D paths is done in the 3D scene (by) or in the 3D model tree with the activated option:

	<T>	Select 3D Path

The last 3D path selection can be canceled by the option:

	<D>	Delete last 3D Path

3D Path as a 3D Path Projection on a Face or Operation Body

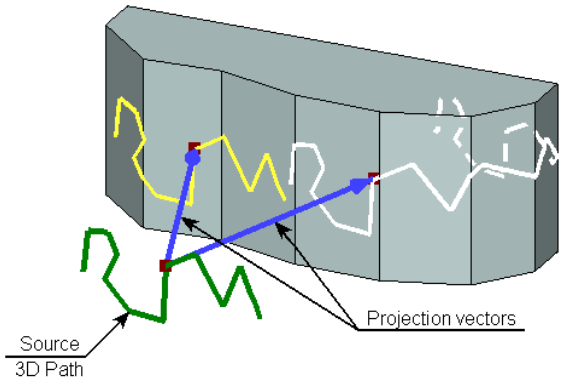
A 3D path can be created as a projection of another already existing 3D path on all faces of an operation body or on a separate face. In the case of mapping on all faces, select the operation body itself, on an isolated face – a specific face.

Projection on a face is constructed as the intersection line between the selected face and a surface created by extruding the source path in the direction of the projection vector. When mapping on an operation body, then projections of the path are created on each face. The resulting elements (that is, the results of mapping the path on each face) are combined in one path. The elements that cannot be connected are discarded.

> When creating a 3D path as a projection, only those pieces of the 3D path are used that fully map on the selected face or body.
>
> When creating a 3D path, the first segment of the source 3D path must be among the resulting mappings.

The vector of mapping (projecting) the source path is defined either with two 3D points, or using some 3D object suitable to define a straight line (this could be, for example, a straight path, an edge, an LCS). If the projection vector is not specified, then:

- When projecting on an isolated face, the normal to that face will be used.
- When projecting on all faces of an operation body, each point of the source path is extruded towards a point on the selected body which is nearest to the former point.



Two 3D Paths - the projectionsof the same source Path on a body's faces.
Blue arrows (not shown in the 3D scene) mark the direction of projection vectors defined by 3D Nodes.

The dashed line indicates the continuation of the projection on the back side of the body.

To create a 3D path as a projection of another 3D path, select the option:

	<J>	Create 3D Path as projection on Face or on Body

The 3D path creation begins with selecting a body or face on which the path will be mapped. To select those, use the following options:

	<B>	Select operation on which 3D path will be mapped
	<F>	Select face that will be used for 3D Path projection

The face or operation body are selected in the 3D scene by . An operation body can also be selected in the 3D model tree by picking the intended operation in it.

The source 3D path is selected by the option:

	<T>	Select 3D Path for projecting onto face

The projection vector can be defined by two 3D points with the help of the options:

	<M>	Select first 3D point of projection direction vector
	<S>	Select second 3D point of projection direction vector

The projecting vector can be defined by a single object using the option:

	<L>	Select projection direction

To undo a specified direction, use the option:

	<K>	Cancel selection of projection direction

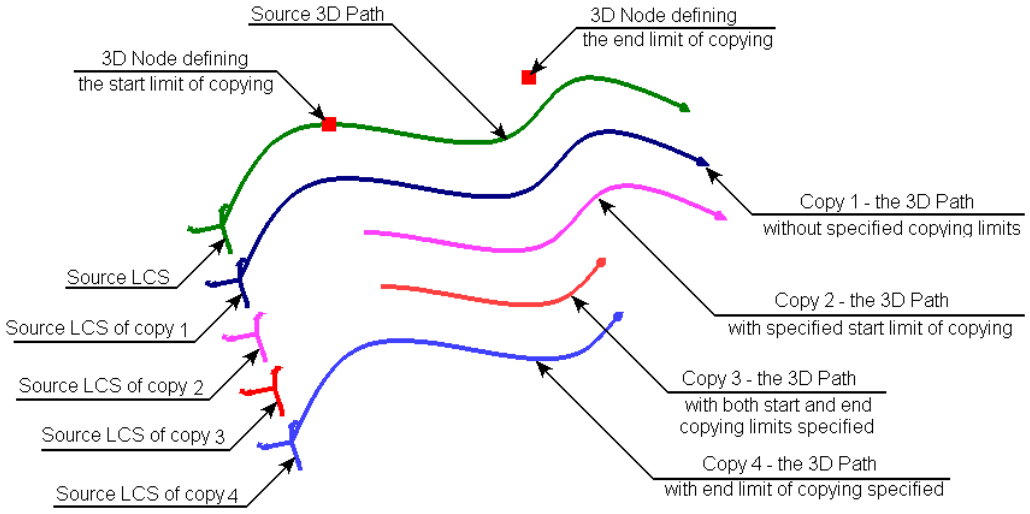
Creating Copy of 3D Path

To create a 3D path as a copy of an existing path, simply select the source 3D path. The new path will coincide in the space with the source 3D path. If any transformations are specified among parameters of the created path, those will be performed with respect to the global coordinate system. Additionally, one

can specify the source in the target coordinate systems. As a result, copying will be done in the direction from the source to the target coordinate system. The transformations defined for the path in this case will be performed with respect to the specified transformation location.

The resulting path could be a copy of just a portion of the source 3D path. To do this, specify the *start* and *end limits for copying*: two 3D points on the source path defining the portion to be copied. If the selected 3D points do not lie on the source path, the boundaries of the portion to be copied are determined by the points on the path that are nearest to the specified points.

You can also specify just one copying boundary. In this case, the opposite end of the path will be used as the second boundary.



To create a copy of a 3D path, use the following option in the main command's automenu:

	<M>	Create Copy

Creation of a path copy begins with selecting the source 3D path. A 3D path is selected either directly in the 3D scene or in the 3D model tree. For this, use the automenu option:

	<T>	Select 3D Path

To select the source and the target coordinate systems, the following options are provided:

	<1>	Select source LCS
	<2>	Select target LCS

If the source or target coordinate system doesn't exist, it can be quickly created using the following options:

	<F>	Select Point for creating source LCS
	<W>	Select Point for creating target LCS

As a result, LCS will be automatically created with the vertices at the specified 3D points and the same orientation of the axes as that of the global coordinate system.

To cancel a selection of the source or target coordinate system, you can use the option:

	<F>	Cancel LCS selection

The copying limits are defined by the following options:

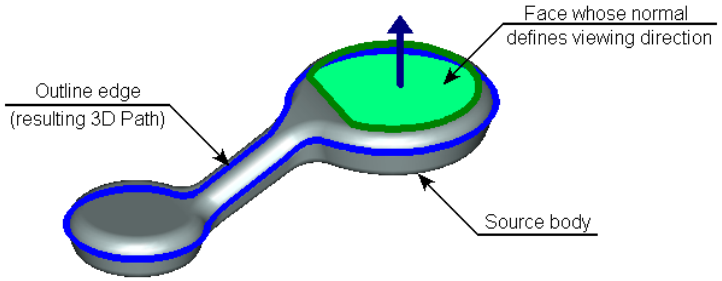
	<S>	Set starting copying limit
	<E>	Set ending copying limit
	<D>	Clear copying limits

3D Path as an Outline Edge

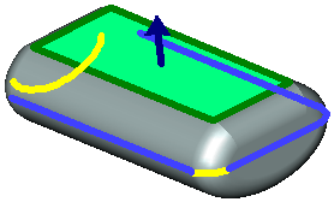
Creating a 3D Path as an *outline edge* can be found useful for defining the separation line when designing diecast molds.

Construction of a body's outline edge can be schematically described as follows: imagine that spatial lines are drawn parallel to the viewing direction through each point on the body's surface. The points on the body surface in which such a line just touches the body (without intersecting it) are the candidate points for an outline edge.

To create a 3D path, the source body and the viewing direction are specified. The system analyzes the surface of the body, searching for points satisfying the outline edge definition. The resulting set of points is united in one curve – the 3D path.

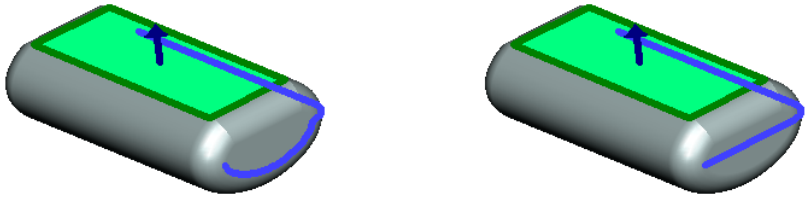


If it is impossible to join all found points into one curve, the system returns a set of separate outline edge segments. The user is offered to select the segment that will be used for building the resulting 3D path. One can select multiple outline edge segments simultaneously. In this case, multiple 3D path will be created rather than a single one, according to the number of selected segments.



Viewing direction is defined by normal to upper face System-identified outline edge segments are shown as yellow and blue lines.

A situation as possible when the vector of the viewing direction is parallel to some of the body's faces. In this case, all points of such face satisfy the definition of the outline edge, that is, the system finds an outline "surface" instead of an edge. This situation is resolved by the system by constructing the outline edge through the first points encountered on such face. As a result, most often, the outline edge is constructed along one of the side edges of such a face. A better shape of the outline edge will be obtained by applying the *"vertical" face processing*. In this mode, the system draws the outline edge straight across the face.



Constructing outline edge without "vertical" face processing

Constructing outline edge with "vertical" face processing

The mode of processing "Vertical" faces is turned on in the command's properties window.

To create an outline edge, in the command's main automenu select the following option:

	<O>	Construct 3D Path as an outline

The body whose outline edge will be constructed is selected by the option:

	<1>	Select Solid Body

The viewing direction is defined either by two 3D points or by one 3D element suitable for defining a vector:

	<2>	Select First point of Direction
	<3>	Select Second point of Direction
	<4>	Select direction

After specifying the source body and the viewing direction, the image of the created outline edge will be shown in the 3D scene.

In the case when the system cannot create a single outline edge, the 3D scene will show one of the identified outline edge segments. To create a 3D path based on it, simply confirm the path creation by .

The full list of found outline edge segments is displayed in the command's properties window. To create a 3D path based on another outline edge segment, it is possible to select the desired segment in the properties window or use the additional option:

	<Tab>	Change outline

This option allows us to loop through the outline edge segments found by the system (when pressing this option the next segment of the outline edge is displayed on the screen). The segment selected in the list is displayed with a *light blue* color in 3D scene.

To create a 3D path by several segments of the outline edge, it is necessary to find all required segments in the list and checkmark them, and then confirm the path creation with the help of .

Creating Offset to 3D Path

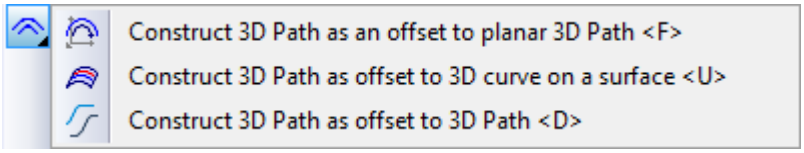
This way of creating a 3D path allows creating 3D curves of the following types:

1. 3D path as an offset from a flat 3D path
2. 3D path as an offset from a 3D curve on a surface
3. 3D path as a 3D offset from a 3D path.

All the methods of creating the 3D path as an offset from another 3D path are united in the automenu under the option:

		Construct 3D Path as offset to existing 3D Path

The drop-down list of this option contains three choices of creating a 3D path as an offset.



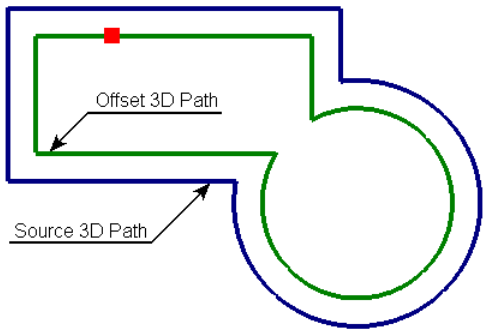
3D path as offset to a planar 3D path

To create a path - an offset to a planar 3D path, select the source path and specify the shift with respect to the source 3D path. The shift can be defined in two ways:

By specifying a 3D node through which the offset should pass. The node must lie in the plane of the source path.

By a numerical value of the shift of the offset path from the source path.

The figure presents an example of creating an offset path with the shift defined by a 3D node.



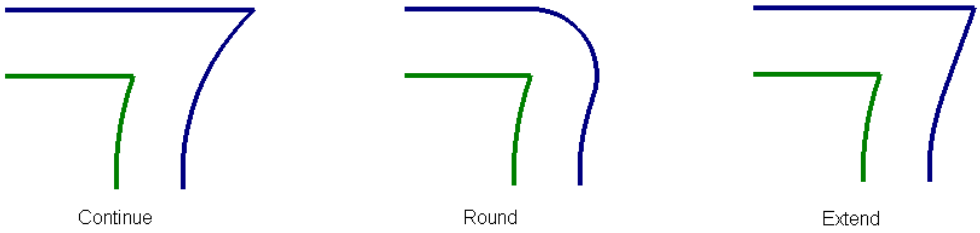
> Creating this kind of a path is similar to creating a 3D profile as an offset to another planar 3D profile (see the chapter "3D profiles").

The gaps introduced between the segments of the offset path can be processed in three ways:

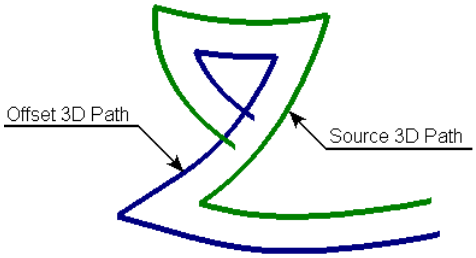
Continue – the curves making up the source path are continued. Straight lines are continued straight, arcs are rounded up.

Fillet – a fillet is created between the end points of the neighboring segments of the path being created.

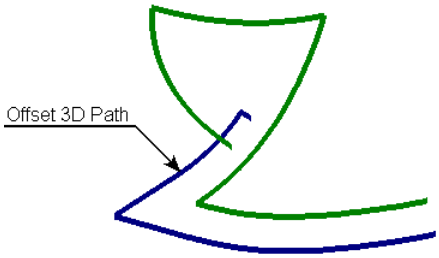
Extend – straight lines are added to the hanging points that are tangent to the path segments at the gaps.



When creating an offset to a path of the complex shape, self intersection loops may occur in the contour of the new path. An additional *loop trimming mode* helps removing those automatically.



Creating 3D Path - an offset to planar 3D Path without trimming loops



Creating 3D Path - an offset to planar 3D Path with trimming loops

To create a 3D path as an offset to an existing planar 3D path, use the option:

	<F>	Construct 3D Path as offset to planar 3D Path

The first step in creating the 3D path is selecting the source planar 3D path by the option:

	<T>	Select planar 3D Path

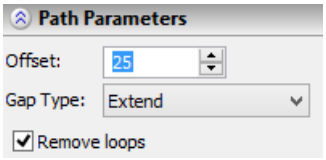
After that, one has to specify the required shift by a 3D node or numerical value. The 3D node is selected by the option:

	<N>	Select 3D Vertex in the plane of selected path

To reject a selected node, you can use the option:

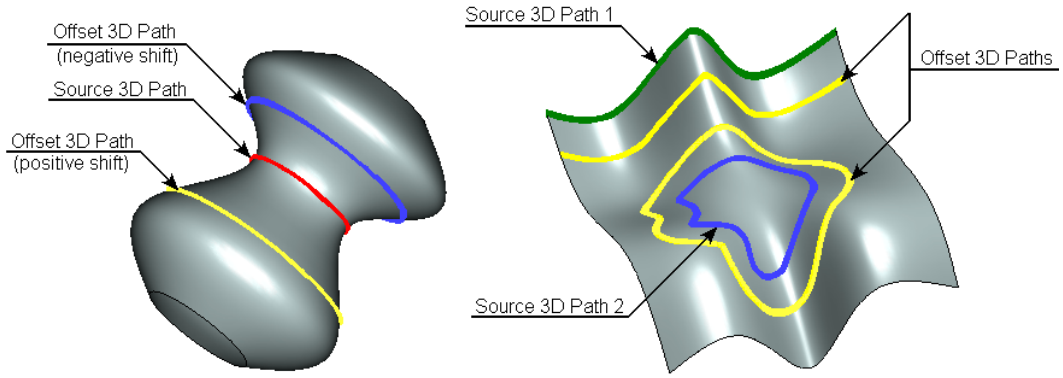
	<K>	Cancel node selection

The numerical value of the shift is specified in the command's properties window. Here, other offset parameters are defined as well. The option of processing gaps introduced by creating the offset path is defined by the parameter **Gap Type**. The mode of deleting loops turns on upon setting the flag **Remove loops**.

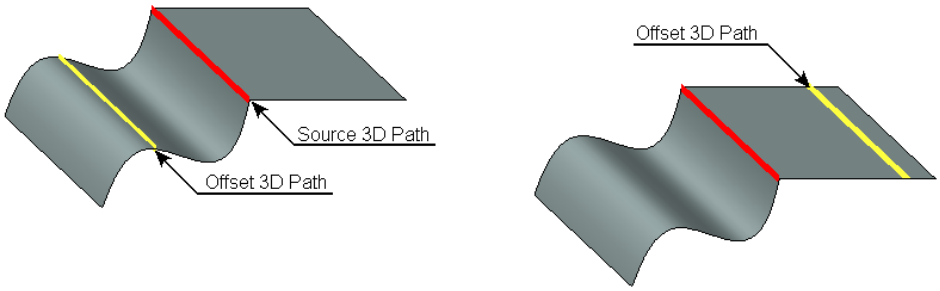


3D path as offset to 3D curve lying on surface

To create a 3D path as an offset to a 3D curve on a surface, select the source curve – a 3D path. The offset will be constructed by shifting the points of the original curve by the specified distance along the surface of the face on which the source curve lies. The source 3D path must necessarily lie on a face of a body – either solid or sheet.

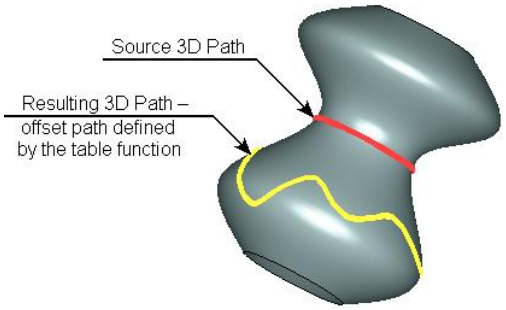


The face on which the selected 3D path lies is defined by the system automatically. Whenever possible, the user is offered to change the system selection (for example, if the 3D path was created on an edge dividing two faces of a body).



Switching face. The source 3D Path is constructed on an edge and belongs to two faces of a sheet body.

Shifting the offset from the source curve can be defined by a numerical value or by a tabulated function. The table definition of the shift distance allows creating an offset with variable shift from the source curve. The shift table determines the position of control points on the source path and their respective shift values.



The accuracy of constructing the resulting 3D path is defined either by the *number of points*, through which the path is created, or by the *offset tolerance*. When using *offset tolerance*, the path is constructed in such a way that the resulting curve-to-chord distance for any given pair of sequential points does not exceed the specified accuracy value. The more use the number of points set for the offset or the higher use the specified accuracy, the more precisely the offset path will be constructed.

To create a 3D path as an offset to 3D curve on a surface, use the option:

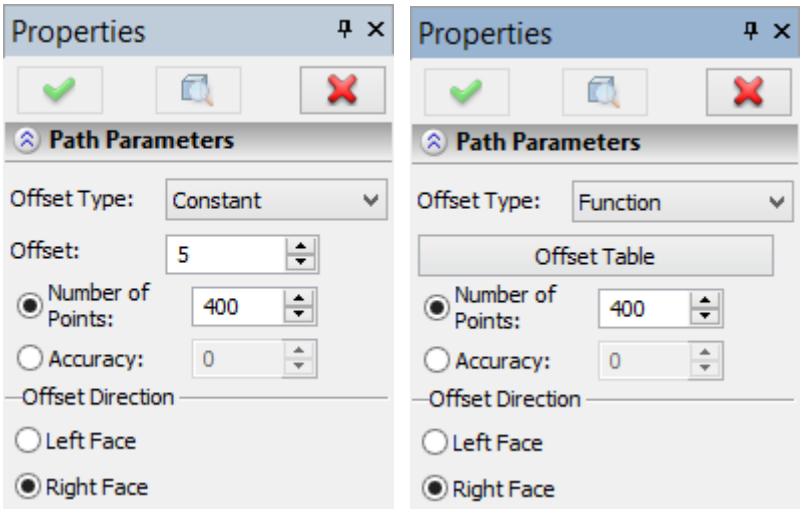
	<D>	Construct 3D Path as offset to 3D curve on a surface

Upon activating this mode, select the source 3D path by the option:

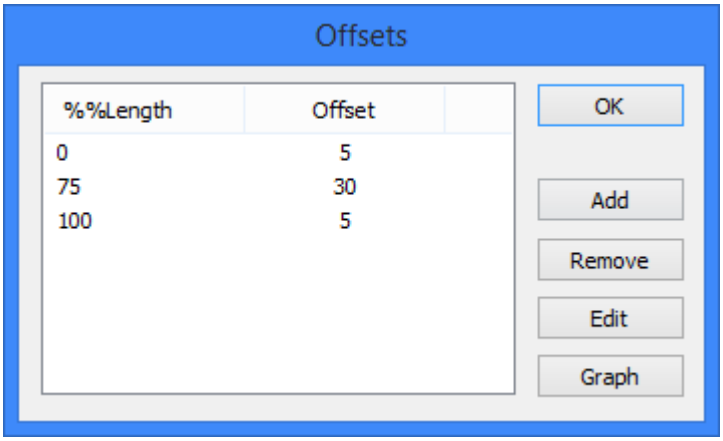
	<Z>	Select path on surface

The offset of an equidistant and its construction accuracy are defined in the properties window or in the path parameters dialog (the option).

To define an offset by a numerical value, set **Offset Type** to **Constant** and enter the desired value in the **Offset** field. To use a tabulated offset function, set the **Offset Type** parameter to the value **Function**. The **[Offset Table]** button appears in the dialog that brings up the window to define the table of offsets.

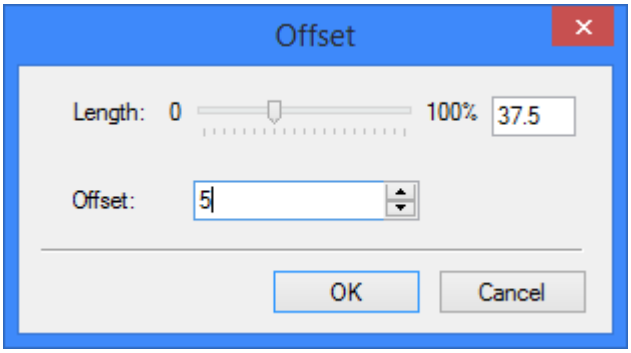


The **Offsets** dialog allows creating or editing the table of shifts:



To add a new row, set the cursor on one of the existing rows in the table and click **[Add]**. The **Offset** dialog will appear on the screen. In it, specify the position of the new control point (the parameter **Length**) and the amount of its offset (the parameter **Offset**).

The **Offset** dialog is also used for editing an existing row in the table (by clicking the button **[Edit]**).



The button **[Delete]** allows deleting the selected row from the table of shifts.

The table of offsets can be also defined using a plot (graph). To do this, use the button **[Graph]**. Upon clicking this button, the window to edit a plot appears. Details about working with plots are described in the chapter "Graphs".

To define the accuracy of constructing the resulting path, decide on the parameter to use for this purpose – **Number of Points** or **Tolerance** – and enter the value of the chosen parameter in the field at the right.

The face on which the offset is being created can be replaced by the option:

	<Tab>	Select other adjacent Face

The **Offset Direction** switch in the offset properties dialog can be used for the same purpose.

3D path as 3D offset to a 3D path

To create a 3D path as a 3D offset, all you need is just to select the source 3D path (of any shape), the offset distance and *offsetting direction*. The offsetting direction will define the direction of offsetting each

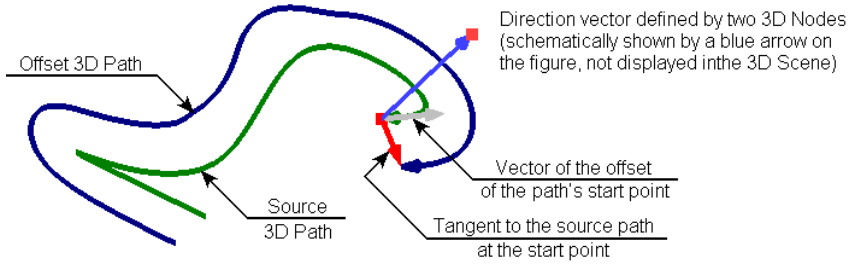
point of the source path. A 3D spline will then be constructed, uniting all such points created by offsetting.

A 3D path created as a 3D offset can be used, for example, in CAM processing for creating tool movement trajectories accounting for the tool radius correction; also, in diecast mold design for creating the separation surface (using the outline edge and an offset to it).

There are two ways of defining the offset direction:

Direction vector. A single direction vector is defined for the whole offset – by two 3D points, a straight edge, a normal to a flat face or workplane.

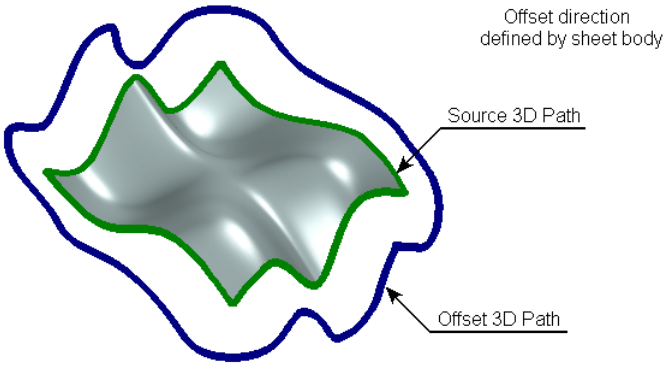
To yield the offset, each point of the source 3D path is shifted by the specified distance along the vector defined as the cross product of the tangent to the path at the given point and the specified offset direction vector. That means, a point being offset is shifted in the direction perpendicular to both the direction vector and the tangent to the source path.



Direction from body. This way of creating a 3D offset implies creation of a 3D Path as a "continuation" of a sheet body surface (tangentially). The offsetting direction is defined by the operation body, whose surface needs to be extended. The source 3D path must lie on the same body.

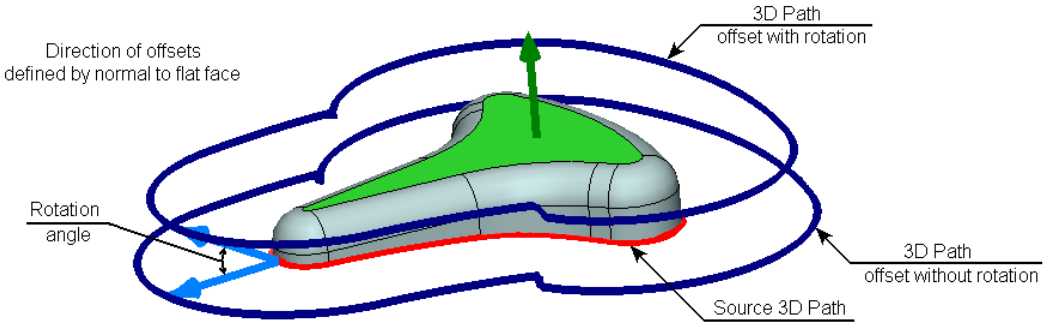
We recommend using a sheet body that lies in one surface, while the source path – a 3D path constructed along the bounding edges of this sheet body. Use of other configurations of the source data leads to an undetermined result of the offset creation.

To make the offset, each point of the source 3D path is shifted by the specified distance along the vector defined as the cross product of the path tangent and the normal to the guiding surface at this point.



An additional provision of *offset with rotation* can be used together with any method of creating an offset. In this mode, the offset vector calculated for each point of the source path is additionally rotated

about the path tangent at that point by the specified angle. The 3D offset with rotation can be useful, for example, in CAM operations – for constructing a guide path defining the tool inclination angle.



To create a 3D path as a 3D offset, use the option:

	<D>	Construct 3D Path as offset to 3D Path

The first step in creating the 3D path is selection of the source 3D path by the option:

	<A>	Select 3D Path

After that, you need to define the offsetting direction.

To define the direction by the direction vector, the following options are used:

	<B>	Select first Point of Direction
	<C>	Select second Point of Direction
	<D>	Select Direction

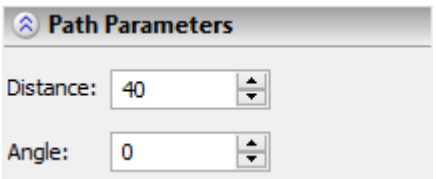
The first two options (and) allow defining the direction vector by two 3D points. The option serves to select a 3D object suitable for defining the direction vector at once.

To define the offset direction from a body, use the option:

	<G>	Select Solid Body

The offset amount is specified in the command properties window or in the path parameters dialog (the option).

The shift of the offset is specified in the **Distance** parameter.

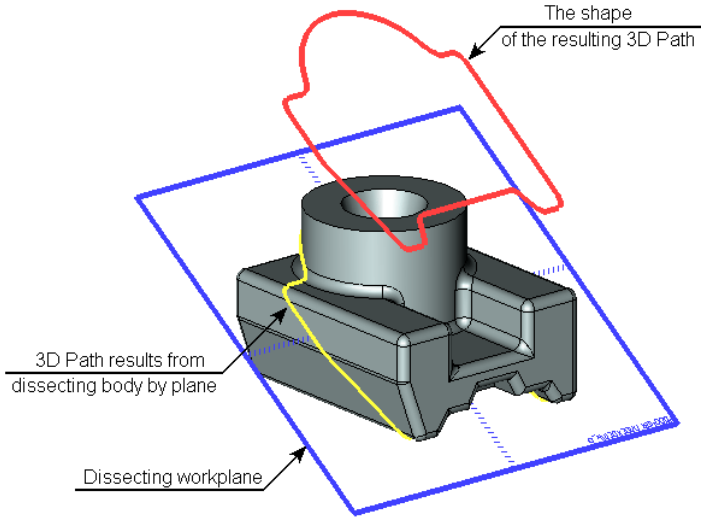


Here, the offset with rotation mode can also be turned on (if necessary). For this, simply enter the desired value of the rotation angle in the field of the **Angle** parameter (the default value is set to "0").

Creating 3D Path Based on Body Section by Plane

To create the 3D path, you need to specify the dissecting workplane and the source body (operation). The path will be created along the intersection contour between the body and the workplane.

> The 3D path is constructed only if a single contour results from dissecting the selected body by the plane. If the source body is solid, the contour will be closed. Dissecting a sheet body results in an open contour.



This mode is initiated by the option:

	<B>	Create 3D Path as intersection of Body and Workplane

The command's automenu provides the options for selecting the dissecting plane and the operation body:

	<1>	Select Section Plane
	<2>	Select Operation

Creating 3D Path by Parametric Variation of 3D Point

The way of creating a 3D path by parametric variation of a 3D point is used for creating trajectories of complex geometry.

A spatial curve of the path results from “sweeping” a 3D point along an arbitrary or specifically defined spatial trajectory. “Sweeping” of the point is realized by multiple copying of the source 3D point. The resulting set of points is joined by a spline, and a 3D path is built. The resulting 3D path can be closed.

> When constructing such path, a general mechanism is used that provides the final result based on parametrical copying of the source element. This mechanism is used in the “By Parameters” operation – when creating a body based on “sweeping” (copying) a profile. It is also used in the

"Parametric Array" operation, where the result of the operation is a set of parametrically varying copies of the source object.

This algorithm is described in most details in the chapter "By Parameters". The current chapter provides only a brief description and some examples of using this mechanism.

To create a 3D path by a parametrically varying point, you need to define the law, according to which the copies of the source 3D point will be constructed.

A so-called **current copy's coordinate system** is calculated for each copy of the point. Copying the 3D point is done in the direction from the source coordinate system to the calculated target coordinate system of the current copy. The user can select one of the local coordinate systems available in the model as the source coordinates. If no local coordinate system is selected, then the source coordinate system will be either the global coordinate system or the coordinate system of the first copy, depending on the method of calculating the current copy's coordinate system.

The current copy's coordinate system (that is, its origin and axes orientation) can be defined by one of two methods:

- *Free copy orientation*. The laws of modifying the coordinate system parameters of the current copy of the 3D point are described by expressions.
- *Copy orientation by 3D curves or surfaces*. To define the spatial position of the current copy's coordinate system, existing 3D curves or surfaces of other bodies are used. By directing the copy's coordinate system, the user controls the position of each copy of the 3D point.

Directing the copy's coordinate system is done via the operation parameters. There are parameters of two types - **driven parameters** and **driving parameters**.

Driven parameters are the parameters whose values are calculated automatically in the process of creating a copy of the point. You can assign independent variables to read the values of such a parameter.

Driving parameters are the entries with the dependencies defined by the user. The entries of the driving variables allow entering arbitrary expressions that can include any system variable. For example, by including a variable representing the ID number of the 3D point's copy in such an expression will make the value of this parameter dependent on the copy number.

Some operation parameters can be sometimes driving, and sometimes driven. The leading parameter **Copy number** is always driven. It necessarily requires an independent variable that would be assigned the current value of the copy number while calculating the operation. The copy number automatically changes during the calculation from 1 up to the specified amount with the increment of 1.

Calculation is done in the following order: first, the copy number is incremented, then the driving parameters are calculated, after which the rest of driven parameters are calculated (if any), and, finally, copying is done. All copies of the 3D point obtained in this way are united in one spline.

The icon for activating this mode is provided in the main automenu of the 3D path creation command:

	<P>	Create 3D Path by parametric change of 3D Node position

Once the mode is active, the following actions become available:

	<R>	Select 3D Node
	<1>	Select Curve defining position of coordinate system
	<2>	Select 3D Curve defining X direction of coordinate system
	<3>	Select 3D Curve defining Y direction of coordinate system
	<4>	Select Surface defining X direction of coordinate system
	<5>	Select Surface defining Y direction of coordinate system
	<6>	Select source LCS
	<7>	Cancel selection of elements defining coordinate system

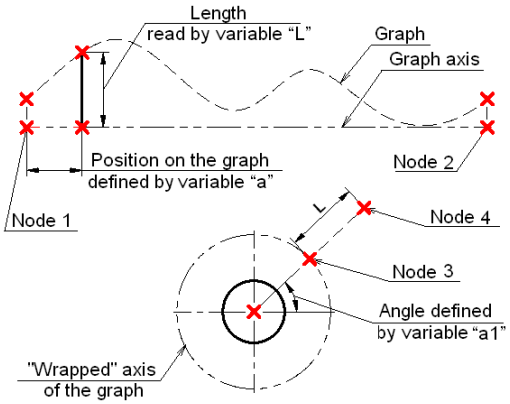
As was mentioned earlier, one can distinguish two ways of creating a path by parametric variation of a 3D point. Let's review examples of each of them.

Free copy orientation method

When creating a path by this method, all that is required is selecting a 3D node, specifying the number of copies and assigning an independent variable to the "Copy number" parameter. By using the variable assuming the value of the current copy number, one can create certain dependencies in the driving the variable entries, as well as in 2D or 3D constructions, that, in turn, define the position of the source 3D point.

Example 1

Let's review the use of a parametric path on the example of creating a model of a cam. This example can be found in the library *"Examples"*, the folder *"3D Modeling/Paths/Cam.grb"*. The source data is represented by a graph defining the cam surface.



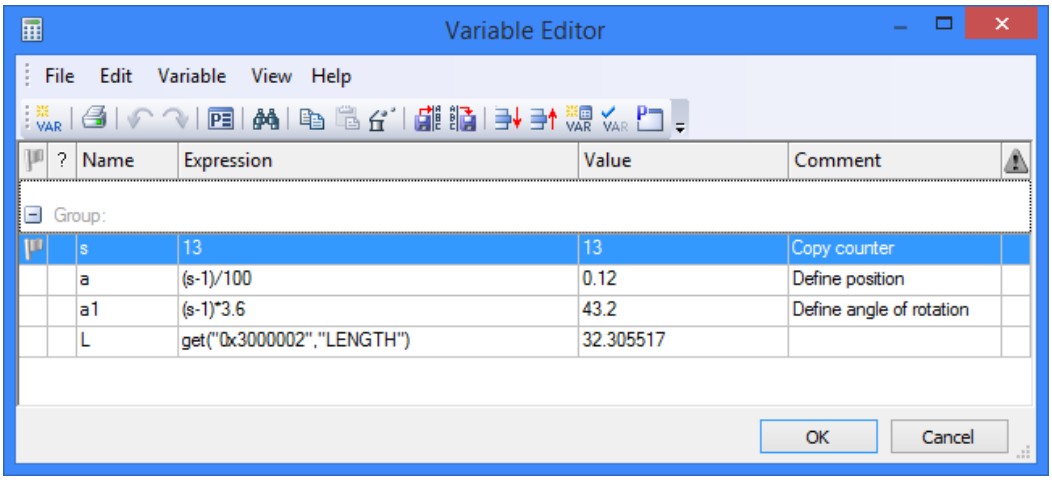
A parametric segment is constructed between the nodes 1 and 2. Its position is defined by the value of the variable *"a"* in the range from 0 to 1, inclusive. As the variable *"a"* changes, the line will move from point 1 to point 2. The resulting line will cost the axis of the graph and the graph curve. A segment can be constructed between these points. Its length will be equal to the current value of the function represented by the graph. The read value of the segment length is stored in the variable *"L"* (by means of the function get).

To construct a 3D Path repeating the contour of the cam surface, one needs to create such a 3D node that will change its spatial position according to the law defined by the graph as the variable changes that is assigned to be the copy counter. Such a node can be conveniently created by adding a few additional 2D constructions based on already available data.

To get the cam of the desired size, the axis of the graph needs to be "wrapped" onto a circle of the specified radius. In the given example, the constructed circle has the radius of 50 mm. A new line needs to be created through the circle center at an angle to the horizontal. The variable *"a1"* shall be assigned to the parameter defining the slant angle of this line.

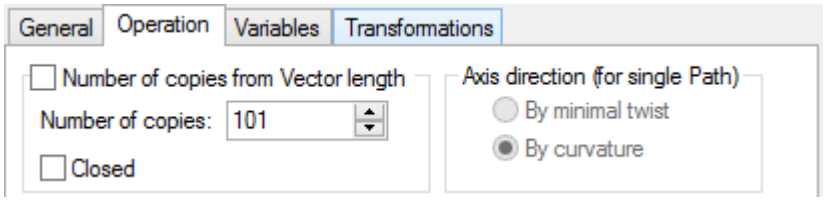
The node 3 is constructed at the intersection of the circle and the slanted line. The node 4 is constructed at the distance *"L"* from the node 3 along the slanted line. A 3D node is constructed based on the 2D node 4. For that purpose, a "Front view" workplane was created on an existing page.

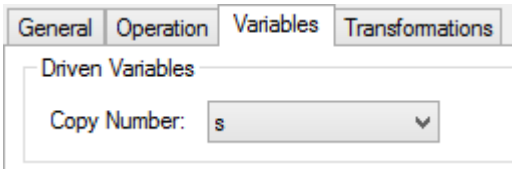
Now, we need to relate all variables in such a way that the 3D node would rebuild according to the copy counter changes. This can be achieved by introducing the following expressions instead of fixed values of some variables, that will maintain the dependency from the copy number:



Since the values of the **Copy number** parameter always start from one, sometimes a (–1) adjustment is necessary in the expressions employing the **Copy number** variable.

At this point, the parametric 3D path can be constructed. We need to define two parameters in the parameters dialog: enter the number of copies and assign the variable to act as the copy counter. In the given example, the "Copy number" parameter is assigned the variable *"s"*.

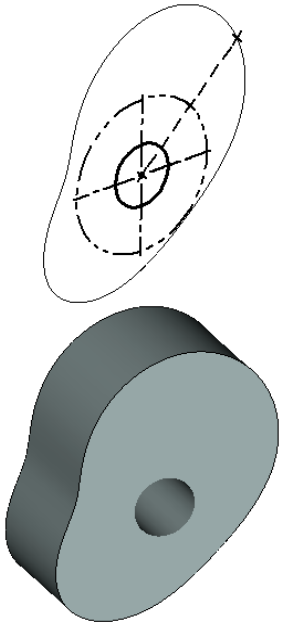




The first copy of the 3D node will coincide in its position with the 101-th. As a result, a closed 3D path will be created. Its geometry will correspond to the cam shape.

Next, to create the cam body, extrude the path in the normal direction by 50 units and create a solid body based on the obtained sheet surface using "Sew" operation.

Finally, one can make a cylindrical hole along the cam's axis (by extruding a circle with automatic subtraction from the main body).

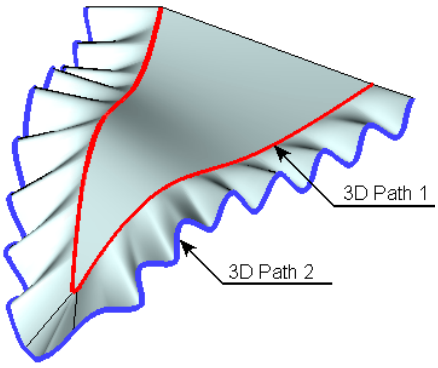


Method of orienting copy by paths or surfaces

The method of orienting a copy by paths or surfaces is convenient when other means fail that would define the law of varying copy parameters by expressions. In such cases, it could be convenient to use various 3D curves or surfaces for defining the position and orientation of the current copy's coordinate system.

Example 2

As an example of using a parametric 3D path with orientation of the copy by paths or surfaces, let's consider building a 3D path defining the outer border of a "waves" surface. On the figure, it is denoted as 3D path 2.

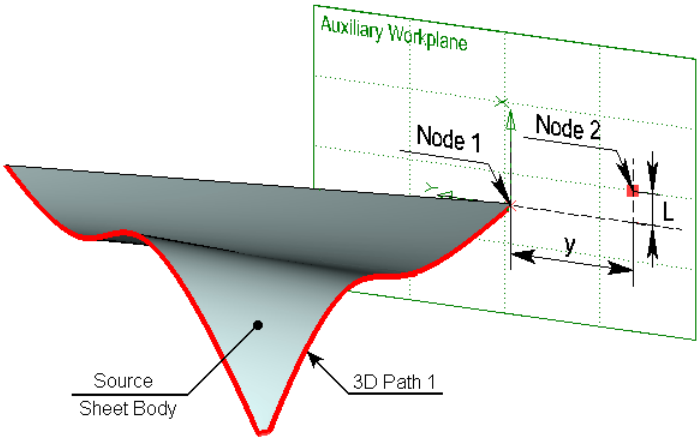


To construct such a path, we need:

- the source 3D node whose position will be changing depending on the number of the copy being created in the process of building the path;
- a 3D path defining the position of the coordinate system origin for each copy;
- a surface, defining the orientation of the X-axis of each copy's coordinate system.

The path being created shall repeat the shape of the outer border of the source sheet body. Therefore, to define the origins of the copies' coordinate systems, we would need to use a 3D path built on the edges of the source sheet body – the 3D path 1.

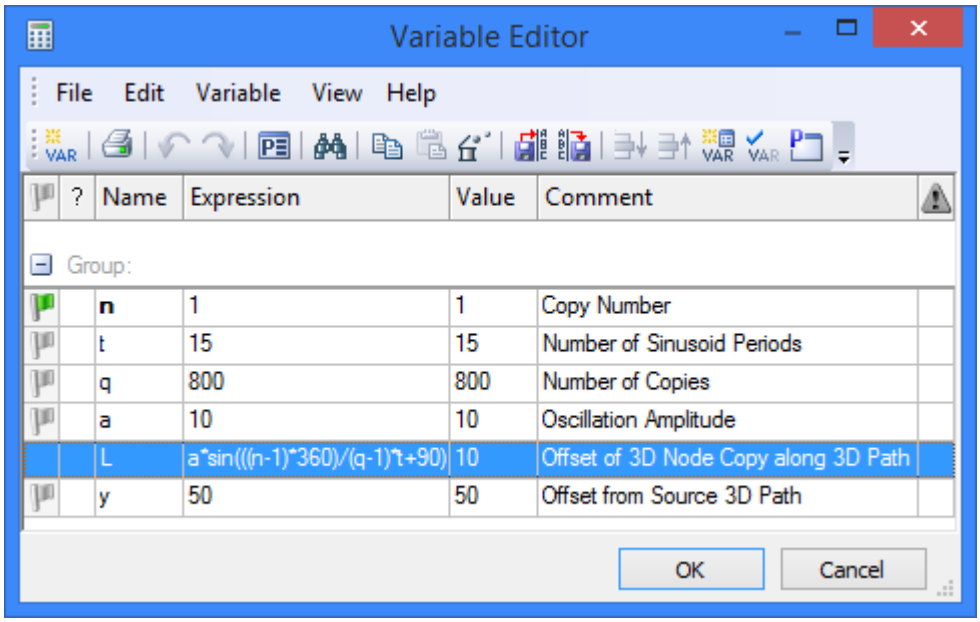
An auxiliary workplane is constructed in order to get the source 3D node, which is perpendicular to the 3D path 1 and is passing through one of the vertices of the source sheet body.



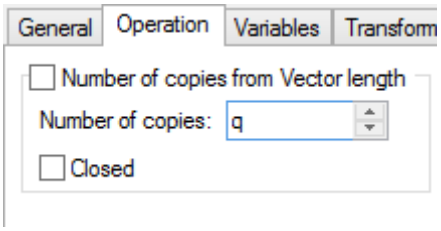
The necessary 2D constructions are created on the workplane. The sought 3D node is created based on a 2D node on the workplane – the Node 2.

Node 2 was constructed as follows: first, the Node 1 was created on the workplane with the coordinates (0,0) (corresponding to the vertex of the source sheet body through which the workplane passes). A horizontal line is drawn through this node. A vertical line is drawn at the distance *"y"* from Node 1, perpendicular to the horizontal line. Node 2 lies on this line at the distance *"L"* from the lines intersection point.

The value of the *"L"* variable is defined in such a way as to vary with the changes in the copy counter of the 3D path. Therefore, as the 3D path creation progresses, the position of the 2D Node 2 changes by the sinusoidal law with the current copy number, and so does the based on it 3D node.



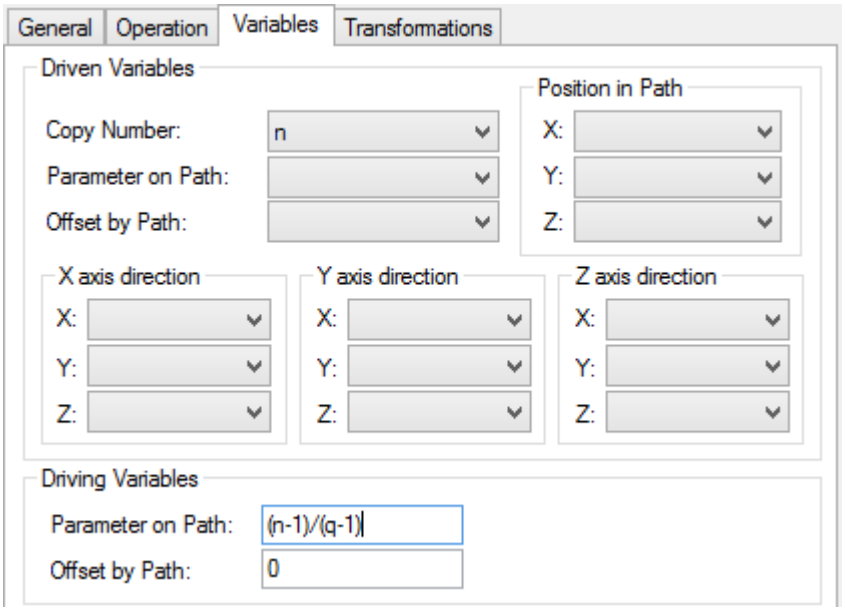
To define orientation of the X-axis of the copies' coordinate systems with the progress of the parametric path creation, we need to specify the source sheet body itself. If no surface is selected, then the axis orientation will be defined based on the curvature vector of the 3D path 1, whose direction may change abruptly.



The following needs to be defined in the path parameters dialog: the number of copies (the variable *"q"*), the copy counter variable (in the given example, this is the variable *"n"*) and the driving parameter "Parameter along path".

Once 3D path 2 is created, the based on it "waves" surface can be created by "Loft" operation.

This example is located in the library *"Examples"*, the folder *"3D Modeling/Path/ Parametric path with copies orientation by path and surface.grb"*.



3D Paths Based on 2D Elements

Hatches, 2D paths and 2D splines can be used for a path creation, that are located on the pages associated with the workplanes or worksurfaces. When using a 2D element lying on a page of a workplane, the resulting 3D path will be flat. A 3D path based on a 2D element from a worksurface will repeat the shape of the used surface.

Creating 3D Path Based on Hatch Contour

To create a 3D path, select a 2D hatch on a workplane or surface, whose contour is intended for the path. By default, the plane (surface), in which the created 3D path will lie, coincides with the workplane or surface, whose page hosts the source 2D hatch.

For more precise positioning of the 3D path in the space one can specify a 3D node through which the path plane or surface must pass. The plane (surface) of the path is shifted by parallel translation. If that's not enough, the position of the 3D path can be further adjusted by specifying a 2D node (on the workplane hosting the hatch) that shall be aligned with the selected 3D node.

This method of creating a 3D path fully corresponds with creating a 3D profile based on a hatch contour. The detailed description of this capability is available in the chapter "3D profiles".

To create a 3D path by this method, select the following option in the command's main automenu:

	<H>	Create 3D Path from Hatch

After activating it, the following options appear in the automenu:

	<H>	Select Hatch

	<M>	Select 3D Node for 3D Path plane fixing
	<N>	Select Node to be moved to 3D Node
	<K>	Break link with 3D Node
	<F>	Break link with 2D Node

Some items may be unavailable at different creation stages. The last icons become available only after specifying the 3D path and its snapping points.

The first step of creating 3D path (and often the only one) is selecting the source hatch. The hatch is selected in the 2D window or on the active workplane, while the option is active.

The workplane or surface with which we edge can tour will be associated is determined automatically by the system. If other workplanes or surfaces also reference the same drawing page, the system selection can be altered by the option:

	<W>	Select other workplane

This option allows to loop cyclically through workplanes or surfaces up until the desired one is selected.

The 3D node defining the position of the path in the space can be selected by activating the option .

The 2D node on the page of the source hatch is selected with the help of the option . The selected nodes can be rejected by the options and .

Creating 3D Path from 2D Paths

To create a 3D path, several 2D paths are selected subsequently. The paths must lie on a 2D page (pages) associated with workplanes or surfaces and be open.

The system automatically determines the workplane or surface for each selected 2D path. If the automatic selection is incorrect, the user can fix it manually. A subsequent 3D segment of the path being created is constructed based on the 2D path and the workplane (surface). All resulting segments are joined as follows: the beginning of the second segment is connected to the end of the first one, the beginning of the third – to the end of the second, and so on. In other words, this method can be considered a variation of one of the above-described methods – creation of a 3D path as a sequence of 3D paths.

For more precise placement of the 3D path in the space, one can specify a 3D node to which the plane (surface) of the first 3D path segment will be snapped (that of the one corresponding to the first specified 2D path). Additionally, to adjust the 3D path position, you can specify a 2D node on the page of the first 2D path. This 2D node will be aligned with the selected 3D node.

Creating a 3D path by this method is done with the help of the option:

	<C>	Create 3D path from 2D Paths

Upon activating this mode, a number of auxiliary options appear in the command's automenu for selecting all required objects.

To select 2D paths, use the option:

	<C>	Select 2D Path
	<D>	Cancel selection of 2D path

The option allows rejecting the last selected 2D path.

The system automatically finds the workplane or surface for each selected 2D path. A different workplane or worksurface can be selected (if any exist on the given 2D page) using the option:

	<W>	Select Other Workplane

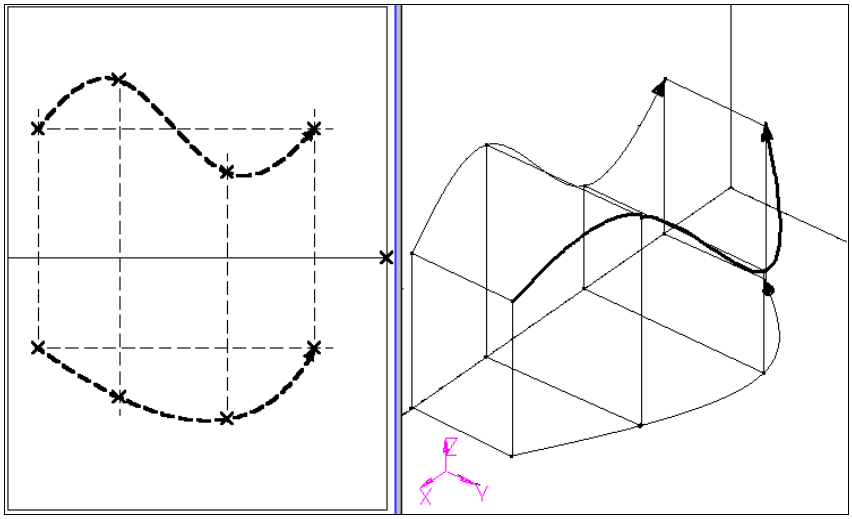
The nodes for exact spatial positioning of the 3D path being created are selected by the following options:

	<M>	Select 3D Node for 3D Path plane fixing
	<N>	Select Node to be moved to 3D Node
	<K>	Break link with 3D Node
	<F>	Break link with 2D node

Create 3D Path Using Two Projections

The 3D path is created by two projections on the main workplanes. The projections of the 3D path are represented by 2D paths.

> The 2D paths being selected must fully and uniquely define the position and direction of the 3D path in the space. Otherwise, the placement of the 3D path in the space will be incorrect or the system will output a message about an error and a failure creating a 3D path.



This example is located in the library *"Examples"*, the folder *"3D Modeling/Path/3D Path by two 2D Paths.GRB"*.

This method of creating a 3D path is activated by the option:

	<Q>	Create 3D Path using two projections

Upon calling this option, the following options appear in the automenu:

	<F>	Select first projection (2D Path)
	<S>	Select second projection (2D Path)

These options allow selecting two projections of the path being created in the 2D view window. The workplanes on which the projections will be placed are selected by the system automatically. A different workplane can be selected (if any exists on the given 2D page) using the option:

	<W>	Select Other Workplane

ROUTE

The route is a special type of a 3D path, that is composed of straight segments. The adjacent straight segments are connected by arcs of a specified radius. This type of a 3D path is mostly intended for defining a spatial trajectory used for creating pipelines.

TYPES OF ROUTE SEGMENTS

For each segment of a route, you can specify the type that determines the way of its construction:

- **Spline by polygonal line**. Spline of arbitrary degree based on control polygonal line. Control points are used for construction of control polygonal line. The spline itself does not pass through these points (except the first and the last points). The degree can be specified in parameters of the spline.

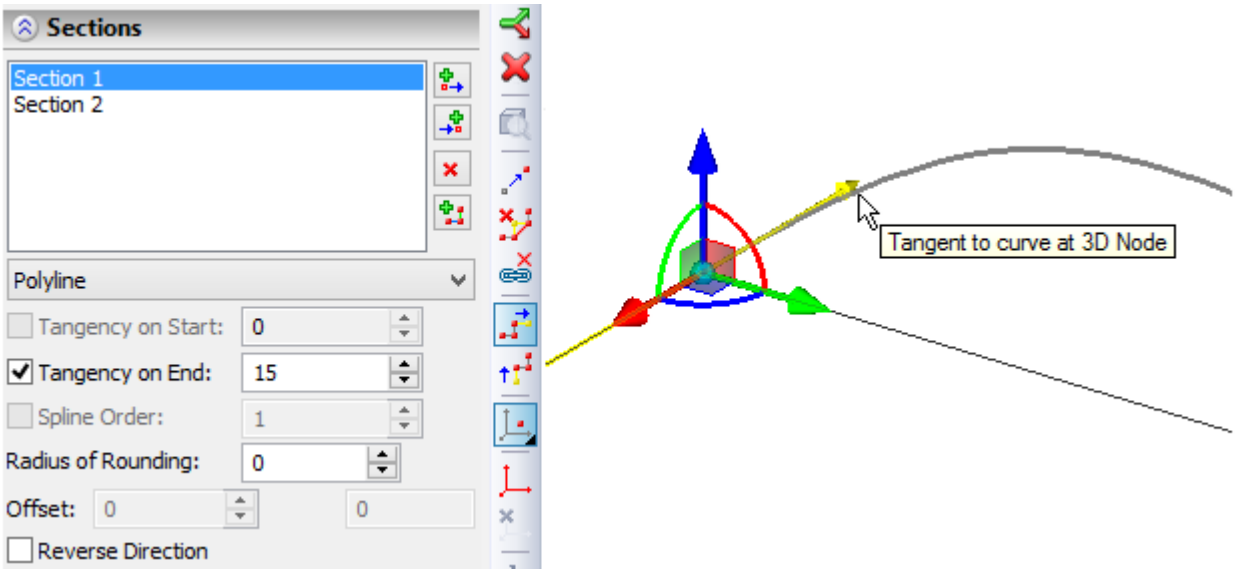
    - **Spline degree** is a *degree of polynomials of a piece-continuous function* that describes the spline being created. The maximum admissible value of the degree must be smaller by one than the number of spline points. For a spline containing no fewer than four control points the degree of 3 is selected by default. The higher degree of a spline, the more "rigid" it becomes. This means that it weakly reacts to the translation of an individual control point. Splines of lower degrees pass closer to the control points and the change in their location affects the shape of a spline more significantly.

- **Spline by points**. Spline passing through all given points. When creating a 3D spline through the nodes, it is possible to additionally specify spline direction vectors at the boundary points of a segment (start and end points); a tangent line at other points of a spline.

- **Polyline**. It constitutes a spatial polygonal line composed of segments of the lines that join the points of the initial set. Polyline can be both open and closed (when creating a closed polyline the first 3D point of the initial set is used twice – for determination of the beginning and the end of the path);

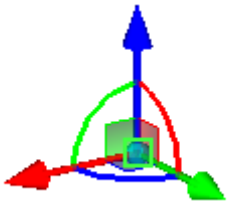
CONNECTING ROUTE SEGMENTS

If it is necessary to combine in one 3D path the segments of different types the system automatically constructs a transition from one segment to the next one. The initial point of a segment might not coincide with the end point of the previous segment. In this case, between the segments the transition zone is constructed in the form of a curve joining these segments. If possible, the system will construct a smooth fillet. The «**Tangency at the start**», «**Tangency at the end**» parameters allow us to specify the radius of curvature at the tangency points. Values of these parameters can be modified with the help of manipulator in 3D window or with the help of the numeric value in the properties window.



USING MANIPULATORS

When adding a new point of the 3D path the **LCS manipulator** appears in the scene that allows us to change location of the point with respect to the selected attachment (absolute coordinates, 3D node, vertex, etc.). By default directions of the manipulator axes coincide with the directions of the global coordinate system axes.



Manipulator provides for the following transformations:

- translation along the axis (X, Y, Z). To enable translation, move the cursor closer to the axis arrow.

- translation in the plane (XY, XZ, YZ). To enable translation, move the cursor closer to the image of the plane.

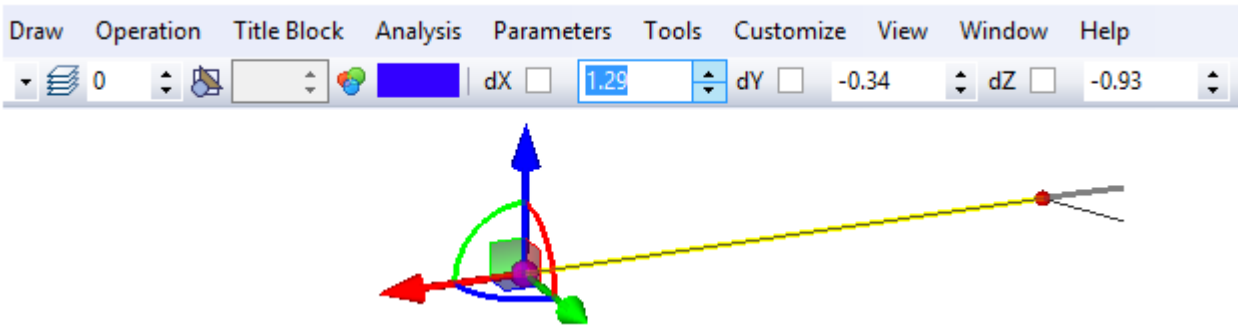
- translation by three axes. To enable translation, move the cursor closer to the sphere at the center of the manipulator.

- rotation about the axis (X, Y, Z). To enable rotation move cursor closer to the arc (the color of the arc corresponds to the color of the axis around which the rotation will be carried out).

Manipulator can work in one of the two modes:

- numeric specification of the transformation value. To work in this mode move the cursor of the mouse closer to the manipulator (it will be highlighted and the tip will appear on the screen) and translate the

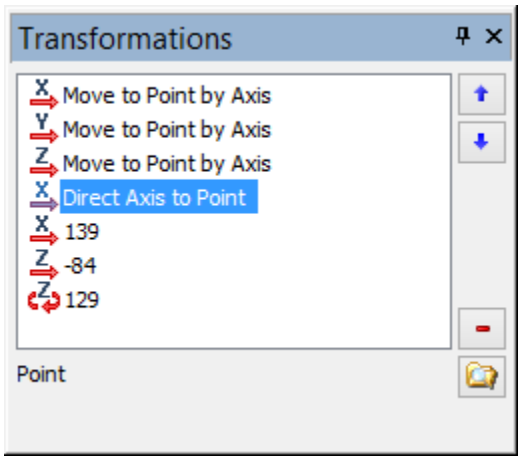
mouse with the pressed left button. Precise value of the transformation can be specified on the system toolbar.



- transformation is prescribed by snapping to the existing object of the model (for example, translation up to a 3D node). To work in this mode move the cursor of the mouse closer to the manipulator and click . After that the object snapping will be working upon translation of the cursor. In this way it is possible to specify the translation up to a point, edge midpoint, etc.

Any change in the location of the manipulator is reflected in the auxiliary window «Transformations». For each point of the path a separate list of transformations is saved. Translations along the axes are summed up. Transformations with the use of object snapping are recorded as a separate line. Transformation selected in the list can be deleted. The object snapping can be redefined by indicating a new object in 3D scene.

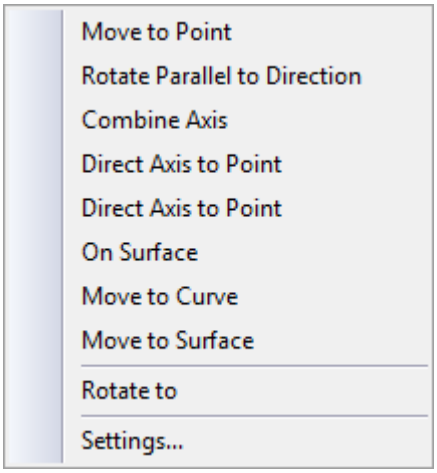
The order of transformations can be changed by moving transformation up and down along the list with the help of buttons with arrows. However, it is important to take into account the fact that an arbitrary change of the order might lead to the incorrect result.



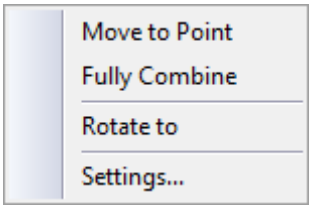
Snapping manipulator to the elements of the model. Using context menu in work with the manipulator

When selecting one of the manipulators mentioned above the following commands will become available in the context menu:

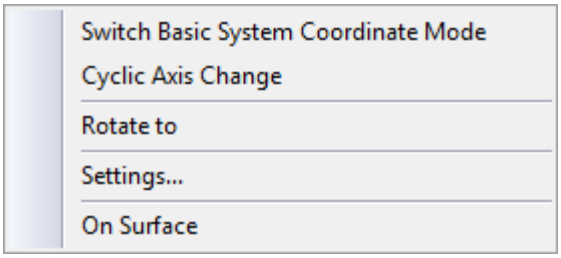
- for translation along the axis



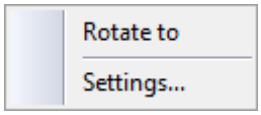
- for translation in the plane



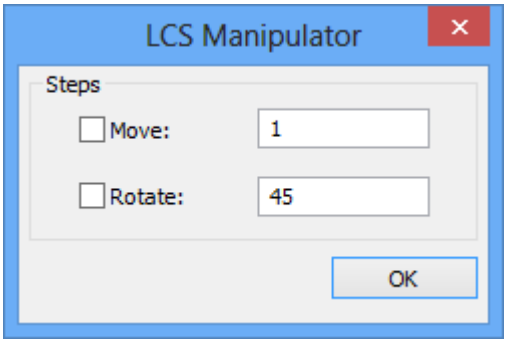
- for translation along three axes



- for rotation around the axis

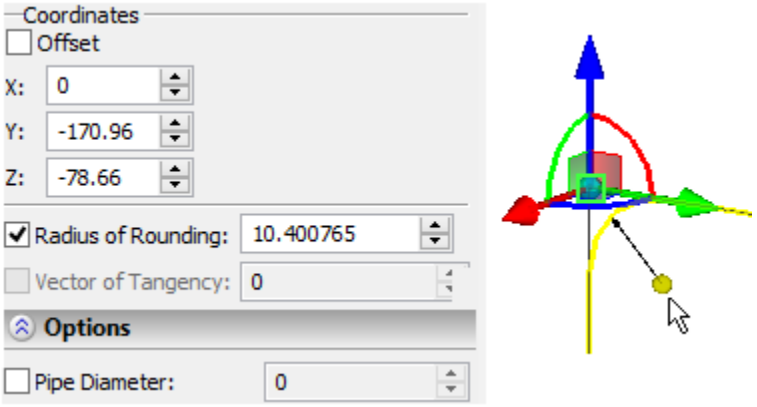


The option of the «**Settings**» menu defines the step with which the distance and the angle change upon translations and rotations of the manipulator.



Manipulator of rounding

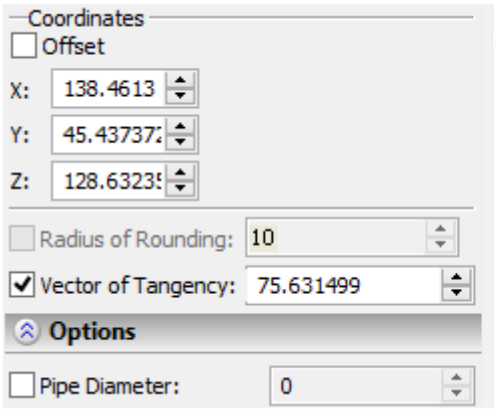
Manipulator that controls rounding of the path is available at any point of the segment, except the start and the end points. Translation of the manipulator changes the value of rounding in the selected point. For input of the numeric value the «**Rounding**» field in the properties window is used. By default all points of the segment have the same value of rounding – the «**Radius of rounding**» field.

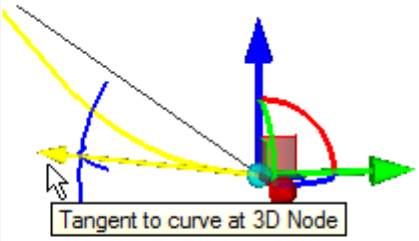


For the manipulator of rounding there is a limitation – minimum radius, i.e., the manipulator with the radius as small as desired cannot be specified. For input of small values of rounding, the field in the properties window must be used.

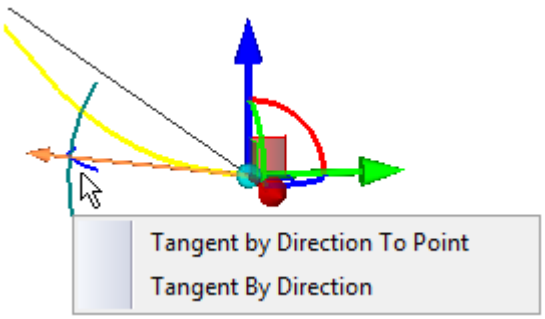
Tangency manipulator (available only for a spline by points)

- **tangent to a curve at a point**. The length of the tangent vector is specified.





- **direction of tangent**. Two manipulators in the form of arcs that allow us to specify direction of the tangent. For these manipulators the two commands: «Tangent along the direction up to a point» and «Tangent along the direction» are available in the context menu.

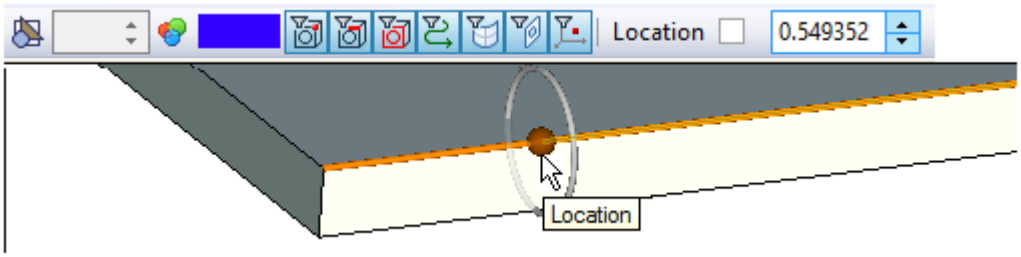


Tangent in the direction up to a point allows us to select a point that defines the direction of the tangency vector. Tangent along the direction allows us to select an object that defines the direction of the tangency vector. Direction can be specified by an edge, normal to a surface, axis of the coordinate system, etc.

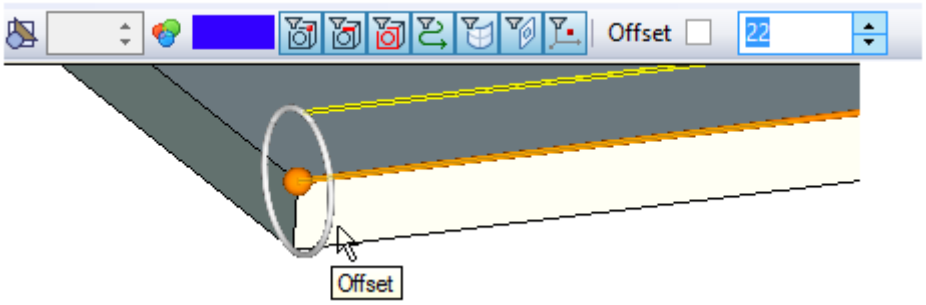
Manipulator for a segment along the path\edge

If a part of the segment is specified as an edge or 3D path, the following manipulators will be displayed in the scene:

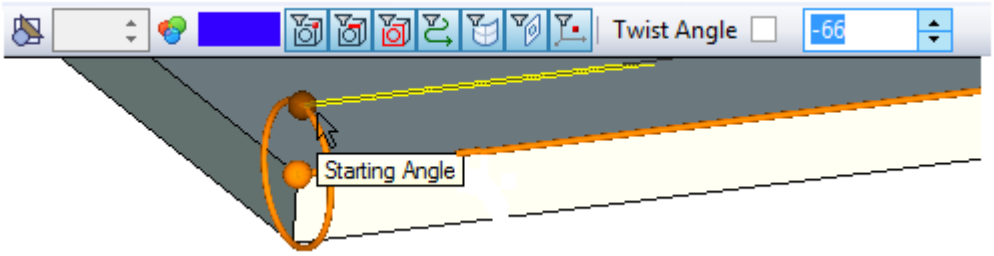
- **Location**. Specifies the shift from the ends of the selected object. The precise value can be specified on the system toolbar. In the context menu of the manipulator the «Position in the nearest point» and «Delete a point» commands are available that allow us to specify\cancel the snap of the shift to a point.



- **Offset** (available only for straight segments of the path). Specifies the shift of the segment of the path from the selected object. The precise value is specified on the system toolbar.



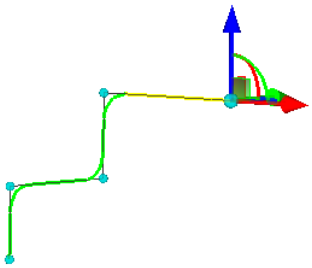
- **Starting angle** (available only for straight segments of the path). Specifies the angle of rotation around the selected edge/path. Precise value of the angle is specified on the system toolbar.



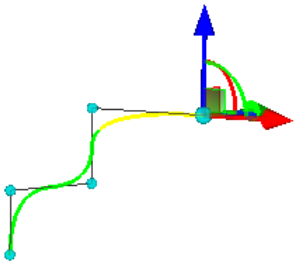
- **twist angle** (available only for straight segments of the path). Specifies the angle of rotation of the end point with respect to the start point of the segment. The selected edge/path is considered as the axis of rotation.

Segment by Points

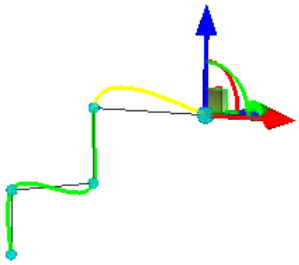
The segment by points is created when calling the command by default. In the scene the manipulator is displayed which must be translated to the start point of the path. To determine location of the point the attachment to geometric objects (vertices, edges, etc.) or specification of absolute coordinates/shifts can be used. By indicating sequentially the set of points the path's segment of various shapes can be created:



polyline



spline by polygonal line

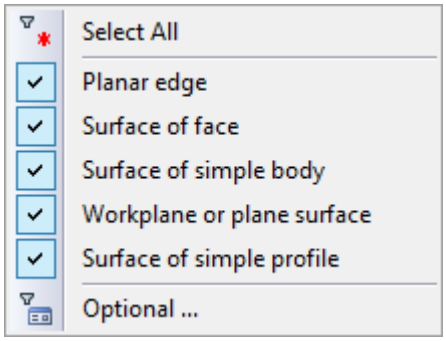


spline by points

Rounding of the path can be specified via the common value «Rounding» or individually for each point of the segment. The options for addition or removal of points of the segment are available in the automenu and properties window. There is a capability of dividing the segment into two parts at indicated point.

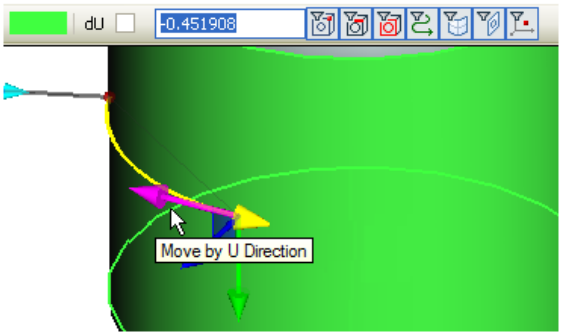
SEGMENT BY SURFACE

To create a segment by surface it is required to indicate the object of one of the types:

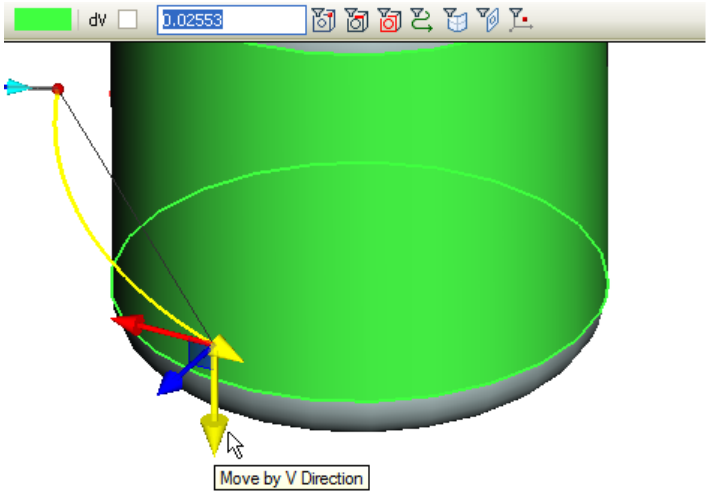


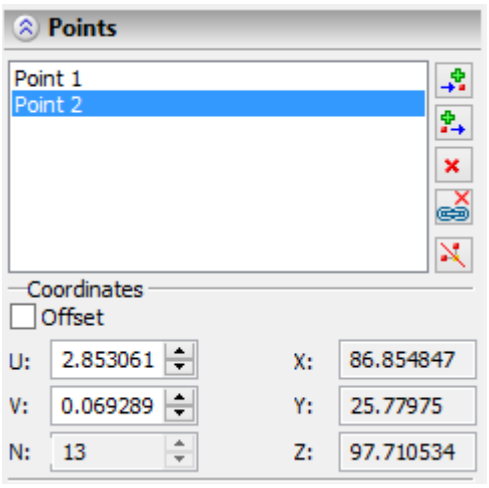
The start point of the segment is created on the surface at the place of the mouse click. By adding the next point, its location can be changed with the help of manipulator. The red arrow (U axis) allows us to translate the point across the selected surface. The value of the translation is specified on the system toolbar.

If necessary, the point of a segment on the surface can be snapped to the node that belongs to the selected face. To do so, select the red arrow of the manipulator, and from the context menu invoke the «**Move up to a point**» command and indicate the point on the surface.

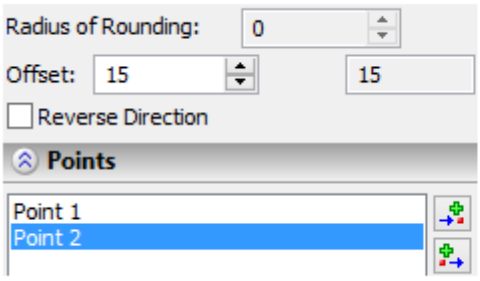


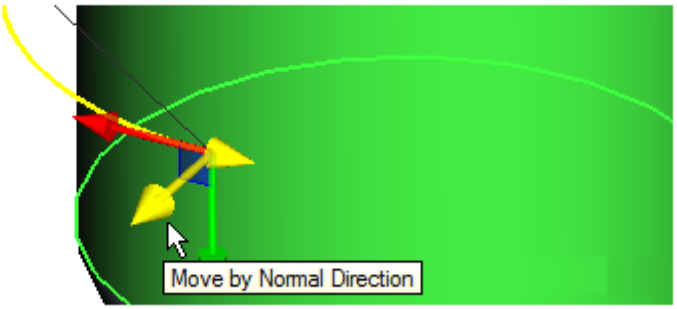
Green arrow of the manipulator (V axis) moves the point across the surface along the second coordinate V. The values of U, V coordinates at each point of the segment are displayed in the corresponding fields in the properties window.



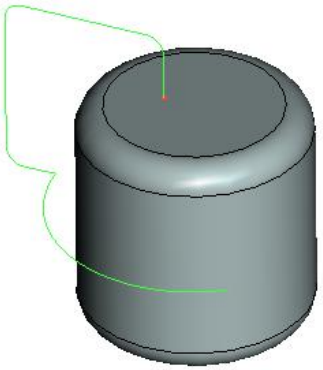


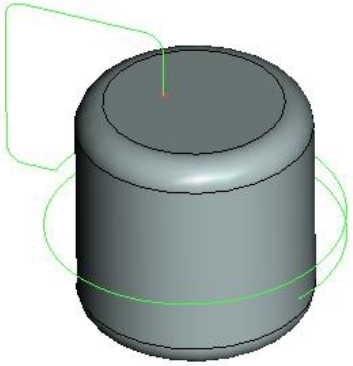
Blue arrow (Z axis) moves the segment being created along the normal to the surface. The precise value of the translation is indicated in the properties window in the «Shift» parameter. When changing the shift the segment equidistant to the one lying on the surface is constructed.





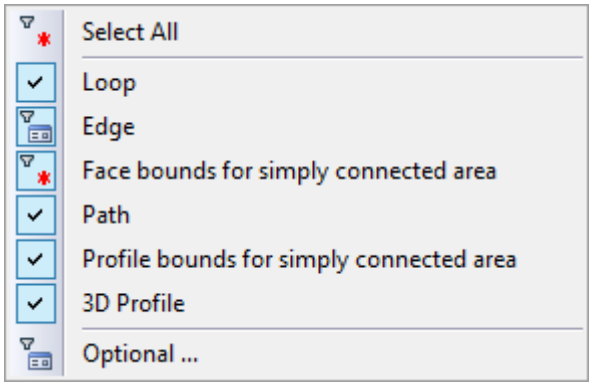
By changing the values of the parameters mentioned above it is possible to obtain both the simple segment of the path and the segment consisting of several windings around the selected face.



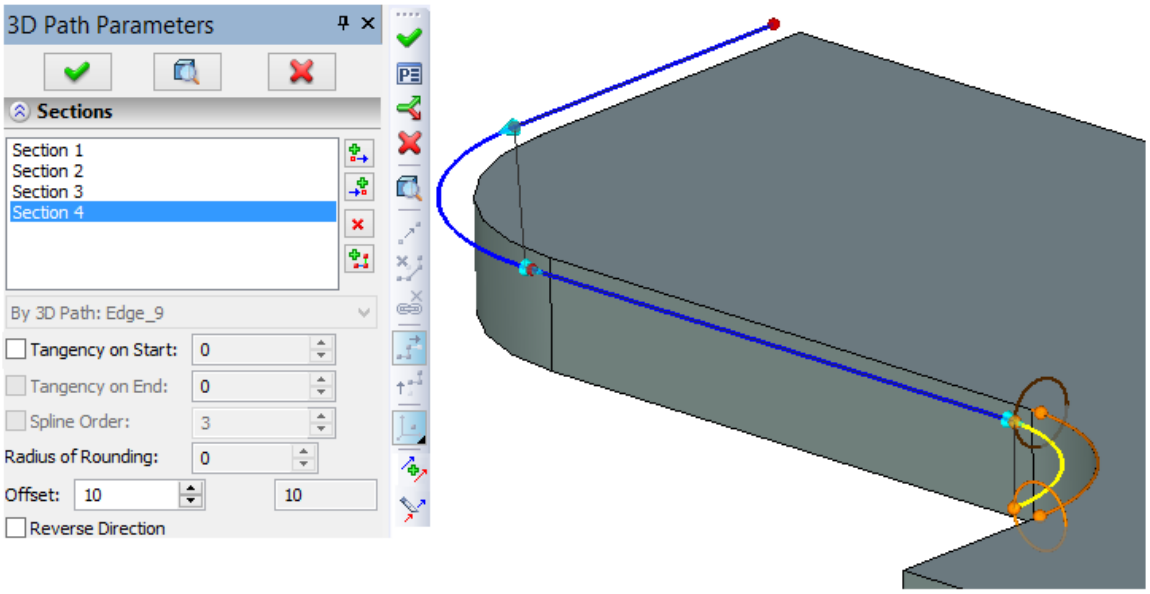


SEGMENT BY PATH OR EDGE

To create a segment by path or edge it is required to indicate the object of one of the types:



After selection of the object, the created segment of the path will entirely coincide with it. With the help of manipulators described above location of the segment and its start and end points can be modified. When creating the path by the set of edges (paths) it is required to add a separate segment for each of them.

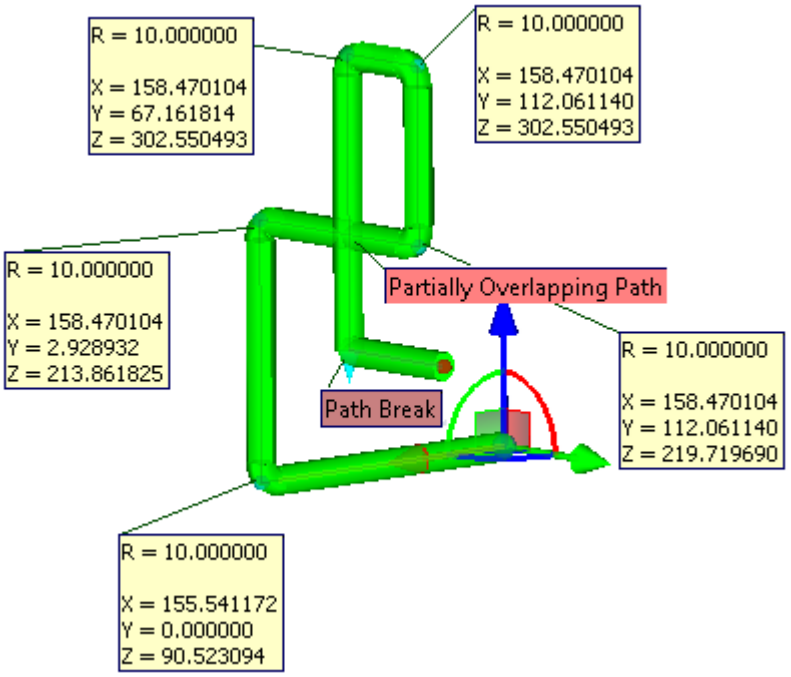


DIAGNOSTICS OF WRONG CONSTRUCTIONS

For intuitive diagnostics of errors that arise during construction of the path, the following parameters can be enabled in the properties window:

- «Show bends»
- «Show radius of curvature less than».

In addition, the system automatically shows diagnostics about the self-intersection of the path, about partial coincidence of segments of the path right after the input of the new point or the segment.



CREATING OF ROUTE

To create the route there is a separate command:

Icon	Ribbon
	3D Model → Construct → Route
Keyboard	Textual Menu
<3PP>	Construction >Route

The same command is invoked from the path creation command with the help of the option of the automenu:

	<B>	Create/edit path

Input of new points

By default the following option is enabled in the command's automenu:

	<H>	Mode of adding points after the active one

After invoking this command the first segment with the start point is automatically created in the route. This point becomes *active*. Manipulator is located at the zero point of the global coordinate system.

In general, a point is considered active when it is highlighted in the list of points in the properties window. The active point can be changed by selecting another point in the 3D view window or in the list of the properties window.

Several possible actions are available:

Action	**Result**
Pressing the key «Enter» or the option	Editing of the active point is finished and the new point is created *after* the active point. The new point is initially located in the same coordinates as the point created just now. The new point becomes active.
Translation of the manipulator	Location of the active point is being changed. To confirm the current location of the active point and creation of the new point press the «Enter» key or select the new 3D point.
Selection of 3D point of the model (3D node, vertex etc.)	Location of the active point is confirmed, the new point with the snap to the selected object is created.
Pressing the «Esc» key	Cancel the last action (translation, snapping, creation of new point).
Pressing the «Del» key or the option	Delete active point
Completion of path creation (option)	A route is created. If location of the active point coincides with the previous point, it will not be included into the path.

If it is required to add new points before the active one it is possible to switch the mode with the help of the option

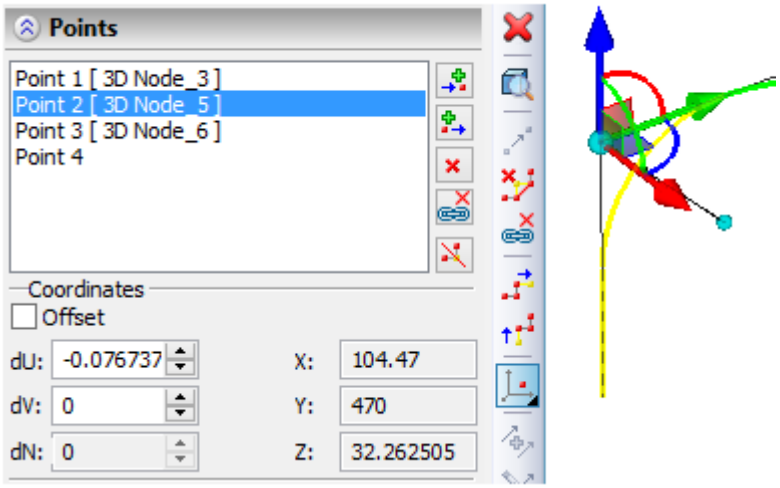
	<J>	Mode of adding the points before the active one

In this case the actions and the results will be the same as described above. The only difference is that the new point is added not after but before the active one. If the first point of the path is active, the new points are added to the beginning of the segment.

In the properties window next to the list of points there are the buttons which also allow us to add the points before or after the active point, remove the active point. Pressing the or buttons confirms location of the active point and creates the new point.

Editing of points

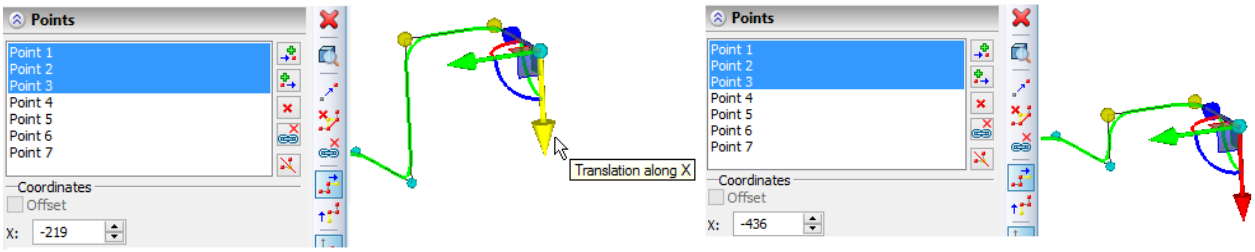
If it is required to change location of already existing point, the options must be disabled. To edit location of the point, select it in the list of points. The point becomes active. Manipulator will be translated to the active point.



Several possible actions are available:

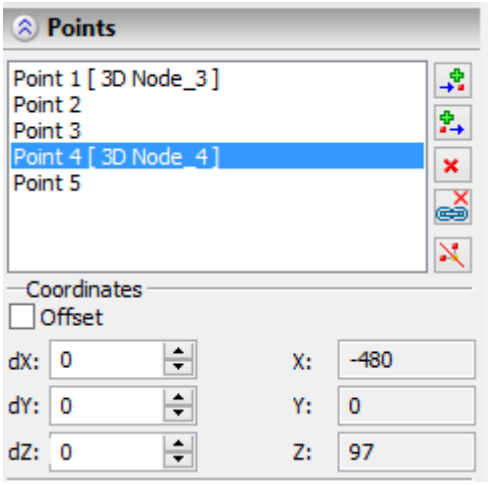
Action	**Result**
Translation of manipulator	Location of active point is being changed.
Selection of geometric snapping (3D node, vertex, etc.)	Location of active point is defined by snapping to the selected object.
Selection of another point of the segment	Editing of the point is finished, another point becomes the active one.
Pressing the «Esc» key	Cancel the last action (translation, snapping).
Pressing the «Del» key or the option	Delete active point
Pressing the «L» key or the option	Deletion of snap of the point to the geometric object. After cancelation, location of the point is stored in absolute coordinates
Completion of path creation (option)	Changes in the path are saved.

There is a capability of multiple editing of points. Multiple selection is carried out in the properties window, in the «Points» list. After that, translation of the manipulator will affect location of all the selected points.

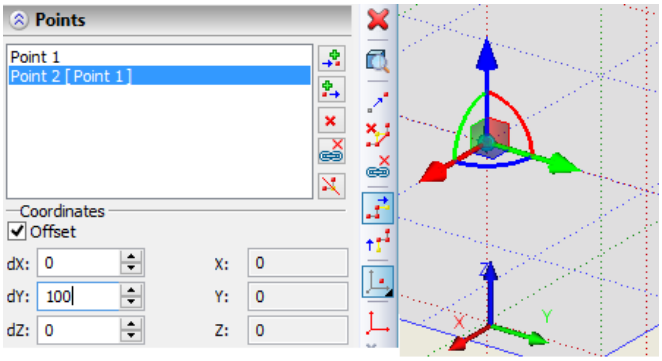


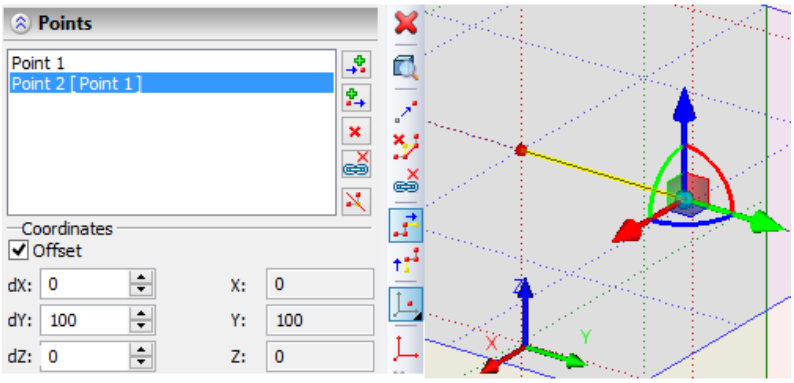
Specifying coordinates and offsets of a point

The absolute coordinates of the active point are shown in the properties window. For points of the segment by surface in addition to the X, Y, Z fields the U, V, W coordinates are filled in. For points of the segment by edge the U, V, R coordinates are filled in. The values of coordinates can be changed manually.



When creating or editing a point it is often more convenient to specify the values of the shifts but not their absolute coordinates. For input of this data the group of parameters «Offset» is used. When creating the new point its location initially coincides with the point created just now. By specifying the value of the shift, we translate the active point with respect to the previous one. If the shifts are specified upon the editing of the point, then the result will be its translation relative to its current location.

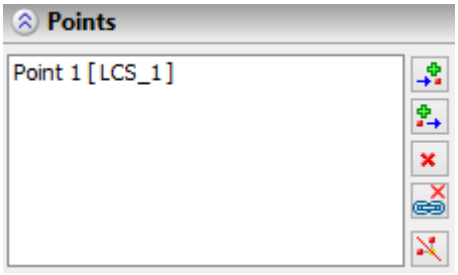




Segment points can be specified in the coordinates of an arbitrary LCS. To select\cancel LCS in automenu use options:

	<C>	Select LCS
	<V>	Deselect LCS

After selecting a coordinate system, it will be used for the current point and all points that are created after the current one. LCS deselection transfers the coordinates of the current point to the global coordinate system. Segment of the path can contain any combination of coordinates definition.



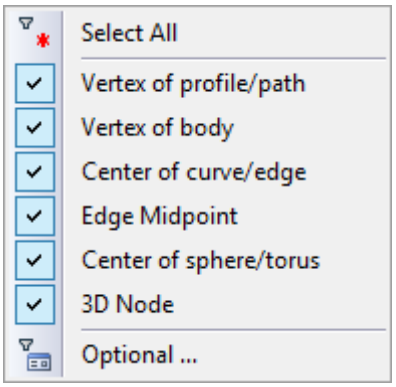
Moreover, for each point the value of the radius of rounding (for polyline) and the tangency vector length (for a spline by points) are stored.

By default the following option is active in the automenu:

	<F>	Select a point

That allows us to carry out the snap of the points of the path to geometric objects.

As a point, an object of one of the following types can be selected:



Creation of segments

By default the system creates segments by points. If it is required to construct a path by using geometry of already existing bodies one of the following options is used.

For creation of the segment of the path by path/edge the following option is used:

	<P>	Construct segment by path/edge

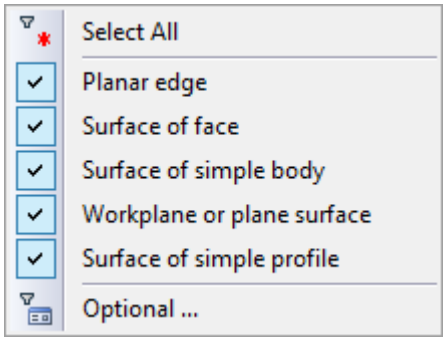
After selection of this option the user can select the edge of already existing body or 3D path which will define the trajectory of the segment being created.

Direction of the segment of the path of this type can be changed to reverse with the help of the «**Reverse direction**» flag. It is convenient to use this parameter when adding new segments in order to provide for the same direction of the path on all segments of the route.

To create a segment by surface the following option is used:

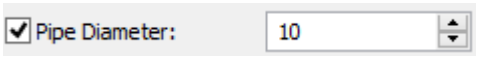
	<S>	Construct segment by surface/face

After adding the segment, it is required to indicate an object of one of the following types:



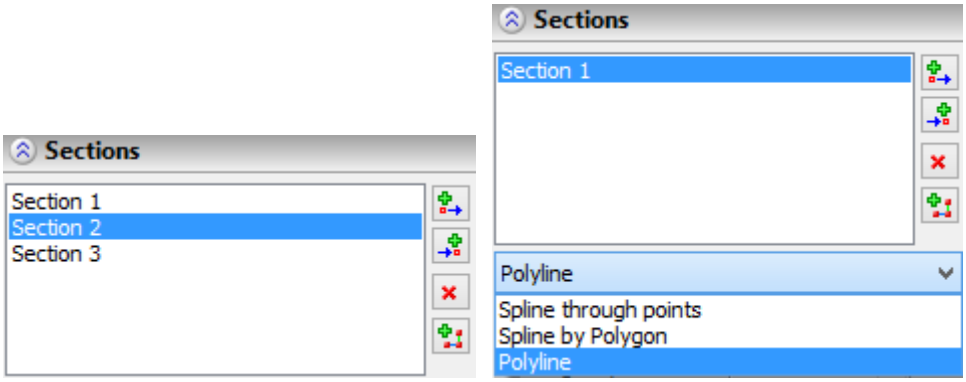
The start point of the segment is created at the place of the mouse click. After that, having added the new point, it can be translated with the help of the manipulator or by specifying coordinates U, V in the properties window.

In the properties window the **Pipe diameter** parameter can be specified. If the value is specified, then in the process of specification of segments and points of the path in 3D scene the preview of the route is dynamically displayed. If the value is not specified, then just a 3D path is displayed.



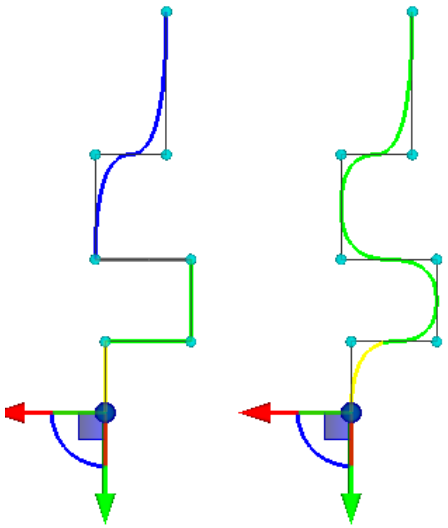
ADDING NEW SEGMENT OF ROUTE

The list of route segments is displayed in the properties window. To create a new segment the «Add to the end», «Add to the beginning» buttons can be used. To remove the current segment – the «Delete» button. When deleting the segment from the middle of the list the preceding and the following segments are connected via a transitional zone. In the field, located below the list, the type of the curve is displayed (for a segment by points) or geometric object that is a parent (for segments by edge or surface). The type of the segment by points can be modified with the help of the drop down list.

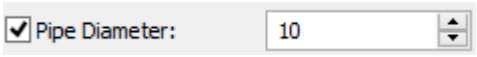


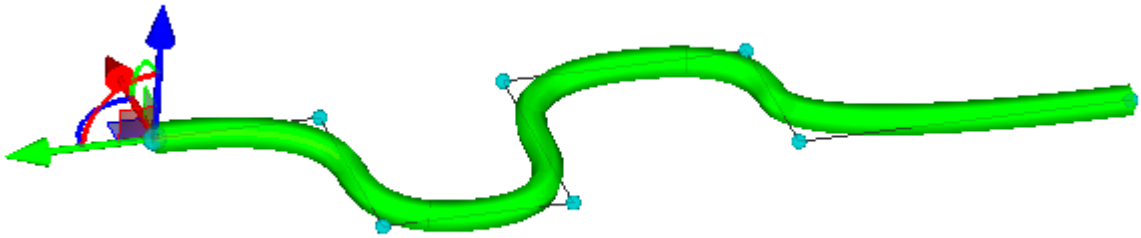
COMBINING SEGMENTS OF ROUTE

Selected segment of the route can be combined with the next segment with the help of the button located next to the list of segments. If the next segment has a different type, then the combined segment inherits the type of the first segment.



In the properties window for a path the «**Pipe diameter**» parameter can be specified. If the value is specified, then in the process of specification of segments and points of the path in 3D scene the preview of the route creation operation is dynamically displayed. If the value is not specified, then just a 3D path is displayed.





Existing 3D path can be combined with the other path with the help of the option:

	<V>	Merge two paths

The option is available only when **editing** an existing path. After calling the option, you must specify the second path. The neighboring points of selected paths will be joint on merging. If the paths are parallel to the joint points are determined randomly.

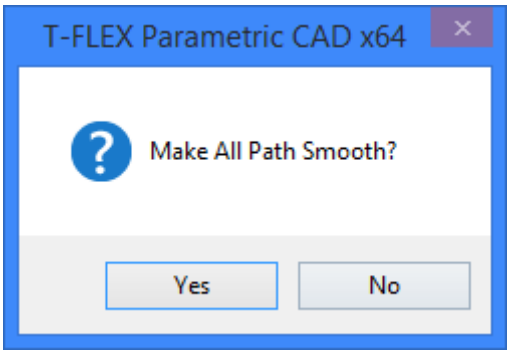
Existing 3D path can be divided into two paths with the help of the option:

	<D>	Split path

Additional Options

Create smooth path. Smooth transition zones will be added when creating new sections of a path.

When this flag is used for the path that contains multiple sections, you are prompted to smooth transition zones. To do this, select "Yes" in the following message box:



Mark point with curvature less than. When using this option together with the definition of minimum curvature radius, system will indicate points at which this parameter is less than the specified value.

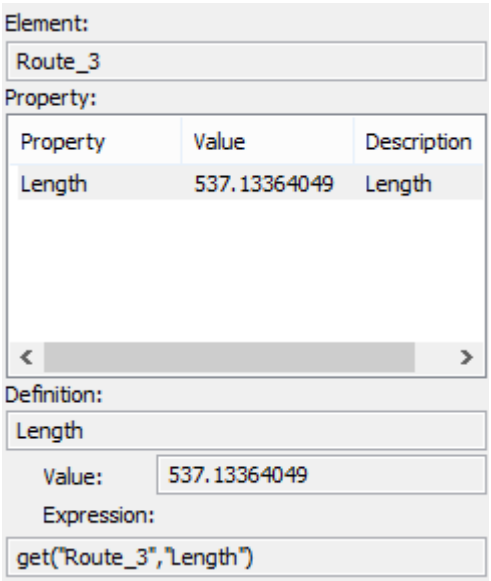
Mark breaks. With this option, the system will mark points at which there is a break in the path, i.e. violated conditions of smoothness (tangent to the right is not equal to the tangent to the left) .

Check self-intersections. This option controls the dynamic definition of intersection points of the path sections. The points of intersection are marked with “Partial overlapping of path”." Such points will prevent path creation. Using this option increases system response time, so it is recommended to add verification of the intersection just before the completion of the path creation.

Length of Route Path

To get the value of the route length, use the command "Measure". Measurement command can be called from the context menu of Pipeline operation.

Length is contained in *Length* parameter.



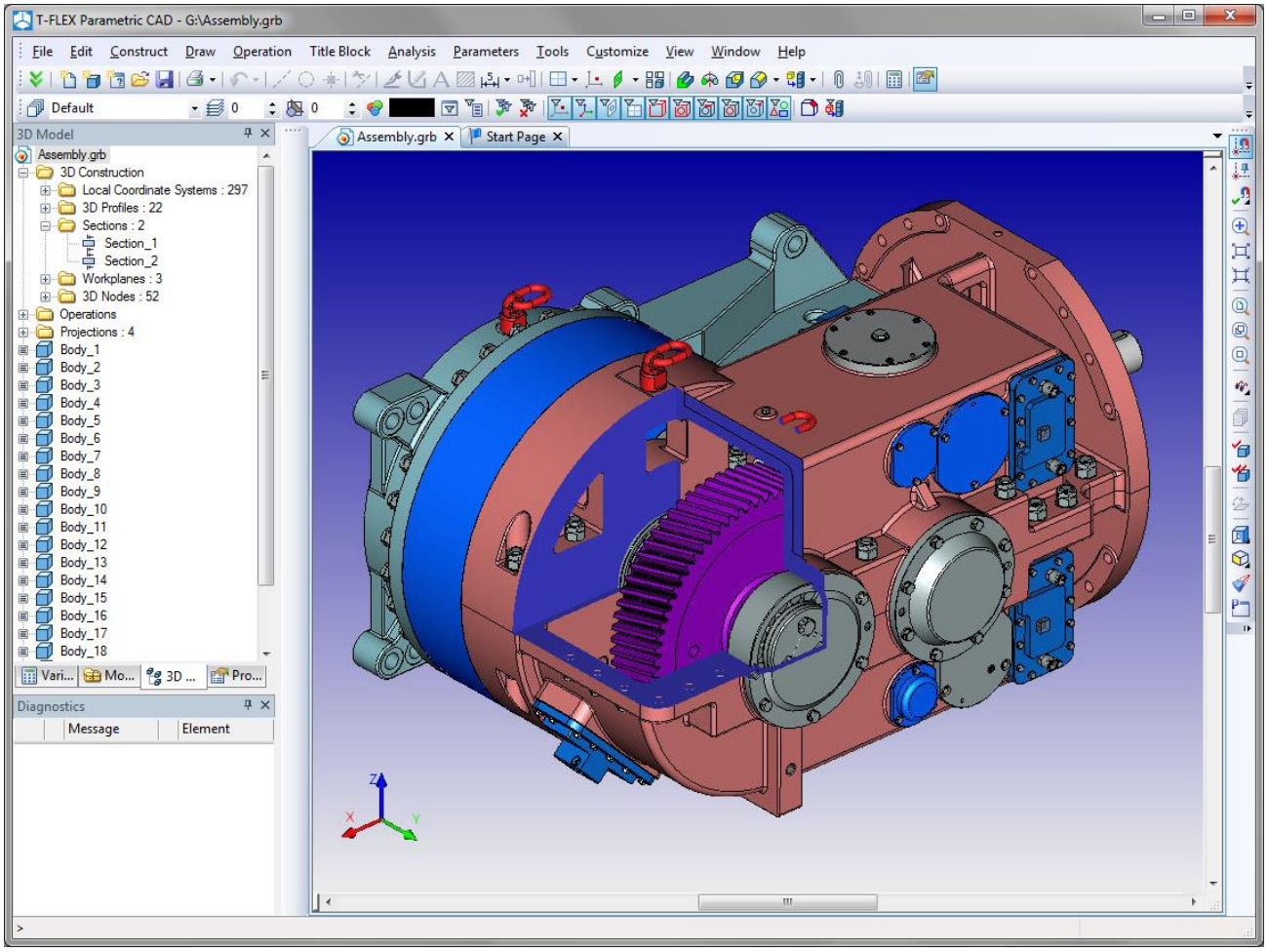
To call the dialog box of path parameters use option:

	<P>	Set entity parameters

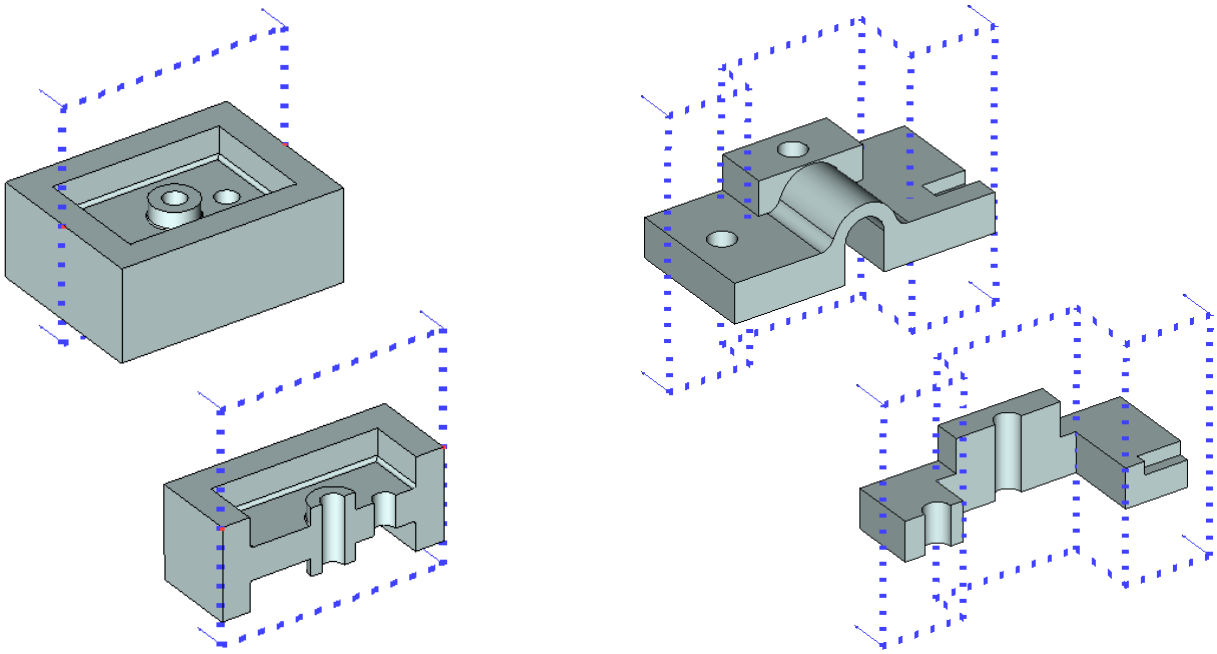
The dialog box of route parameters contains two tabs - "General" and "Transformations". The contents of these tabs is described in detail in the section "General Parameters of 3D Elements".

SECTIONS

The 3D section is an auxiliary 3D construction element in T-FLEX CAD. It is used as a secting object when creating 2D projections (for obtaining local area views and sections) in the "Cut by section" operation. The 3D section can be used when viewing objects of the 3D scene, helping quickly examine the inner model structure without changing it.



The section is displayed in the 3D scene by dotted lines indicating the boundaries of the 3D section. The arrows on the image of a 3D section indicate *the view direction*. The view direction of the 3D section defines the part of the model that would remain as a result of applying the section to a 2D projection, a model in the 3D scene or when cutting by section.



Main Approaches to Creating 3D Sections

3D section can be defined by the following methods:

- ✓ With an octant (trihedral angle);
- ✓ With a planar angle (dihedral angle);
- ✓ On the basis of a plane;
- ✓ With a parallelepiped
- ✓ As a result of extrusion of the specified curve in a certain direction. The initial curve can be specified:
  - based on a 3D view;
  - based on 2D constructions on the workplane;
  - based on a 2D projection.

3D section by octant and planar angle

To create a 3D section with an octant and a planar angle, it is sufficient to indicate the location of the secant trihedral or dihedral angle in the space. To specify the location of the section, use the special manipulator in the form of the LCS to which the view of the secant angle is attached. Position of the secant angle can be snapped to points of the 3D model and also to LCS.

The created 3D section is displayed in the 3D scene in the form of a dashed rectangle coinciding with the X-Y plane of the manipulator of the section. The boundaries of the rectangular section are imaginary and

only used to display the section in the 3D scene. The direction of the sight is parallel to the Z-axis of the manipulator of the section and has an opposite direction.

3D section based on a plane

When creating a 3D section of the given type, the location of the secant plane is specified with the help of the special manipulator. It constitutes the LCS to which the "draft" of the secant plane is attached. The location of the secant plane can be specified and controlled with the help of the manipulator and commands of its context menu.

The secant plane can also be specified by simply indicating the 3D object that defines the required geometric plane (planar face, planar edge, workplane, etc.).

The view direction of the 3D section is set by the system arbitrarily, and the user can change it, if necessary.

The boundaries of the 3D section in the 3D window are purely imaginary and are used solely to display the section in the 3D scene. The model is slashed by such section throughout the entire range of the section plane. Direction of the sight of the 3D section is determined by the system arbitrarily but the user has always a possibility to modify it.

3D section based on a parallelepiped

To create a 3D section with a parallelepiped it is sufficient to indicate dimensions and location of the secant parallelepiped. Note that when applying the section of the given type everything that is situated **outside** the parallelepiped will be cut off.

To specify location of the section, the special draggers in the form of LCS are used to which two opposite vertices of the parallelepiped are snapped. Location of the secant parallelepiped can be snapped to points of the 3D model, and also to LCS.

Created 3D section is displayed in 3D scene as a dotted parallelepiped.

3D section as a result of extrusion of specified curve

3D section can also be specified as a list body obtained as a result of extrusion of the specified curve (line of the section) in a certain direction. To do so, it is possible to use one of the three variants of creation of 3D section:

- ✓ based on 3D view;
- ✓ based on 2D constructions on a workplane;
- ✓ based on 2D projection.

These approaches differ in the way of defining the source curve of a 3D section and the direction of its extrusion. The view direction of a 3D section is arbitrarily defined by the system, regardless of the creation technique, however, the user can always flip it.

The boundaries of such 3D sections are imaginary only in the direction of extrusion of the initial curve. The extents of the section in the other direction are precisely determined by the boundaries of the

source curve. Therefore, when creating a 3D section, make sure that the base curve fully intersects the 3D model. Otherwise, only a portion of the model will be sected that is intersected by the base curve of the 3D section.

> The 3D sections, created based on 2D projection and 2D constructions on a workplane, by default are not displayed in the 3D scene. To make such section appear in the 3D window, set the flag **Show Image in 3D View** in the section parameters.

3D Section Based on 3D View

The curve that guides creation of a 3D section is specified by a set of 3D points (3D nodes or body vertices). The selected 3D points are joined by straight segments. The 3D section is constructed by extruding the resulting polyline in the normal direction to the screen (at the time of selecting the first point).

> Before creating a 3D section, you need to a pointing and the 3D scene to make it perpendicular to the desired direction of the section creation. This can be done, for example, by using the command **3VP: Set View Parameters**, selecting the desired view position with its help. Or, if the 3D section is to be made perpendicular to some face or flat edge of the model, you can use the command **3VN: Rotate Normal to Selected Element**.

A section, created based on 3D view, is displayed only in the 3D window.

3D Section Based on 2D constructions on a Workplane

The base curve of a 3D section is defined by 2D elements lying on workplane:

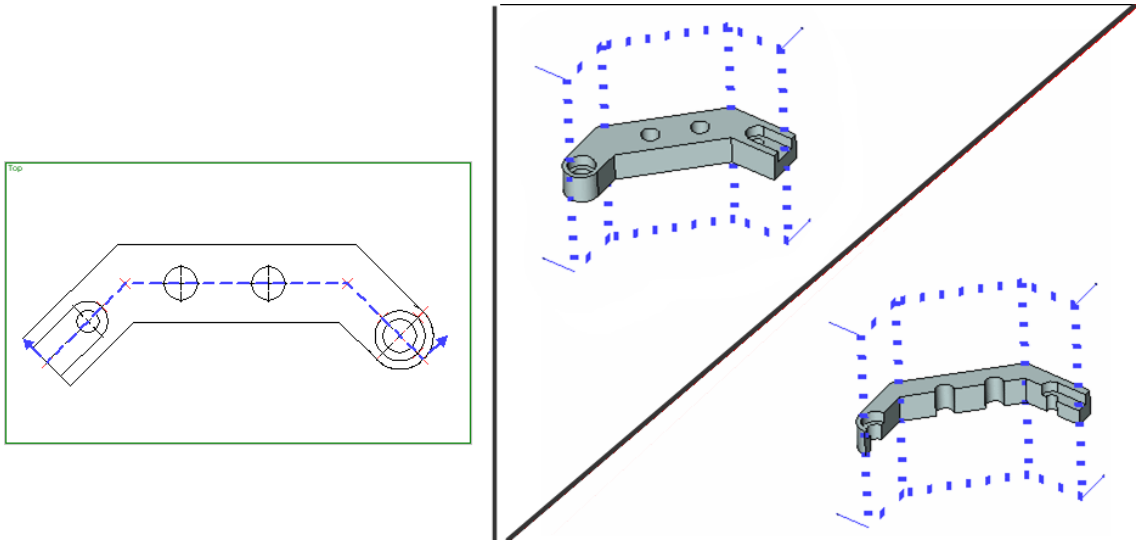
- ✓a set of 2D nodes (two and more nodes);
- ✓a 2D path;
- ✓a view notation (2D section), created by the command **SE: Create Section**.

The resulting flat curve as extruded perpendicular to the workplane, that is referenced by the source 2D elements.

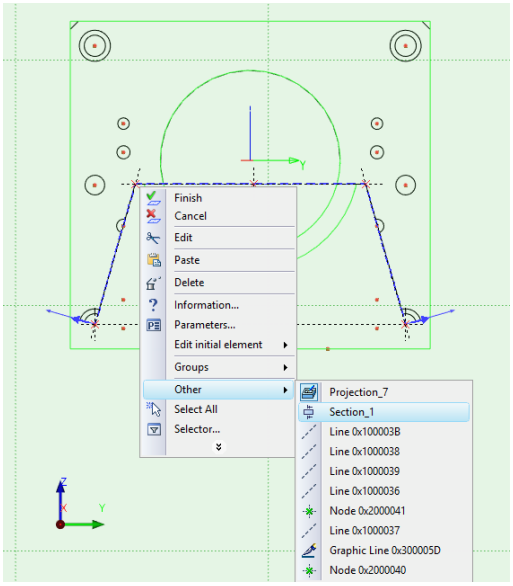
When using 2D nodes, the curve is created as a polyline (the specified nodes are connected by straight segments). The 3D section will be composed of planar segments in this case.

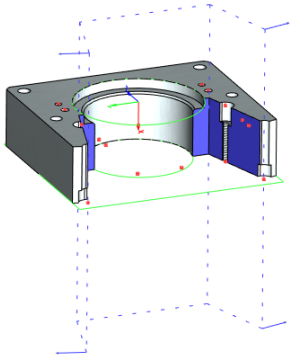
If a 2D path is selected, the base curve of a 3D section will exactly follow the path shape. The 2D path can be of arbitrary shape. Therefore, 2D paths allow creating 3D sections, containing segments of selected circle or spline surfaces.

The 3D sections, created based on 2D nodes and 2D paths, are displayed in the 2D view window by a dotted line with the arrows at the ends, indicating the view direction of the section.



A view notation is commonly used, when the 3D section is created based on existing 2D drawing. The base curve of the 3D section will repeat the shape of the 2D section, created in the command **SE: Create Section**. Such 3D section is not displayed in the 2D window. You can find out about its presence in the drawing from the 3D model tree or from the context menu for the 2D section.





Creation of 3D Section Based on 2D Projection

This way of the section creation is almost exactly identical to creating the section based on workplane. The section is constructed perpendicular to the direction of projecting the source 2D projection. The base curve of a 3D section is defined by 2D elements one the projection: a set of 2D nodes, 2D path, view notation.

> Such 3D section can be created automatically in the 2D projection creation command when constructing a local view or section based on a 2D view notation.

Selection of Objects of the Section

When creating a section, it is possible to specify a set of objects of the section. The Bodies, operations, welded joints can be selected as such objects of the section. The selected set of objects can be used in one of the three modes:

- ✓section is applied only to the selected objects;
- ✓section is applied to all elements of the model except the selected ones;
- ✓section is applied to all elements of the model (the specified set of objects is removed).

Applying Section to 3D Model

A 3D section can be used for viewing a 3D model. This helps quickly examine the inner structure of the model and evaluate the shape of the model of the setting with this 3D section.

When viewing a model with the help of 3D section, the body is "slashed" by the 3D section into two parts. The 3D scene displays only the portion have the disappointed up by the view direction arrows of the 3D section. In this way, no actual separation is performed, the model itself is left unchanged, what is changed is just the way of its rendering in the 3D scene.

To apply a section to the model, call the command **View > Apply Section**. A dialog box will be displayed, listing all sections existing in the document. Select the desired section in the list (it will become highlighted both in the 2D window and in the 3D scene) and press the button **[OK]** in this dialog. The section will be applied to the model.

There is another way of calling this command. In the command waiting mode, right click in the 3D view window. In the coming up context menu, select the command **Apply Section...**. As a result, the described dialog box will appear.

A section can also be applied to the model from the context menu of the particular 3D section (by right clicking). In this case, you do not have to select the section. The section will be applied to the model, whose context menu was called.

> The **Apply Section...** command is unavailable for complex 3D sections built by extrusion of the specified curve (on the basis of a 3D view, constructions on the workplane or a 2D projection). If such a section consists of several segments, the section can not be applied to the 3D model.

Canceling section is done by the respective command that can be launched from the context menu of the 3D section (by selecting it in the 3D window or in the 3D model tree).

CREATING SECTION

to create a section, use the command **3SE: Construct Section**:

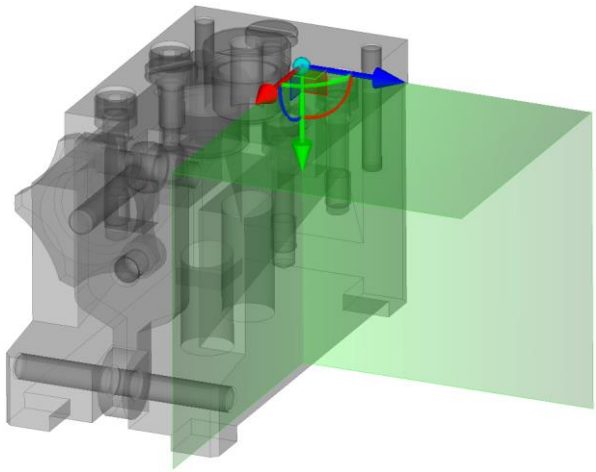
Icon	Ribbon
	3D Model → Construct → Section
Keyboard	Textual Menu
<3SE>	Construct > Section

Upon calling the command, the automenu will offer both the general system options and the options for selecting a method of creating the 3D section:

	<W>	Construct Section using Workplane
	<V>	Construct Section using 2D Projection
	<R>	Construct Section using 3D View
	<L>	Create Section by Plane
	<O>	Create Section by Octant
	<A>	Create Section by Plane Angle
	<B>	Select Bodies for Section
	<E>	Construct Section by Parallelepiped

Creation of 3D Section by Octant/Planar Angle

After selection of the or options, in the 3D scene a preliminary image of the secant angle with the manipulator in the form of the LCS at the base appears. Location of the secant angle is defined by translation/rotation of the manipulator.



The manipulator constitutes a view of the coordinate system. The coordinate axes are connected with each other by colored arcs and rectangular “areas”, the center of the manipulator is marked with a

sphere. Different parts of the manipulator are the active elements intended for specification of the following transformations:

- ✓the axes of the manipulator are intended for specification of translations along individual axes;
- ✓rectangular "areas" connecting axes of the manipulator in a pairwise manner are intended for translations across the XY, XZ or YZ planes;
- ✓arcs of the manipulator are intended for specifying rotations;
- ✓the center of the manipulator is used for an arbitrary translation (without fixing to axes).

When pointing at the coordinate axis, the user can proceed to specifying a translation along the selected axis. When pointing at the arc, the system prompts the user to specify a rotation in the corresponding plane. In the latter case, in addition to that, the axis of the manipulator closest to the cursor is selected so that the user could specify the rotation by directing the selected axis at the specific element of the model. When clicking the center of the manipulator, the manipulator itself and the secant angle that is linked to it start to dynamically move following the cursor.

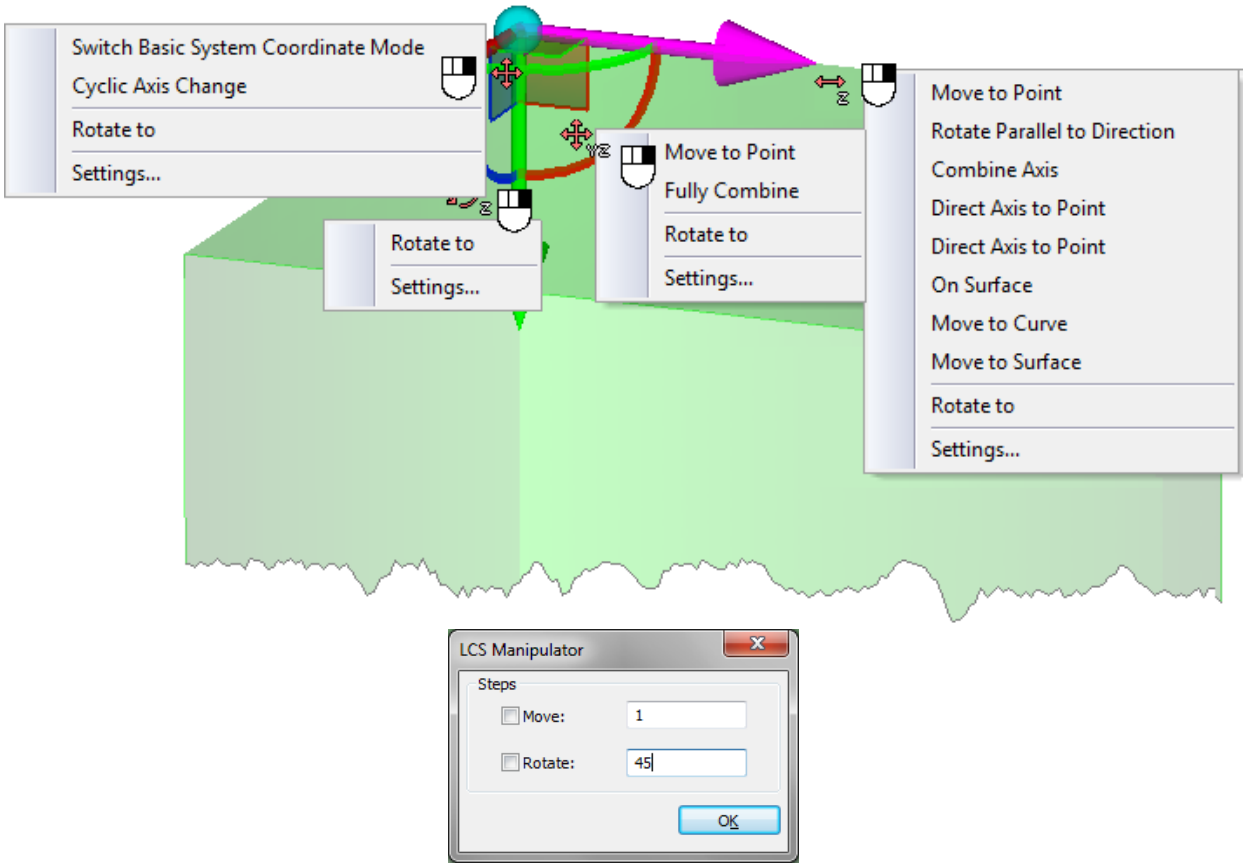
To specify a transformation with the help of the manipulator, the user must choose the active element of the manipulator by pressing . Next, without releasing the pressed button, the user can move the cursor. Transformation will be specified when the mouse button is released. It is also possible, after selection of the manipulator, release the mouse button, move the cursor and select a new element (specify a location in the space), to fix the transformation, by clicking again.

To snap the secant angle to LCS the following option of the automenu is used:

	<F>	Select coordinate system

After invoking the option it is required to indicate the desired LCS.

The context menus of the elements of the manipulator offer additional capabilities of defining orientation of the secant angle. For example, with their help it is possible to direct the selected axis of the manipulator at a 3D point or translate the axis up to the specified surface. The option that is common to all menus – "Settings…" – invokes the dialog for specifying the step of the translation of the manipulator.



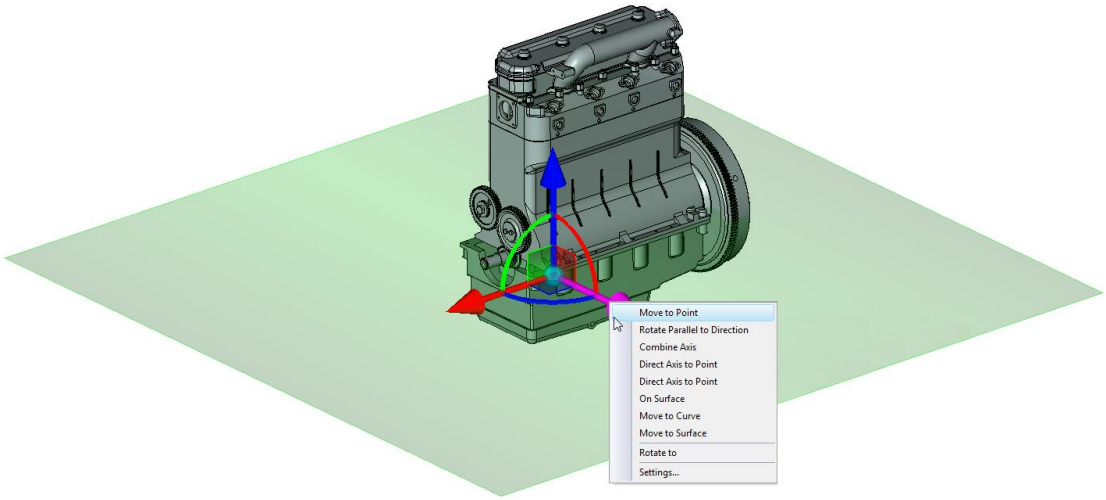
After specifying the required location of the secant angle in the space, it is required to confirm its creation with the help of in the properties window or in the automenu.

Creating a 3D section based on a plane

After selecting the option , the following two options will appear in the automenu:

	<L>	Select Plane
	<Tab>	Switch view direction
	<F>	Select LCS
	<B>	Select Bodies for Section

The green plane – a «workpiece» of the secant plane, and the special manipulator in the form of the LCS for controlling the location of the plane appear in the 3D scene. With the help of the manipulator and commands of its context menu, it is possible to specify and control the location of the secant plane.



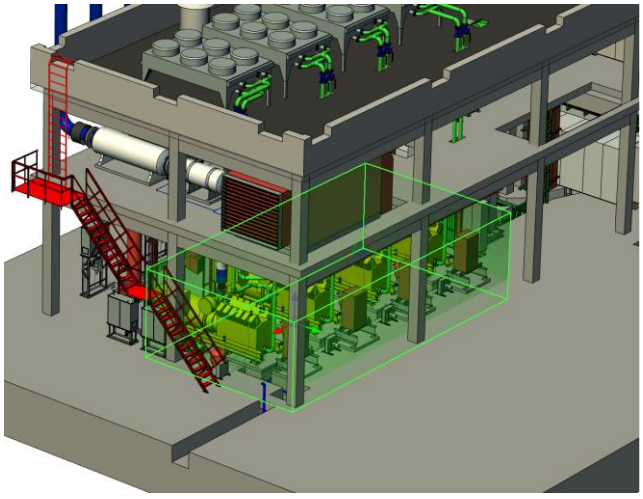
To fix the secant plane to an already existing 3D element (a face, a plane, etc.), it is possible to use the option. The drop-down list of the option contains the filters to select 3D objects that define a plane. It is also possible to fix the plane to different 3D points when moving the dragger by using double pressing of the mouse.

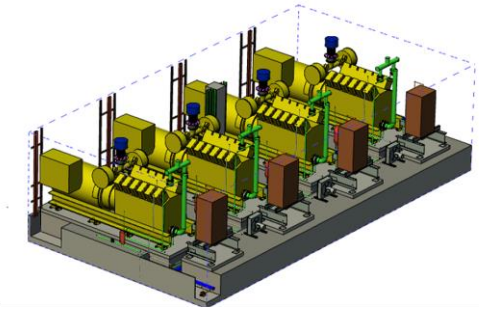
The option toggles the view direction of the 3D section being created.

Once the plane and the view direction of the section are selected, confirm the section creation by clicking in the properties window or in the automenu.

Creation of 3D Section with Parallelepiped

After selection of the option, will appear a preview image of the secant parallelepiped with two draggers in the form of LCS in the lying opposite vertices. To specify the required location and size of the parallelepiped it is required to translate/rotate the draggers.





The master vertices of the parallelepiped can be snapped to different 3D points when moving the draggers by double pressing of .

To specify the initial coordinate system, one of the vertices of the parallelepiped can be snapped to LCS with the help of the option:

	<F>	Select coordinate system

The dragger of the vertex will be superposed with the selected LCS.

Coordinates of the second vertex of the parallelepiped (associated with a second dragger) are measured with respect to the coordinate system.

Creating Section Based on 3D View

Upon selecting the option , the automenu will provide the additional options for selecting 3D points, the source curve of the section being created, and for switching the view direction of the section.

3D points can be selected by the following options:

	<N>	Select 3D node
	<V>	Select Vertex

You can select as many 3D points as necessary using these options. If some point was minced when specifying the points, cancel the selection and define the points anew. To cancel the selection of a set of points, use the option:

	<K>	Eliminate all Section Nodes

To flip the section view direction, offered by the system, use the option:

	<Tab>	Switch view direction

Upon selecting 3D points and view direction of the section, confirm its creation by the option in the properties window or in the automenu.

Creation of Sections Based on 2D constructions on a Workplane

A section can be constructed in this way either in the 2D window (on the page of the 2D drawing related to the workplane), or in the 3D window on the activated workplane.

Upon activating this mode by the option , select 2D elements defining the source curve for the section. This can be done by the following automenu options:

	<N>	Select Next Section Node
	<C>	Select 2D Path
	<M>	Create 2D Section for creating 3D Section

The workplane used as the base for the 3D section construction, is automatically selected by the system. In the cases, when another workplane can be selected (for example, if several workplanes are located on the given 2D page), the automenu will provide the option for changing the workplane:

	<W>	Select Other Workplane

To flip the view direction of the section, as in the previous case, use the option:

	<Tab>	Switch view direction

Upon selecting the source curve, the workplane and view direction of the section, confirm its creation by the option in the properties window or in the automenu.

Creation of Section Based on 2D Projection

Creation of a 3D section in this way is done by the option . Creating such section is exactly identical to the previous method (creating a 3D section based on workplane).

> The 3D section based on 2D projection can be created by the system automatically (without calling the command **3SE: Construct Section**) within the command of creating the 2D projection when constructing a local area view or section based on a 2D view notation.

Selection of Objects of Section

The set of objects of a section is specified with the help of the properties window.

> The list of objects can be specified both directly when entering the **3SE: Construct section** command (before selection of the section creation method), and during the stage of creation of the 3D section of any type. In the former case, the specified set of objects is automatically applied to all sections that are created during the current session of work with the command. In the latter case, the specified set of objects acts only for the specific section.

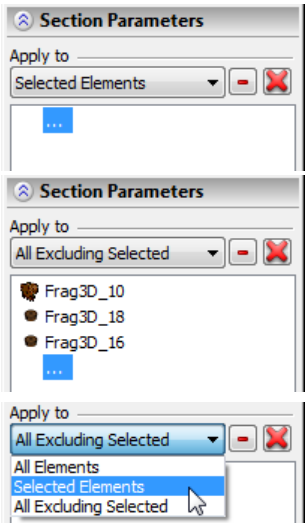
Place the cursor into the "..." line and press . The command automatically transfers to the mode of selection of objects of the section. In the automenu the following option will become active:

	<B>	Select Bodies for Section

The drop-down menu of this option contains the list of filters of the objects being selected (Body/operation/welded joint).

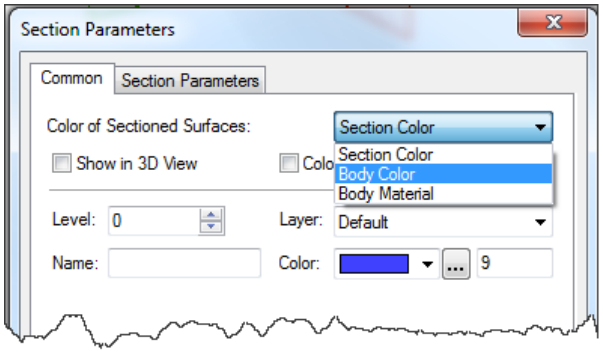
Objects can be selected in the 3D window or in the tree of the 3D model.

The way of interpreting the specified set of objects (Apply to all elements; Apply only to selected elements; Apply to all elements except selected ones) is set up with the help of the drop-down list **Apply to**.



Defining Parameters of 3D Section

Defining parameters of a section is done by the parameters dialog (the option).



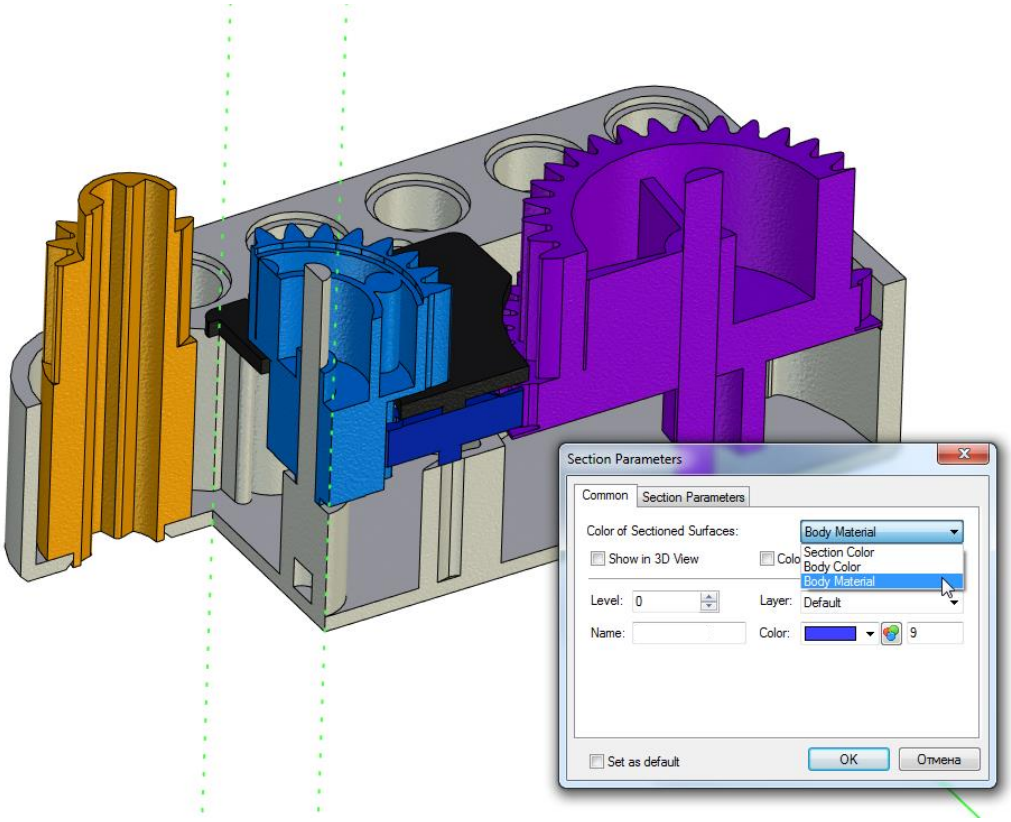
Body color. When this flag is turned on, then the surface generated as a result of applying the section will be displayed in the same color, as the model. When the flag is cleared, color is used specified in this dialog.

Show on 3D View. Defines the display of the section plane in the 3D view.

Section color.. This color is used for drawing the section in the 2D and 3D windows. Additionally, this color is used for painting surfaces resulting from slashing the model, when the **Color from solids** flag cleared.

Body material. Section is displayed using the material of the body according to material's texture, color and other characteristics.

It is possible to get different colors when sectioning multicolor assemblies. If parameter value is set to "Body Material", the visual properties of material including texture will be used.



Basic Operations of 3D Modeling

EXTRUSION

Extrusion operation creates solids by sweeping the shape-defining element – the extrusion contour. Extrusion is a most popular operation across solid modeling systems.

The operation supports two types of extrusion: extruding along an arbitrary vector, and extruding in the normal direction to the base contour surface. The extrusion type determines the direction assignment and the ways of extrusion creation.

The extrusion contour can be made of wire or sheet geometry objects. The result of extruding is a sheet or solid body, depending on the type of the contour geometry.

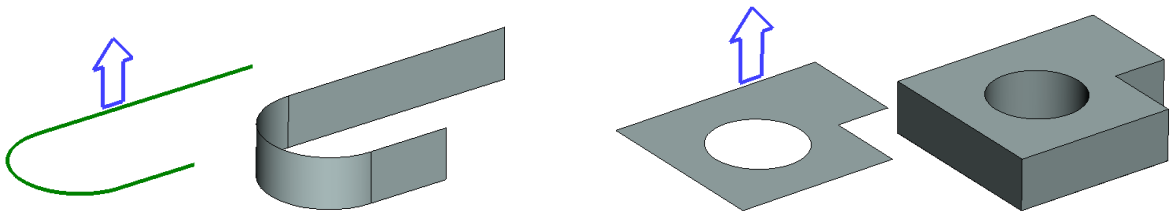
The extrusion length (or extrusion start and end) can be defined by various means, including bounding by other model elements.

MAIN CONCEPTS. OPERATION CAPABILITIES

Extrusion Contour

The extrusion contour can be made of wire and sheet geometry objects. Several elements can be selected simultaneously. In this case, the elements are joined first, and then extruded together. However, only same-type objects are allowed in the selected group of elements, either all wire or all sheet objects. These include 3D profiles and faces, or paths and edges.

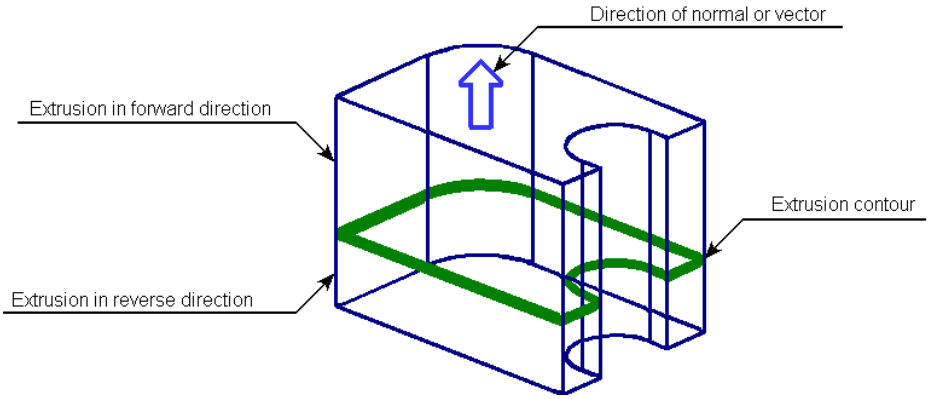
The result of extruding a wire contour is a sheet body. Extrusion of a sheet contour makes a solid body.



Extrusion Direction

The trajectory of sweeping the contour for extrusion is defined by the extrusion direction. Depending on the extrusion type, the direction can be defined by the normal to the original contour surface or an arbitrary vector.

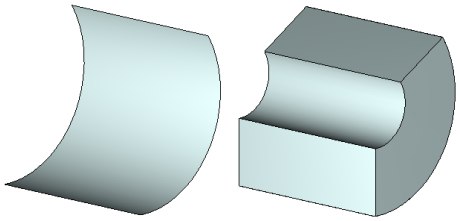
In either case, there are the forward and reverse directions of sweeping the contour. The forward direction coincides with the normal direction or the extrusion vector. Reverse is the direction opposite to the forward one. Two-sided extrusion is also supported, in which the contour is swept in both directions. By default, the system creates extrusion in the forward (“first”) direction.



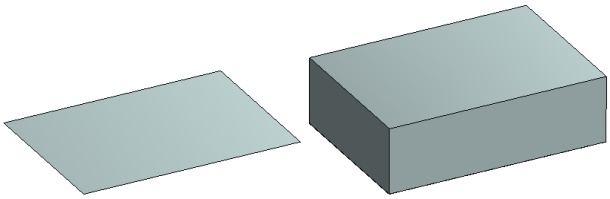
Extruding in normal direction

Extruding along the normal is the system default when selecting a sheet or flat wire contour. Each point on the contour is swept along a separate line defined by the normal direction to the contour surface at the point. In this case, extrusion can be described as an offset from the original contour defined by the specified length of the extrusion.

In real life, flat contours are extruded most often, with the same normal direction at all points of the contour. For practical purposes, one can think of the flat contour in this case being translated parallel to its original position along one common line defined by the single normal to the contour plane. Therefore, extrusion normal to the flat contour can be viewed as a special case of extruding along an arbitrary vector.



Extruding a non-flat contour



Extruding a flat contour

Extruding along an arbitrary vector

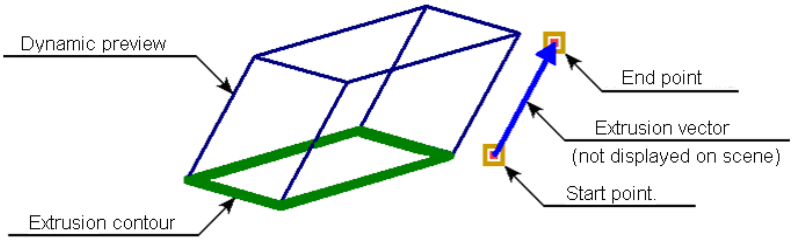
When the extrusion direction can't be defined by the normal (for example, if a non-flat wire contour was selected) or extrusion is desired in a specific direction, then its definition may include an arbitrary direction vector.

The vector of the extrusion direction can be defined in two ways:

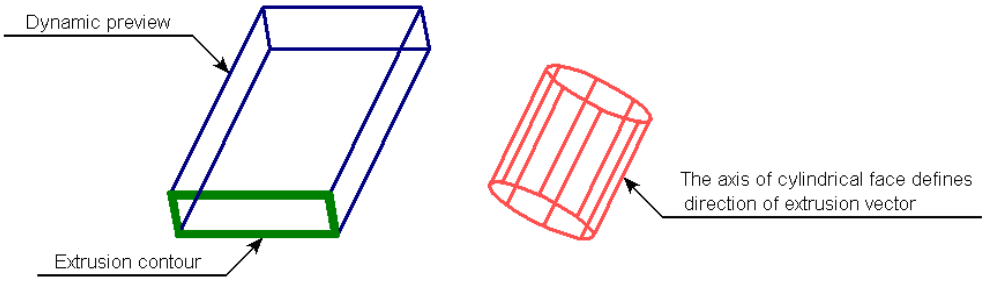
1) By specifying a pair of 3D points (the vector start and end);
2) By selecting some element of the 3D model suitable for defining a vector in the desired direction (regardless of the length).

Two 3D points define the start and end of the extrusion direction vector. 3D points can be defined by 3D nodes and vertices. One can also select other elements defining the desired point. For example, one can select an edge – in this case, the edge midpoint will be used as the 3D point. Selecting a coordinate

system defines the point at the origin. Selection of a face, which is part of a sphere or torus – yields the point at the center of the sphere/torus, and so on.



In the latter approach, the selected 3D object defines the vector of extrusion direction. The suitable kinds of objects are flat curved edges, workplanes, flat faces, surfaces (the direction is determined as the normal to the selected element), straight edges (the vector coincides with the selected edge), coordinate systems (the direction is one of the axes), faces/surfaces that are part of a cylinder or a torus (the direction of the vector is the direction of the face/surface axis).



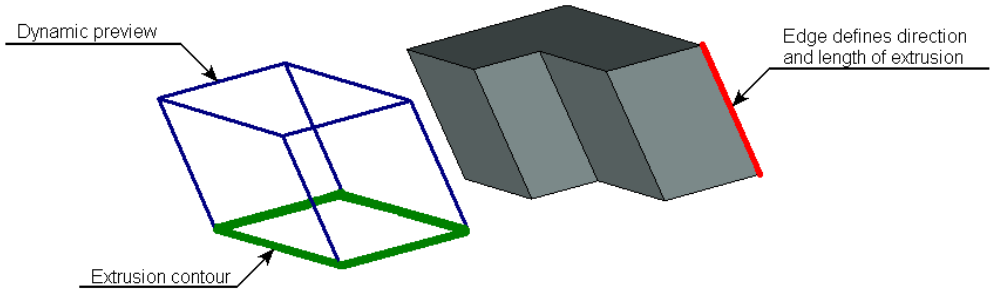
Defining Extrusion Length

The length of an extrusion can be specified in one of the three ways:

- ✓ By the length of the defining vector of the extrusion direction;
- ✓ By an arbitrary numerical value, measured from the original contour position;
- ✓ By an offset from a 3D point or a plane;
- ✓ By extrusion bounds.

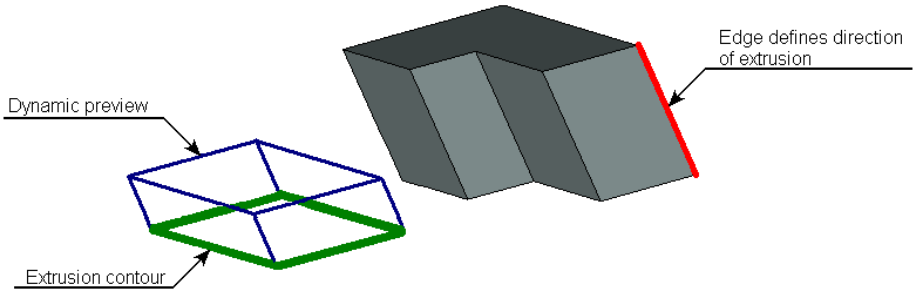
Defining extrusion length by length of direction vector

The extrusion length can be derived from the length of the defining vector of the extrusion direction in the cases when the direction is defined by two 3D points or a straight edge. The length of the extrusion vector in this case is defined by the distance between the specified points or the length of the selected edge. Extruding is performed from the original contour position straight by the amount equal to the vector length. The length of extruding in the reverse direction, if necessary, can be defined by an arbitrary number or set equal to the length of extruding in the forward direction.



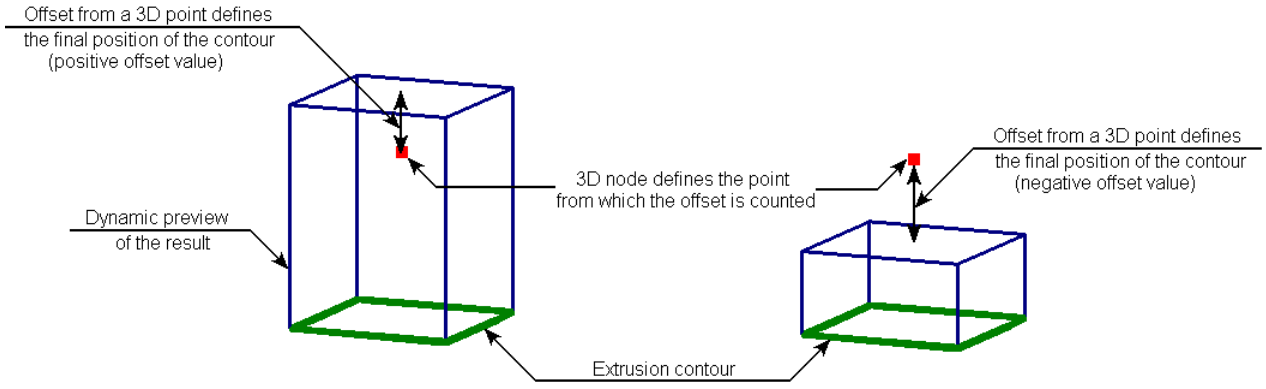
Defining extrusion length by numerical value

Regardless of the ways of defining extrusion direction, the extrusion length in the forward direction (from the original contour position) can be defined by an arbitrary numerical value. The extrusion length in the reverse direction, if necessary, is defined by an arbitrary numerical value or is set equal to the length of extruding in the forward direction.



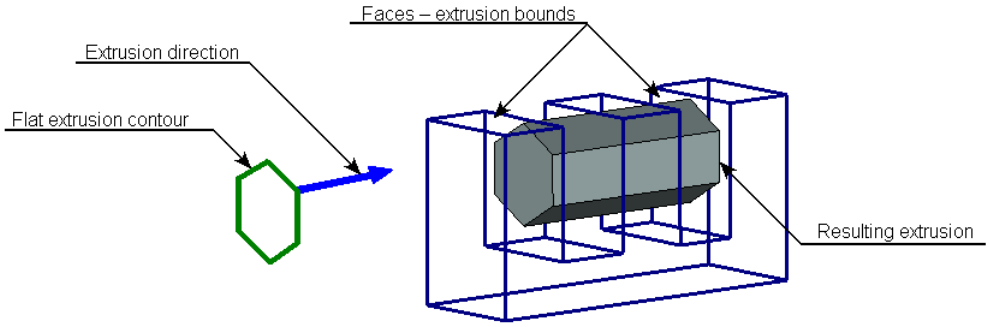
Defining extrusion length as an offset from a 3D point or a plane

An extrusion length can be defined as an offset from an arbitrary 3D point or a plane only in the case of a flat contour being extruded in the normal direction. The extrusion is performed from the original contour position to the position defined by the offset.



Defining extrusion length by extrusion bounds

When specifying the extrusion length with the help of bounds the extrusion is performed from one boundary to another.



Types of Bounds

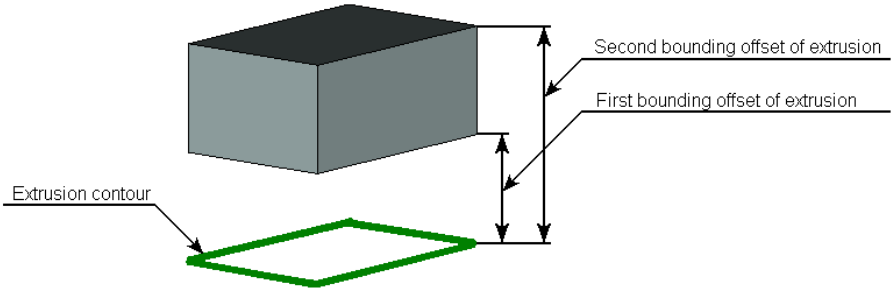
The bounding conditions can be of the following types:

Offset from the original contour plane	The body being created is bounded by a plane parallel to the original profile, at the specified distance from it
Bounding by a point	The body being created is bounded by the specified 3D point
To surface	The body being created is bounded by the specified surface
To face	The body being created is bounded by the specified face
To body	The body being created is bounded by the specified solid body
To nearest face	The body being created is bounded by the nearest face of the specified solid body in the given direction
Through all body	The body being created is bounded from both sides by the specified solid body
Through all body by direction	The body being created is bounded on one side by the original profile position, whereas on the other side - by a specified solid body
To sheet	The body being created is bounded by the specified sheet body

For the extrusion to be created successfully to the specified bounds, make sure that the extrusion contour fully reaches the 3D element specified as an extrusion bound.

Offset from the original contour plane

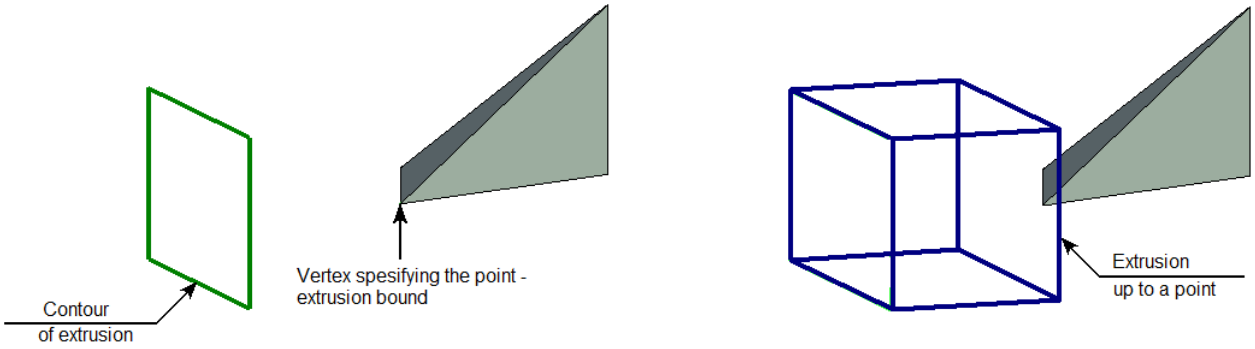
An offset from the original contour plane can be specified as one of the extrusion bounds. In this case, the extrusion is bound by the specified distance from the original contour.



Bounding by point

A geometric 3D point can be specified as a bound of extrusion in the straight or reverse direction. As a 3D point, it is possible to specify a 3D node, a vertex of the body, the midpoint of an edge or any other 3D object that is capable of defining a point.

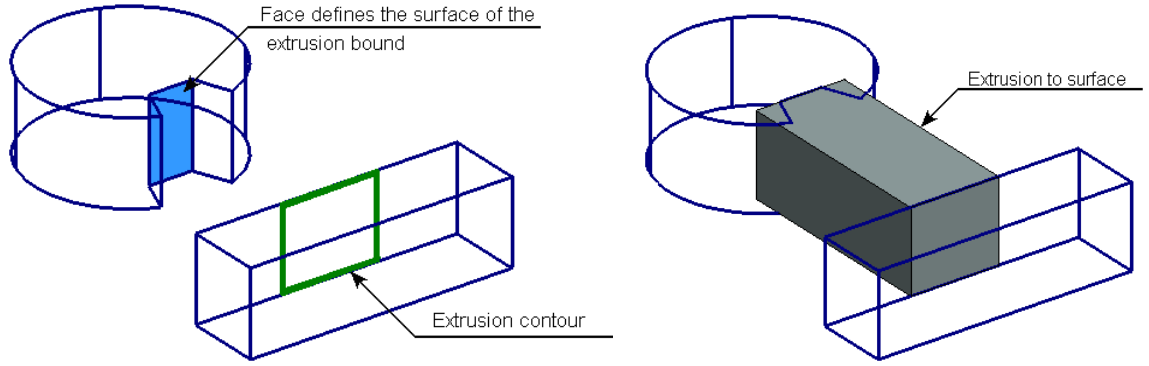
The extrusion being created will be bounded by the given point.



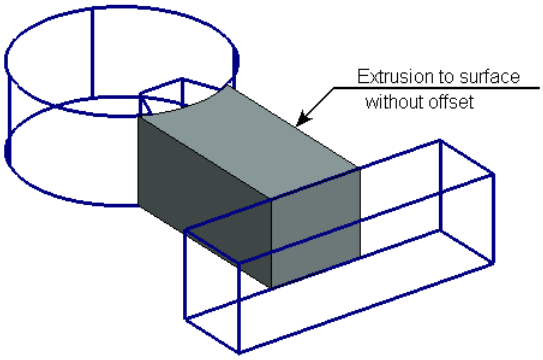
Bounding by surface

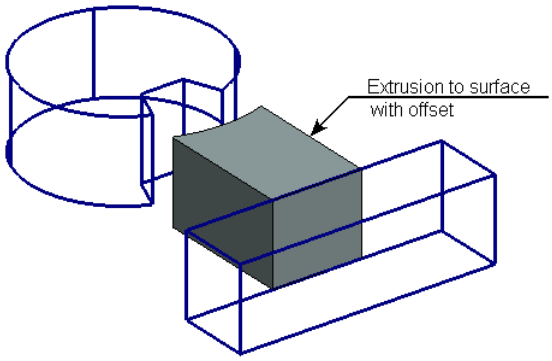
One of the bounds can be defined by a workplane or surface. Also allowed is a face, flat edge or simple body (a body with all faces on one surface) – in this case the extrusion bound is defined by the geometrical surface where the selected element belongs.

The extrusion being created is bounded by the whole surface. If the surface was identified by an element belonging to it, the surface is extended beyond the element limits whenever possible. Some complex surfaces, such as an underlying surface of a body face defined by a spline, like those created in the “Loft” operation, can’t be extended. In this case, the surface limits coincide with the limits of its defining object.



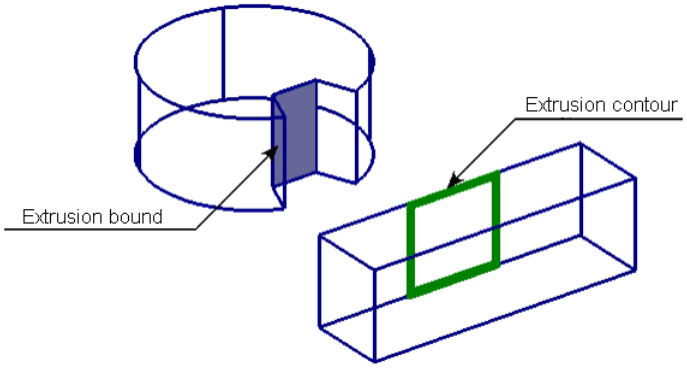
Additionally, one can impose an offset from the specified surface (positive offset is at the outer side of the surface).

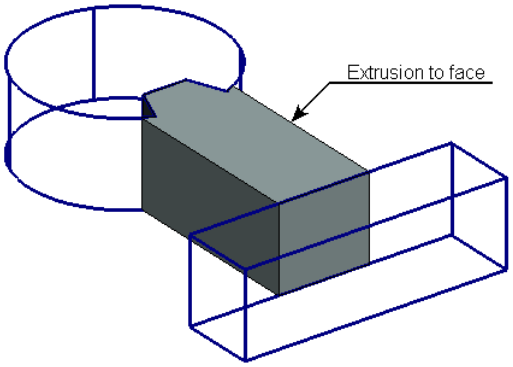




Bounding by face

If “to face” bound is selected, the body being created is extruded in the specified direction and is bounded by the specified face. If the contour only partially intersects with the selected face, the adjoining faces are used for bounding as well.

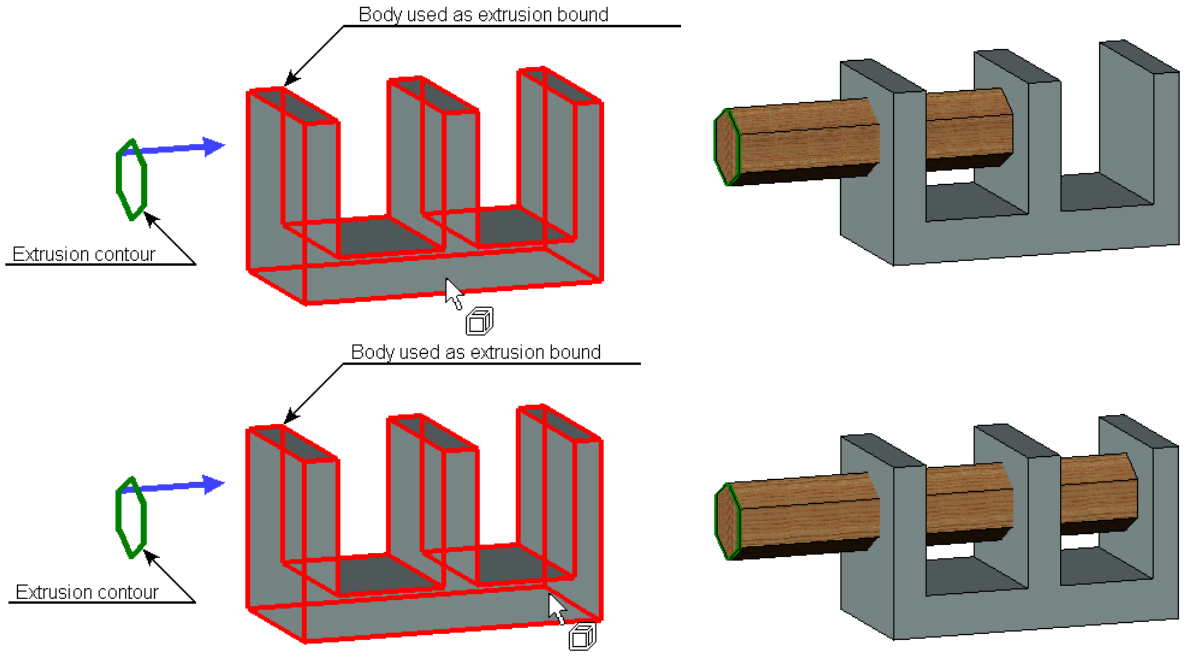




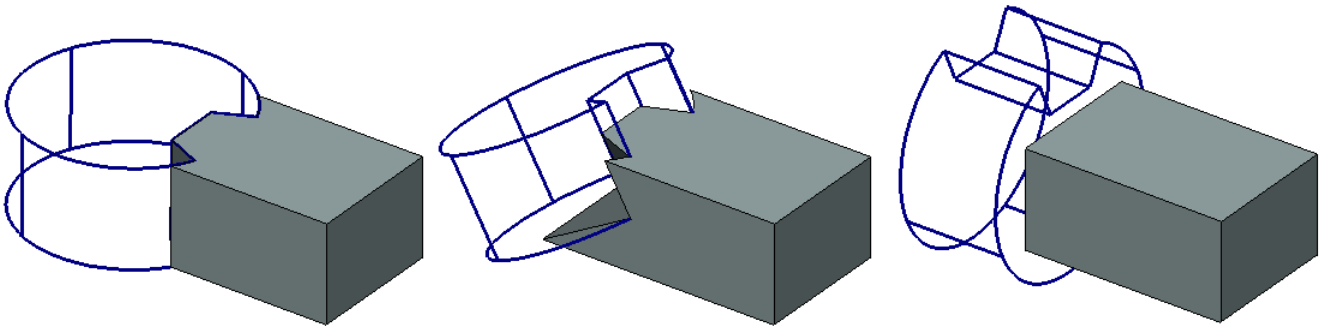
Bounding by body

A solid body can be specified as an extrusion bound. The body being created is extruded in the specified direction until it intersects with faces of the specified body. Bounding of the extrusion is done in the same way as in the case of a specific face bound.

If the bounding body intersects with the body being created several times, the intersection is processed that was nearest to the position of the cursor when clicking the body for selection.



Bounding by body is a more robust technique then bounding by a specific face, because different faces may act as the extrusion bounds upon changes in topology of the selected object. The face selected as a bound at the time of extrusion creation may later shift or disappear altogether.

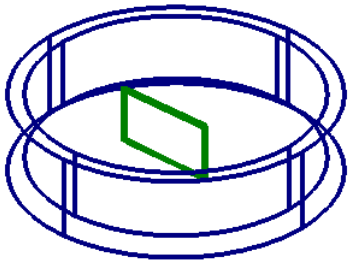


Bounding by nearest face

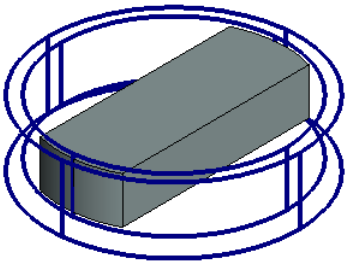
A special case of extruding to body is extrusion **to nearest face of the body** in the specified direction. In this case, it is always the first intersection with the selected body in the specified direction of extruding (forward or reverse).

The following conditions need to be observed when defining this kind of bounding:

1) The specified extrusion bounds (at least one of which is the "To nearest face") must be on different sides of the original contour. An exception from this rule is the case when the second extrusion bound is defined by an offset from the original contour plane.
2) The body, whose nearest face will be the extrusion bound, must be in the forward direction from the original contour, when it defines the first bound, and in the reverse direction, when defining the second bound.



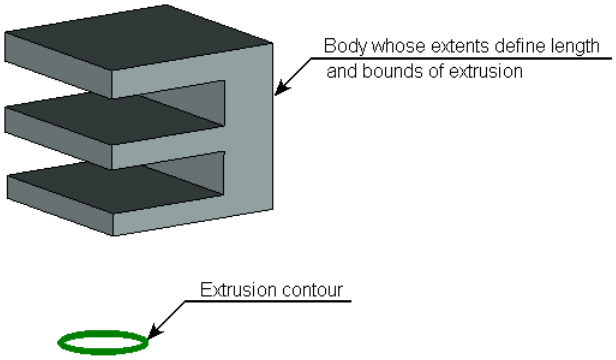
The same cylindrical body is selected for defining both bounds of the "Nearest face" type

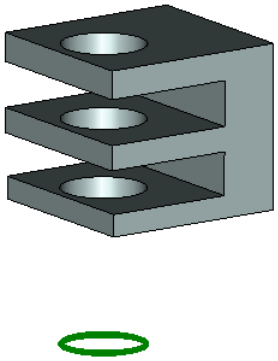


Resulting extrusion

Through all body

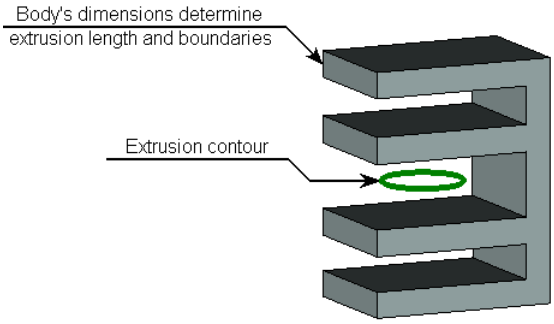
Yet another variation of extrusion bounding by intersection with a solid body is "**Through all**" bound type. What is special in this case is simultaneous definition of both extrusion bounds and required simultaneous imposing of Boolean subtraction or intersection. The amount of extruding is defined by the size of the selected bounding body. This body will be used as the target body for the Boolean. This technique can be used, for example, when creating through holes in a body.

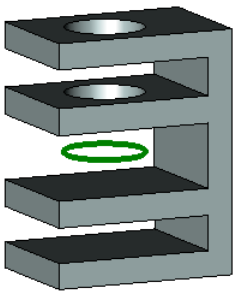




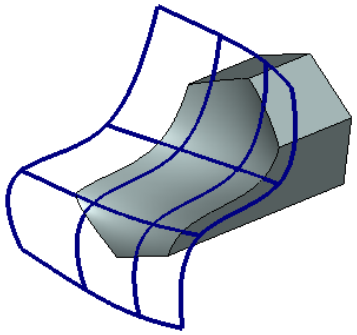
Through all body by direction

This method of bounding an extrusion is very similar to the previous one. The only difference is that in this case the extrusion is performed only in the forward direction from the source contour position. Therefore, the resulting body is bounded on one side by the source contour position, whereas on the other side - by the selected bounding body. As in the previous case, this method requires a simultaneous Boolean subtraction or intersection.





Bounding by sheet body



Bounding of an extrusion by a sheet body is done similar to the general case of bounding by a solid body. An operation (surface), face or a 3D profile can be selected as an extrusion bound. The body being created is extruded in the specified direction until the intersection with the specified sheet body.

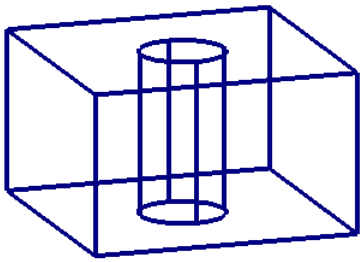
If the sheet body intersects with the one being created several times, the intersection is processed that was nearest to the position of the cursor when clicking the body for selection.

Add-on Extrusion Features

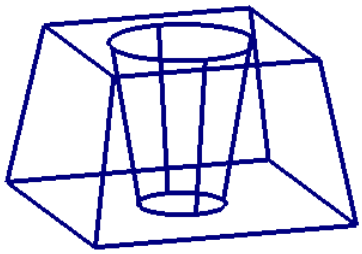
Creating tapers

Tapers can be applied automatically on the side faces while creating an extrusion. All side faces of the body being created are slanted from the original position by the specified angle.

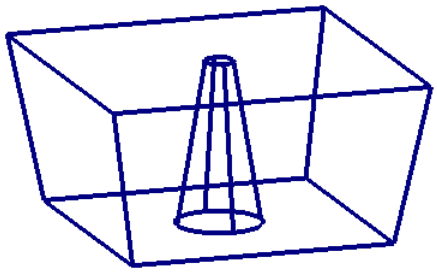
The direction of tapering depends on the specified angle and body type. If the resulting extrusion is a solid body then positive angle values correspond to tapering toward the inside of the body. For the opposite result, specify a negative tapering angle.



Solid extrusion without tapering



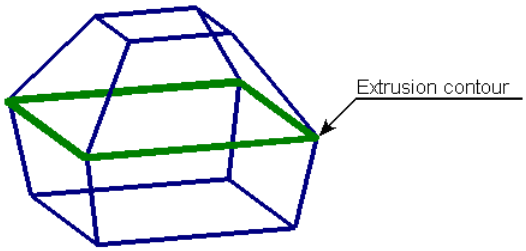
Solid extrusion with tapering
Taper angle is positive



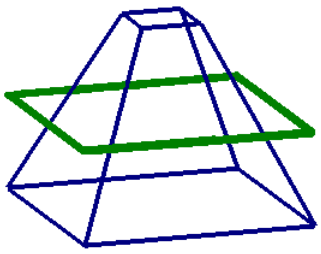
Solid extrusion with tapering
Taper angle is negative

When creating a sheet body, the choice of taper direction is left out to the system. If the system choice is different from the desired, simply change the angle sign.

The taper angle can be defined either separately for each extrusion side (if the extrusion length was defined by a numerical value or direction vector length), or once for the whole extrusion (if the extrusion length was defined by specifying bounds).



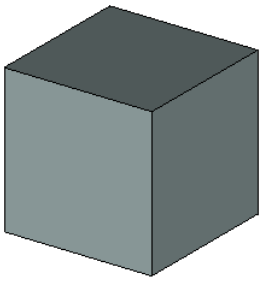
Separate tapers are created in forward and reverse directions of extrusion whose length is defined numerically



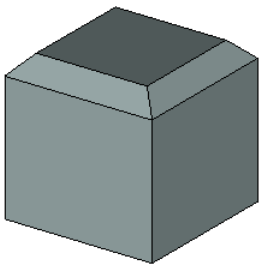
Uniform taper is created on bounded extrusion

Creating blend

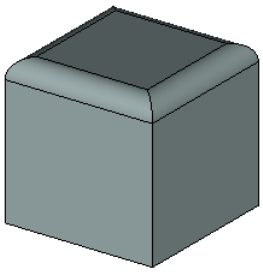
While extruding, the created side edges can be instantly blended. In the case of solid body extrusion, rounds or chamfers can also be applied at the end faces of the resulting extrusion.



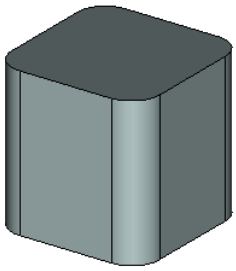
Resulting extrusion without blends



Extrusion with chamfer applied to one of the end faces



Extrusion with rounding of the end face edges



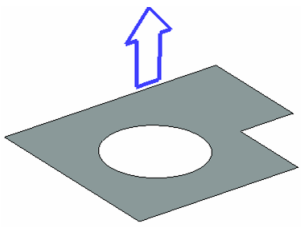
Extrusion with rounding of the side edges

Creating thin walls

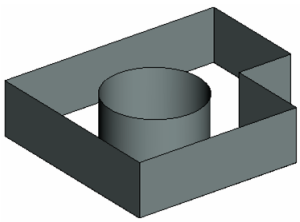
Use of an additional mode of thin wall element creation yields a different extrusion. As a result, a body is created similar to that created by the "Shell" operation. This mode is only available when extruding flat contours.

As any flat contour is extruded in the thin wall mode, a sheet body extrusion is created initially. Thereafter, it is processed depending on the selected wall type:

No thickness – the resulting extrusion is left as is (used for creating a sheet body based on a sheet contour; for a wire contour, the result is the same as in regular mode);

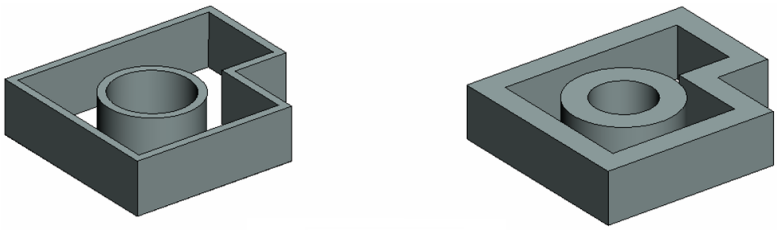


Original sheet profile



Sheet body

With thickening the side walls by various means (inward, outward, symmetric, double-sided with different thickness in the outward and inward directions from the original face) – this creates a thin-wall solid body from the sheet body by thickening the walls.



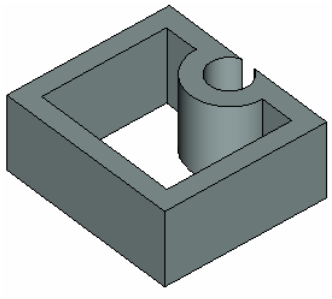
Thin-wall solid body

The new faces in the thin-wall body are created by extruding an offset contour of the original one. The shape of the faces depends on the choice of handling the gaps when creating an offset contour:

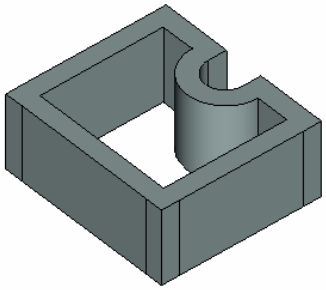
By curve – The offset curves of the original contour are prolonged until intersect.

By line – The offset curves of the original contour are extended at their free ends by tangent line segments.

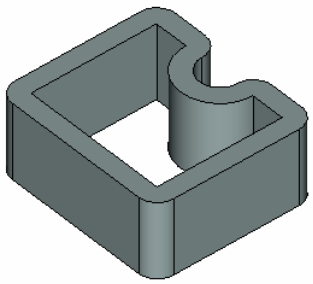
Round – The free ends of the neighboring offset curves are connected by rounds.



Outward thin-wall body creation with gaps handling "by curve"



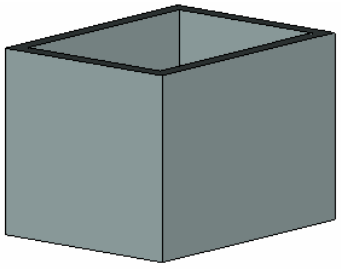
Outward thin-wall body creation with gaps handling "by line"



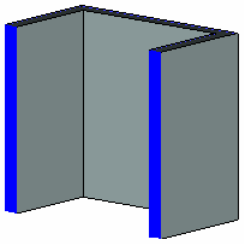
Outward thin-wall body creation with "round" gaps handling

Creating "bottom" and "top"

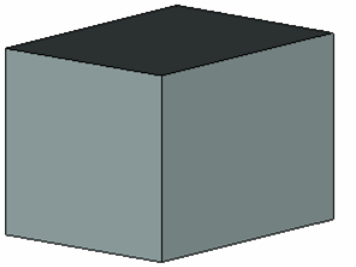
As a thin-walled body is created, only side walls exist. If necessary, one can automatically add "bottom" and "top" – the capping walls of specified thickness to the resulting body.



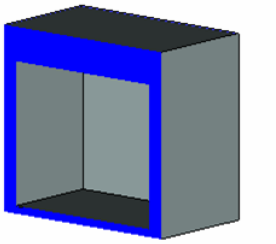
Resulting extrusion of a wire contour with thickening of side walls



Resulting extrusion of a wire contour with thickening of side walls (clip plane view)



Resulting extrusion of a wire contour with thickening of side walls and adding "bottom" and "top"

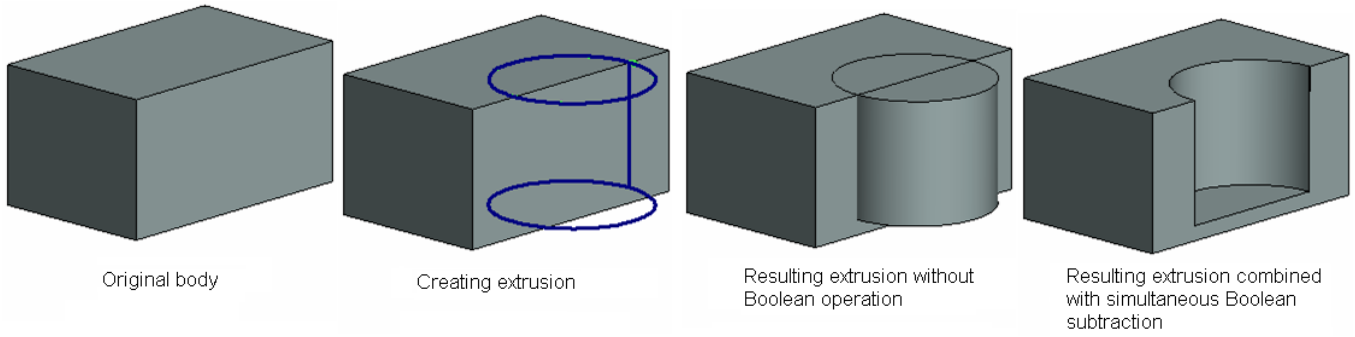


Resulting extrusion of a wire contour with thickening of side walls and adding "bottom" and "top" (clip plane view)

This feature is not available when boundaries are specified for the extrusion.

Using created body in Boolean operation

To speed up modeling, an option is provided for combining an extrusion being created with a simultaneous Boolean operation.



CREATING EXTRUSION OPERATION

Extrusion operation is created by the command **3X: Create extrusion**. The command can be called in one of the following ways:

Icon	Ribbon
	3D Model → Create → Extrusion
Keyboard	Textual Menu
<3X >	Operation > Extrusion

The automenu and the property window are used together in the command. These work together and enhance each other. The state of the automenu depends on the current step in defining the operation and on the purpose and type of the objects being selected.

The following steps are to be done to create the operation:

- ✓Select extrusion contour
- ✓Define extrusion direction (optional when extruding in normal direction)
- ✓Define the length or extrusion bounds (optional in some cases)
- ✓Define additional parameters (taper, blend, thin-wall mode, create Boolean operation) (optional)
- ✓Confirm operation creation

Selecting Extrusion Contour

To create the operation, first define the extrusion contour. Upon calling the command, the following option automatically activates in the automenu:

	<R>	Select Contour

This option contains a list of filters for selecting 3D element allowed as extrusion contour: loop, edge, face, 3D profile and other wire and sheet objects. When selecting elements, pay attention to the active filters. Use of filters is described in details in the chapter “Basic 3D terms and principals of modeling with T-FLEX CAD”.

To define the extrusion contour, move the cursor over the desired element in the 3D window. The pointed at element will be pre-highlighted, and the cursor gain a tooltip with the element name. Click to select.

> 3D profiles that are based on hatches can be selected by picking the original hatch in the 2D window.

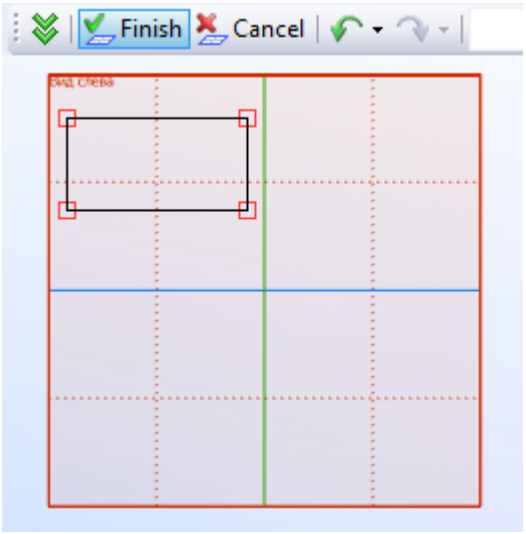
Upon selecting the extrusion contour, the option will appear in the automenu for rejecting the contour:

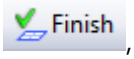
	<H>	Cancel contour selection

Calling the command with automatic contour selection

A slick way of handling an extrusion is calling the command with automatic selection of the extrusion contour.

Enter the "drawing on workplane" mode (see the chapter "Workplanes") and draw the extrusion contour.

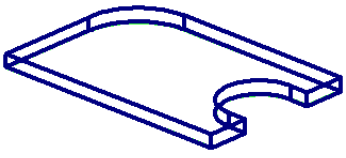


Then, without leaving the active workplane mode via 

, call the command **3X: Create extrusion**. The system will then automatically create a 3D profile based on the "continuous"-type graphic lines or a hatch, and will select it as the extrusion contour.

Defining Extrusion Direction

Upon selecting a sheet or flat wire contour, the system automatically offers to create an extrusion in the normal direction. This is indicated by the wireframe preview of the body being created, displayed on the 3D scene. There is no need in this case for defining the extrusion direction. The user can alter the kind of extrusion simply by defining a different extrusion direction by specifying a direction vector.



If a preview is not displayed on the 3D scene, extrusion in the normal direction can't be created from this contour.

The following option lets define extrusion direction vector by a 3D model element:

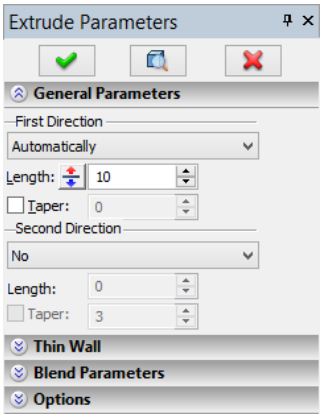
	<D>	Select Direction

This option lets select a 3D element suitable for defining the extrusion direction vector. The pull-down list under this option contains filters for selecting respective objects. When selecting elements, pay attention to the active filters.

The following automenu options let define the extrusion direction vector by two 3D points (activate sequentially):

	<F>	Select starting Point of Extrusion
	<S>	Select ending Point of Extrusion

A 3D point can be defined by selecting a 3D node, a vertex on a body, profile or path. Besides, selecting an edge yields the edge midpoint as the 3D point (for straight edges) or the center (for edges that are full circles, arcs or ellipses). Selecting a face that is part of a sphere or torus yields the sphere/torus center as the point. A coordinate system selection yields the origin as the 3D point.



The set of the elements allowed for selection is defined by the state of the selection filters in the pull-down lists of the mentioned options.

Besides the automenu options, directions can be defined via the property window dialog. The button in the dialog helps quickly change the direction of the selected vector or the surface normals to the opposite.

The last option in the direction-defining group of the automenu rejects the vector selection:

	<K>	Reset Direction selection

Defining Extrusion Length

The means of defining the extrusion length is selected in the section **General Parameters** of the command property window. This section contains two groups of parameters – **First Direction** and **Second Direction**. Each group contains a list of options for the ways of defining the length in the specified direction or the types of bounding conditions.

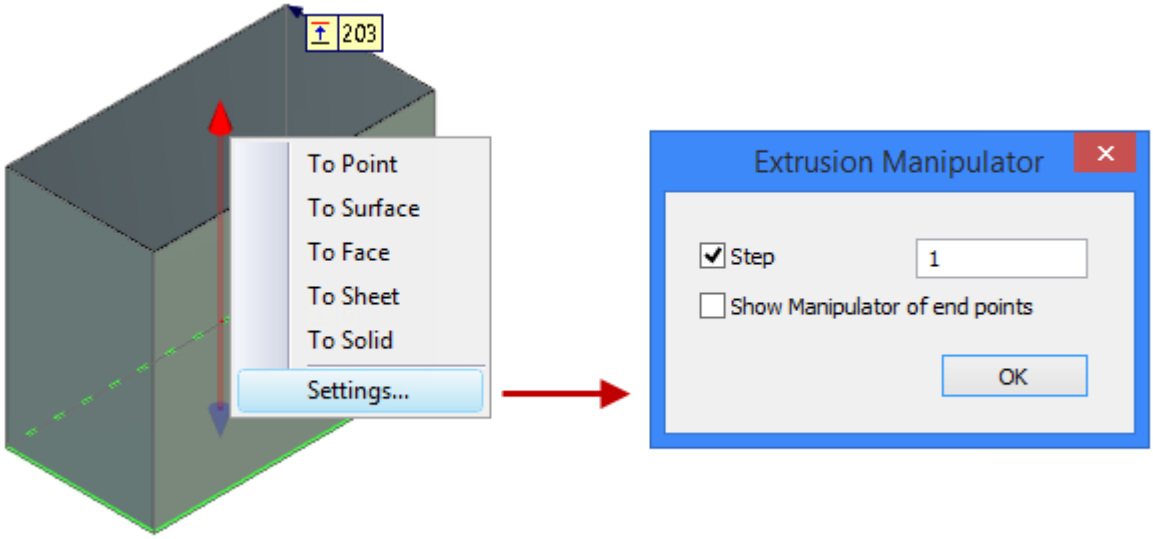
Manipulator of extrusion

To specify the length and also the direction of the extrusion, it is possible to use the special manipulator

in the form of a large two-directional arrow colored in two colors. Positive direction of the extrusion is marked with red color, negative – with blue color. Manipulator has the context menu (invoked by pressing) in which the additional options are available:

- ✓ setting the step of the manipulator;
- ✓ displaying additional elements of the manipulator – small spheres at the arrow ends (used for specifying the direction of the extrusion with two 3D points);

selection of the type of the extrusion bounds (of the bounding surface, face, Body).



All elements of the manipulator (the axis of the arrow, arrow ends in the form of cones or the hidden by default points-spheres at the arrow ends) have their own functionality. Clicking the sphere at the arrow end transfers the command into the mode of specifying the first and the second points of the extrusion direction. Clicking the axis of the arrow transfers the command into the mode of selection of the extrusion direction. Clicking the arrow end (cone) or dragging the arrow allows us to specify the bounds of the extrusion by the numerical value or via fixing to a geometric point.

Defining extrusion length by direction vector length or numerical value

The default option for extruding in the first (forward) direction is **"Automatically"**. It allows the system to automatically choose a means of defining the extrusion length in the first direction (depending on the way of defining the extrusion direction).

When defining the extrusion direction by a vector, the system tries to define the extrusion length in the first direction by the vector length. This is possible only when the vector is defined by two 3D points or a straight edge. The input box of the extrusion length in the first direction becomes inaccessible for editing. The numerical value displayed in this box reflects the value derived from the vector length. To override this mode, one can force the extrusion length input mode (in the first direction) by a numerical value. To do this, simply change the option of defining the length to **By value**.

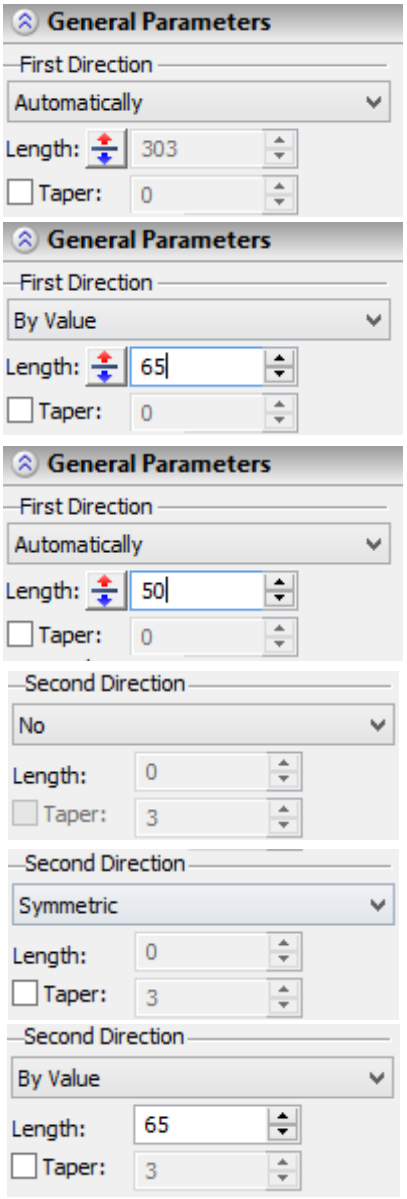
When defining the direction vector by other means (for example, by the axis of a cylindrical face or normal to a surface), as well as in the case of normal extrusion, the extrusion length can't be defined by the length of the direction vector. In this situation, the system will offer to enter the extrusion length as a numerical value. The **Length** input box will become accessible for input. In it, specify the desired value of the extrusion length (in the first direction).

By default, extruding is not performed in the second direction. The **Second Direction** group has the option **No** set in this case. The input boxes of the extrusion length and taper in the second direction are inaccessible.

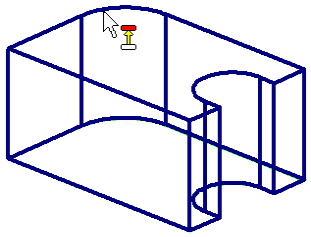
To perform extruding in the second direction, select another option of defining the length from the list. To have the extrusion length in the second direction equal to the extrusion length in the first direction, select the option **Symmetric**. The **Length** input box stays inaccessible in this case.

To define the extrusion length in the second direction by an independent numerical value, select the option **By value**. The value input box will become accessible.

The wireframe preview on the 3D scene shows the resulting extrusion with the specified parameters. Besides, it can also be used for inputting the value of the extrusion length both in the first and in the second direction.



As the cursor approaches the preview of the body being created, it gets marked as (for extruding in the first direction) or (for extruding in the second direction). Dragging the mouse with the depressed dynamically changes the extrusion length. The numerical value of the current length is reflected in the property window.



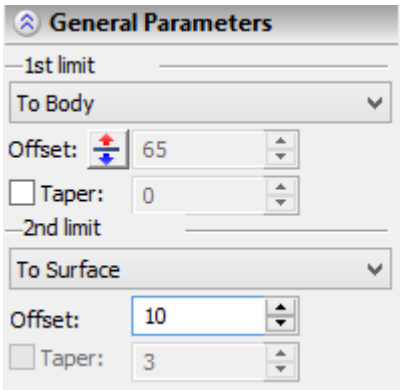
The sampling of mouse input depends on the amount of zoom into the object. For example, by zooming in one can refine the sampling step by 10 times.

Note that in the mode of defining the length of for extrusion in the first direction by the length of the extrusion vector, the wireframe preview on the 3D scene allows to change dynamically only the length in the second direction. In the case of the extrusion length in the second direction being defined by the option "Symmetric", the length in that direction can't be modified by the mouse either.

Defining extrusion bounds

The mode of defining the length by boundaries is represented with the following items in the lists of methods to define the extrusion length: **Through All**, **Through All by Direction**, **To Surface**, **To Face, To Body, To Sheet, Offset, To Nearest Face**.

Defining extrusion bounds is done by combined use of the property window and the automenu. In the general parameters of the property window, select the type of each bound in the first and second direction. Parameters of each bound can also be defined there, if necessary.



The names of the groups ("First Direction", "Second Direction") for specifying bounds are essentially arbitrary. The bound positions are generally not related to the extrusion directions. The exception here is the bounding condition of the "To nearest face" type that is defined for a particular extrusion direction.

After selecting the bound types, one of the following options become available in the automenu:

	<T>	Select 1st limit of extrusion
	<B>	Select 2nd limit of extrusion

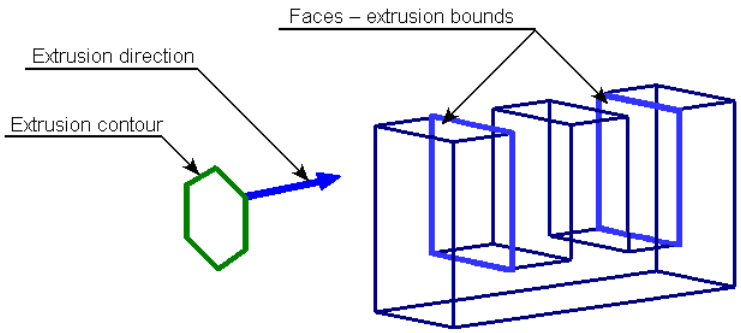
These options are used for selecting 3D elements defining the bounding conditions. These options may have pull-down lists containing filters for selecting 3D elements. The presence and contents of a list are determined by the specified type of the respective bound.

The last option in this automenu group rejects the selection. Depending on the state of the command, this may be one of the following icons of rejecting a bound:

	<U>	Discard 1st extrusion limit
	<U>	Discard 2nd extrusion limit
	<U>	Discard extrusion limits

The option is available while defining the first bound (under the active option). The option is available while defining the second bound (under the active option). In other states of the command (for example, under the option of contour or direction selection), the option is displayed for rejecting both bounds .

When using bounds, the extrusion direction vector is displayed in the 3D window by a blue arrow. The elements selected as bounds are also outlined (by default, in *blue* color).

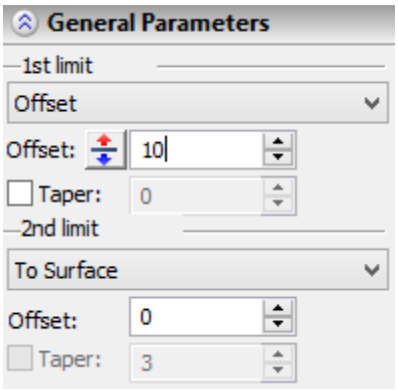


The wireframe preview of the extrusion being created won't be displayed in this case. To preview the resulting operation, use the preview option in the automenu or the respective button in the property window:

	<F5>	Preview Operation Result

The type of the first bound can be selected among extrusion parameters in the first direction. The following options can be selected from the list:

Offset. This is the condition type "Offset from the original contour plane". In the extrusion parameters, enter the value of the offset from the original contour plane in the "**Offset**" input box.



To surface. This is the condition type "To surface". On the 3D scene, select a worksurface/workplane, face, flat edge, simple body or profile. The list of elements allowed for selection is defined by the state of the filters of the option .

If necessary, one can enter the value of the offset from the specified surface in the **Offset** input box.

To face. This is the condition type "To face". On the 3D scene, select a bounding face for the extrusion.

To body. This is the condition type "To body". On the 3D scene, select a bounding body for the extrusion. If the bounding body intersects with the extrusion several times, the intersection is processed that was nearest to the position of the cursor when clicking the body for selection.

To nearest face. This is the condition type "To nearest face". On the 3D scene, select a body with the nearest to the extrusion face serving as the extrusion bound.

The specified body must be located properly. For the first bound, the selected body should be in the first extruding direction from the extrusion contour. For the second bound, the body should be in the second direction.

Through all, **Through All by Direction**. These are the conditions of the types "Through all body" and "Through all body by direction", respectively. These boundary conditions do not require the second bound definition.

If any of these types of boundary conditions is used, then the mode of creating a Boolean operation ("Subtraction" or "Intersection") is enabled automatically. The body, whose extents define the extrusion length, is treated as the target body for the Boolean.

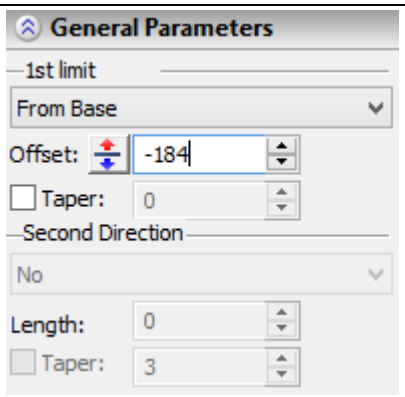
To sheet. This is the condition type "Bounding by sheet body". On the 3D scene, select a sheet body, face or 3D profile. The list of elements allowed for selection is defined by the state of the filters in the option .

The type of the second bound is defined among the extrusion parameters in the second direction in the same way. The list of 3D elements allowed for selection for each type of the bounding condition is defined by the state of the option .

Defining extrusion length by an offset from a 3D point or a plane

This method of defining an extrusion length is used, if the **From Base** item is chosen in the properties window or one of the following options is selected in the automenu:

	<Ctrl+N>	Select Starting Point of Extrusion
	<Ctrl+P>	Select Plane setting starting position of Extrusion



Both options have drop-down lists with the filters to select geometrical objects defining a 3D point or a plane.

A point or plane selection can be canceled using the option:

	<Ctrl+F>	Cancel selection of Extrusion Base

Defining Add-on Operation Features

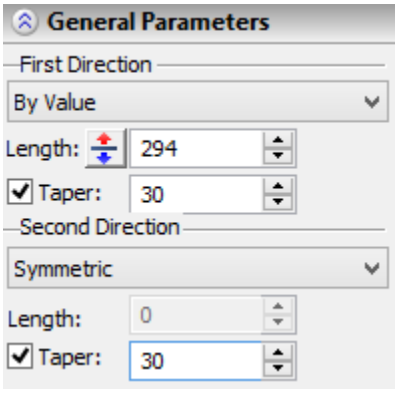
The result of add-on extrusion features is not reflected on the preview of the body being created on the 3D scene. To preview those, use the option .

Creating tapers

The value of the taper is entered in the property window. In the case of defining the extrusion length by an arbitrary numerical value or by the length of the direction vector, the separate values are specified for the tapers in the first and second direction.

To create a taper in either extruding direction, check the **Taper** item in the respective group (**First Direction**, **Second Direction**). The input box then becomes accessible for entering a numerical value for the taper.

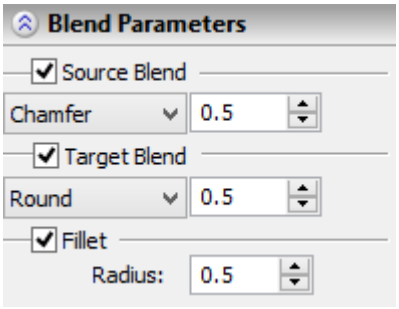
If the extrusion length is defined by bounds then tapering is uniform across the extrusion extents. Its numerical value is defined in the group of the first extrusion bound (the **First Direction** group).



Creating edge blends

The parameters of various blends can be defined in the **Blend Parameters** section of the property window:

The **Target Blend** and **Source Blend** groups define blending parameters on the end faces of the extrusion. To create a blend, check the respective item, select a blending type from the list – **Round** or **Chamfer** and specify the amount of blending.



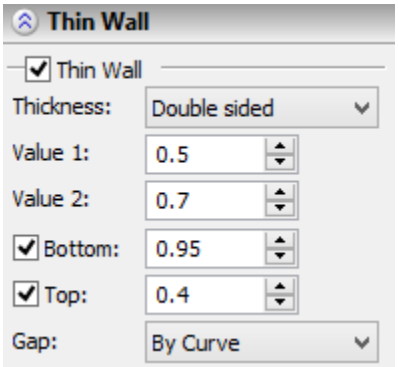
> The source face is considered the first end face when moving through the extrusion along the first extruding direction.

The **Fillet** flag sets the mode of rounding side edges of the extrusion. The rounding radius is defined by the **Radius** parameter.

Creating thin walls, "bottom" and "top"

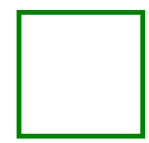
The parameters of the thin-wall extrusion can be defined in the section **Thin Wall** of the property window. To turn on the mode of thin-wall body creation, check the item **Thin Wall**. With the item checked, the following parameters of this mode become accessible:

Thickness. Defines the way of thickening the walls of the thin-wall body being created. The following options can be selected from the list:

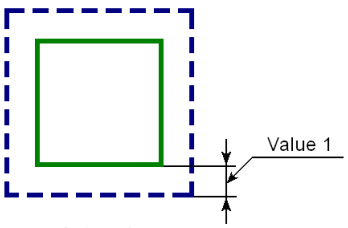


No. Without thickening of the side walls (the resulting extrusion will be a sheet body).

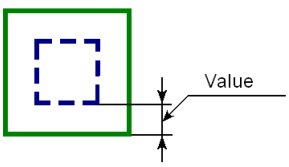
Outward, Inward, Symmetric, Double sided. Define various ways of thickening. The result of using each of these options is shown on the diagrams. The wall thickness is defined by the parameters **Value 1** and **Value 2**.



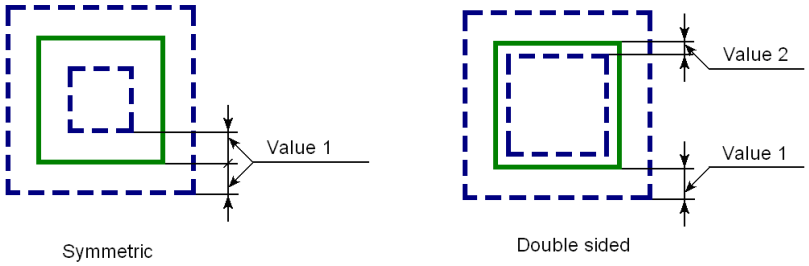
No (no thickening)



Outward



Inward



Symmetric Double sided

The ways of handling the gaps between the created offset faces are defined by the parameter **Gap**: **By line**, **Round**, **By curve**.

To create the "bottom" and "top", check the respective items in the dialog and specify the desired thickness of these elements.

Using the created extrusion in a Boolean

To create a Boolean operation, do the following:

Turn on the Boolean creation mode by the automenu option:

	<Ctrl+B>	Select original body for Boolean operation

A Boolean is created when the icon is pushed.

Select the type of the operation from the pull-down list under the above option:

	<Ctrl+'+'>	Addition
	<Ctrl+'-'>	Subtraction
	<Ctrl+'*'>	Intersection
	<Ctrl+I>	Smart mode

Operating principles of the Smart mode:

- ✓If the body being created and the existed selected body have intersection of volumes - the type of Boolean operations: subtraction.
- ✓If the body being created lies entirely within the selected body - the type of Boolean operations: subtraction.
- ✓If the body being created touches and selected body touch each other - the type of Boolean operations: addition.
- ✓If conditions set out above are not met, or a selected body lies inside the created, or error occurred determining the type of penetration, then, Boolean type is not defined - the Boolean operation will not be created.

Select the target body for the Boolean (optional in some cases), using the automenu option:

	<Ctrl+T>	Select Target Body for Boolean

If only one body exists in the scene, it is selected automatically. The new body created by the extrusion operation becomes the tool body of the Boolean.

Upon confirming operation creation, first the body is created by the extrusion operation, and then the specified type Boolean is performed.

ROTATION

Rotation operation creates solids by rotational sweeping of the shape-defining element – the rotation contour about an axis in space. The surface of the profile can be arbitrarily positioned with respect to the axis, but is not allowed to intersect the axis. Flat contours may not be orthogonal to the axis of revolution.

The rotation contour can be made of wire or sheet geometry objects. The resulting rotation is a sheet or solid body, depending on the type of the contour geometry.

The axis of revolution can be defined by any suitable 3D elements: 3D nodes, edge, axis of a surface of revolution, coordinate axis, etc.

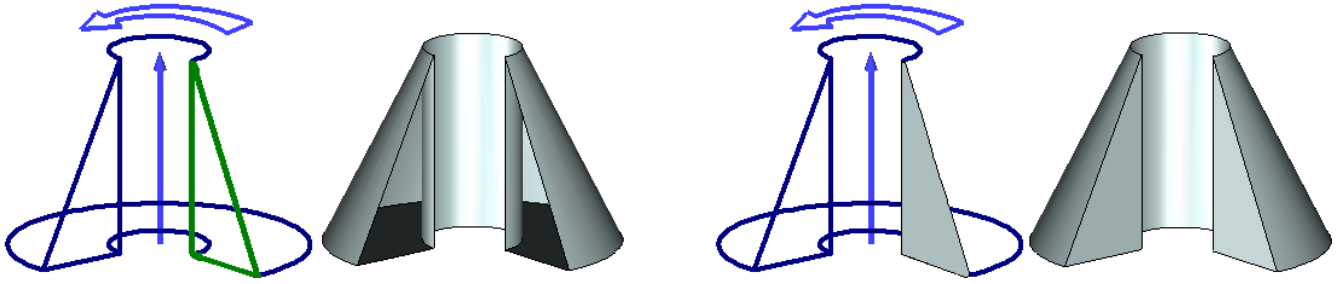
MAIN CONCEPTS. OPERATION CAPABILITIES

Rotation operation has much in common with the extrusion operation, since both operations belong in kinematical operation group. Both operations have the same rules of selecting the original contour, create bodies of the same types and share the same additional functionalities. Therefore, in this chapter we won't be concerned with the rotation operation capabilities that are common to both operations and were already described in the "Extrusion" chapter.

Rotation Contour

The rotation contour can be composed of one or several wire or sheet objects. If selecting several objects, only same-type objects are allowed, either all wire or all sheet objects.

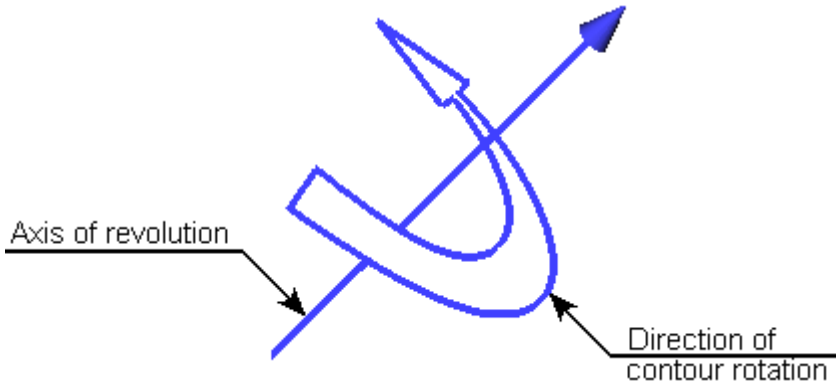
The result of rotation of a wire contour is a sheet body. Extrusion of a sheet contour makes a solid body.



Axis of Revolution

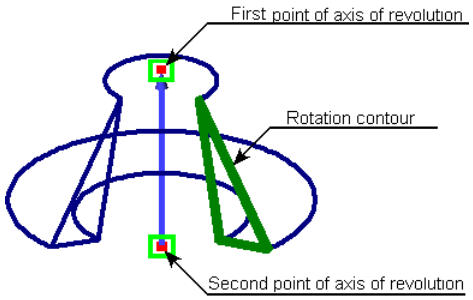
The axis is represented by the vector. The specified contour is rotated about this axis. The direction of contour rotation is defined by the right-hand screw rule: rotation is counter clock-wise as watched in the arrow direction.

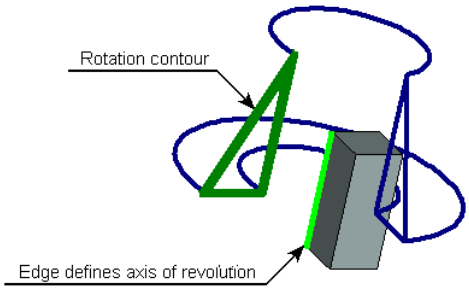
The axis can be defined either by a pair of 3D points for the axis to pass through or by selecting some 3D model element, suitable for defining a line.



If a 3D node was selected as the first point that was based on a 2D node, then only one point is enough. In this case, the direction is defined by the normal to the workplane hosting the 2D node.

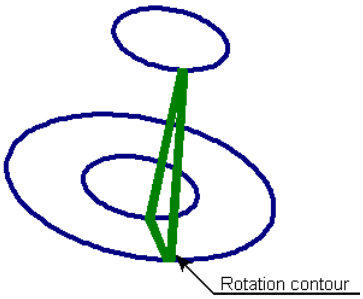
The axis can be defined explicitly by following 3D objects: flat curved edges that are arcs (in this case the axis of the arc edge is used as the axis of revolution), straight edges (the axis coincides with the edge), coordinate systems (one of the axes is used), surfaces/faces that are part of a cylinder or torus (the axis of the surface/face is used as the axis of revolution), various curves (the axis is defined by the normal or binormal to the curve at the specified point). The direction of the axis is defined by the direction of the selected element.



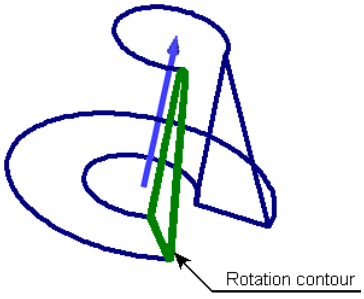


Angle of Contour Rotation

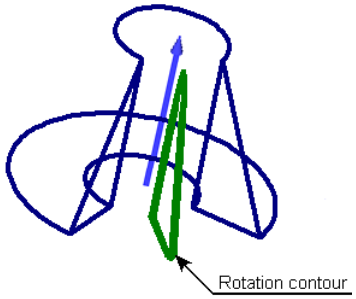
Rotation is performed from the original contour position by the angle specified by the user, no greater than 360 degrees (full turn). Additionally, the start angle of rotation can be specified. In this case, the contour will be rotated about the axis starting from the position defined by the start angle of rotation, for the amount of the specified rotation angle.



360-degree rotation (full turn)



Arbitrary-angle rotation



Arbitrary-angle rotation with non-zero start angle

The start angle and the rotation angle are defined either by numerical values in the operation parameters dialog or by specifying additional 3D points.

Add-on Features of Rotation Operation

Additional features of the rotation operation include:

Filleting of side edges of the resulting body. Besides, for bodies created by rotating by less than 360 degrees, rounds or chamfers can be applied on the end faces;

Thin-wall body creation mode;

Using created body in Boolean operation.

Detailed description of these features is provided in the "Extrusion" chapter.

CREATING ROTATION OPERATION

To create a body by rotation operation, use the command **3RO: Create Rotation**. The command can be called as follows:

Icon	Ribbon
	3D Model → Create → Rotation
Keyboard	Textual Menu
<3RO>	Operation > Rotation

The following steps are to be done to create the operation:

1. Select Rotation contour;
2. Specify axis of revolution;
3. Define the start angle and rotation angle (optional in some cases);
4. Define add-on feature parameters (optional, performed as described in the "Extrusion" chapter);
5. Confirm operation creation.

Contour and axis of revolution selection can be automatic if calling the command in the active workplane drawing mode. In this case, the system will automatically create a 3D profile based on the continuous graphic lines or a hatch and select it as the rotation contour. If only a single dash-dotted line was created on the workplane, it will be automatically selected as the axis of revolution.

Rotation Contour Selection

To create the operation, first define the rotation contour. Upon calling the command, the following option activates automatically in the automenu:

	<R>	Select Contour

To select a contour for the rotation, move the cursor over the desired element in the 3D window. The pointed at element will be pre-highlighted, and the cursor gain a tooltip with its name. Click for selection.

3D profiles based on hatches can be selected by picking the original hatch in the 2D window.

After selecting the rotation contour, the option will appear in the automenu for rejecting the contour:

	<H>	Cancel contour selection

Defining Axis of Revolution

After selecting the rotation contour, define the axis of revolution.

To define the axis by an arbitrary 3D model element, use the automenu option

	<A>	Select axis of revolution

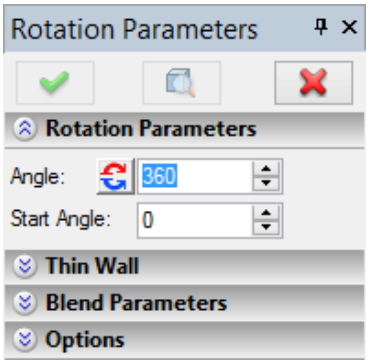
This option allows selecting a 3D element, suitable for defining the axis of revolution. The pull-down list of this option contains appropriate filters for selecting various objects. Pay attention to the active filters when selecting elements.

To define axis of revolution by two 3D points, use the following automenu options (activate sequentially):

	<F>	Select 1st Point of axis
	<S>	Select 2nd Point of axis

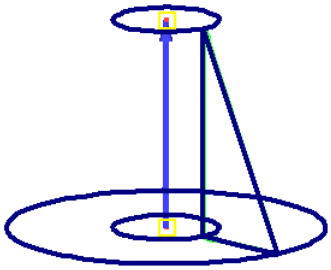
A 3D point can be defined by selecting a 3D node, a vertex of a body, profile or path. Besides, an edge can be picked – in this case, the 3D point will be defined by the edge midpoint (for straight edges) or center (for edges that are arcs, circles and ellipses). Selecting a face, which is part of a sphere or torus, yields the point at the sphere/torus center. A coordinate system selection yields its origin as the 3D point.

The set of elements allowed for selection is defined by the state of selection filters in the pull-down lists of the d270escribed options.



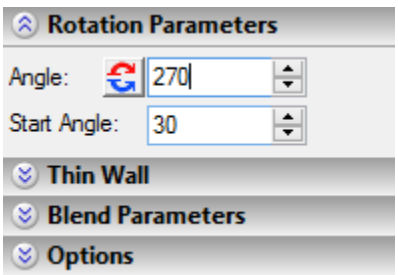
The axis can be defined by a single point by selecting as the first point a 3D node based on a 2D node. In this case, the axis is defined by the normal to the workplane hosting the 2D node.

Upon defining the axis, a wireframe preview of the resulting rotation is displayed in the 3D window, with an arrow showing the position and direction of the axis of revolution.



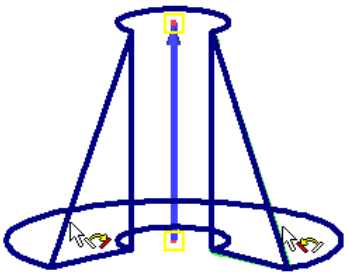
Defining Start Angle and Rotation Angle

The start angle and rotation angle are defined in the command property window in the section **Rotation Parameters**. By default, the system offers a 360-degree rotation (full turn of the contour) and zero start angle (the contour will be rotated from its original position).



If necessary, the default parameters can be modified. The rotation angle is defined by the parameter **Angle**. To change the start angle of the rotation, enter the desired value in the **Start Angle** input box.

The wireframe preview on the 3D scene shows the resulting rotation with the current parameters. Both the start angle and rotation angle can be defined directly on the 3D scene. As the cursor approaches the preview of the body being created, it gains a mark (for the start angle) or (for the rotation angle). Dragging the cursor with depressed dynamically sets the value of either angle. The numerical value of the angle is displayed in the property window.



Rotation parameters can be defined by additional 3D points selected with the help of the following automenu options:

	<1>	Select starting point of Rotation
	<2>	Select ending point of Rotation

These options become sequentially accessible after defining the axis of revolution. Both options contain pull-down lists with filters for selecting elements suitable for defining a 3D point.

When using 3D points, consider the type of the rotation contour. If the contour is flat then the specified points define the start and end positions of the contour plane in the rotation (the contour plane will be passing through the specified points in these positions). Therefore, the first point defines the start angle, and the combination of the two – the rotation angle. One can only specify the first point, and then enter the rotation angle numerically in the parameters dialog box.

When using 3D points for a non-flat contour, always define both points. Note that in such case, the point positions define the rotation angle only, as the start angle is always set to zero (the rotation starts at the original contour).

After defining the 3D points, dynamic angle modifications are no longer available on the 3D scene. The respective parameter input boxes also become inaccessible in the command property window.

Selection of 3D points can be cancelled by the automenu option:

	<K>	Cancel selection of points for Rotation

Using the created rotation in a Boolean

To create a Boolean operation, do the following:

Turn on the Boolean creation mode by the automenu option:

	<Ctrl+B>	Select original body for Boolean operation

A Boolean is created when the icon is pushed.

Select the type of the operation from the pull-down list under the above option:

	<Ctrl+'+'>	Addition
	<Ctrl+'-'>	Subtraction
	<Ctrl+'*'>	Intersection
	<Ctrl+I>	Smart mode

Operating principles of the Smart mode:

- ✓If the body being created and the existed selected body have intersection of volumes - the type of Boolean operations: subtraction.
- ✓If the body being created lies entirely within the selected body - the type of Boolean operations: subtraction.
- ✓If the body being created touches and selected body touch each other - the type of Boolean operations: addition.
- ✓If conditions set out above are not met, or a selected body lies inside the created, or error occurred determining the type of penetration, then, Boolean type is not defined - the Boolean operation will not be created.

Select the target body for the Boolean (optional in some cases), using the automenu option:

	<Ctrl+T>	Select Target Body for Boolean

If only one body exists in the scene, it is selected automatically. The new body created by the rotation operation becomes the tool body of the Boolean.

Upon confirming operation creation, first the body is created by the rotation operation, and then the specified type Boolean is performed.

BOOLEAN OPERATION

Boolean operation creates a new body based on two or more existing bodies. As a result of the operation, a new body is created that is a combination of the source bodies.

GENERAL CONCEPTS. OPERATION CAPABILITIES

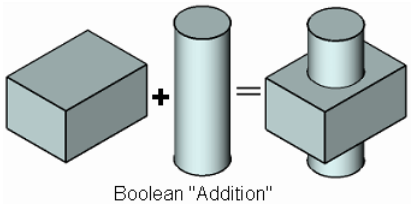
Types of Boolean

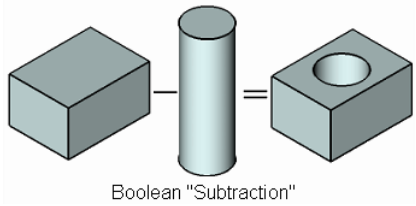
There are three types of Boolean operations:

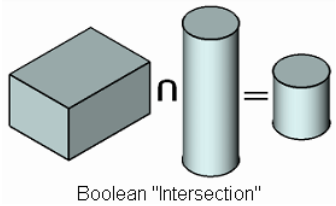
Addition. This operation results in a body which is the union of all pieces of the bodies participating in the operation.

Subtraction. This operation subtracts one body from another body.

Intersection. This operation results in the body which is the intersection of the participating bodies, being their common part.

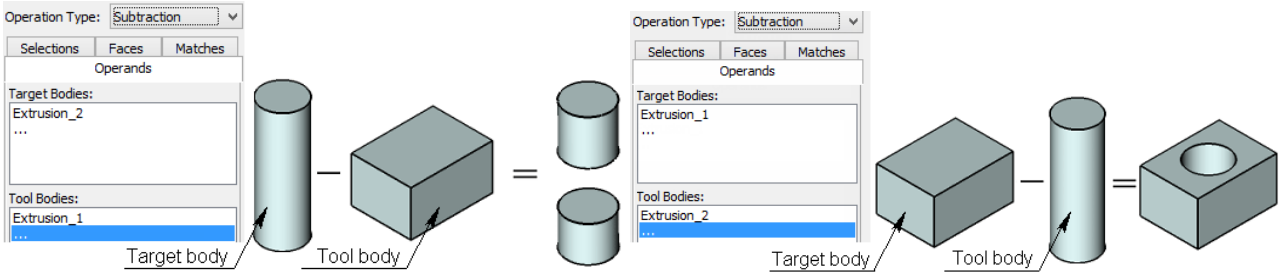

Boolean "Addition"


Boolean "Subtraction"

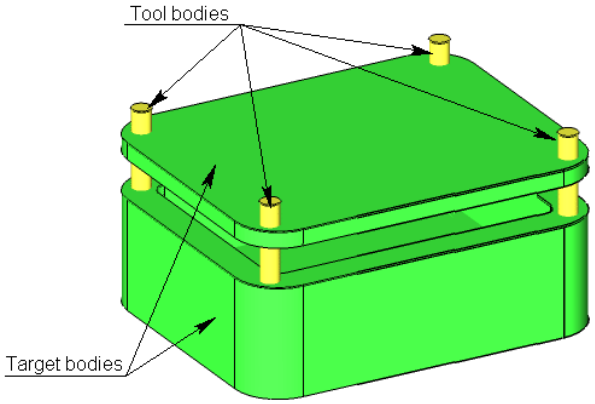

Boolean "Intersection"

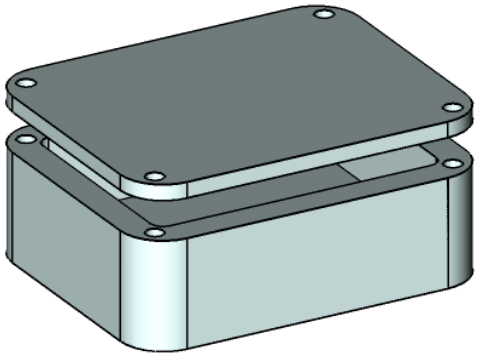
Boolean Operands

The bodies combined in a Boolean are called operands of the Boolean operation. Operands of a Boolean are the target and the tool. The bodies that are worked on are called “Target” of a Boolean. The bodies used for the manipulations are called “Tool”. Different result is obtained, depending on which of the body was the target and which was the tool.



A Boolean can also be defined on a set of bodies, some of which are selected as the target and other as the tool. In such an operation, all targets are united in one body, and all tools are united in another one. Then, a Boolean is performed on these bodies. This results in a single multiple-connected body.





Result of the Boolean "Subtraction" on the target and tool sets (a multiple-connected body)

Results of Boolean Operations

Operands can be either solids or sheet bodies. By selecting various combinations of operands and using different options, you can get a variety of results of a Boolean operation.

> Note that in certain cases, inadmissible combinations of operand types in a Boolean will not produce a valid result.

Below, we review common operand combinations and results of the respective Boolean operations.

Selecting type of the resulting body

In some cases, a Boolean can produce either a solid or a sheet body. The user can select the type of the resulting body with the help of the parameter “Create body”.

> This complementary option allows selecting one or another type of the resulting body, if possible under the rest of conditions. If the selected body type cannot be generated by the operation, this parameter setting will be ignored.

Sheet body treatment

A sheet body used as an operand in a Boolean, can be treated in the command as the sheet body proper or as a semi-space. Different results will be obtained, depending on the treatment of the sheet body and on whether it is used as the target or the tool.

If a sheet body operand is treated as a semi-space, then the vectors normal to the sheet body surface will be displayed in the scene. Direction of those vectors specifies which semi-space is selected, determining the operation result.

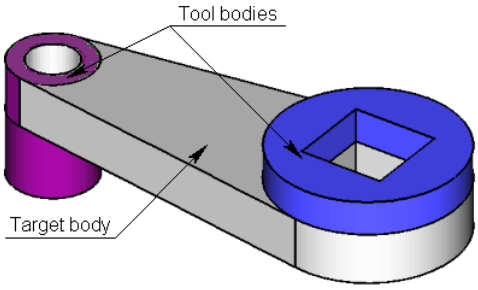
"Addition" operation

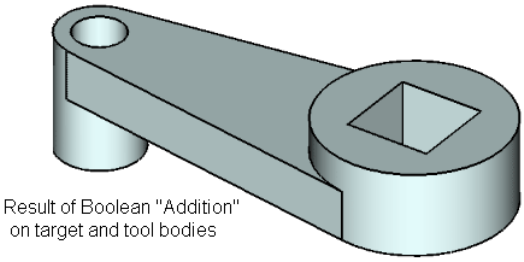
The following are the results of the Boolean "Addition" per different operand type combinations:

Target	Tool	Result
solid	solid	solid
sheet (semi-space)	solid	sheet
solid	sheet (semi-space)	solid
sheet (semi-space)	sheet (semi-space)	sheet (with corners)
sheet	sheet	sheet (sewn)

Solid + Solid

If both the target and the tool are solid, then the operation results in a single body uniting all parts of the bodies subjected to the operation.

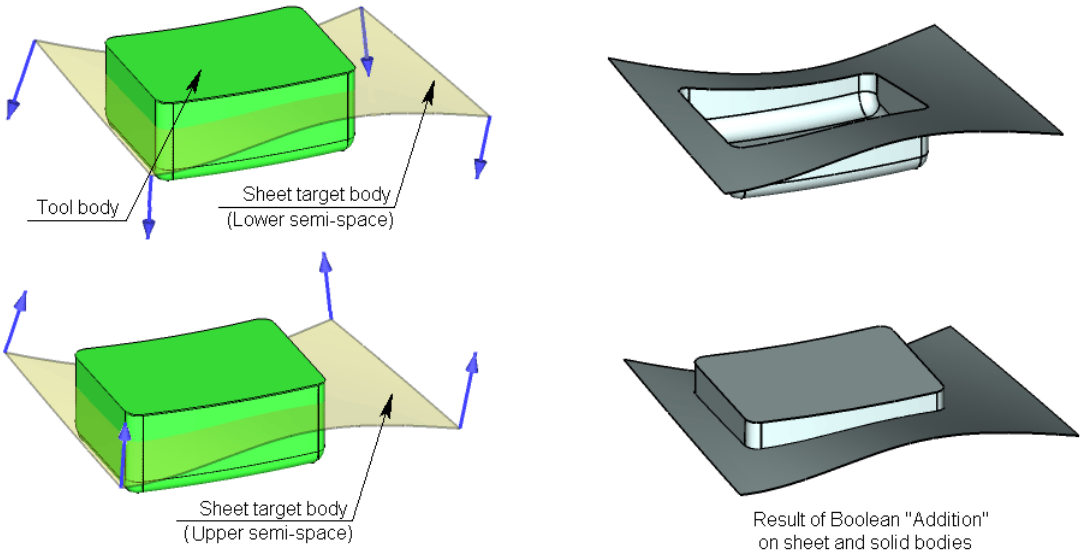




Sheet + Solid

The operation results in a sheet, which is "stamping" of the sheet with a portion of the solid body. Depending on the selected semi-space, the "stamping" is created either on the bottom or on the top part.

> Note that the sheet must fully intersect with the solid.



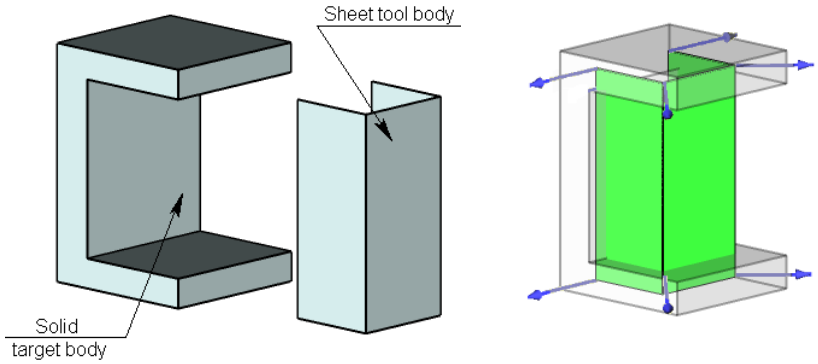
Solid + Sheet

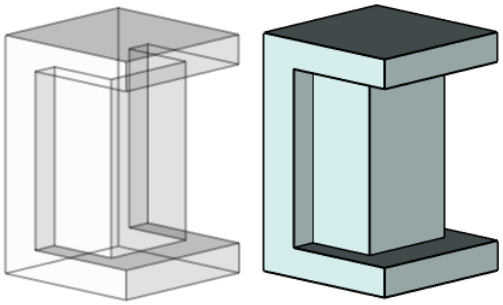
If a solid is selected as the target, while a sheet - as the tool, then the default result of the Boolean "Addition" will be a solid. The faces of the sheet body will create an additional volume for the solid.

The sheet body must be defined as a semi-space in this instance of the command.

> Note that this type of the Boolean operation requires, that the sheet body formed a closed volume when united with the solid. Otherwise, the operation cannot create a valid result.

When selecting a sheet body, pay attention to the normal vectors direction. An invalid result can be produced due to certain normal directions. In such a case, you need to specify the other semi-space by the parameter "**Operands treatment**".

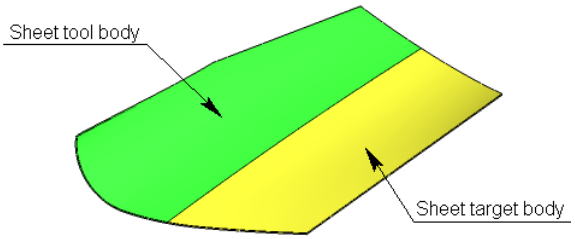


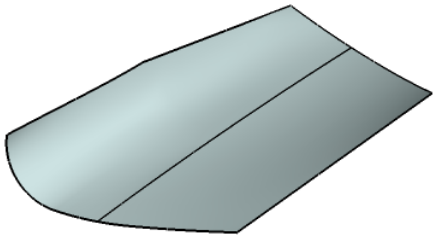


Result of Boolean "Addition"
on solid and sheet bodies

Sheet + Sheet

Adding two sheet bodies results in a sheet. For successful operation creation, the faces of the sheet bodies must either have a common edge or coincide within some region. This functionality is similar to the "Sew" operation. The difference is in that the Boolean "Addition" does not eliminate gaps between the bodies, while the "Sew" attempts to create a new continuous surface.





Result of Boolean "Addition"
of sheet bodies

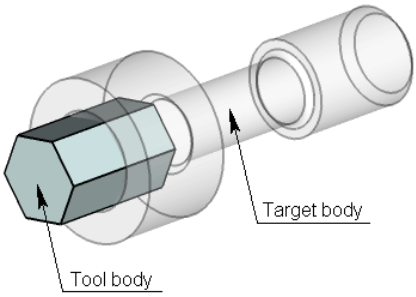
"Subtraction"operation

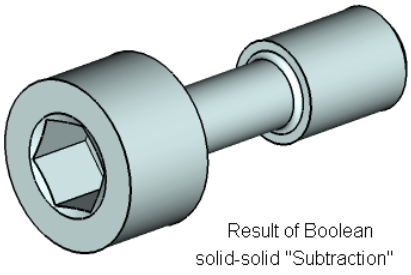
Following are the results of the Boolean "Subtraction" for different operand type combinations:

Target	Tool	Result
solid	solid	solid
sheet	solid	sheet
solid	sheet (semi-space)	solid
sheet (semi-space)	sheet (semi-space)	sheet (with corners)

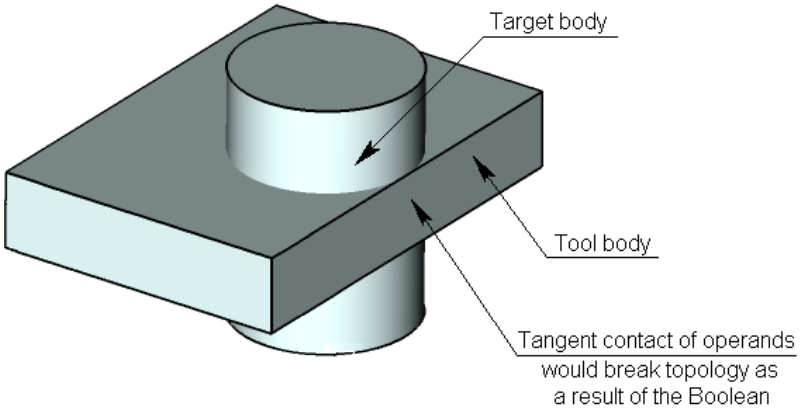
Solid - Solid

The result of the Boolean "Subtraction" of one solid from another one will be the target body with the removed tool portion of the volume.



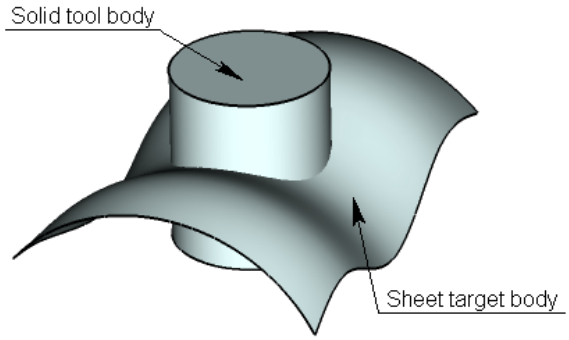


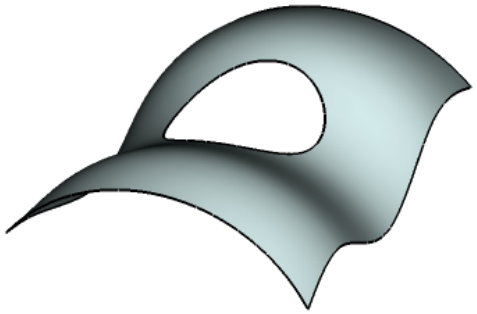
When creating the Boolean "Subtraction", the system guards from a situation in which the target body would have portions with zero-thickness walls. This would break the model topology.



Sheet - Solid

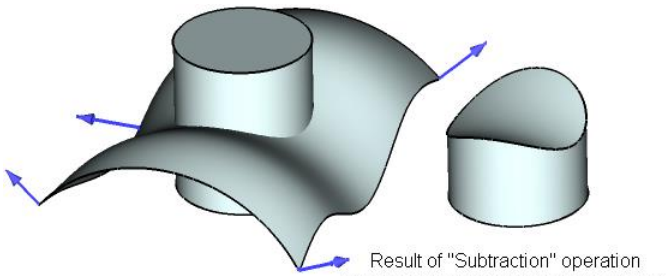
When a solid is subtracted from a sheet, an opening is created in the sheet body, repeating the shape of the solid tool.

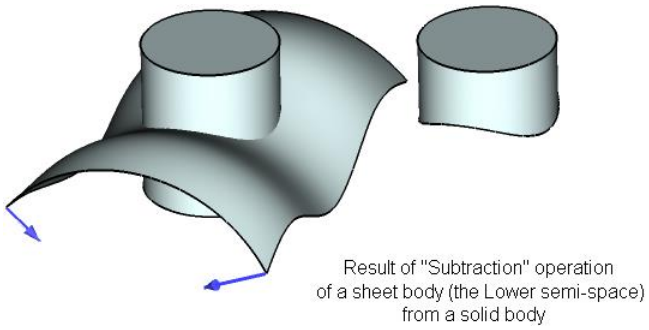




Solid - Sheet (semi-space)

Subtracting a sheet body from a solid leaves a part of the solid body cut off by the sheet. In this case, the sheet body must be defined as a semi-space. Depending on the selected semi-space, either the upper or the lower portion of the solid remains. This type of the Boolean is analogous to the "Cut" operation.





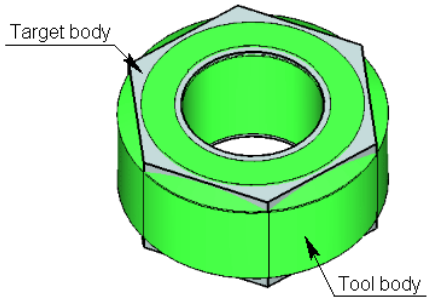
"Intersection"operation

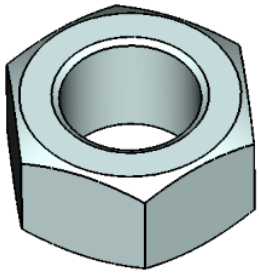
The results of the Boolean "Intersection" operation on various combinations of the operand types are:

Target	Tool	Result
solid	solid	solid
sheet	solid	sheet
sheet (semi-space)	solid	solid
solid	sheet (semi-space)	solid
sheet (semi-space)	sheet (semi-space)	sheet (with corners)

Solid ∩ Solid

The result of intersection of two bodies is the body that is the common part of all operands.

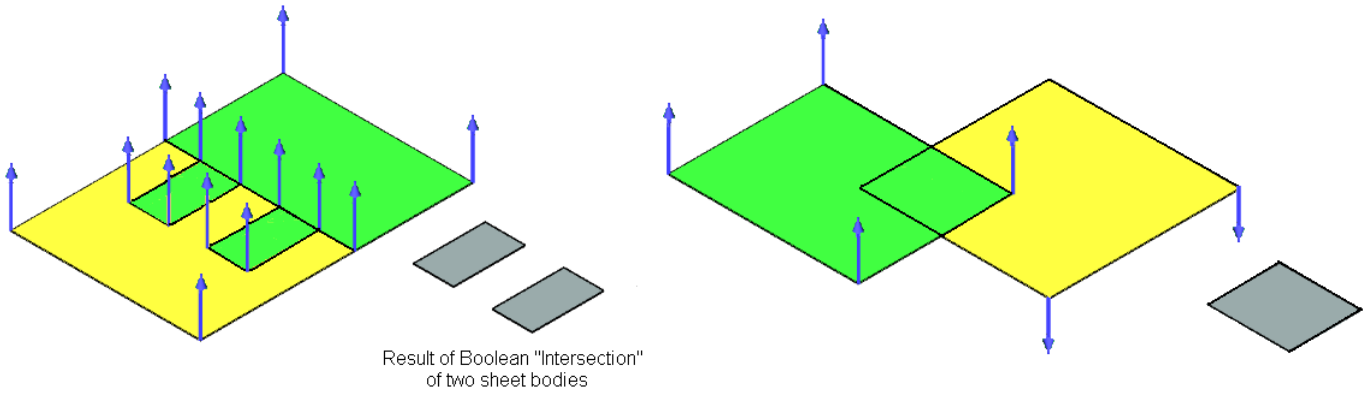




Result of Boolean "Intersection" of solids

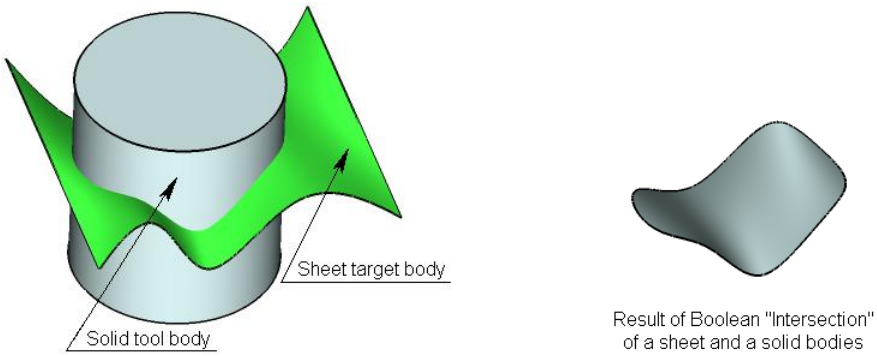
Sheet (semi-space) ∩ Sheet (semi-space)

If sheet bodies coincide within some region, then their common portion is left as a result of the Boolean "Intersection".



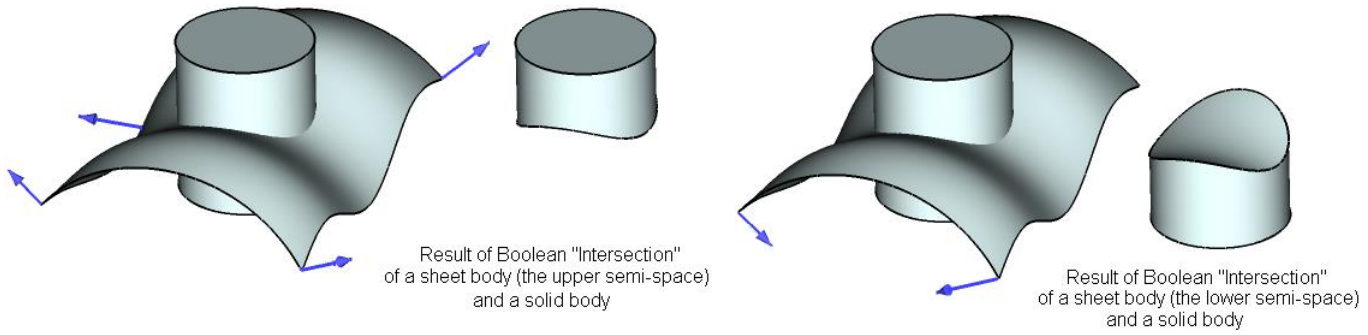
Sheet ∩ Solid

If a sheet is selected as the target while a solid as the tool, then the Boolean "Intersection" operation will result in creation of a sheet which is the common part of the two operands.



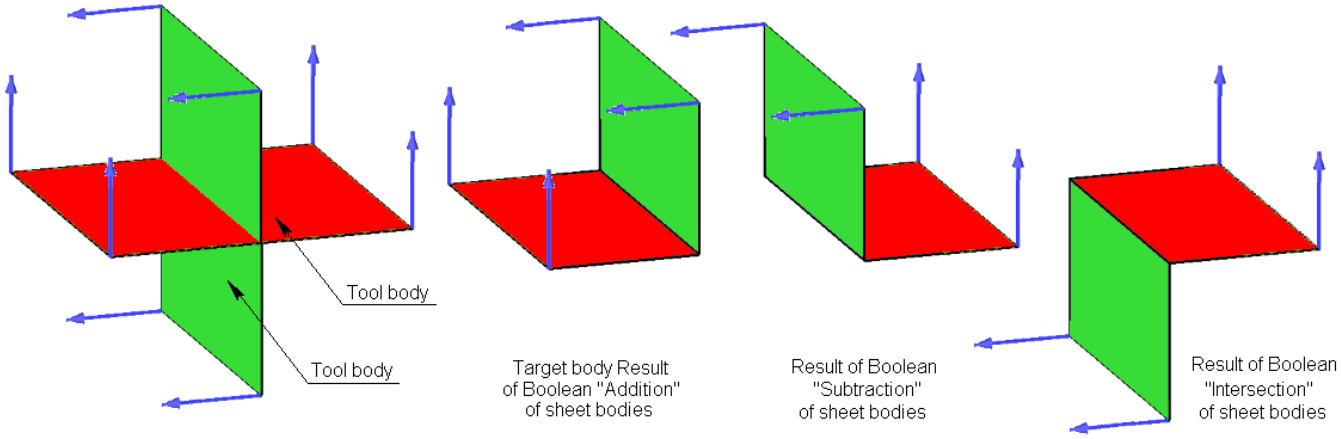
Solid ∩ Sheet (semi-space)

The result of performing the Boolean "Intersection" of a sheet body treated as a semi-space, and a solid, will be the portion of the solid body cut off by the sheet. The result of performing this operation is similar to the result of the Boolean "Subtraction" of a sheet (semi-space) from a solid. The difference is in that the result of the "Intersection" operation is the other part of the solid. Note that the order of selecting operands does not affect the result.

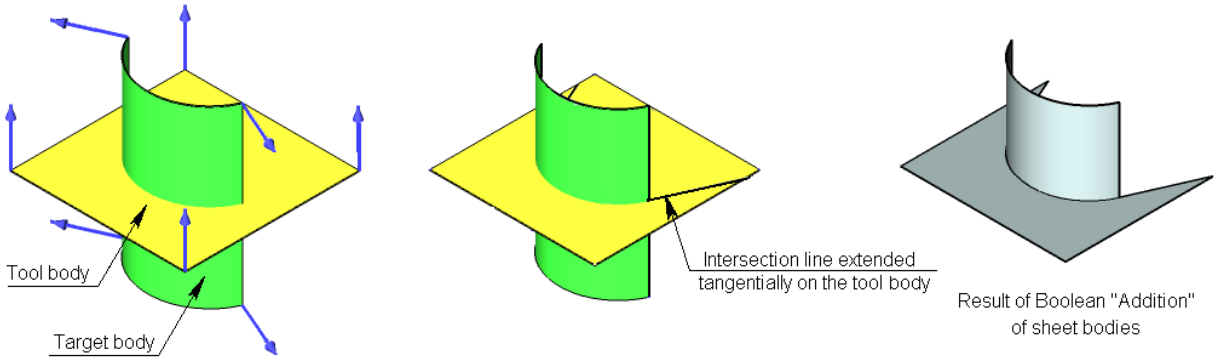


Boolean operations on sheet bodies (semi-spaces)

One indication to using Boolean operations on sheet bodies is creation of corners. In this case, the sheets specified in the command must intersect and be treated as semi-spaces. Depending on the normal directions of the sheet surfaces and the type of the used Boolean operation, different sheet bodies are produced as a result. Note that the normal directions are preserved on the resulting body.



If one sheet does not fully intersect the other one, the Boolean won't work. To fix the situation, an option is provided among the Boolean operation parameters for extending edges of the target or the tool up to the next edge or the body boundary. In this case, the intersection line between these two bodies will be extended tangentially up to the selected edge or to the boundary of one of the operands.



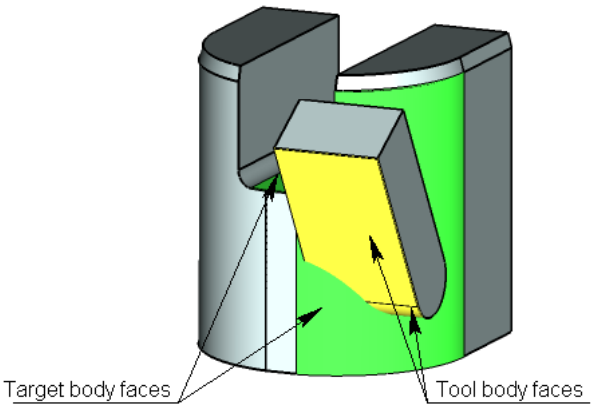
Global and Local Boolean Operations

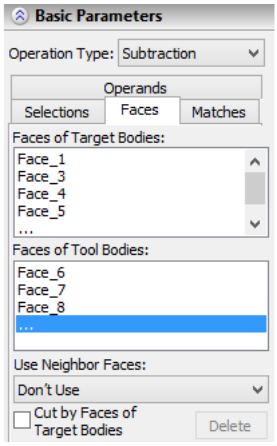
If sheet or solid bodies are selected as operands, we will call such instances of the Boolean operation “Global”. When creating such a Boolean operation, the system automatically determines which faces of the bodies are involved in the operation, and finds their intersections. If a Boolean involves geometrically complex bodies or the large number of operands, then the search for all intersections may noticeably delay the command execution.

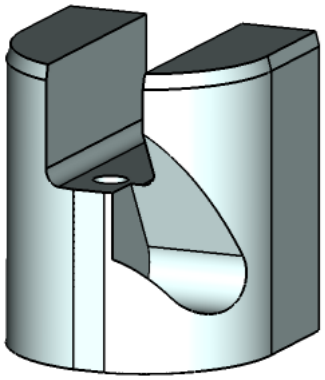
Besides the “Global” Boolean operation, T-FLEX also introduces the “Local” Boolean operation. Unlike the “Global” Boolean operation whose operands are always bodies, the “Local” Boolean operation allows selecting faces, besides the body operands. The operand faces can be selected with the help of the tab

"Faces" in the operation's parameters. In some cases, use of the "Local" Boolean operation saves considerable time on the model regeneration, as compared with the "Global" Boolean operation, since the intersections will be searched only among the specified faces when processing the operation. For example, to make a hole in a rather complicated body, you can specify the faces to be intersected in the operation parameters if you know in advance that just few faces will be intersected.

You need to be careful when selecting faces for the "Local" Boolean operation. An invalid body may result from incorrect selection of faces.

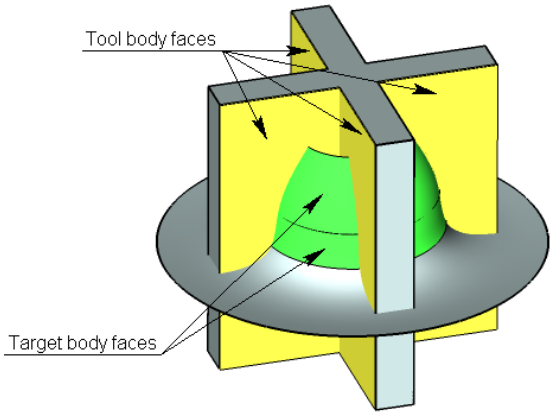


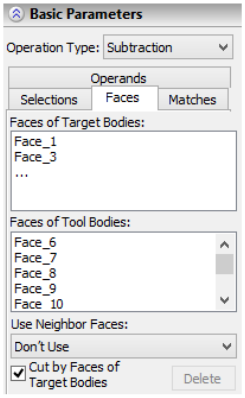


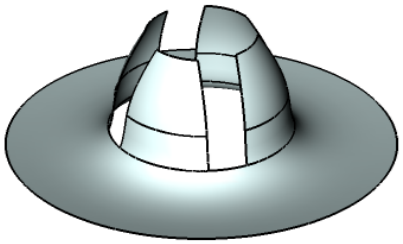


Result of local Boolean "Subtraction"

Selecting specific faces for the operands of a "Local" Boolean operation with the option "Cut by Faces of First Operands" turned on, allows creating bodies like those shown on the diagram below. In this case, the intersection lines between the operands' boundaries will be determined first, and then closed loops will be formed along these intersection lines and the boundaries of the target operands' faces, in order to create a Boolean operation.

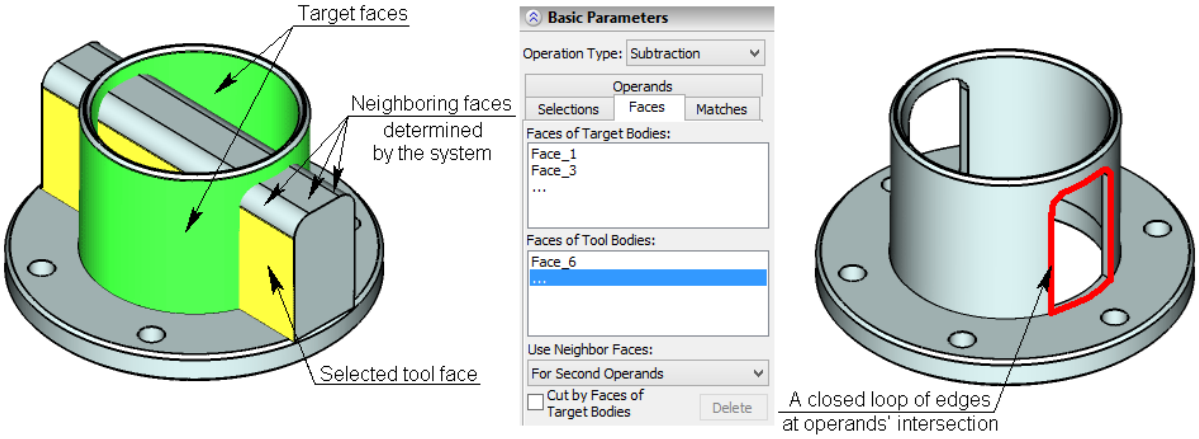




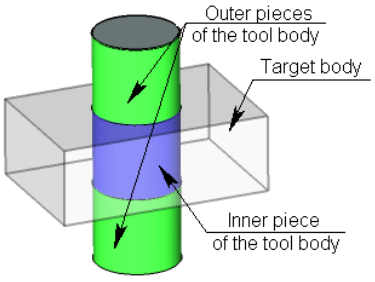


Result of local Boolean "Subtraction"

In some cases, the option "Use Neighbor Faces" can be conveniently used for the target or the tool. All you need is select one face of an operand, and the system will automatically define the neighboring faces in such a way, that the intersection lines between the operand faces form closed loops. In this case, changes in the number of faces making a closed loop due to operand modifications, will not invalidate the result of the Boolean operation.



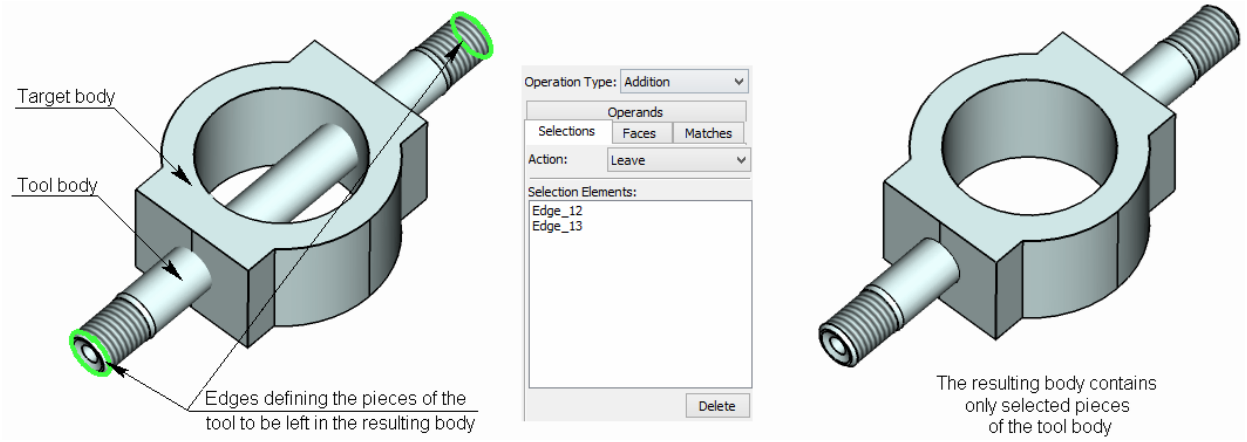
Selective Boolean Operations



When intersected, operand faces create common edges. Those edges divide the bodies of the operands into pieces. There are inner pieces (shared by the operands) and outer pieces of operands. When creating a Boolean operation, you can specify the outer pieces of the operands that will be left in the resulting body, or, on the contrary, excluded from it.

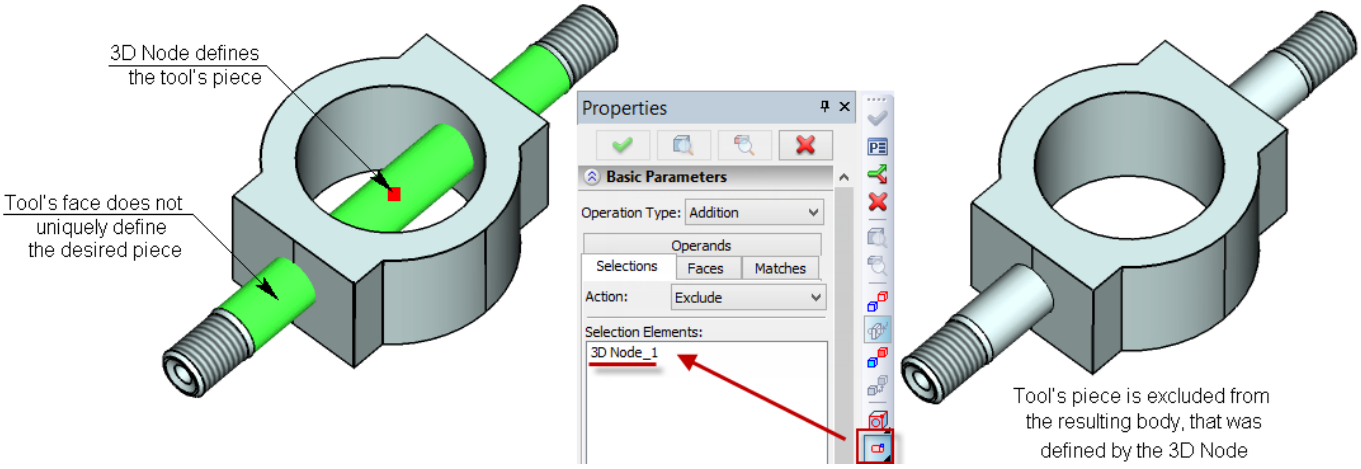
A piece of an operand is defined by topology elements, such as vertices, edges are faces. To select a piece of an operand, you need to specify one of such elements in the operation parameters on the tab "Selections".

The diagram below shows an example of using a selective Boolean "Addition" operation, whose execution results in a body containing only selected pieces of the tool.



In some cases, the selected topology element does not explicitly defines the desired piece. For example, as shown on the diagram below, the selected face belongs to several outer pieces of the tool. To uniquely determined the desired piece, one needs to create a 3D node on this face, and then select this

3D node by the option "Select point that indicates part of operand". In this case, you do not have to select the face.



Setting Element Matches

In the cases when the operands have coinciding regions, the command provides the function of additionally specifying topological correspondence of the operands' regions. The command will select the appropriate algorithm for creating the Boolean operation based on this information.

This function allows:

- ✓ Speed up regeneration of the Boolean operation due to the particular specified type of contact.
- ✓ Successfully create the Boolean operation in the cases when this would be impossible without knowing the match. An example of such situation is a contact between a spline and an analytical surface (a Boolean on a cylinder and a face of a helix on the diagram below).

The command has provisions for four types of matches:

Exact match – the boundaries of topological elements coincide within the specified tolerance.

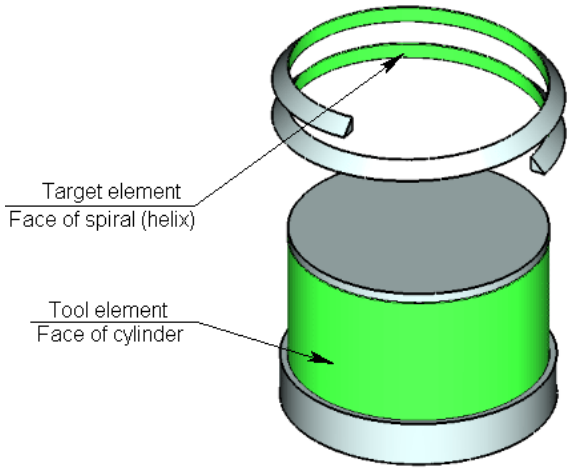
Contained – the boundary of the second element lies fully within the extents of the first element's boundary.

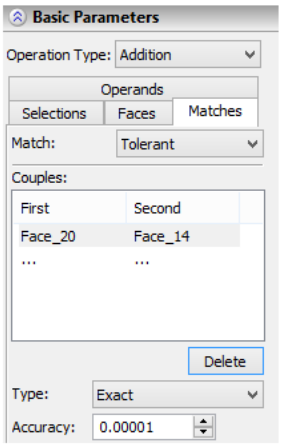
Overlap – the boundary of one element partially coincides with the boundary of the other element.

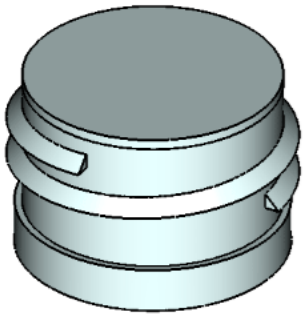
Imprinted – the boundaries of the imprinted topological elements coincide.

The table below presents the description of match types for operands of various topology:

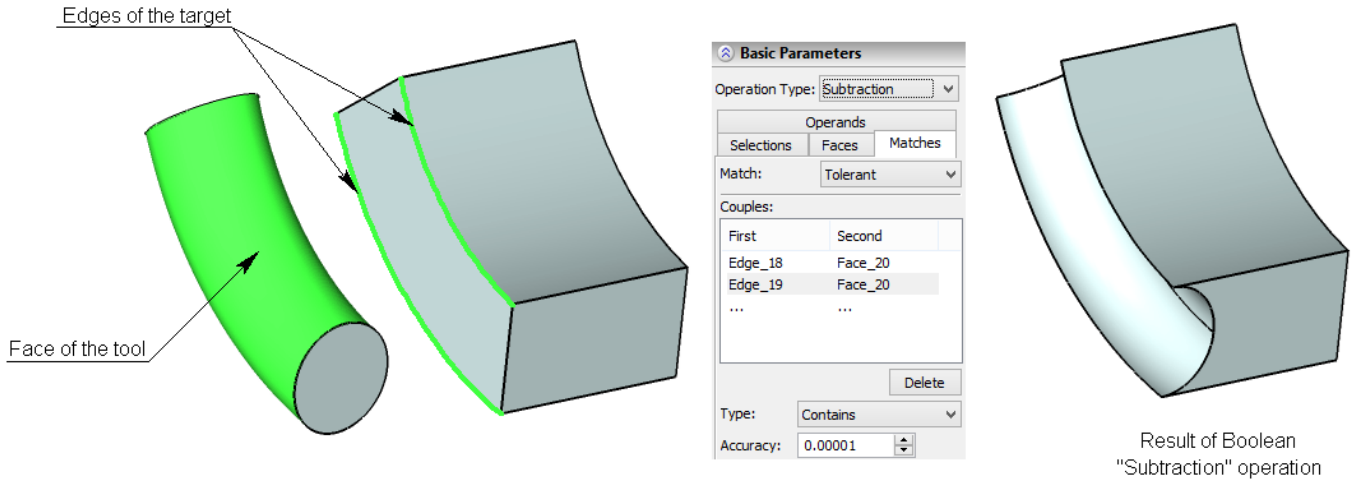
Match Contact type	Exact match	Contained	Overlap	Imprinted
Vertex Vertex	–	–	–	Projections of operands' vertices coincide
Edge Edge	Edge curves coincide within specified tolerance. Respective vertices of edges coincide within specified tolerance.	Edge curves coincide within specified tolerance. Extents of the second edge are fully within the first edge	–	Imprinted operands' edges coincide
Face Edge	–	Edge curve lies on the surface of the face within specified tolerance. Edge lies fully within the extents of a face.	Edge curve lies on surface of the face within specified tolerance. Edge may intersect the face boundary (edges).	–
Face Face	Surfaces of faces coincide within specified tolerance. Each edge of first face has matching edge on second face. Inexact coincidence of edges is allowed.	Surfaces of faces coincide within specified tolerance. Second face is fully contained within first face. No edge of second face intersects boundaries of first face. Edges may coincide.	Surfaces of faces coincide within specified tolerance. Boundary of one face may intersect boundary of the other face. Edges of the faces may coincide.	–





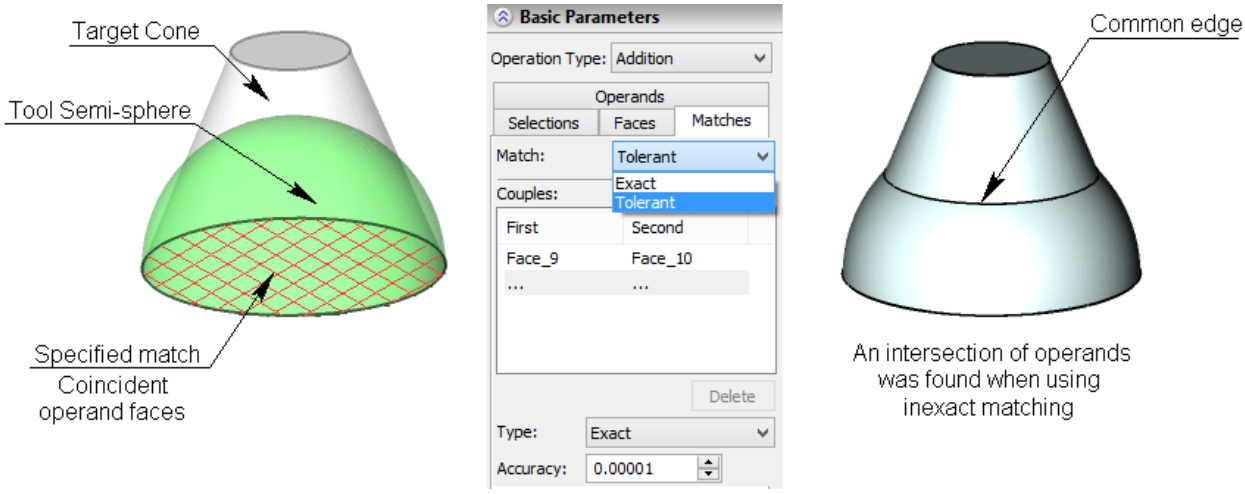


Result of Boolean "Addition" operation



A match can be exact or approximate. An exact match implies that the operands have only the described contacts and no more overlaps or intersections. If the inexact ("tolerant") matching is set, the system will be searching for other intersections of operands, besides the specified contacts. This type of processing takes more time, compared to the exact matching; however, in some cases this is the only possible way of creating a Boolean operation.

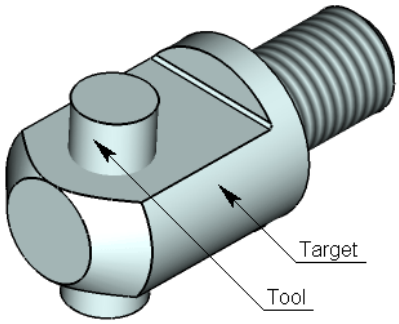
The diagram below illustrates defining a match in a Boolean "Addition" operation on a semi-sphere and a cone with the common base. If the match is defined as coincident faces at the base, and the exact option is used, then the Boolean operation will not be possible to create, since there is another common edge in the intersection of the operands, which is not described among the matches. In this case, the only option is use of the inexact matching that will find this other intersection during the processing.

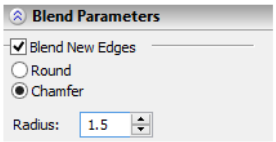


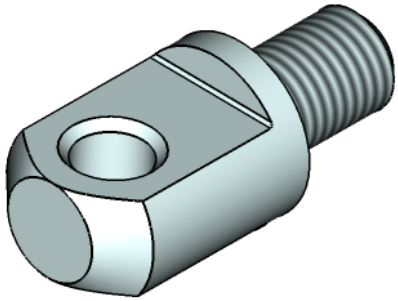
Blending New Edges

Another capability of Boolean operations is blending new edges. By new edges, we mean the edges created as a result of intersecting faces of the operands. As the operands' geometry changes, the system

will automatically track the increasing or decreasing number of new edges and create blending. You can select the type of blending in the command: round or chamfer, and specify the blending radius.





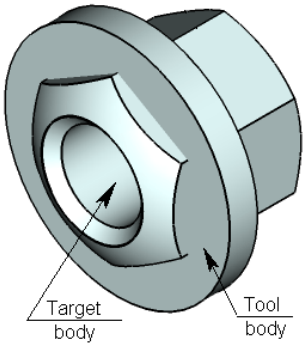


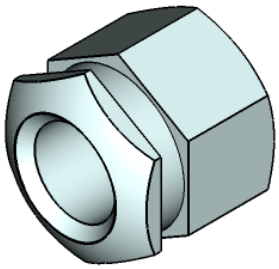
Result of Boolean "Subtraction" operation with chamfers created on new edges

Missing Operands

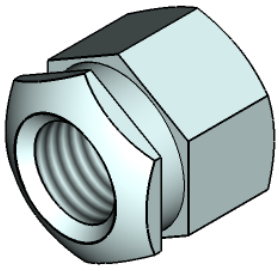
When creating a Boolean operation, you can turn on the option “Allow missing operand”. This technique is often used in parametric models, when one of the model's elements needs to be excluded in some instances. As shown on the diagram below, a nut can be in two instances: with and without grooving. To realize both instances in one model, one can suppress the tool, while maintaining the rest of the consequent operations based on the Boolean (in the example, this is the cosmetic thread on the face of the Boolean operation).

If this option is turned off, then an error will be output when recreating this Boolean operation with the operand suppressed.

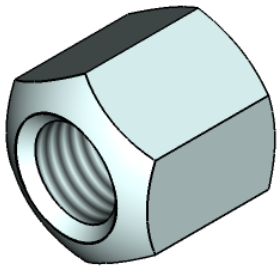




Result of Boolean "Subtraction" operation



Result of applying cosmetic thread on a face of the Boolean (Instance 1)



Result of suppressing the tool body (Instance 2)

Creating Boolean Operation

The Boolean operation creation command can be called in one of the following ways:

Icon	Ribbon
	3D Model → Create → Boolean
Keyboard	Textual Menu
<3B>	Operation > Boolean

The automenu and the property window are used together in the command. These work together and enhance each other. The state of the automenu depends on the step of defining the operation and on the purpose and type of the objects being selected.

The following steps are to be done to create the operation:

- ✓Select Boolean operands;
- ✓Select the type of the Boolean operation;
- ✓Specify operands' pieces for creating a selective Boolean operation (optional);
- ✓Define element matches (optional);
- ✓Define additional parameters (if necessary);
- ✓Define blending parameters (optional);
- ✓Confirm operation creation.

Main Operation Parameters

Selecting operands

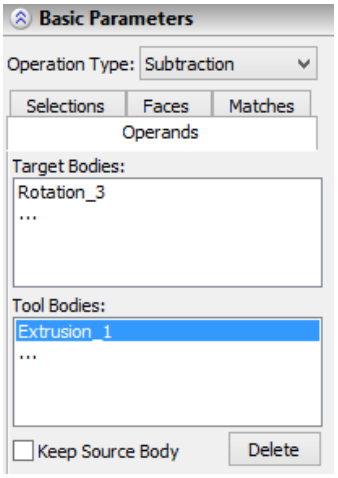
Selecting operands of the global Boolean operation

To select operands of a global Boolean operation, use the following option:

	<1>	Select Boolean operands

At this point, the options become available in the automenu:

	<F>	Select 1st body
	<S>	Select 2nd body



As one of these options is activated, the input focus switches to the respective pane of selecting targets or tools in the property window. Vice versa, as you switch the focus between the selected operands panes in the property window, the respective options activate in the automenu.

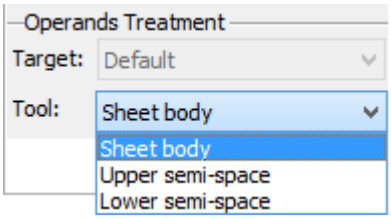
Target and tool bodies can be selected either directly in the 3D scene or in the 3D model tree. Target bodies are highlighted green in the scene, tools – yellow. An operand selected in the property window is highlighted blue in the scene.

To delete any operand from the list, select it in the "Properties" window, and then press the button **[Delete]**.

To cancel selection of all operands, use the option:

	<D>	Cancel selection of all operands

Once the Boolean operation is created, the operand bodies will be removed from the scene, with only the resulting body left. For each operand, you can turn on the option **"Keep source body"**. In this case, the body of the operand with this option engaged will be left in the scene upon creating the operation.



If a sheet is selected as an operand, you need to specify the treatment for this body. There are three ways of treating a sheet body: as a sheet proper, or as an upper or lower semi-space. Depending on its definition by the system, the resulting Boolean operation will be created differently.

A sheet body's treatment can be defined by selecting the appropriate value in the combo box of the operands' treatment parameters.

Selecting operands for the Boolean operation

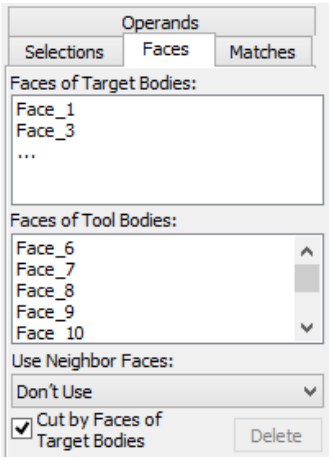
To enter the mode of selecting faces for creating a local Boolean operation, one can use the automenu option:

	<3>	Select local faces

Alternatively, to activate this mode, you can switch to the **"Faces"** tab in the "Properties" window.

At that time, the following options become available in the automenu:

	<F>	Select local faces for first Operands
	<S>	Select local faces for second Operands



Once the operands are selected, you can proceed with selecting bodies' faces for creating a local Boolean operation. You can do this in the scene by the mentioned automenu options or by focusing on the pane of the "Faces" tab.

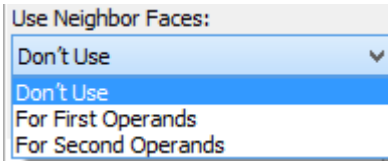
When creating a Boolean operation, you can skip selecting operands and directly proceed to the mode of selecting faces. In this case, the bodies, whose faces are selected, will be automatically added as operands on the respective tab in the operation parameters.

Highlighting of the target and tool faces is analogous to that of the bodies-operands. To remove a face, select it in the list on the "Faces" tab, and then press the button **[Delete]**.

In some cases, you may need to turn on the option **"Cut by Faces of Target Bodies"** to insure the correct operation result.

To cancel selection of all local faces, use the option:

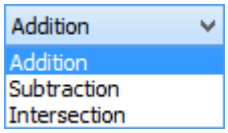
	<D>	Cancel local faces selection



When creating a local Boolean operation, you can engage faces adjacent to the selected face. To do this, select the respective entry in the combo box of the **Use Neighbor Faces** parameter.

Selecting Boolean type

Selection of the operation type is done either via the parameter "Operation Type" in the property window, or by the following options in the automenu:



	<+>	Addition
	<->	Subtraction
	<*>	Intersection

When the Boolean operation is called first time in a session, the Boolean operation type "Addition" is automatically set in the automenu. On subsequent calls to the Boolean, the default button will be the type used in the previously called Boolean.

Selecting operands' pieces

To create a selective Boolean operation, use the following option:

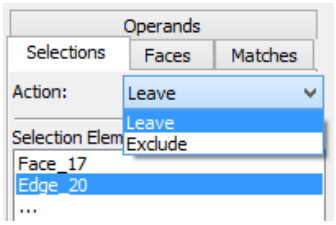
	<2>	Select parts of operands

Alternatively, to activate this mode, you can switch to the **Selections** tab in the "Properties" window. In this case, the following options become available in the automenu:

	<F>	Select element that indicates part of operand
	<H>	Select point that indicates part of operand
	<D>	Cancel selection of elements that indicate parts of operands

With these options, you can select the elements defining the operands' pieces. The names of the selected elements will be added to the list on the "Selections" tab in the "Properties" window.

On the same tab, you can select the action for the selected pieces of the operands: leave those in the resulting body, or exclude. The selected action will work on all selected pieces of the operands.



Setting element matches

To define element matches, use the option:

	<4>	Select matches

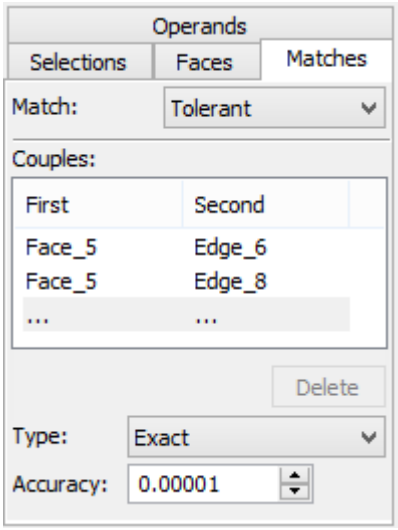
Alternatively, to activate this mode, you can switch to the **Matches** tab in the "Properties" window. Then, the following options become available in the automenu:

	<F>	Select first match Element
	<S>	Select second match Element
	<D>	Cancel match selection

Use these options to select pairs of elements for defining matches. The names of the selected elements are displayed on the "Matches" tab in the "Properties" window. Each pair of elements can be assigned the exact or approximate ("tolerant") matching option. The type of match is selected from the "Type" combo box on this tab.

Note that the set of match types in this combo box can vary, depending on the particular combination of element topologies in a pair (see the table in the topic "Setting element matches").

The matching tolerance can be specified on this tab.



Preview of Boolean Operations Results

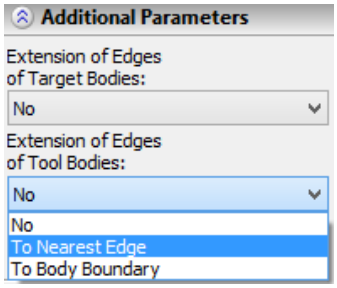
To preview the result of a Boolean operation and also the changes introduced by a Boolean operation it is required to use the preview option in the automenu or a similar button in the properties window:

	<F5>	Preview result of operation
	<Ctrl+F5>	Preview changes introduced by operation

Additional Operation Parameters

Extending operands' edges

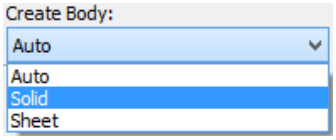
If sheet bodies defined as semi-spaces are used as operands in the command, then edges may need to be extended to achieve intersection of the operands in order for the command to succeed with the Boolean operation creation. To do this, select the type of the object, up to which the edges of the target or tool body are to be extended. This can be done using the respective additional parameters.



Selecting the type of resulting body

If a Boolean operation permits either the solid or the sheet result, then the desired type can be set by the option "**Create body**" in the additional operations parameters.

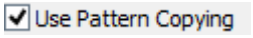
By default, the command will create the resulting body of the preferred type.



As was mentioned earlier, this option is complementary. If the selected type of the resulting body is impossible, this parameter setting will be ignored.

Use Pattern Copying

If a 3D array is used as one of the operands, you can significantly reduce the time of executing the Boolean operation by turning on the parameter **"Use Pattern Copying"**.

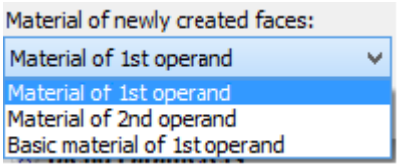


Provision for a missing operand

By turning on the **"Allow missing operand"** parameter in the Boolean being created, you will be able to suppress one of the operands in the future, and still keep the result of the Boolean operation.

Material of newly created faces

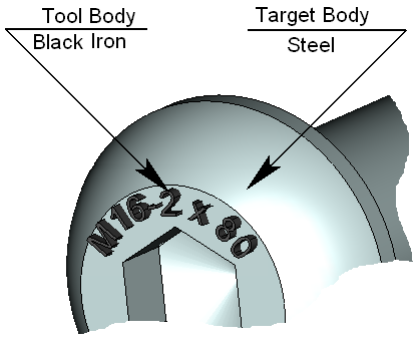
To determine the material to which the result of the Boolean operation is applied, the combination of parameters is used. In addition to material parameters found on the "General" tab and common for all operations (the "Source", "Main", "Covering" parameters), the material assignment parameter for the new faces, which are created as a result of applying Boolean operation, is also taken into account. This parameter is located on the "Parameters" tab and is called **Material of new faces**.

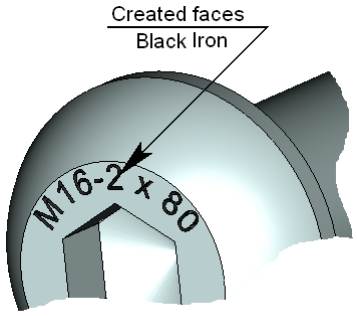


This parameter can take one of the three values:

- **Material of 1st operand**. In this case the main material or the material of the covering (if it is specified) will be assigned for the new faces according to the parameters of the first operand. If there are several first operands and they are with different materials, then on the new faces the material of that operand which participated in formation of these faces will be saved;
- **Material of 2nd operand**. For this value of the parameter, on the new faces of the resulting body the material will be transferred from those faces of the second operand which participated in their formation. This can be either the main material or the material of the covering (if it is specified), and even the material of the **Material overlap** operation. For example, as a result of execution of the Boolean operation "Subtract" between the first operand with the material steel and the second operand with the material black plastic, the

material black plastic will be put on the created faces of the resulting body. If there are several second operands and they are with different materials, on the new faces the material of that operand which participated in formation of these faces will be saved.



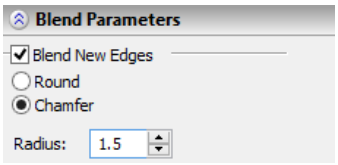


Result of Boolean "Subtraction"

- "**Basic material of 1st operand**". The action of this parameter is analogous to the first parameter ("Material of first operand"), except the case when the material of the covering is specified. In this case the material of the covering is not used on the new faces. For example, if in a part with the main material "Steel" and with covering "Copper" we create a hole with the help of the Boolean operation, the hole inside will appear as steel in accordance with the main material.

Blending new edges

To turn on the mode of blending newly created edges resulting from the intersection of the target and tool bodies, turn on the parameter **Blend New Edges**. After that, a group of parameters becomes accessible for selecting the blend type ("Round", "Chamfer") and amount of blending (the parameter "Radius").

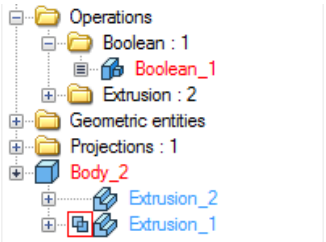


Specifics of Handling Boolean Operations

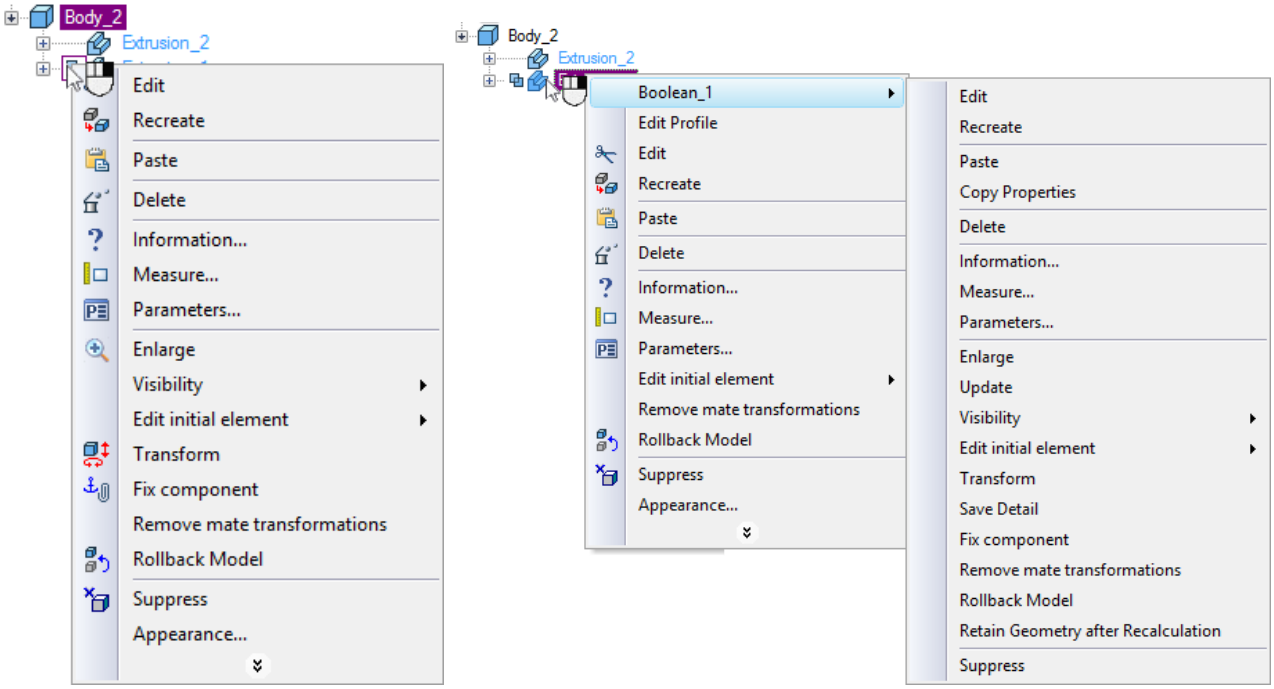
Boolean operation representation in "3D Model" window

Upon creation, a Boolean operation is put in the "Boolean" folder in the "3D model" window. An icon is displayed before each Boolean operation, denoting the operation type (Addition, Subtraction, Intersection).

Booleans appear in a special way in the 3D model tree. When expanding the Body's structure, the Boolean appears as an icon before the tool body. The diagram on the right hand side shows "Boolean_2": an intersection of the target "Extrusion _0" and the tool "Extrusion_1".



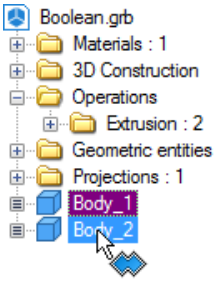
Upon right clicking the icon before a tool operand in the model tree, the Boolean operation context menu appears. Alternatively, the Boolean context menu can be accessed as the context menu of the tool body in the model tree.



Creating Boolean operation in the 3D model tree

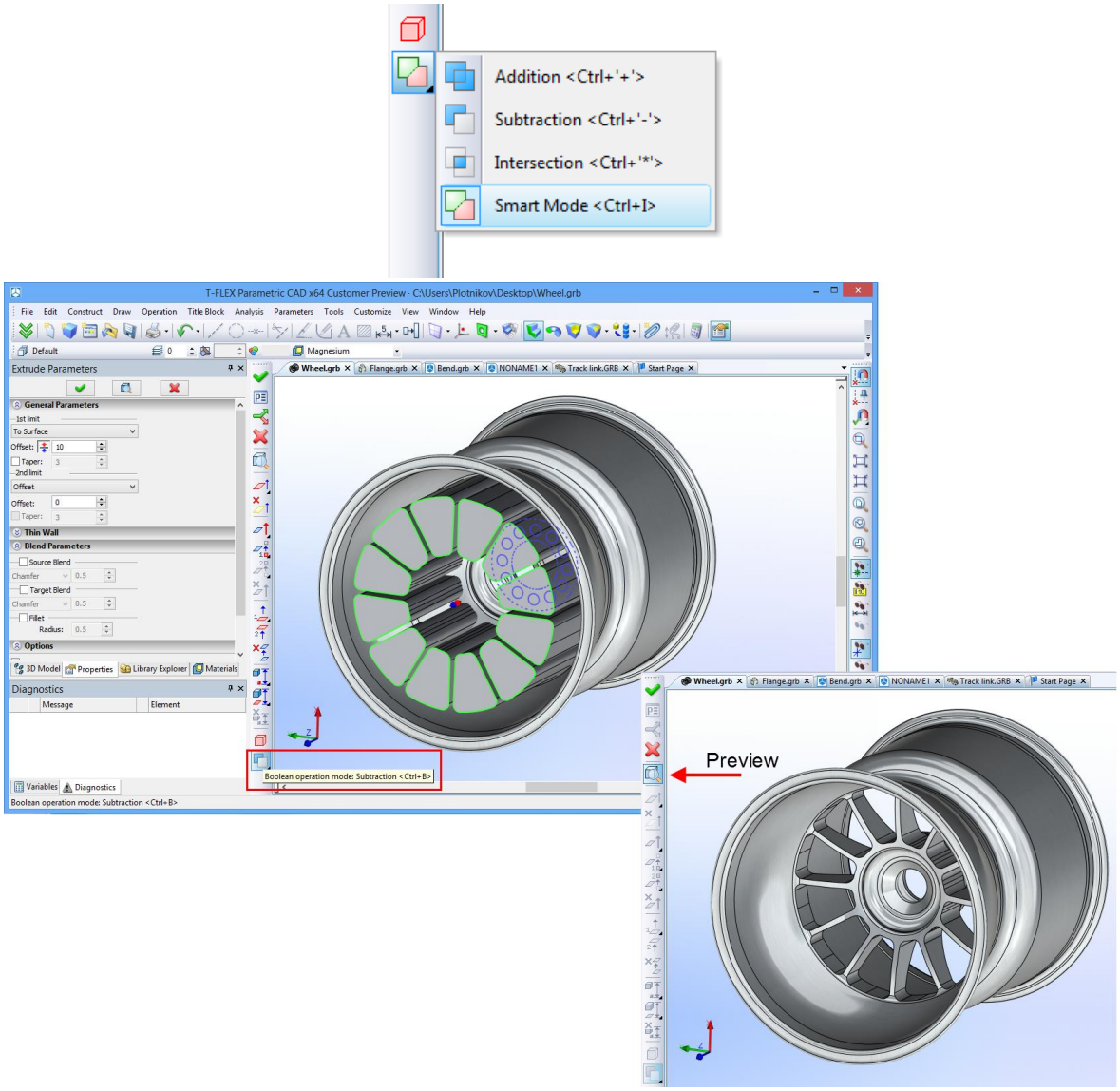
You can create a Boolean in the 3D model tree without explicitly calling the creation command.

Select a body in the 3D model tree, and then Drag&Drop it to another body by dragging the mouse with the left button depressed. This would automatically create the Boolean "Addition". By additionally holding down the <Ctrl> key, the Boolean "Subtraction" will be created. If, instead of <Ctrl>, you hold the <Shift> key, the Boolean "Intersection" will be created. The dragged body is the tool, and the body, to which the other is dragged, is the target.



Automatic Boolean creation combined with creating other operations

Numerous operations of T-FLEX CAD allow automatic creation of a Boolean. The body being created in those operations is always the tool of the Boolean. The target is selected among all existing bodies of the 3D model.



It should be noted that when creating these operations the preview displays the result of the automatic Boolean only upon creation of the operation. When editing the operation the preview is not displayed. If the Boolean operation created automatically does not have a result, then only the operation being created will be seen on the preview.

To create a Boolean operation, do the following:

Turn on the Boolean creation mode by the automenu option:

	<Ctrl+B>	Select original body for Boolean operation

A Boolean is created when the icon is pushed.

Select the type of the operation from the pull-down list under the above option:

	<Ctrl+'+'>	Addition
	<Ctrl+'-'>	Subtraction
	<Ctrl+'*'>	Intersection
	<Ctrl+I>	Smart mode

Operating principles of the Smart mode:

- ✓ If the body being created and the existed selected body have intersection of volumes - the type of Boolean operations: subtraction.
- ✓ If the body being created lies entirely within the selected body - the type of Boolean operations: subtraction.
- ✓ If the body being created touches and selected body touch each other - the type of Boolean operations: addition.
- ✓ If conditions set out above are not met, or a selected body lies inside the created, or error occurred determining the type of penetration, then, Boolean type is not defined - the Boolean operation will not be created.

Select the target body for the Boolean (optional in some cases), using the automenu option:

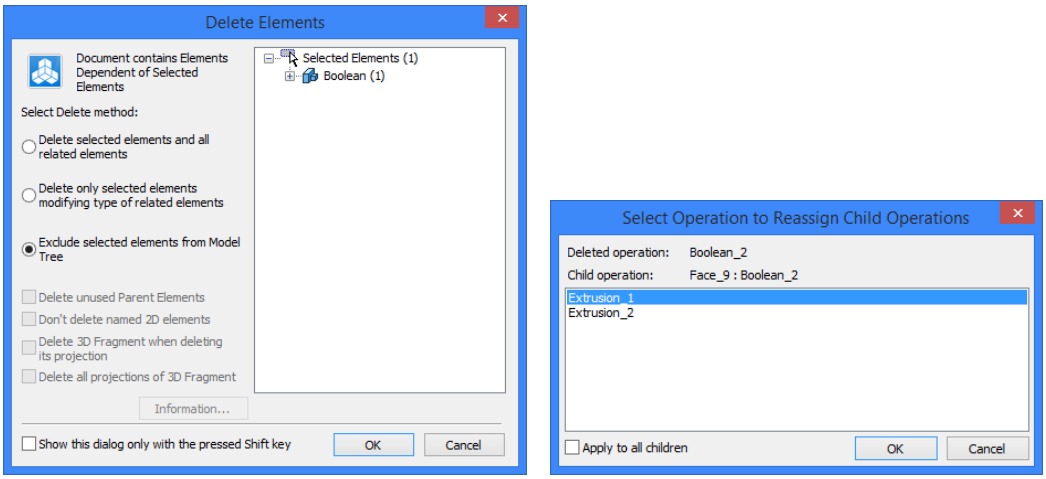
	<Ctrl+T>	Select Target Body for Boolean

If only one body exists in the scene, it is selected automatically. The new body created by the operation becomes the tool body of the Boolean.

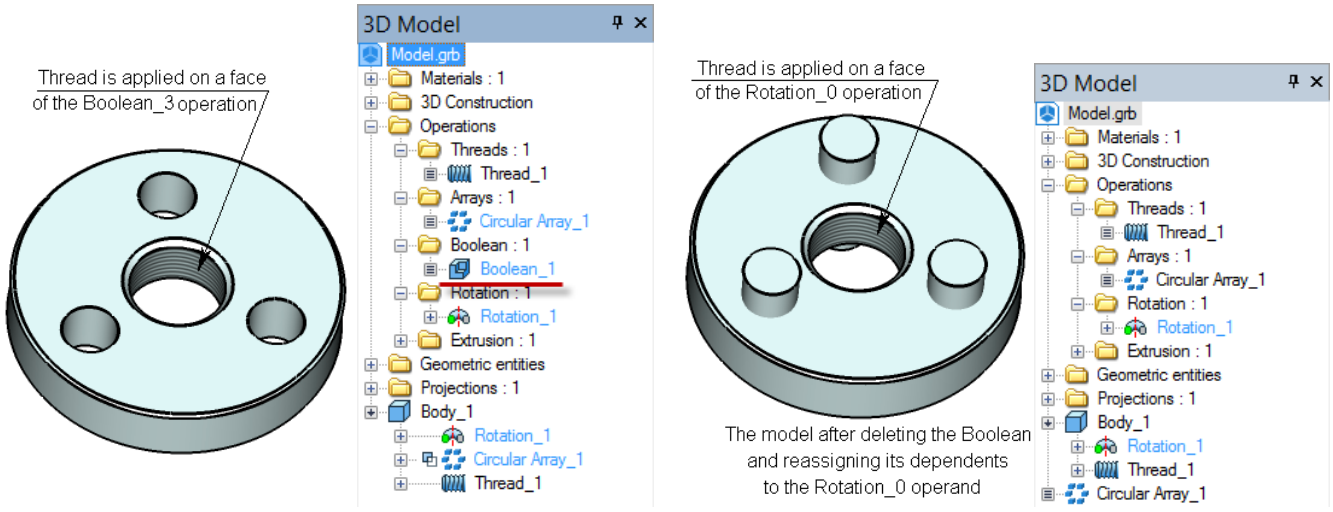
Upon confirming operation creation, first the body is created by the operation, and then the specified type Boolean is performed.

Deleting Boolean operation

If a Boolean is at the top level of the tree, then, upon its deletion, the operand bodies of this Boolean operation will be left in the model. However, if a Boolean operation already has dependents, then the "Delete Element" dialog box will be launched upon its deletion. If you select the option **"Exclude selected elements from Model Tree"** in this dialog box, then another dialog will be launched, "Select operation to reassing child operation", that offers selecting one of the Boolean's operands on which all dependents of the Boolean would be transferred.



The diagram below shows the structure of the Body_1. First, a Boolean "Subtraction" of "Circular Array_1" from the "Rotation_1" operation was done, and then the "Thread_1" was applied on the Boolean's face. Suppose, you attempt to delete the "Boolean_1" with the option of excluding the element from the model tree. Now, if you select the "Rotation_1" operation in the "Select operation to which all references will be re-assigned" dialog box, then the operation "Thread_1" will become a child of the "Rotation_1" operation.



When one of the operands is deleted and the element excluded from the model tree, the Boolean itself will also be deleted.

Boolean operations on 3D fragments

Use of 3D fragments as operands is a unique feature of T-FLEX CAD. This significantly expands capabilities of creating parametric 3D models and assemblies. At the time of inserting a 3D fragment into an assembly, there is a provision of creating a Boolean with the assembly's bodies. For details on capabilities of using a 3D fragment as an operand, refer to the chapter "3D Assemblies Creation" and its

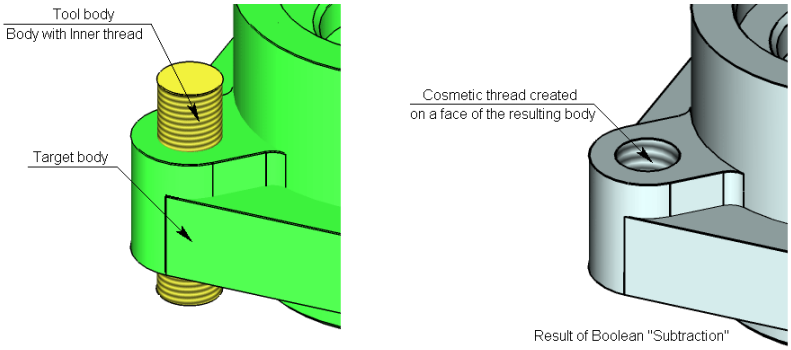
topic “Preparing a T-FLEX CAD document for use as a 3D fragment”, and the chapter “Customizing drawing”, the topic “The tab ‘3D’”.

When you insert a 3D fragment with option for automatic Boolean operation creation, the preview displays not only the inserted fragment, but also the result of the Boolean operation.

Applying thread by Boolean operation

If a cosmetic thread is applied on a face of one of the operands, and this face is in contact with the faces of the other operand, then it is possible to automatically create a cosmetic thread on the face of the resulting body. For example, as shown on the diagram below, a body with a cosmetic thread on its face is subtracted from a base body. In this case, a cosmetic thread is automatically created on the newly created face of the resulting body.

Note that the thread on the face of the tool operand must be “Inner” in order for the thread in the hole of the resulting body to be created “Inner” as well.



BLENDING EDGES

The edge blending operation makes transition between two or more adjoining surfaces belonging to one solid body. Blending can be performed on both solid and sheet bodies.

There are several types of blend:

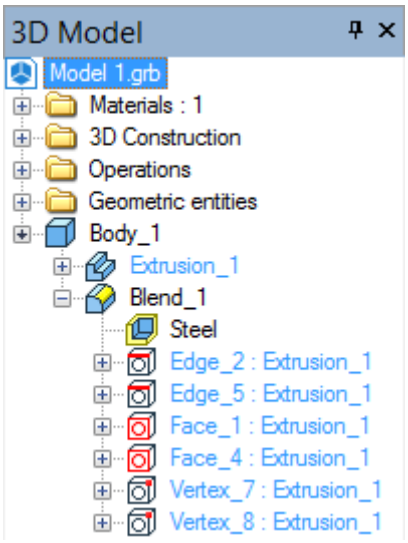
- chamfer (offset; length-angle);
- rounding;
- variable-radius rounding (circular, elliptical).

As the result of the operation, the edge where two faces meet is replaced by an additional surface (the blend surface) providing the required transition from one face to the other.

In the edge blending operation there is a special diagnostics mode available which allows a user to promptly estimate possibility and causes of error appearance.

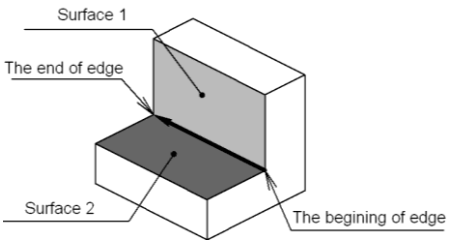
MAIN CONCEPTS. OPERATION CAPABILITIES

The object of the edge blending operation is the edges of the sheet or solid body. Additionally, some vertices may be selected on the involved edges in order to specify particulars of the intended shape. The set of the automenu options allows selection of various objects, besides the edges, such as loops, faces, vertices and
operations. This helps selecting certain common combinations of edges. Thus, for instance, by selecting a vertex, we select all the edges meeting at that vertex. Selecting a loop by the definition means selection of the sequence of edges making this loop. Selecting a face results in selection of all its surrounding edges (single or multiple loops). Selecting a solid automatically makes selected all its edges appropriate for blending.



It is very important to understand how the information is maintained about the original elements subjected to blending. These original elements selected for blending are remembered in the model history. Thus, in case a face was selected, a reference to this face is recorded in the model tree. In this case, the set of edges to blend is built anew at each regeneration. This way of selecting brings the

advantage of having a quite robust operation. If topology of the selected object changes (for example, the number of edges changes), the operation will be correctly regenerated with the new set of edges of the involved object. Otherwise, if a specific edge were selected that would later disappear, then the system would output an error.



Each selected edge has a direction indicated by an arrow. Other blend-defining conditions depend on this direction. The arrow tail defines the edge start. According to an agreement, the surface on the right-hand side of the edge, as looked in the arrow direction, is conventionally called the first blend surface, and the one on the left-hand side – the second.

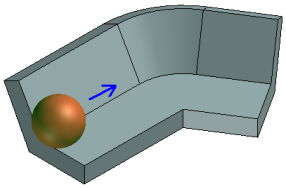
Types of Blend

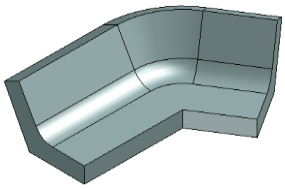
Rounding

Rounding is a most commonly used type of blending. A surface is created along the selected edge, providing a smooth transition from one face to the other. A condition holds that any normal cross section of this surface with respect to the edge is an arc of the specified radius.

To easily realize how the rounding surface is generated, imagine a ball of the specified radius rolling along the selected edge, touching the faces adjacent to the edge. The trace of the ball represents the blend surface for the given edge. The surface may be defined in a single or multiple passes, particularly, if different edges are rounded with different radii. Complex spline surfaces may be constructed to join multiple edge blends.

The operation may add or remove material, depending on whether the angle between the faces of the blending edge is convex or concave.

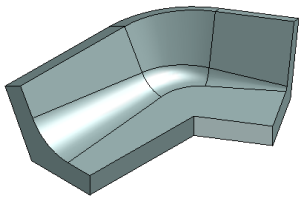


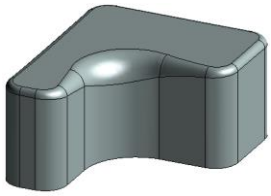


The rolling ball must be touching at least one of the adjacent faces. Otherwise, rounding will fail and the system will output an error.

Variable-radius rounding

This type of rounding creates a surface of a variable radius. A list of critical points is defined. A certain value of the rounding radius can be set at each of these points. Each point position is defined as percentage of the total edge length. Initially, this list contains two points at 0% and 100% of the edge length.

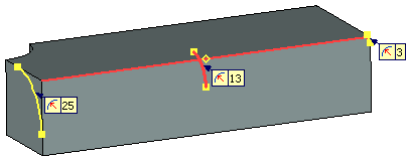


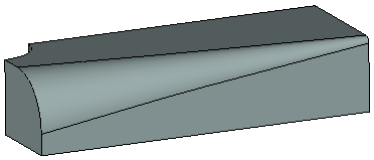


If several edges are selected, then the point positions are defined as percentage of the total chain length for all consequently adjoined edges. The edge joints are not required to be smooth. A single direction is used for the whole chain of edges. It is defined by the direction of the first edge in the chain.

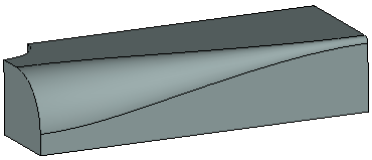
> When defining this type of blending along a closed loop of edges, always make sure that the start and end point radii are equal.

The change in the rounding radius between points can be *linear* or *smooth*. In the linear case, the change in the radius is described with a broken line (the radius changes linearly from one point to another). In the smooth case – the change in the radius is specified by a curve having a horizontal tangent at the start and end points.





Linear change of radius

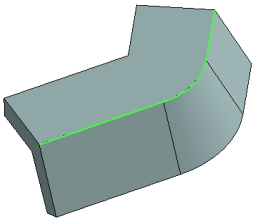


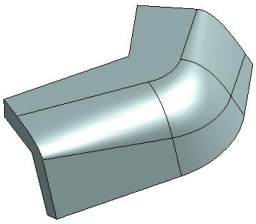
Smooth change of radius

A special mode is provided for processing certain cases, which allows specifying own variable rounding parameters for each individual edge. When processing a smooth chain of edges in this mode, make sure the radii are equal at the meeting ends of the adjacent edges.

Variable-radius elliptical rounding

This type of rounding allows creation of advanced blend surfaces. The sizes and shape of the blend surface is defined by the offsets from the first and the second faces (as conventionally denoted), and by the rho (convexity) value.

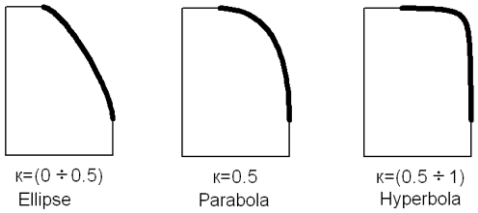




Depending on the rho value of the surface cross section, all resulting surfaces are divided in three groups:

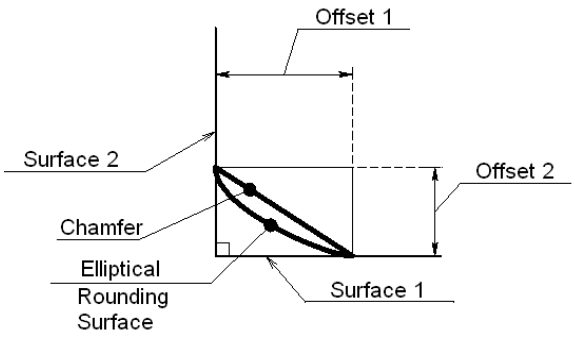
- The first group is with a rho value between 0 and 0.5, excluding the bounds. This results in elliptical shape of the surface cross section.
- The second group is limited to the surfaces with a rho value of 0.5. This rho value corresponds to the parabolic shape of the blend surface cross section.

- The third group lies in the range from 0.5 to 1 (exclusive). In this case, the blend surface has hyperbolic shape.

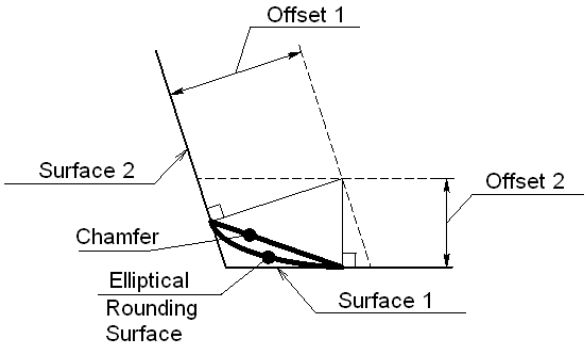


The shape of the surface can be defined at any point on the edge or on the chain of edges. The list is managed in the same way as in the case of the circular variable-radius rounding. The change in the rounding radius between points can be also *linear* or *smooth* as in the circular variable-radius rounding.

The offsets from the faces define the boundaries of the blend surface. The algorithm of constructing the blend surface is common for elliptical rounding and offset chamfer. It is illustrated on the following diagrams:



Construction of the blend surface by offsets in the case of 90-degree angle between the surfaces



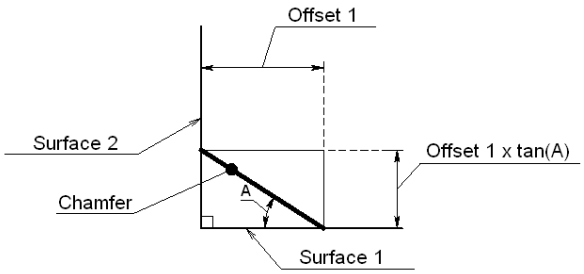
Construction of the blend surface by offsets in the case of an arbitrary angle between the surfaces

- ✓ An offset surface is constructed at the distance Offset_1 from the second face.
- ✓ Another offset surface is constructed at the distance Offset_2 from the first face.
- ✓ The intersection line between the two offsets is projected on the original faces, thus defining the boundaries of the blend surface.

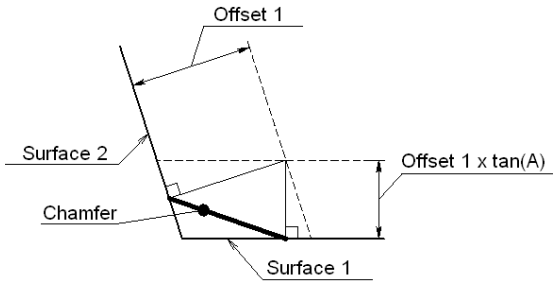
In the special case of blending two flat orthogonal faces, this means the distance from the blended edge to the side boundary of the blend surface is equal to the value of the offset.

Chamfer

A chamfer can be constructed along a selected set of edges. A chamfer can be defined by two offsets from the adjacent faces or by the slant angle and the radius. The algorithm of constructing a chamfer by an offset and an angle is illustrated on the following diagrams:



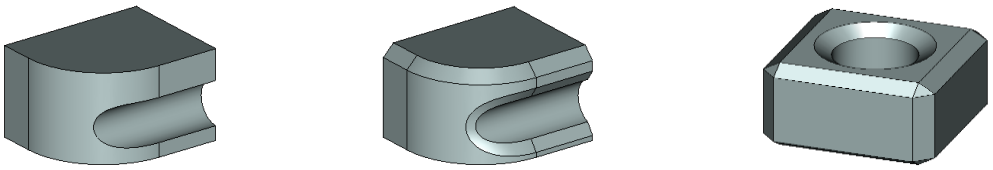
Construction of the blend surface by an offset and an angle "A" in the case of 90-degree angle between the surfaces



Construction of the blend surface an offset and an angle "A" in the case of an arbitrary angle between the surfaces

The second offset is computed based on the angle, by multiplying the value of the first offset by the tangent of the angle. Further computations follow the algorithm of constructing a chamfer by two offsets.

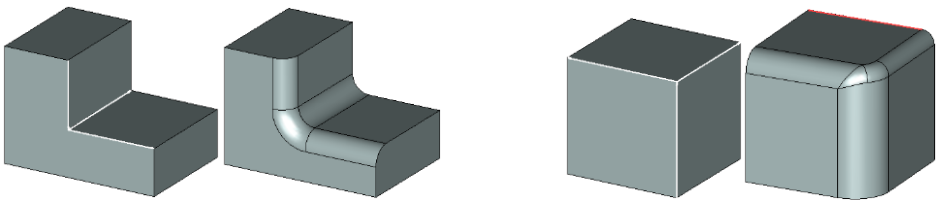
When creating a chamfer, the faces are not required to be flat and the edges straight. They can be of various geometrical shapes (see the diagrams below).



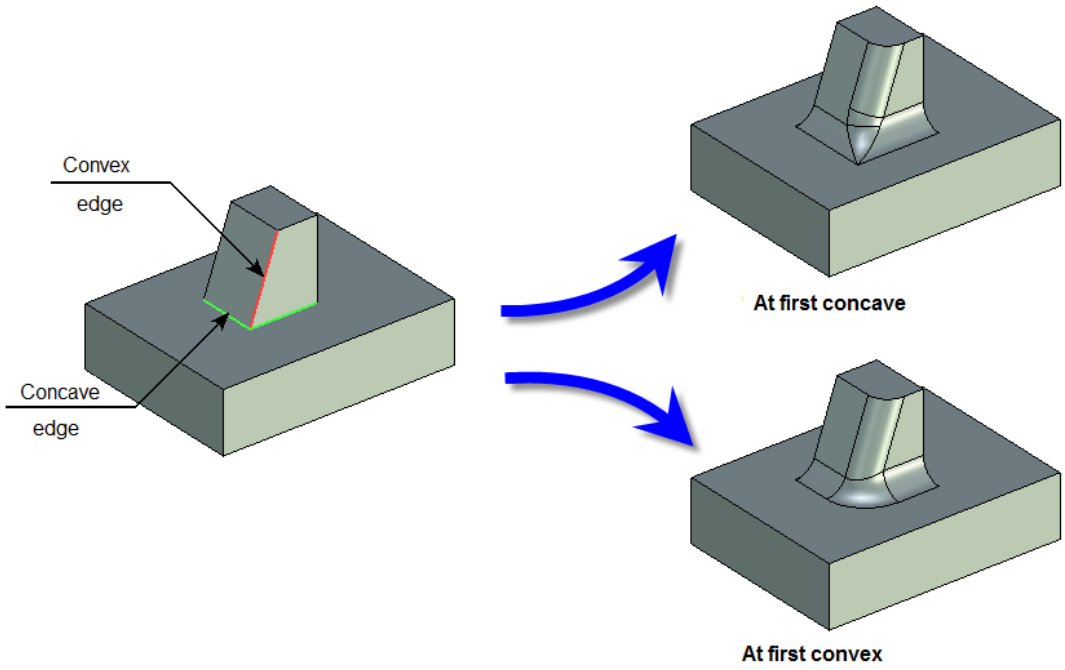
Specifics of Blending Sets of Edges

Automatic vertex blending

A single operation can be used for rounding all edges meeting at one vertex. The edges can be of either the convex or the concave type. Simultaneous rounding of edges automatically involves rounding of the vertex where the edges meet.



The form of the blending geometry at the vertex can be modified by selecting a different order in which the blending of edges is performed: **By default** (the order of the blending of edges is determined by the system itself), **At first concave**, **At first convex** (the order of the blending of convex edges is not determined), **At first with minimum convexity**, **At first with maximum convexity**. It is important to note that in order to control the form of the blending at the vertex, when creating the blend it is required to select, in addition to the edges, the vertex itself.



To avoid the vertex blending, do the edge blending by two blending operations, first blending any two edges meeting at a vertex, and then the third edge. The result in this case will look as shown here:

Sometimes it is impossible to blend all edges at once, and the system outputs en error. In this case, try creating the intended blends by steps.

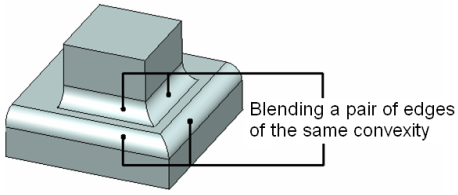
The above implies that proper handling of the operation leads to successful blending of practically any edges of any model.

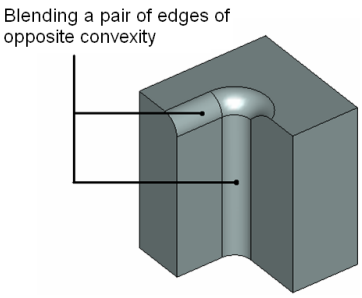
> It is recommended to use as few operations as possible when blending several edges of one solid. Prefer blending a set of edges by one operation, rather than by a number of operations one edge at a time. With this approach, the model regeneration is much faster, and the rounding algorithm most likely performs best.

Blending two of three edges meeting at a vertex

When blending a pair of edges of the same convexity, two blend surfaces are constructed – one per edge. The vertex is not blended in this case; rather, new edges are created at intersections of blend surfaces.

However, when blending a pair of edges of opposite convexity, the vertex where the edges meet is automatically blended.

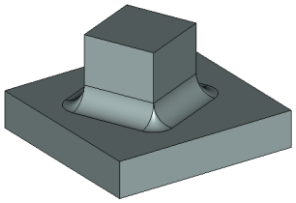


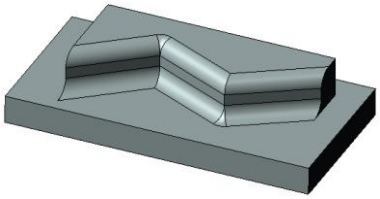


Processing of vertices joining edges of different convexity

Consider the special case when three edges of different convexity meet at a vertex.

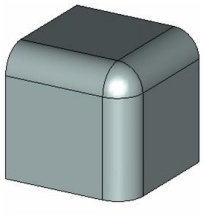
Should it have been just two edges of the same convexity, one could use the option of also blending their common vertex. This would require selecting the vertex as well. The results would have looked like on the diagrams below:



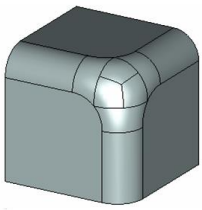


Blending a group of edges with an offset from the vertex

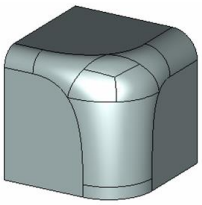
When rounding a group of edges meeting at a vertex, one can specify an offset from this vertex. The following diagrams demonstrate how this offsetting affects the shape of the resulting blend:



No offsets

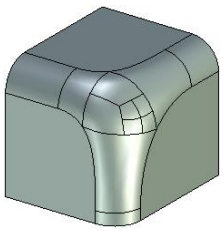


Equal offsets, equal rounding radiuses

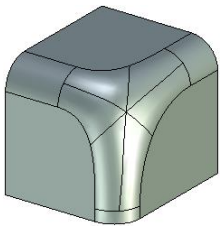


Unequal offsets, unequal rounding radiuses

One can see that the offset defines a zone of smooth transition from one blend surface into the other. The specified radius of rounding is not guaranteed in the transitional zone. Transitional zone can be formed by using transitional surfaces or without them.



Blending with the use of transitional surfaces



Blending without transitional surfaces

The "Offset" parameter can be specified at any edge start and end. However, offsetting can be done only for a set of three or more edges meeting at a vertex. Some of the meeting edges may be directed away and some towards the vertex. That said, make sure the offsets are defined on the vertex side for all edges. If an edge connects two vertices, each subject to offsetting, then the offsets are to be defined for both the edge start and end.

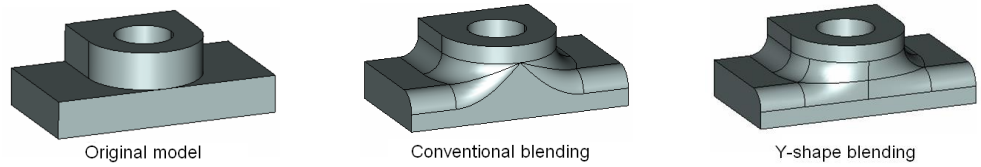
In some cases, T-FLEX CAD system allows automatic synchronization of the "offset" parameters for the edges with opposite direction. This happens, for instance, when selecting a group of edges by picking a vertex. (Detailed description of the steps for defining offsets follows).

Special dynamic draggers are provided for marking offsets on the edges while defining the operation parameters. Their presence and location help easily realize the offset parameters and modify, if necessary, by the mouse.

Special Processing of Edge Blends

Y-shape

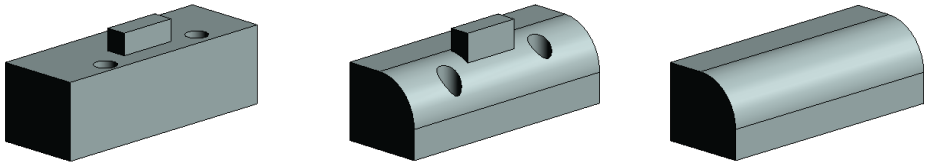
Suppose, two edges of opposite convexity are tangent at a vertex with three or more faces meeting there. In such a case, a special procedure is available for blending the vertex - the so-called Y-shape blending. In this way, surfaces come out lesser curved in the transitional zone around the vertex, then in the conventional way of blending.



The presented way of processing a vertex is only applicable for creating constant-radius or variable-radius rounds.

Topology deletion

In the cases when the rounding surface completely embraces other topological entities, two ways are possible of processing the operation. The system can keep all topological entities, trimming or extending all necessary pieces of the surfaces involved. Alternatively, only the blend surface can be left, with all embraced objects deleted.



Overflow processing

In real situations, an overflow may quite often occur between the rounding surface and some elements of the solid geometry that are not the adjacent faces of the rounded edge. In such cases, the system may use special options for processing various types of overflow. Four types of overflow are processed:

- ✓ **Smooth overflow** – when the rounding surface smoothly merges into another rounding belonging to the solid body being processed. The surface is deformed in the transition zone in order to provide tangency to both the original face and the surface being merged into.
- ✓ **Keep cliff edges –** when the rounding surface "creeps" over an edge not subject to rounding. The rounding surface is constructed near the zone of contact with the edge by the rolling ball method. The surface is swept by a ball rolling on top of the "constraining" edge, touching one of the original faces. When processing an overflow that keeps cliff edges, you can additionally engage or disengage the algorithm for processing the cases when only a portion of the surface being blended (on the sides - **cliff end**) "creeps" on the cliff edge. Disengaging more complex algorithms that are not required would allow speeding up to some extent the model regeneration.
- ✓ **Notch –** the rounding is constructed in such a way that the element affected by the overflow is ignored. The rounding surface is not deformed in this case, and the surfaces of the notch element are trimmed or extended to the rounding.
- ✓ **Inside tight** – some portions of the model subjected to blending are represented by the surfaces with so small curvature radius that the blending ball of the specified radius cannot roll through while maintaining contact with the sidewalls. In such a case, the system can use special processing for providing blending with the specified radius in the curved zone:
  - **Tight blends** – in this case, the system will process the curved zones using the fastest algorithm. The algorithm assumes that the surfaces in these zones are created solely by blends of constant radius. If the type of the original surface is different and the new blend cannot be created, an error is output.
  - **Tight faces** – the system processes the curved zones, assuming, that the zone of small curvature radius fully extends through some face of a sidewall.
  - **Tight partial** – this is the most complex, but also versatile, algorithm. It allows processing the cases, when the zone of small curvature radius covers the portion of a sidewall of the blend. In this case, the blend surface is additionally subdivided in order to provide correct processing.

With an option engaged, the system processes the respective type of overflow when occurs. If various types of overflow occur between the rounding being created and several other rounds then the system finds an optimum solution. The order of applying possible solutions in this case will be first the smooth overflow, then the overflow with cliff edges, and, finally, the notch. The type of processing of the curved zones depends on the selected algorithm.

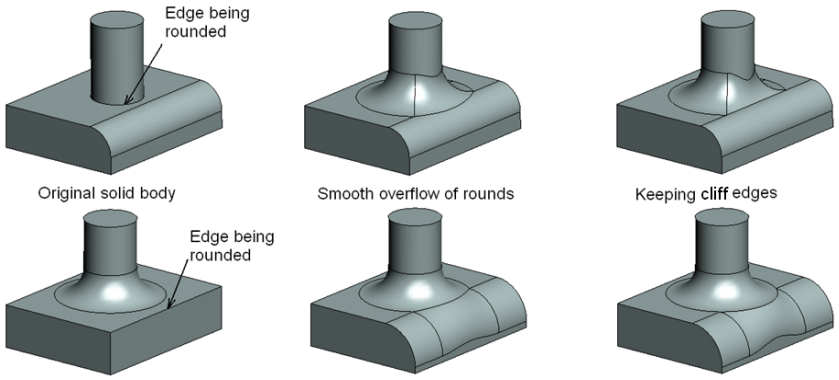
The user can control the system behavior when overflow occurs. The user usually gets involved when the results are different from expected. You can set the overflow options in any desired combination.

> The system attempts applying all three types of processing overflow in the presented order. If any of the overflow types is turned off, it is excluded from the list and is not applied.

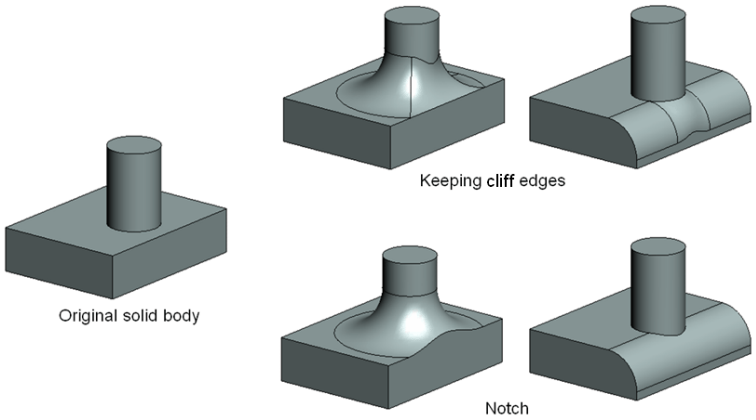
If all three options are turned off when rounding overflow occurs against the body faces and edges, then the system gives up rounding and outputs a message about a failing rounding.

The following set of examples demonstrates the operation capabilities when using various types of overflow processing:

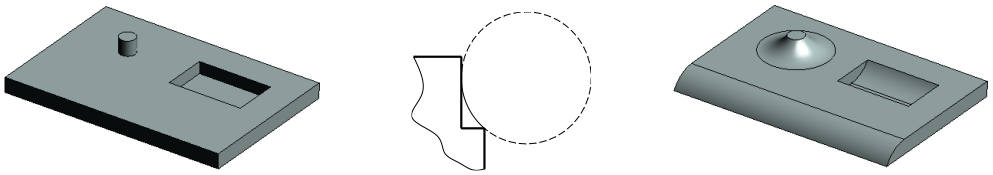
- ✓Overflow processing example: switching the smooth overflow option on and off. The "**Notch**" option is not applicable in this example.



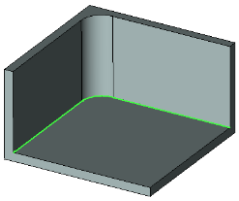
- ✓Overflow processing example: switching on/off the option "**Keep cliff edges**". The "**Smooth overflow**" option is not applicable in this example.



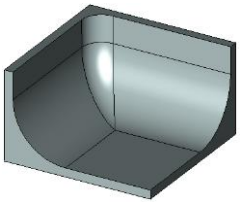
- ✓Example of constructing a blend surface with the option "**Keep cliff edges**" turned on. This is the situation when the blend surface of the specified radius cannot be made tangent to both faces of the edge being rounded. If the option is turned off, the system will output a message about a failing rounding.



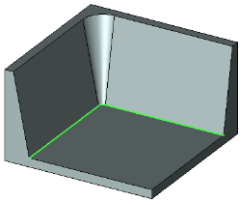
- ✓Examples of processing zones of small curvature radius on blend sidewalls based on various algorithms. In the areas where the blend radius is created then the curvature radius of a sidewall of the blend, the system makes a correction of the blend surface.



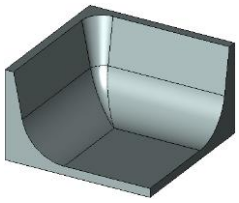
Unprocessed original



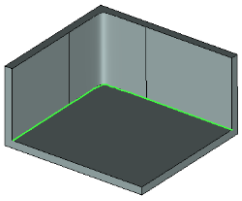
"Tight blends" processing



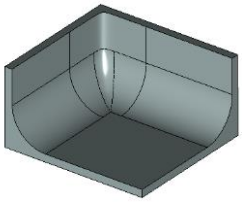
Unprocessed original



"Tight faces" processing



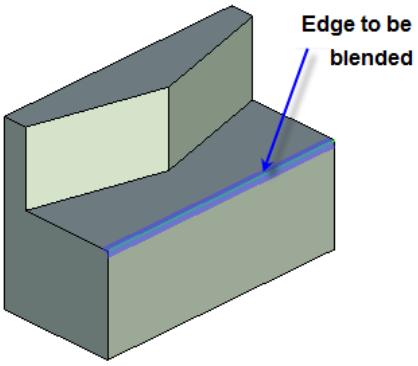
Unprocessed original

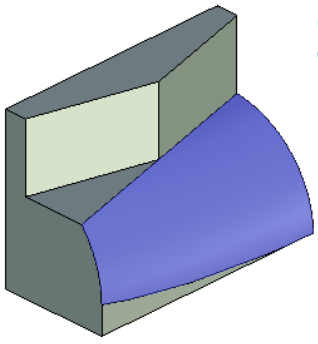


"Tight partial" processing

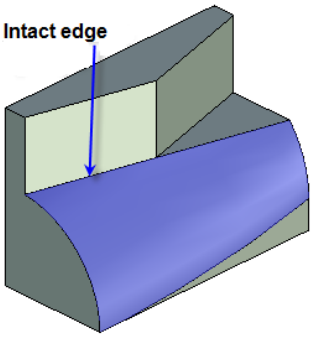
Explicit specification of intact edges

When the blending is created, it is possible to specify the edges of the body which must remain intact upon blending. As a result, the blend surface will contain these edges.





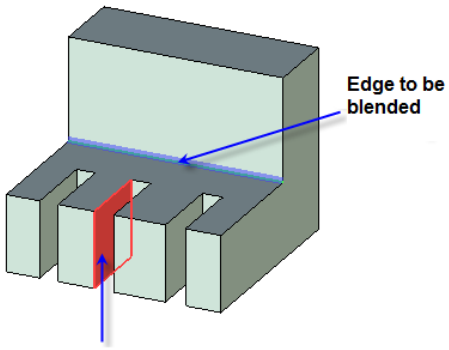
Result of blending by default

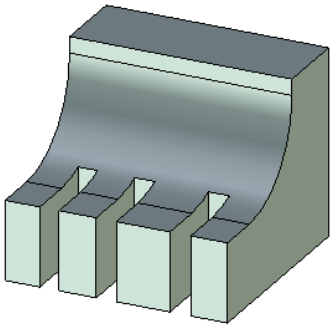


Result of blending when the intact edge is specified

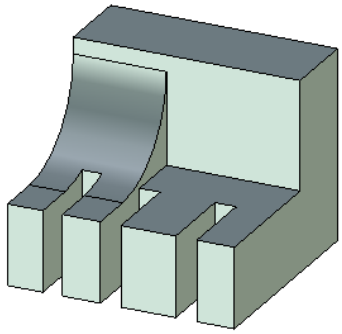
Specifying bounding geometry

Specification of the bounding geometry allows us to bound the blending by the geometry of an arbitrary face, a plane or a list body. This functionality allows us to perform the blending in various complex configurations.





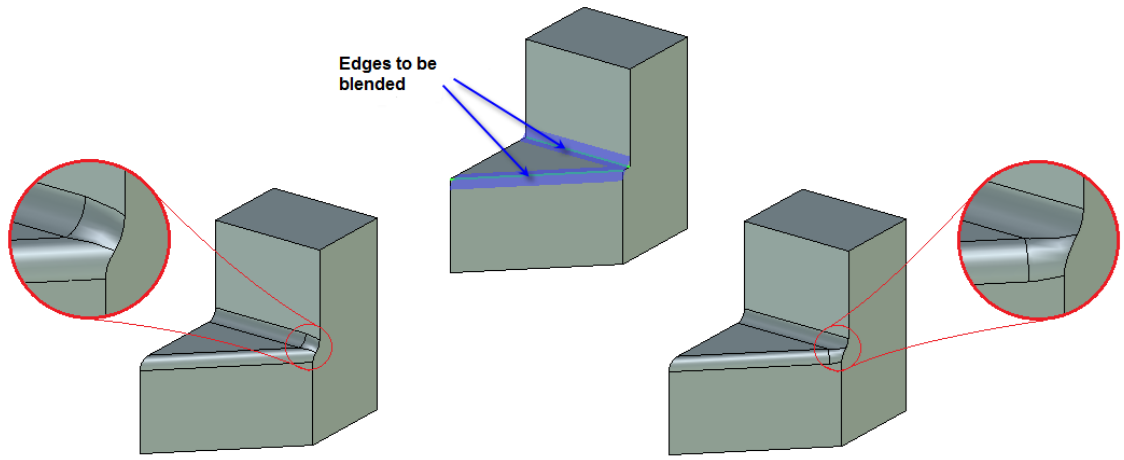
Result of blending without constraints



Result of blending with the use of bounding face

Specifying the order of creation of blending when blend surfaces overlap on several edges

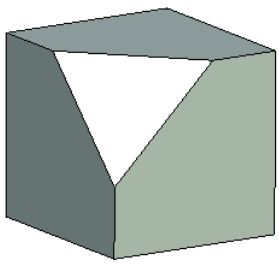
When the blend surfaces overlap on several edges of different convexity, it is possible to achieve the desired behavior of these surfaces by changing the order in which the blending will be performed. The order of the blending is determined by the selected priority of the edge processing: at first concave or at first convex.



Result of blending with different order of edge processing

Specifying chamfer at the vertex

For a vertex at which three edges meet it is possible to create a chamfer without specification of the blending on adjacent edges. This option works without specification of radii of the blending; it works simply by selection of the vertex with three edges. It constructs the triangle through the points that are shifted from the vertex at a given distance and cuts off the body with this triangle. This mode also works for the vertices at which more than three edges meet. In this case, if it is possible, a plane is also created, and if it is not possible – a smooth surface covering the domain.

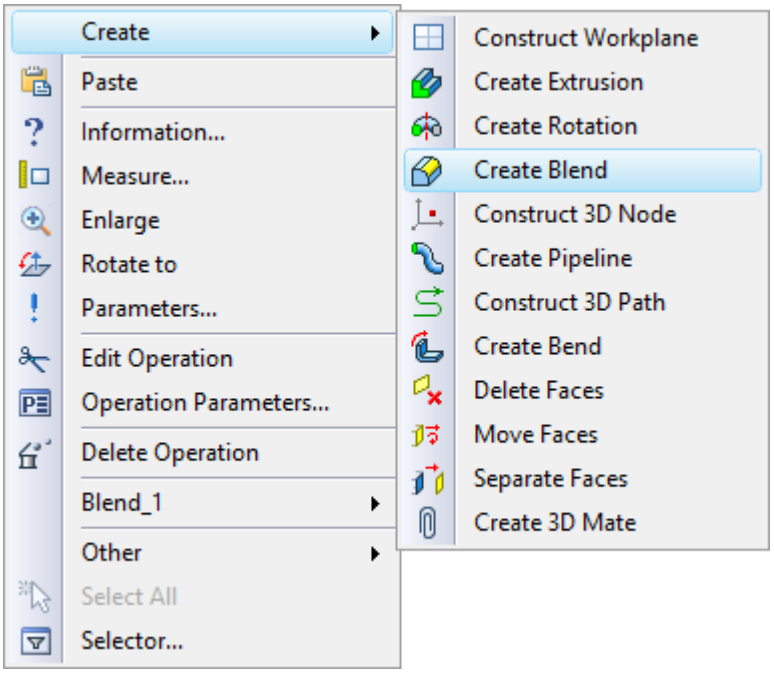


Rules of Defining Operation

The command "**3DE: Create Blend**" is used for blending edges:

Icon	Ribbon
	3D Model → Create → Blend → Edge
Keyboard	**Textual Menu**
<3DE>	Operation > Blend >Edge

The command can also be called from the context menu on the right-click after selecting any of the elements appropriate for blending. These are faces, edges, loops, vertices and operations.



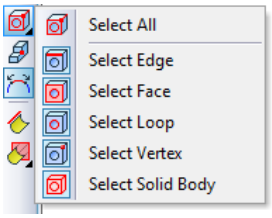
The following sequence of actions is to be done for creating a blend operation:

- ✓ Select elements to blend.
- ✓ Define blend parameters.
- ✓ Confirm the operation.

How to Select Objects

Upon launching the command, a general-purpose option is active in the automenu. It allows selecting any appropriate objects for the operation:

	<E>	Select Faces, Edges, Vertices



The drop-down list of the given option contains the filters for selection of various 3D objects: edges, faces, loops, 3D vertices and bodies.

Selection of elements is done in the 3D window. Move the cursor over the desired object. The element to be selected will then get pre-highlighted, and a tooltip will pop up next to the cursor identifying the element. Click to select.

Let us consider, in more detail, the use of each selection filter from the list of the option:

Select Edge. This option sets the system for edge selection. It can be used together with the smooth chain selection mode, activated by the option:

	<S>	Smooth edge chain selection mode

A smooth chain is a set of edges making an open or closed continuous smooth curve (a C^2 curve).

When using this mode all edges are selected automatically that can make a smooth chain together with the edge being selected.

Select Vertex. This option sets the system for selection of vertices. When selecting a vertex, all edges meeting at this vertex are selected. The zero offset from the selected vertex is automatically defined for the thus selected group of edges. If the directions of the meeting edges disagree, the **Start** and **End** parameters are synchronized so that for all edges the offsets are defined in the selected vertex.

This filter can be used in the mode of selection of vertices without automatic selection of edges. This is necessary for creating some special cases of blend, such as Y-shape blend and processing of vertices joining the edges of opposite convexity. Here is the option turning this mode on and off:

	<Z>	Allow vertex selection without edge selection

In this way, selecting a vertex does not cause selection of edges. The selected vertex is added to the list of elements for blending.

Select Loop (a closed contour). The system prompts us to select the closed contour of edges (loop). The user is not required to define start and end of the edges belonging to a closed loop. In this case, the blending parameters are set simultaneously for all edges. Individual parameters can't be defined for any single edge in the loop. By the convention, the first face of the blend is considered the face surrounded by the loop. This will be also the first face for each edge in the loop. Remembering this convention is important when creating chamfers.

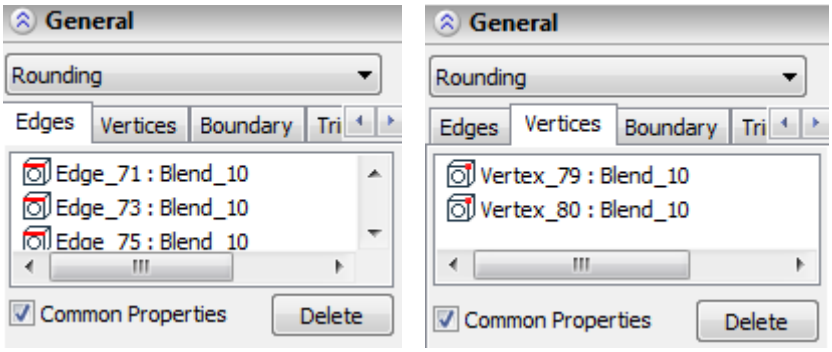
Select Face. The system prompts us to select the faces. As in the previous case, same parameters are used for all edges of the selected face. The selected face is also considered the first face of the blend for each edge.

Select Solid. The system prompts us to select the bodies (operations). Upon selecting an operation, all edges of the solid appropriate for blending are added to the selected list. This is a very different way

from the two previous options. It allows defining individual settings for different edges of one selected solid.

The objects selected for blending are added as a list to the special fields of the dialog of the operation's properties. Edges, faces and loops are added to the list on the "Edges" tab. Vertices selected with the help of the option (without selection of edges) are added to the list on the "Vertices" tab.

The user can cancel the selection of objects for blending with the help of the **[Delete]** button in the command's properties window or by repeated selection of elements in the 3D window.



Defining Operation Parameters. Using Draggers

The operation parameters can be defined at any step of creating the operation. The operation parameters can be defined in the **property window**, which supports the transparent input mode, or the **parameters dialog box**, that requires confirmation before the changes take effect. The property window works together with helper graphic widgets called "draggers".

The property window of the "Blend Edge" operation consists of three sections: **General Parameters**, **Overflow Processing**, **Options**. Some of the controls in the parameter window may get hidden or replaced by others. This depends on the type of blending or the type of the element being processed (edge or vertex).

We recommend to begin defining parameters after selecting the elements for blending. As was mentioned above, the selected edges, faces, and loops are displayed in the list on the "Edges" tab. The vertices selected with the help of the option (without selection of edges) are added to the list on the "Vertices" tab. Blending parameters can be defined for a particular object by selecting it from these lists. Originally, blend properties are set same for all elements. This is signaled by the flag **"Common Properties"**. Individual blending parameters can be defined for each element in the list after clearing this flag.

The blend type can be selected from the pull-down list. Depending on the blend type, the appropriate controls appear for defining parameters attributed to this type.

Using draggers

Draggers are helper graphic widgets for dynamically defining main numerical parameters of the blend in a transparent fashion. Draggers appear automatically once the objects for blending have been selected. Their quantity, look and size depend on the type of rounding, the kind of selected objects and the type and values of the parameters modified by the draggers.

The draggers are displayed in various colors. By default, the active dragger (the one being manipulated by the mouse) is drawn red, all the rest (inactive) are yellow.

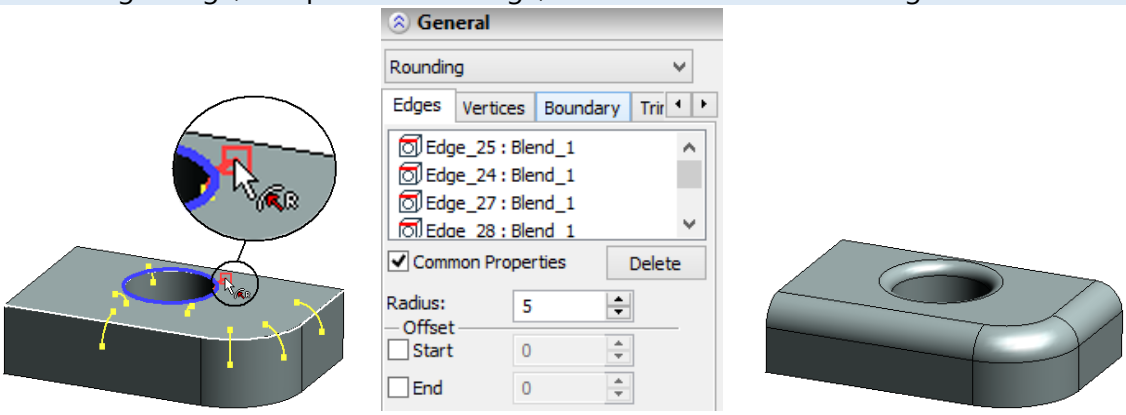
> Using draggers for blending of many edges at once may cause delays in system responses, especially on low-performance computers. To speed up the system in such cases, the draggers can be turned off. This is controlled by the respective checkbox item in the "**Options**" section of the property window.

The following paragraphs describe all kinds of manipulators for each blend type.

Defining constant-radius rounding

- ✓ Select objects for rounding. It is convenient to do this first. However, if necessary, objects can be selected at any step of defining the operation.
- ✓ Set the type of blend to **Rounding** in the pull-down list.
- ✓ Set the radius of the rounding. Initially, it can be roughly set by the draggers.

> The draggers for marking and modifying the rounding radius are displayed at each selected edge: one per a straight edge, two per a curved edge, and four on each closed edge.

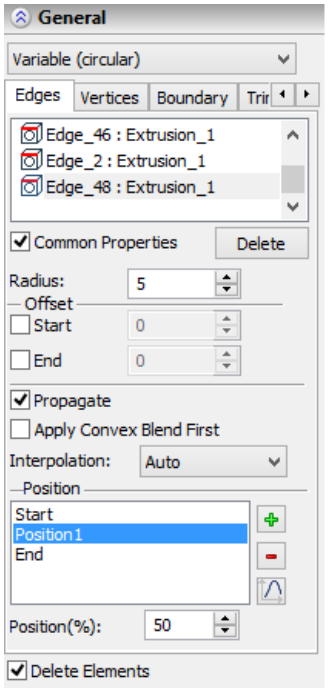


Initially a single value of the radius is set for all edges processed by one operation. Moving one dragger results in all the rest instantly adjusting by the same amount. To define individual properties of some edge, select it in the list and unset the flag "**Common Properties**". Then, an individual value of the radius can be entered in the "**Radius**" input box. The radius-controlling draggers on this edge will become independent from the rest.

- ✓ Confirm the operation creation with the button .

Defining variable-radius circular rounding

- ✓ Select edges for rounding.
- ✓ Select the type **"Variable (circular)"** in the pull-down menu of the property window. The necessary controls will appear for defining the rest of parameters.
- ✓ Next, the set of intermediate points needs to be defined, and radii specified in each point.



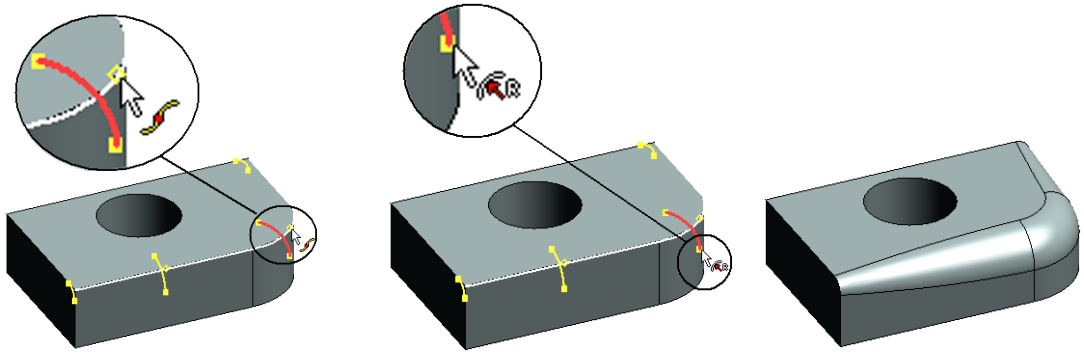
Important: initially, the system tries to compose one chain (not necessarily smooth) of all selected edges. If this is impossible, then several chains will be composed. Within each edge chain, the set of points is distributed along the full length of the chain.

The system can automatically compose the chains from the edges marked with the flag **"Common Properties"**. If the flag **"Common Properties"** is off for some edge then the whole set of points will be distributed along this edge only, and stay strictly specific to this edge.

> If several edges are processed simultaneously that make a smooth chain, while the **"Common Properties"** flag is off, make sure the radii on each side of the edge joints are equal.

The list of points defining the radii is input in the pane. Initially two points exist in the list – "Start" and "End", at 0% and 100% of the edge (chain) length respectively. These points are always present, and their positions are not modifiable.

To add a new intermediate point, press the **[Add]** button. A new dragger will appear on the edge controlling the new point position and the blending radius at this point. By moving the dragger handles, one can vary both parameters of the point being processed.



The current position of the intermediate point is displayed in the **Position (%):** input box. An exact value can also be entered here from the keyboard.

To adjust the radius value at an intermediate point, first select this point from the **Position** list. Then, the radius value can be entered in the **Radius:** field.

Position of the points may be specified with the help of a graph. Use button in the properties dialog for this purpose. This button will open the graph editing window. The range of definition by X is 0-100 corresponding to the percentage from the length. The value area is limited with positive numbers only. See more details on operations with graphs in chapter "Graphs".

- ✓ Specify the type of the radius change – **Linear**, **Smooth**, or a special option – **Auto**;

  The type of the radius change is selected from the drop down list of the parameter **Interpolation**. The option **Auto** is set by default. In this case:

  a. if for the current rounding only start and end points are specified, the radius change will be calculated by the linear algorithm;
  b. otherwise (when internal points are specified) a smooth algorithm of the radius change is applied.

- ✓ Confirm the operation creation by pressing the button .

Defining variable-radius elliptical rounding

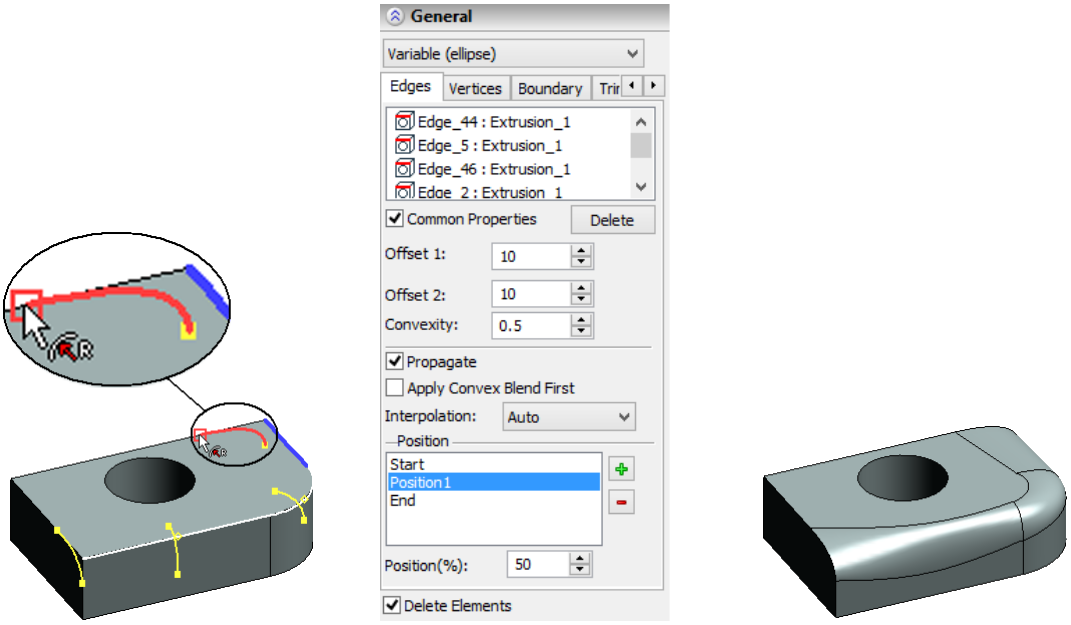
- ✓ Select edges for rounding.

- ✓ Select the type **"Variable (ellipse)"** in the pull-down menu of the property window. The necessary controls will appear for defining the rest of parameters.

- ✓ Next, the set of intermediate points needs to be defined, along with the offsets and rho (convexity) values at each point.

Elliptical rounding is based on special points, just like the previous, variable-radius, rounding. The point defining techniques are similar to those described for the variable-radius circular rounding (excluding possibility of setting positions of intermediate points with a graph). The difference of this blend type is in the ways of defining the shape of rounding.

In this way of blending, two offsets are defined at each point defining boundaries of the blend at the given cross section, and the rho (convexity) value of the cross section curve.



In general, without use of draggers, the following procedure is recommended for defining the blend surface parameters at the points:

Select the desired edge in the list of selected elements.

Enter the point list pane.

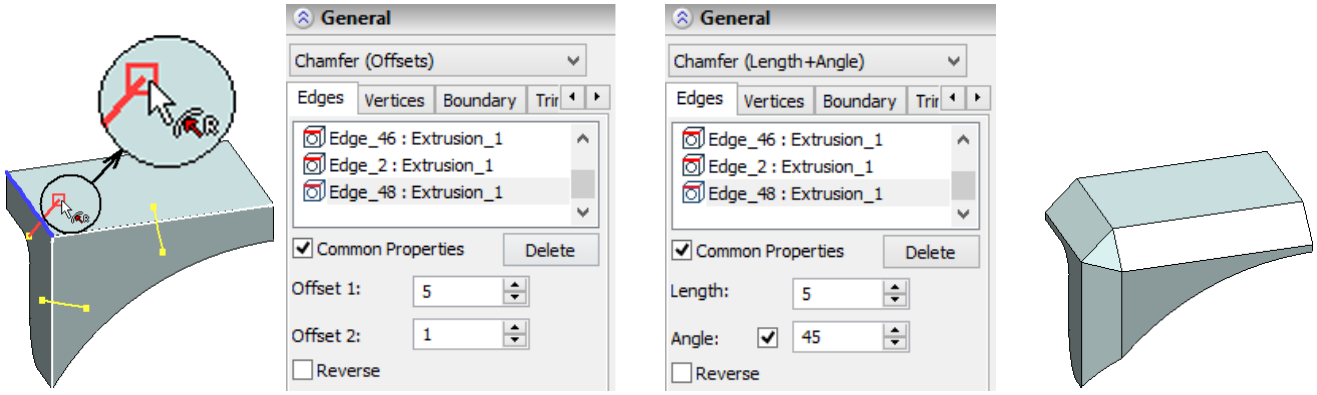
Add required number of intermediate points; define their position on the edge or chain of edges.

Go through the points of each edge, defining the offsets and rho (convexity) values.

- ✓ Specify the type of the radius change – **Linear**, **Smooth**, **Auto** (in the same way as for creating variable-radius circular rounding);
- ✓ Confirm the operation creation by pressing the button .

Defining chamfer

- ✓ Select edges for the chamfer.
- ✓ Select the blend type **"Chamfer (Offsets)"** or **"Chamfer (Length+Angle)"**.
- ✓ Define chamfer parameters. The set of parameters is different for various chamfer types. The easiest way is to roughly set up values using the chamfer dragger. Exact values can be entered in the property window.



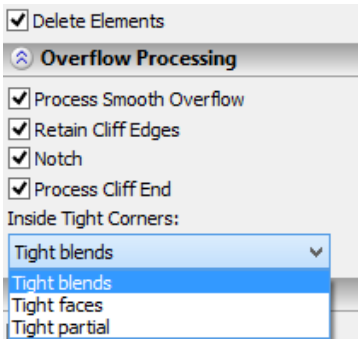
When defining the offset chamfer, one dragger handle defines the first offset, the other second. The edge direction determines which side of the edge is the first, and which is the second. The edge direction can be flipped by toggling the "**Reverse**" flag. This flag works individually on each edge and does not depend on the "**Common Properties**" setting.

When defining chamfer by angle, the second dragger handle will be driving the "**Angle**" parameter. A special flag can be set in the property window against the "Angle" item that will block angle modification by the second dragger handle. In this way, moving either dragger handle will drive the first offset while keeping the angle fixed.

✓ Confirm the operation by .

Manipulating the special blend options

Topology deletion. A special flag "Delete elements" in the property window controls topology deletion. This functionality is available for all types of blend.



Overflow Processing. Processing overflows is defined by the respective flags at a separate section of the window. All these flags are turned on by default. With such settings, the system attempts to apply all possible overflow processing types in the order the options appear.

An algorithm of processing zones with small curvature radius is selected from the combo box.

Propagating blend on tangent edges. The smooth chain selection functionality is available for all types of rounding. The edges smoothly adjoining to the selected one will be also blended. Blend propagates in the order of the elements on the "**List**" pane. This functionality searches for a smooth continuation of the currently blended edge. A smooth chain may exist originally or form as new edges are created on the blend boundary. In the case of successfully defining a smooth chain the system attempts to propagate the blend along the whole chain.

The option described above has a similar purpose. However, when selecting an edge with this option, all elements of the identified chain are placed on the "**List**" pane. When using the "**Propagate**" flag, the list may not contain all edges of the blended chain, rather, could be just one edge. This is important in view of possible topology changes of the model when the smooth chain contents may change. In this way, one can select just a single "most stable" edge of the whole chain to be blended.

Defining Y-Shape blend

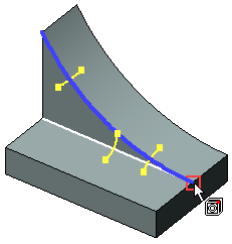
- ✓ Select edges subject to blend.
- ✓ Select the vertex intended for the Y-Shape blending. To select a vertex without selecting edges use the automenu option:

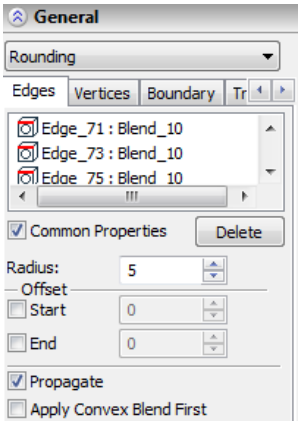
	<Z>	Allow vertex selection without edge selection

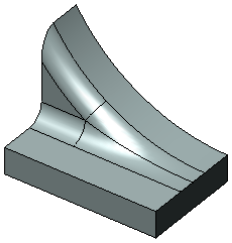
- ✓ In the list on the **Vertices** tab highlight the vertex and set the **Y-Shape** flag for it.

If Y-Shape blend can't be constructed in the given vertex, the system will attempt conventional blending, ignoring the **Y-Shape** flag.

- ✓ Specify the blending radius for the edges. This can be done by using the respective draggers or by entering in the "**Radius**" input box that becomes accessible after highlighting the edge in the list of selected edges on the "**Edges**" tab.







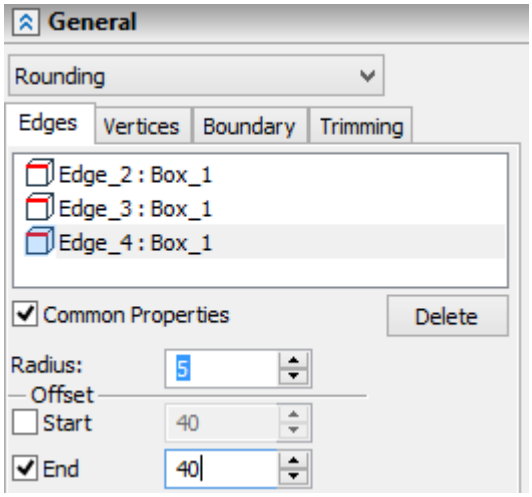
- ✓ Confirm the operation by .

Defining chamfer at the vertex

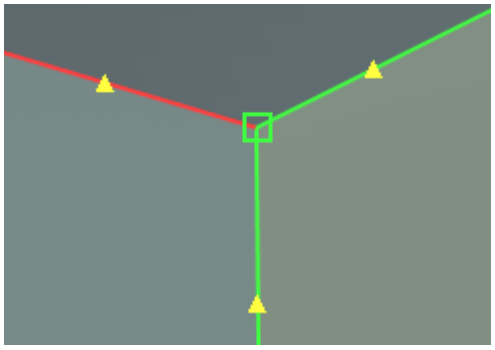
To create a chamfer at the vertex, it is the most convenient to follow the algorithm described below:

- ✓ Select a vertex for a chamfer. All edges that adjacent to the vertex will be selected on the **Edges** tab.
- ✓ You should specify offset for the edges.

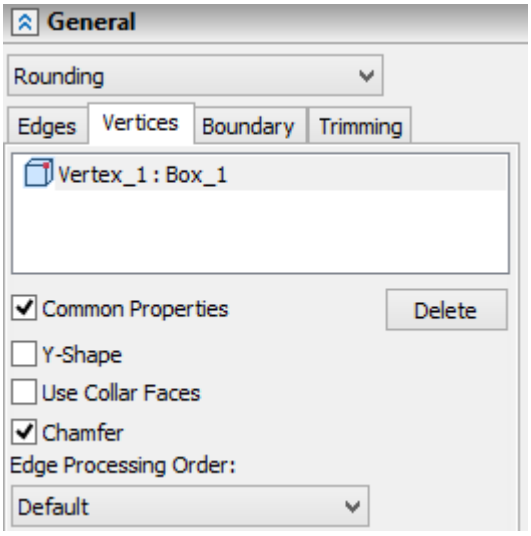
The **Start** parameter will be active for two edges, and **End** will be active for one edge. You should specify the value for the offset in the field near the parameter.

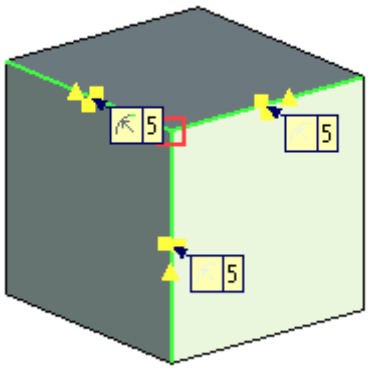


Manipulators that display offset values will be displayed in the scene near the selected vertex.

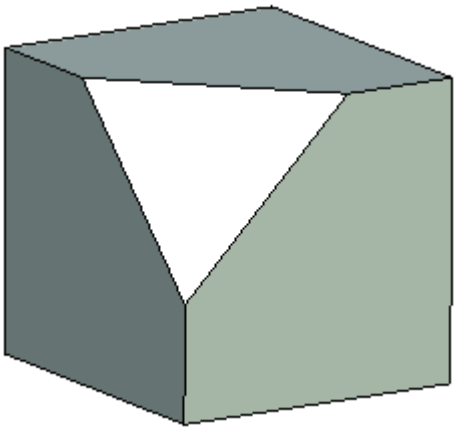


✓ On the **Vertex** tab you should select a vertex for the chamfer. You should activate **Chamfer** flag for it.





✓ Confirm creation of the operation by .



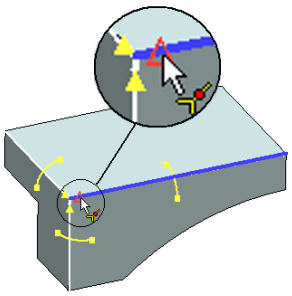
Defining a blend with spherical vertices and offsets form vertices

The vertex offset parameters can be defined for all types of edge blends, except chamfer. The following steps are recommended if planning to make a blend with vertex offsetting:

✓ Selection of objects for blending (the edges) is to be done by vertex selection where the vertex offsets are expected. Make sure to turn off the option to allow edge selection when selecting vertices.

The zero offsets from the selected vertex are automatically pre-set when selecting edges in this way. This is indicated by the special draggers displayed near the vertex.

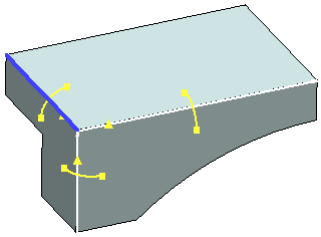
The list of selected edges on the **Edges** tab lists only the edges adjacent to the vertex. The vertex itself is not in the list.

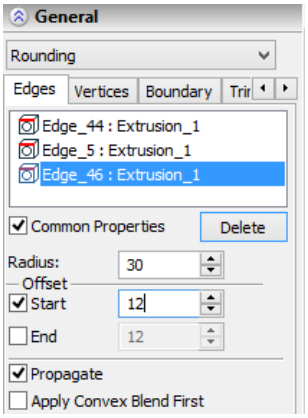


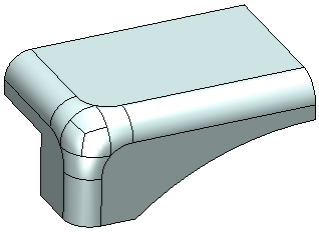
✓ Specify the offset values. This can be conveniently done using the offset-driving draggers.

Initially, the flag **Common Properties** is set for all edges. This allows entering equal offsets for all edges meeting at the vertex. Modifying one offset causes all the rest edges assume the same value. This is seen by the appearance of all offset-driving draggers.

To define different values of the offsets, turn off the flag **Common Properties**. This does not have to be done for all edges at once. Individual properties can be set only for some particular edges by turning off only their flag **Common Properties.** For the rest, the common properties will still be used.



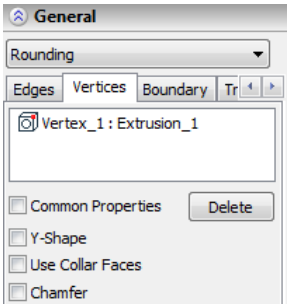


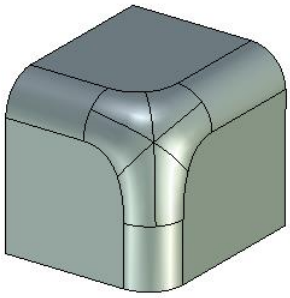


The offset values are displayed in the property window under the items **Start** or **End** - depending on where on the edge is the offset. Exact values can be typed in these input boxes as well. Before each item there is a checkbox activating the **Offset** parameter at the start or at the end of the edge. As we were selecting the edges in the described way (by selecting the vertex), the system was automatically flagging the appropriate ends for the edges meeting at the vertex. This was done according to each edge orientation in the group. Some edges have the offset at start, others at the end, so that all of them have the offset from the vertex where they meet.

For ultimate control, one can manually define the offset parameters using the described flags. Meanwhile, draggers can be used for previewing offset characteristics.

- ✓ Define the radius values for the edges to blend. Any blend type is supported, except for chamfer. The blend parameters are assigned to edges in one of the described above ways, depending on the blend type.
- ✓ If it is required to construct the blending without transitional surfaces, it is necessary to additionally select the vertex itself. This can be done by enabling the option. The selected vertex will be added to the list on the "Vertices" tab. For the selected vertex the **Use Collar faces** flag will become.





- ✓ Confirm the operation by .

Processing vertices joining the edges of different convexity

This special processing is only applicable for constant-radius rounding.

Suppose, three edges meet at a vertex, two of the same convexity, and the third one of the opposite. To blend the two edges of the same convexity with vertex processing, additionally select this joint vertex. To select the vertex alone, use the following special option,

	<Z>	Allow vertex selection without edge selection

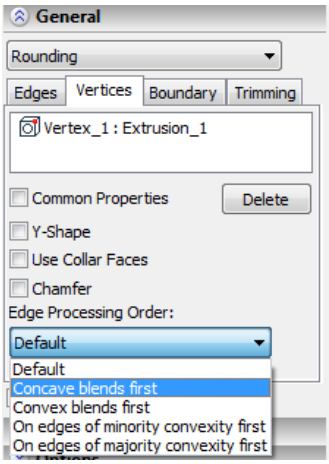
With the option active, select the vertex. The vertex will be added to the list on the **Vertices** tab of the property window. The rest of parameters are defined in the way described for the **Rounding** blend type.

Defining the order of blending of edges upon blending of vertices

To enforce automatic blending of a vertex, in the general case, it is necessary to select all edges meeting at the given vertex. This is sufficient for creation of blending. However, if the option for selection of the order of blending of edges is expected to be used, it is required to additionally select the vertex itself with the help of the option. For the selected vertex, the **Edge processing order** parameter will become accessible.

An approximate algorithm for creation of the blending in this case is the following:

- ✓ Select objects (edges) for blending. It is possible to select the edges explicitly or indicate the vertex at which they meet, when the option is disabled.
- ✓ Specify the value of the radius for the edges to be blended. Specification of parameters of blending for edges is carried out according to the general rules described above for the type of blending being employed.
- ✓ If the form of the blending geometry at the vertex is expected to be controlled, it is necessary to additionally select the vertex itself. This can be done by enabling the option. The selected vertex is added to the list on the "Vertices".
- ✓ For the selected vertex the **Edge processing order** parameter will become accessible. From the drop-down list it is possible to select one of the following options: **By default**, **At first convex**, **At first concave**, **At first with minimum convexity**, **At first with maximum convexity**.
- ✓ Confirm creation of the operation by .

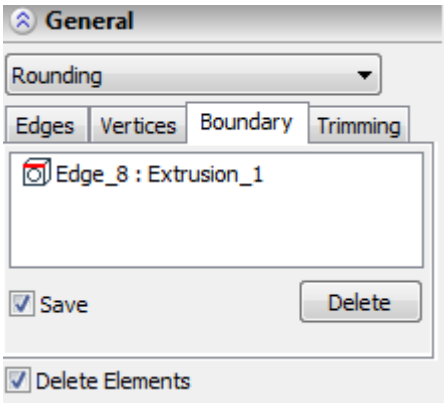


Defining intact edges

To specify intact edges, use the option:

	<L>	Select intact edges

Selected edges are added to the list on the **Boundaries** tab.

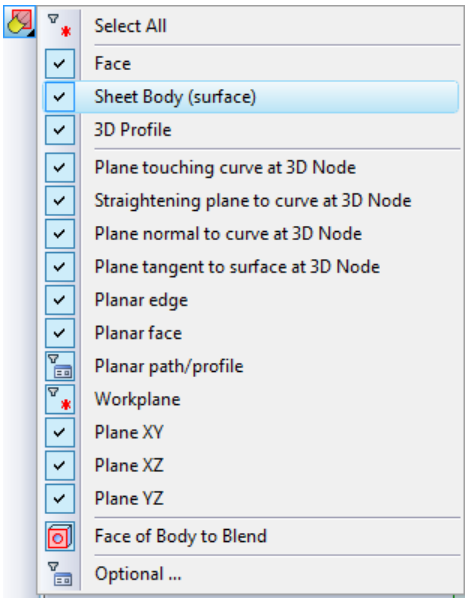


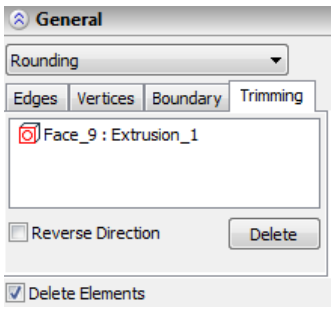
Defining bounding geometry

To specify bounding geometry, use the option:

	<T>	Select element defining the surface or the cut-off plane

The drop-down list of this option contains a combination of filters for selection of various 3D objects that are capable to define the cut-off surface. The selected object is added to the list on the **Cutoff** tab.

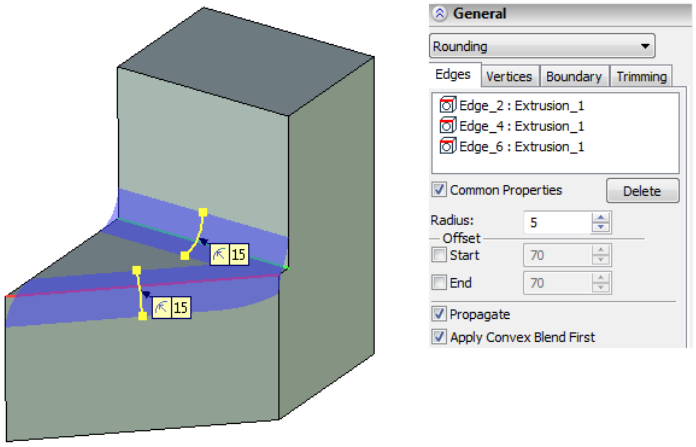




Additional flag called **Reverse direction** allows us to change the location of the blending being created with respect to the specified bounding plane/surface.

Defining the order of creation of blending when the blend surfaces overlap on several edges

The order in which the overlapping blend surfaces are created can be controlled with the help of the **At first concave** flag on the "Edges" tab of the command's properties window.



Diagnostics Mode

The special diagnostics mode in the edge blending operation allows a user to determine the possibility and causes of the error appearance while creating the blend.

> The absence of errors in the diagnostics mode is the necessary but not the sufficient condition for successful blend creation. The absence of errors in the diagnostics mode does not guarantee the successful completion of the operation creation.

The following option of the automenu is used for entry into the diagnostics mode:

	<D>	Diagnostics

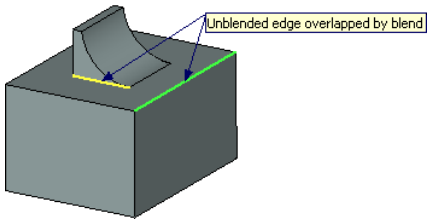
This option is available for selection right after selecting the objects for blending.

> In the preview mode and in the mode of viewing the changes made by the operation the option for starting the diagnostics mode is also available if, in these modes, the problems arise upon the model regeneration.

When entering into the diagnostics mode, the system automatically analyzes the specified objects and the blend parameters. The objects, the blending of which may cause problems, are marked in the list with a exclamation point – .

> In the diagnostics mode in the system's properties window all fields except the list of the blend objects are locked.

When selecting a marked object (edges, vertices, faces) from the list, the edges identified with the selected object are highlighted on the model in the 3D window. Also, a marker with the text describing the error appears. In some cases, the geometric element of the 3D model creating the problem is marked additionally. A user can select several objects at once to evaluate several problems simultaneously. To do it, make the selection with the help of <Shift>+, <Ctrl>+. Upon the entry into the diagnostics mode, all problematic objects are selected.



The blendable edges are shown with green color, whereas the geometric element of the model leading to a problem is shown with yellow color

If in the diagnostics mode several faulty objects are selected from the list, then selecting the marker with the help of without moving it at a significant distance will cancel the selection of the remaining objects.

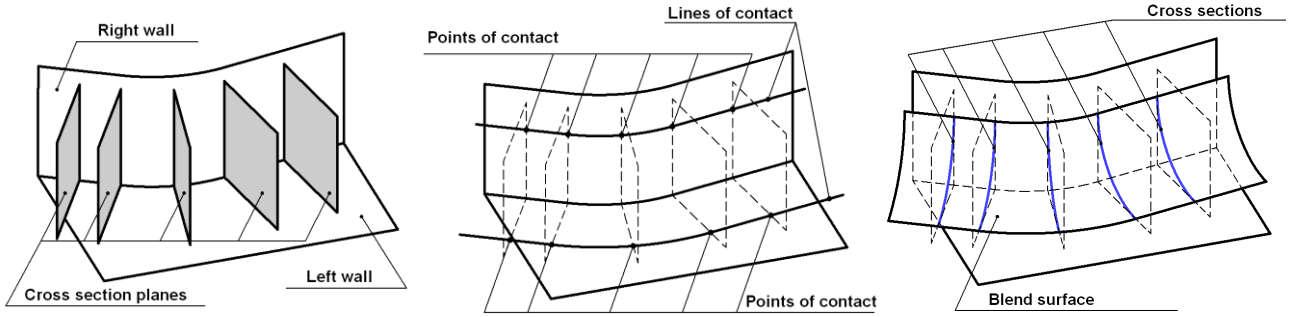
When exiting the diagnostics mode, the marking of the faulty elements with the special icons in the list is retained for convenient correction of the discovered problems.

BLENDING FACES

The face blending operation lets create a blend surface between one set of smoothly meeting faces to another. The sets of the blended faces are not required to have common edges (intersect). Blending can be done over solid bodies, as well as sheet bodies. The command has numerous options for controlling the shape of the blend surface, defining trimming conditions, bounds, etc. Do not consider this operation as an alternative to the edge blending operation. These two approaches have their advantages and limitations, well complementing each other.

MAIN CONCEPTS. OPERATION CAPABILITIES

The two sets of faces for blending are conventionally called the *right* and the *left wall*. In most general case, these can be arbitrarily positioned in space and have arbitrary geometry of the surface. The system constructs the blend surface between the right and the left wall in several steps.



Step 1 Step 2 Step 3

First, the set of planes is computed for holding the blend cross sections. The position of each plane depends on the existence and the shape of the auxiliary direction guide, as well as on the computational algorithm of a particular blend type. One can specify various blending effects by manipulating the orientation of the cross section planes.

Next, two points are defined on each plane that will be the bounds of the future blend surface. One point belongs to the right wall, the other – to the left. These will be referred to as the points of contact of the blend surface. The sets of the points of contact on the right and left walls make the two lines of contact. The position of the points in each cross section depends on the shape and mutual situation of the walls, as well as on the computational algorithm of the particular blending mode.

Finally, the system computes each cross section of the blend surface spanning the two points of computed in the previous step. The shape of the blend cross section depends on the positions of the points of contact and on the selected geometrical algorithm.

The final result may also be affected by a number of additional functions and boundary conditions.

Direction Guide

The direction guide defines positions of the blend cross-section planes. 3D paths can be used as guides. The direction guide is used for constructing most types of blends. Often, when all parameters appear to be defined yet the confirmation button is inaccessible, most probably the direction guide is needed.

If the guide is too short for constructing the whole blend surface, the system extends the guide by straight segments tangent to the 3D path at the ends. If the guide is longer than required, the system will attempt automatically define the necessary portion if the 3D path.

Sometimes, the operation can be created without use of a guide, as in the case of simple rolling ball blending.

Blend Types

There are two types of blends. Each type involves its own algorithm of constructing cross section planes for computing the blend surface.

Rolling ball blend

In this type of blending, the cross section planes are made orthogonal to the right and left walls. This blend type does not require a guide in the “Constant radius” and “Constant width” modes.

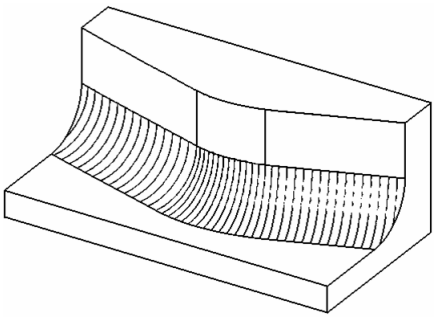
In the “Constant radius” mode, a notional ball of the specified radius defines the points of contact as it touches the right and left walls. This ball “rolls” all the way along the right and left wall and leaves two lines of contact. The cross section planes pass through the points of contact and the center of this notional ball at a given instant.

Rolling ball blend is the only type supporting the “Constant width” mode. In this case, the system automatically positions the cross section planes for best fit to the requirements of the specified blend mode. The guide is not required in this case either.

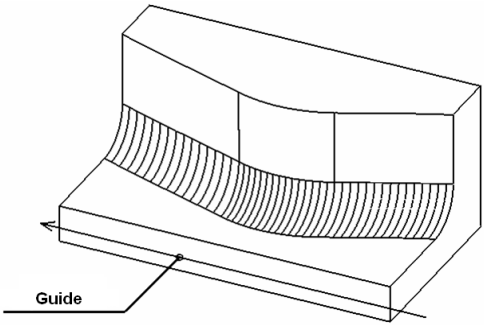
The “Variable radius” mode always requires using a guide. A current point on the guide is used to define the nearest points on the blended surfaces. Normals to the right and left wall surfaces are constructed at these points. The cross section plane passes through the current point on the guide. The plane normal is computed as a cross product of the two surface normals from the above. In the special case of flat wall surfaces, true to say that the cross section plane passes through the current point on the guide and is oriented orthogonal to either wall.

Disc blend

A plane is constructed orthogonal to the guide through each point on the guide. A current blend surface section is computed in each such plane.



Rolling ball blend



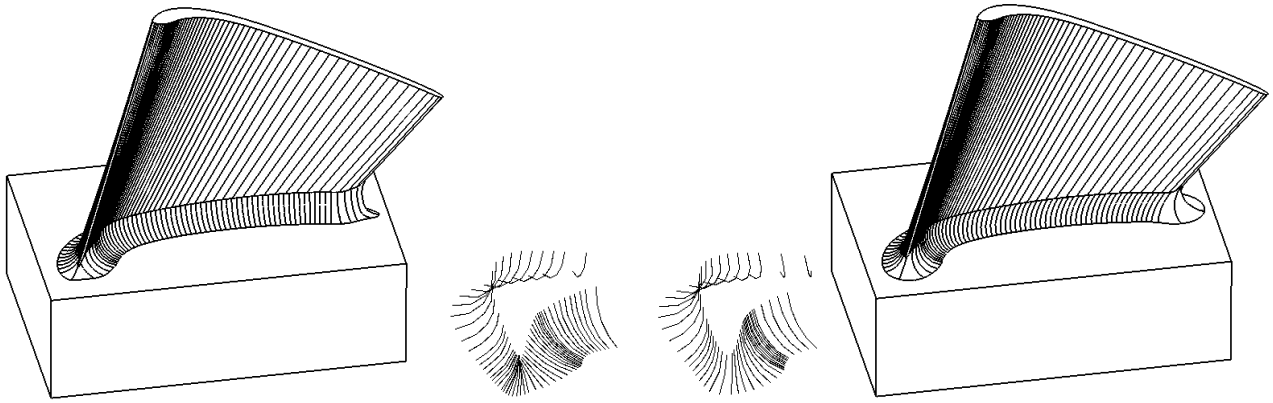
Disc blend

Isoparametric blend

This is an advanced type of blend. Isoparametric blend is a special type provided for turbine design. It is used to make a blend between a turbine blade and a base. The isoparametric blend is best for this kind of parts.

The isoparametric blending is similar to the disc type. However, the blend cross-section planes are oriented along the isoparameter lines of the blade surface in this case. Because of this, the blade surface ought to be necessarily the first set of faces (the left wall).

The direction guide must lie on the left wall (the one that defines the isolines). A dependency is established between the points on the guide and the blade surface. Then the specified cross-section blend surface is constructed tangent to the current isoline.



Rolling ball blend

Isoparametric blend

The diagram on the left-hand side shows a rolling-ball blend with self-intersection zones. The right-hand side diagram demonstrates an isoparametric blend with no flaws.

Blend Modes

Blend modes define the ways of computing the blend surface bounds in the current cross section. Only the bounding points (points of contact) are computed in the cross sections of the blend surface. What will be constructed in between the bounding points depends on the chosen shape of the blend surface.

Constant radius

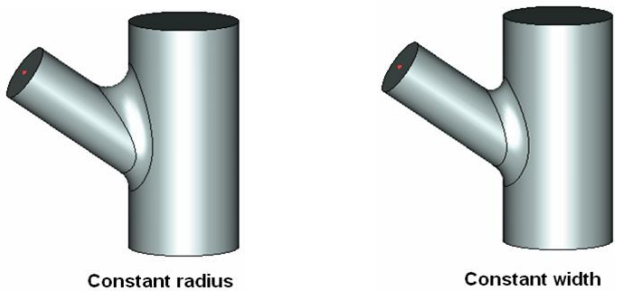
In this mode, the blend surface is constructed of the constant shape across the extent of the blend. The term "radius" in the mode name is just a convention. In fact, variations of this type include, besides the conic-shape roundings, chamfers and surfaces of smooth curvature (see below "Shapes of blend surfaces"). This depends on the specified blending shape. With the conic shape of the cross section selected, a surface is constructed with each cross section being a circular arc of a specified radius.

Variable radius

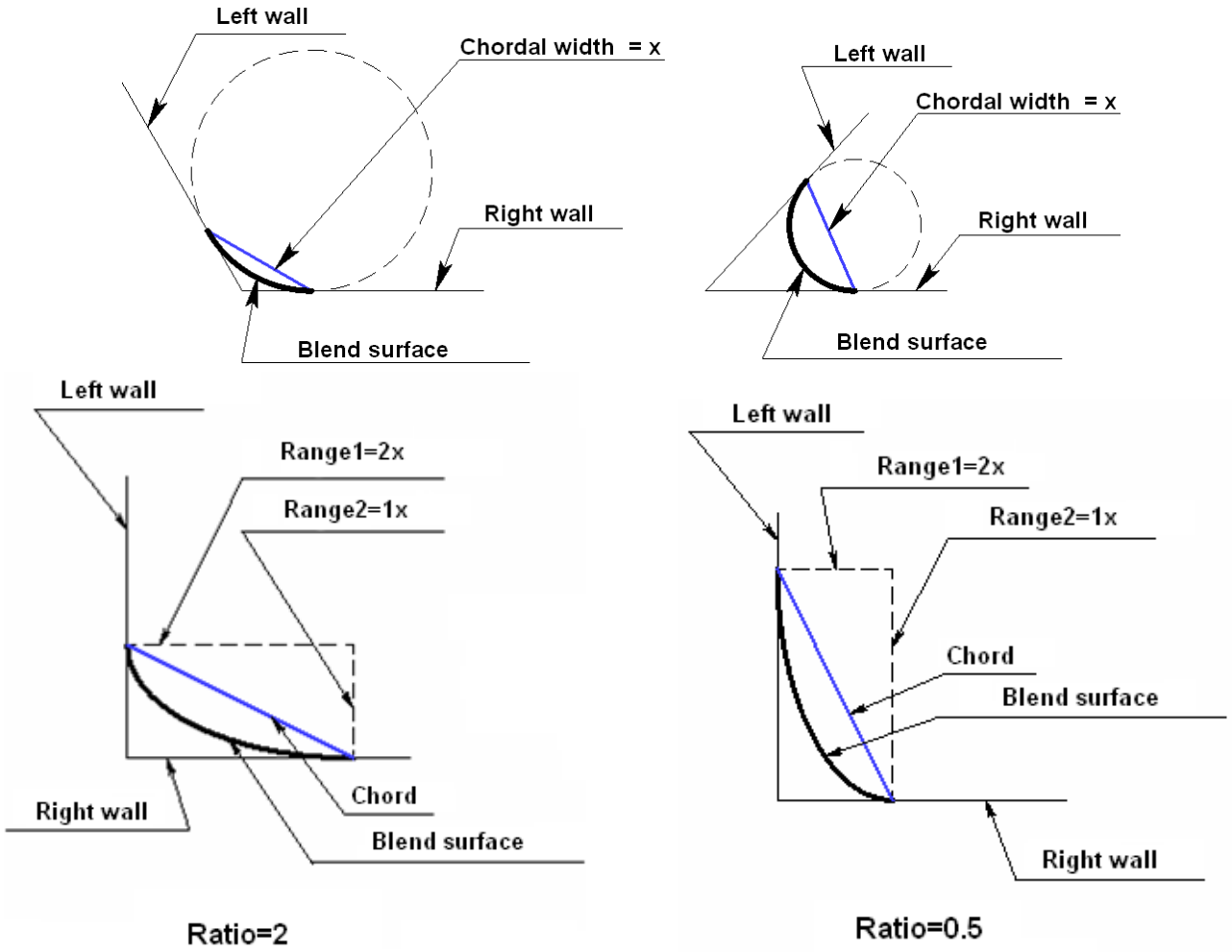
This blending mode supports blend surfaces with the shape parameters varying across the extent of the blend. All cross section parameters are controlled via special auxiliary points distributed along the blend surface. The points of contact are computed in the variable-radius mode based on two ranges defined for each such auxiliary point. The same approach is used for computing the points of contact by the ranges, as in the command "Blend Edge". The same chapter (see the topic "Variable-radius elliptical rounding") describes and illustrates the approach in details. The following is different in variable-radius blending by the "Blend Face-Face" operation from the variable-radius elliptical rounding by "Blend Edge". First, it is the way of constructing the cross section planes. Second, it is availability of other cross section shapes (besides the conic one).

Constant width

In this blending mode, the blend cross sections are constructed based on the constant chordal width condition. These are the chords connecting the points of contact in cross sections. This type of blending is especially desirable when the angle between the blended faces varies across the extent of the blend. In this case, one may want to specify approximate amount of cross section additional material to be maintained along the blend.



"Constant width" mode only works with "rolling ball" type blend.



[In this blending mode, the parameters of the shape are the chordal width and the ratio of the ranges for the bounding points of the blend surface. A ratio of ranges other than 1 makes a blend with a constant chordal width and asymmetrical cross section.

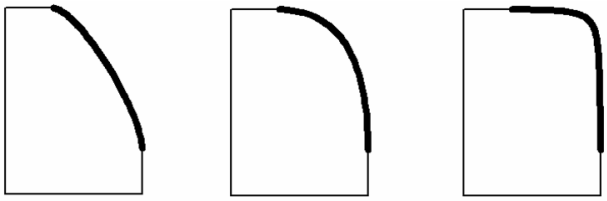
In constant width mode, the system automatically determines the shape of a conic cross section.

Shapes of Blend Surfaces

Three different shapes of blend surfaces are supported.

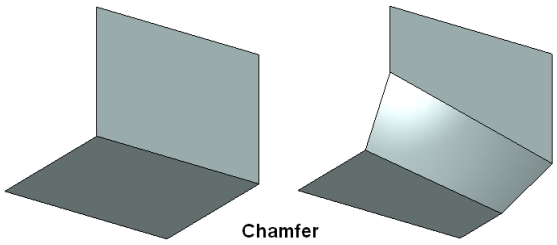
Conic

Conic is one of the shapes found in sections of a cone by a plane. These are a circle, an ellipse, a parabola and a hyperbola. In the constant radius mode, if "conic" shape is selected then the blend cross sections are arcs of the specified radius. In the variable-radius mode, the shape of a cross section is dependent on the "skew" parameter representing the rho value of a conic. Different values of this parameter yield an ellipse, a parabola and a hyperbola.



Chamfer

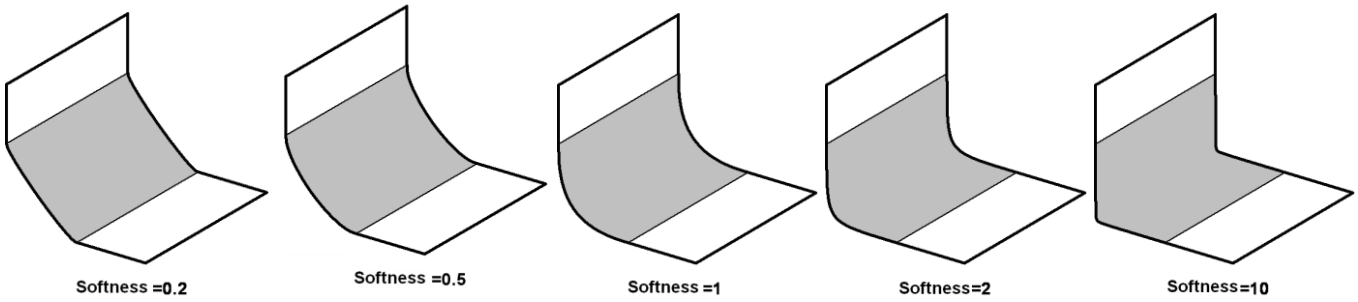
The chamfer-type blend cross section is defined as follows. The points of contact, computed in each cross section, are connected by a straight line. Chamfer is supported for all blend types and blending modes.



Continuous curvature shape

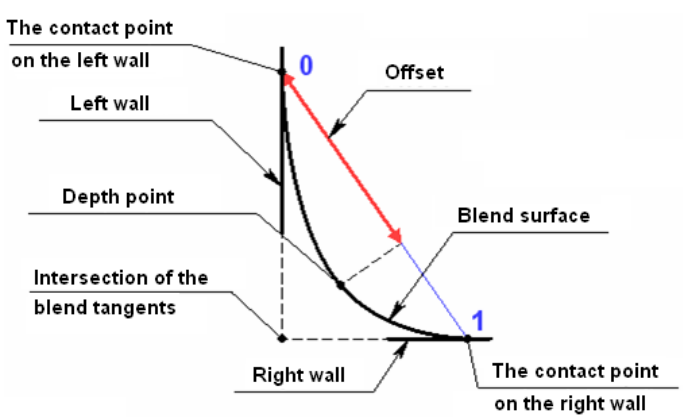
This way of blending provides continuity of the surface curvature where the blend surface meets the walls. In other words, the blend surface has the same curvature at the points of contact as the respective wall.

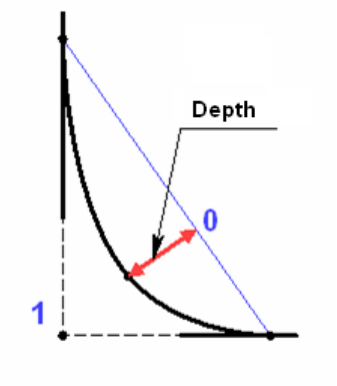
In **"Constant radius"** and **"Constant width"** blending modes the shape of the surface depends on a single "Softness" parameter controlling the smoothness of the transition from one wall to the other.



The value of the softness greater than zero is allowed, with the maximum unlimited. With the softness values approaching 0, the blend cross section becomes closer to a chamfer. However, if a true chamfer is required then the "Chamfer" setting for the blend shape is recommended instead.

The **Variable radius** mode provides two parameters for controlling the blends with continuous-curvature cross sections – "Offset" and "Depth".



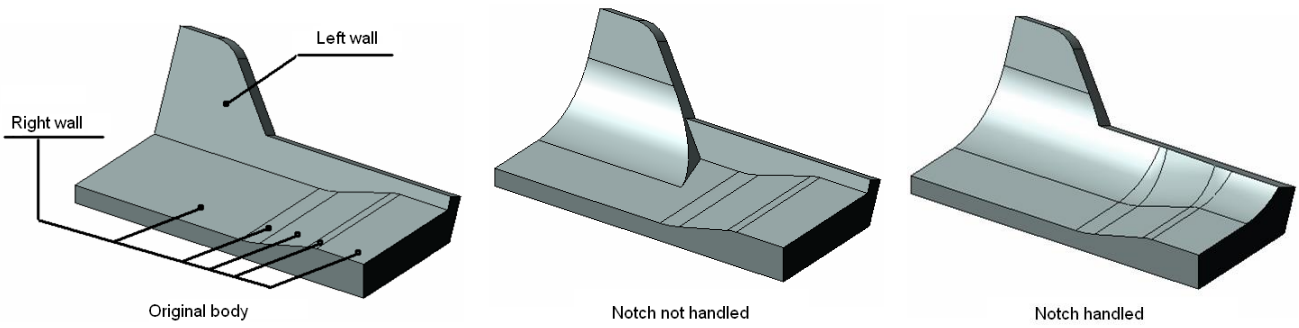


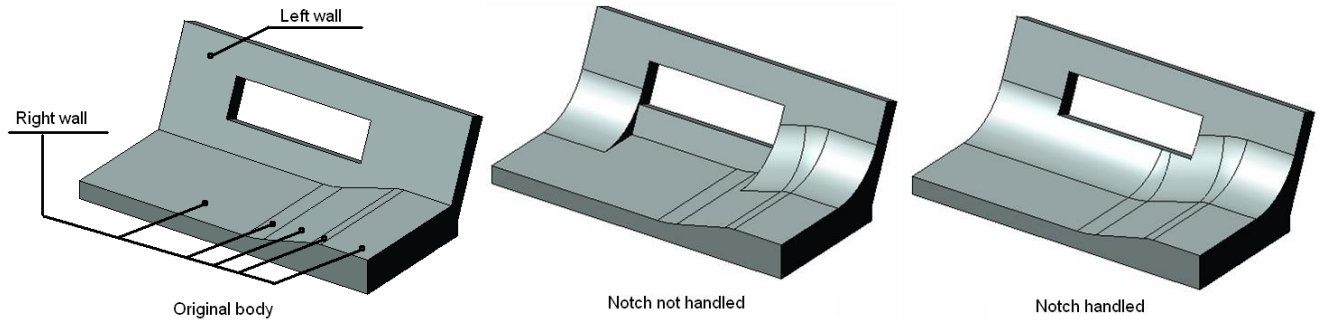
These two parameters together define the deepest point on the current blend cross section – the "Depth point". The "Offset" defines the position of the "depth point" between the two contact points. The parameter varies from 0 to 1, exclusive. The depth point approaches the left wall, as the offset approaches zero and the right wall with the offset approaching one. The "Depth" defines the position of the "depth point" between the chord connecting the two points of contact, and the intersection of the wall tangents. The parameter varies between 0 and 1, exclusive. With values near 0, the "depth point" approaches the chord between the points of contact, while with values approaching 1 – the intersection of the right and left wall tangencies.

Special Capabilities

Notch

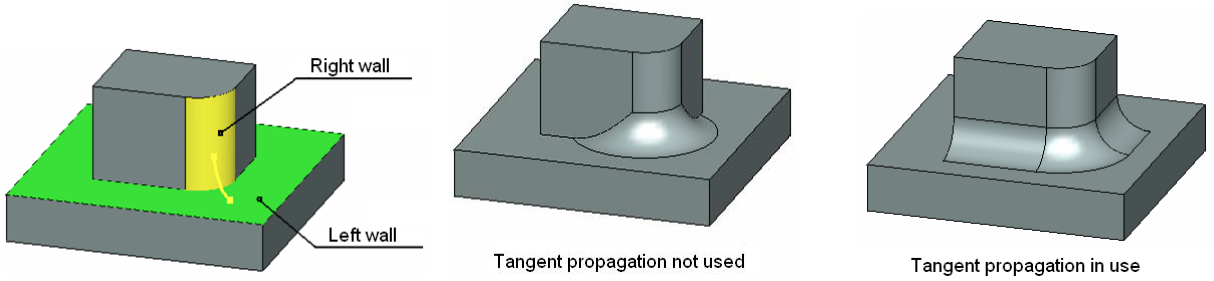
Sometimes, the walls to be blended may include topology insertions (Notches) that cross the line of contact of the blend surface. A special option "Notch" helps identify such insertions and let blending extend in the affected area trimming the blend surface with the faces adjacent to the wall. Without this option, the blending would stop before the affected area, where the line of contact is crossed.





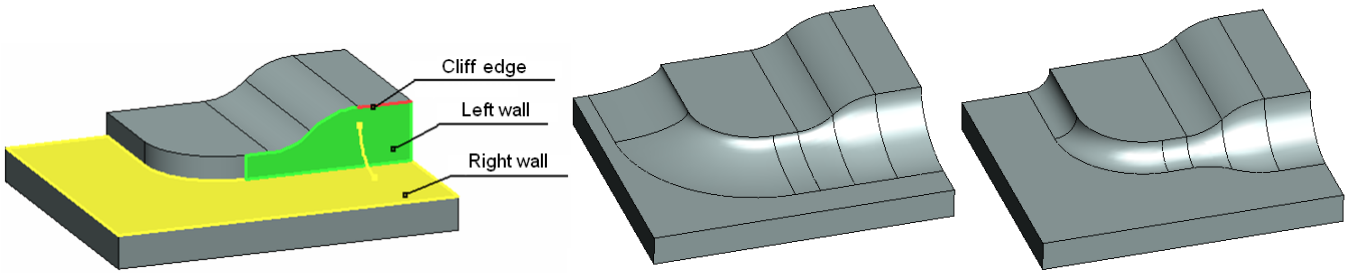
Tangent propagation

This blend option allows extending the blend surface on the adjacent faces smoothly connected to the selected walls. This simplifies the operation definition by letting select just one face per a wall. The system will attempt propagating the blend surface on all adjacent faces as far as possible.



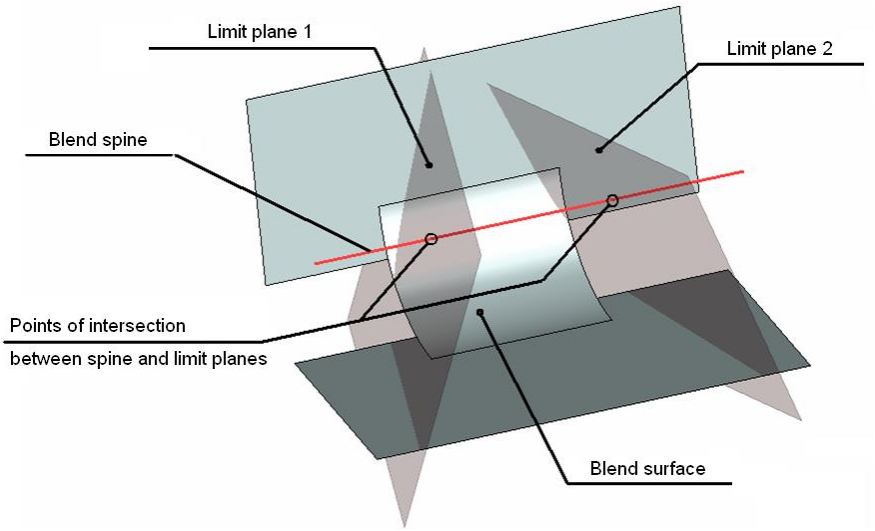
Propagate option can't be used for disc and isoparametric blend types.

Propagate option is applicable when defining boundary conditions (see details below). This allows specifying just one edge for defining boundary conditions if all bounding edges connect in a smooth chain.



Trimming with planes

Flat faces and workplanes can be used for limiting the blended area. One limit plane divides the blend surface into two parts leaving just one part in the scene. Total of two limit planes can be defined. The two planes will surround from the two sides the central portion of the.



It is important to keep in mind that by dividing the blend surface the limit plane is not trimming it. The blend surface is divided along a cross section plane at the point of intersection between the limit plane and the blend spine.

Blend end trimming

When creating a face-face blend, there is a choice of trimming options for the end boundaries of the blend surface. The geometry of the walls themselves is not disturbed by trimming the blend surface. Total of four trimming options are available:

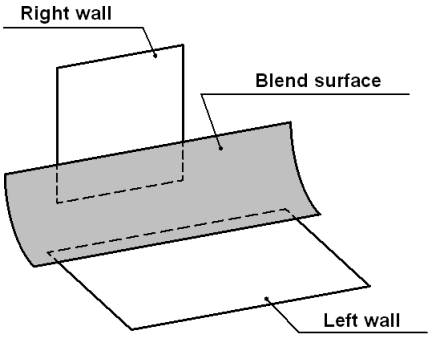
No trimming. When trimming is off, the blend surface is extended beyond the ends of either wall.

Short trimming. The blend surface is trimmed by the end boundaries of the shorter wall.

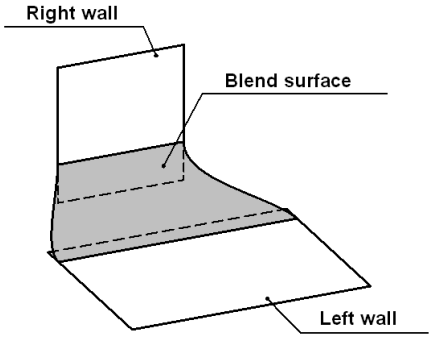
Long trimming. The blend surface is trimmed by the end boundaries of the longer wall.

Faces to wall trimming. The blend surface is trimmed by the end boundaries of both walls. Meanwhile, the side edge of the blend surface is constructed such as to provide tangency with both side edges of the trimming walls.

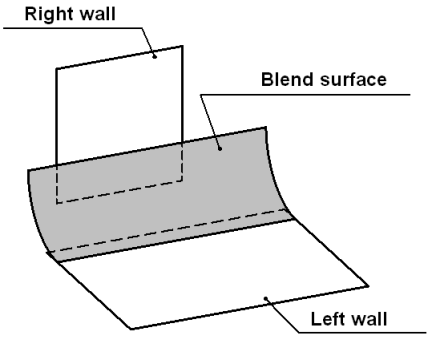
First three options can be used for blending sheet bodies only.



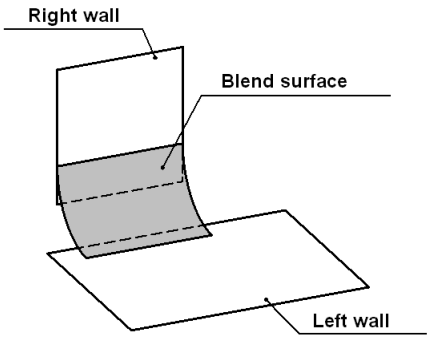
No trimming



Faces to wall trimming



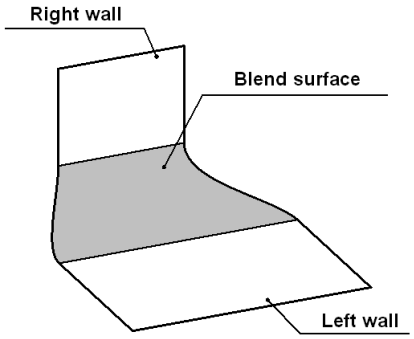
Long trimming



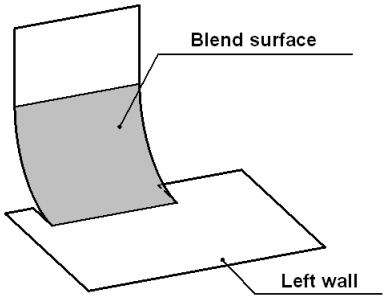
Short trimming

Wall trimming

In addition to blend surface trimming, the walls can be trimmed as well along the line of contact.



Faces to wall and both wall trimming



Short trimming and both wall trimming

Selecting the resulting topology

The face-face blend operation can be stored in the model structure in several topological variations. The choice of a topological solution is combined with the wall-trimming interface (the previous topic). Total of four solutions are available:

“Separate body” means neither wall will be trimmed and the blend surface will stay as a standalone sheet body. The objects selected for the walls to blend will remain in the model structure in the original state. The blend operation will be represented by a separate sheet body – the blend surface.

“Trimmed on both walls” trims both walls along the line of contact without attaching the blend to any wall. This solution has a practical meaning only for sheet bodies.

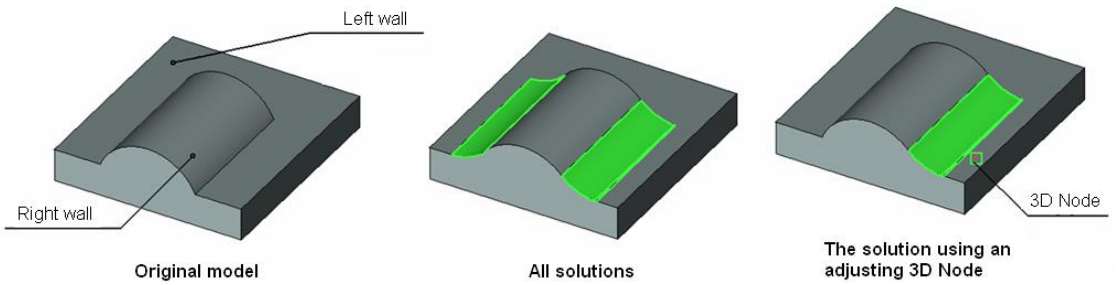
The model structure will reflect the face-face blend operation alone at the top level. The operations whose faces were selected as walls will be written in the model history one level deeper. Meanwhile, the blend operation will be represented by three sheet bodies. This means that if, for instance, the operation **“3SD: Divide Solid”** is applied to the blend then three separate sheet bodies will be created, the two trimmed walls and one blend surface.

"Attach to the left wall" trims both walls and attaches the blend surface to the walls. If blending was done between two different sheet bodies then these bodies are united into one by the blend. The operation used as the left wall will further be used as the reference in the model structure.

"Create body" trims both walls, adds the blend to the walls and creates a solid body if possible. This solution can only be used together with the "Faces to wall" trimming option. This option is similar to the previous. The specific of this option is the system attempting to create a solid body. This is done if a check is satisfied for the resulting geometry to form a closed volume.

Solution using an adjusting point

In some cases, several blend surfaces can be constructed for the same set of walls. One can accept them all. However, a particular solution can be specified. The system accepts the solution nearest to a selected node.



Processing curved zones (Inside tight)

The algorithm's for processing curved zones allow creating blend's in the areas where the value of the curvature radius of the sidewalls is less than the blend radius. This option is available only for rolling ball blending by a conical section of a constant radius. This functionality is described in more details in the chapter "Edge blend".

Boundary Conditions

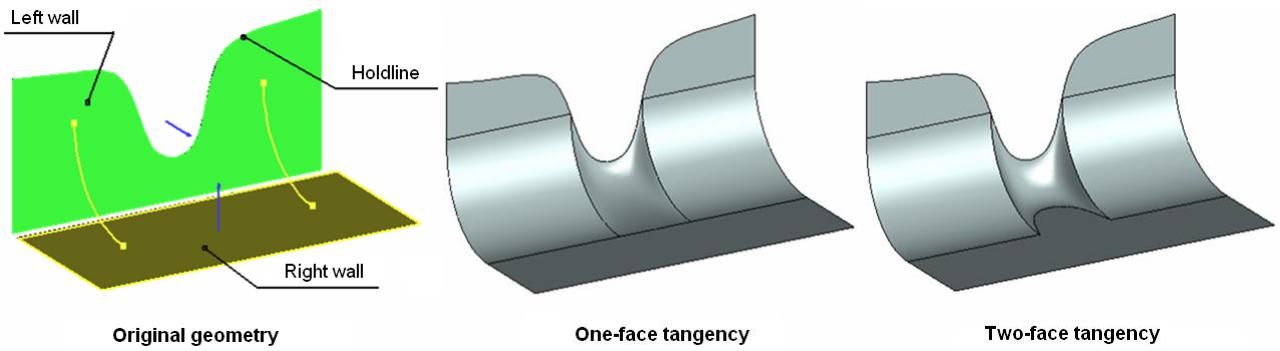
Tangency to faces

In the cases when a blend surface overflows beyond the blended faces (of the right or left wall) then the system can adjust the blend surface to keep edges overflown by the blend surface. This results in shifting the line of contact of the blend surface in the area of the edge overflow to actually make it coincide with the edge. The blend surface adjustment is done while maintaining the tangency condition at the wall surface in the points of contact.

If the blend surface can't be fit within the extent of the walls, two types of constraints can be specified - "One-face tangent" or "Two-face tangent".

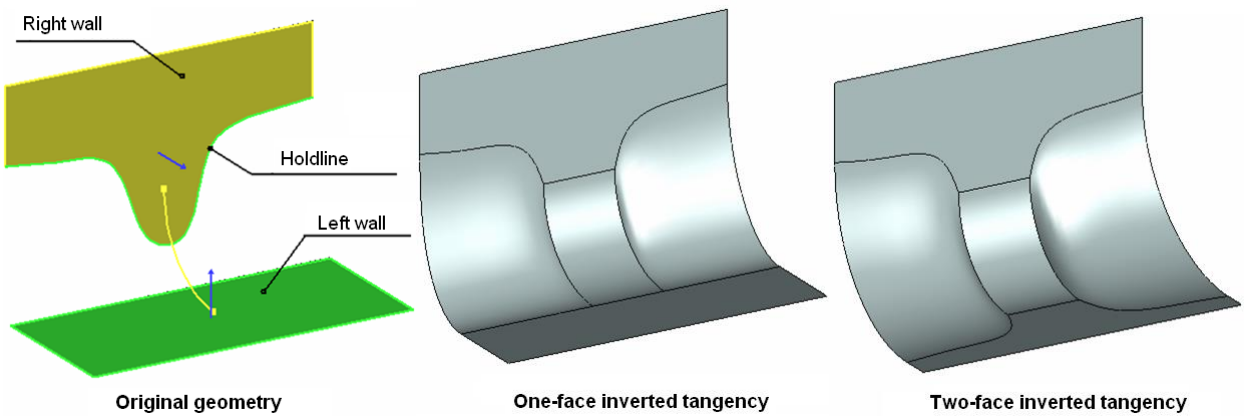
The boundary condition is defined by selecting an edge or a set of edges belonging to one or another wall, called "holdline".

If one face tangency is elected then the opposing line of contact of the blend surface is not disturbed. It will satisfy the defining rule of the shape of the surface (as, for instance, the radius). If two-face tangency is elected then the opposing line is distorted symmetrically.



Inverted tangency to faces

The boundary condition is defined by an edge or a set of edges belonging to one of the walls (the holdline). With too small radius values, when the blend surface can't be constructed, the system shifts the line of contact to the holdline. The blend surface is tangent to the faces of the wall along the line of contact. If the back tangency to one face is elected, then the opposing line of contact of the blend is not disturbed. It will satisfy the defining rule of the surface shape (as, for instance, the radius). If inverted tangency to two walls is elected then the opposing boundary is distorted symmetrically. Thus, the blend surface is no longer defined by the radius. Rather, it is deformed by the boundary conditions wherever imposed.

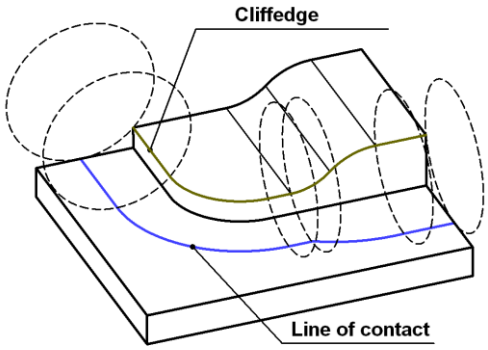


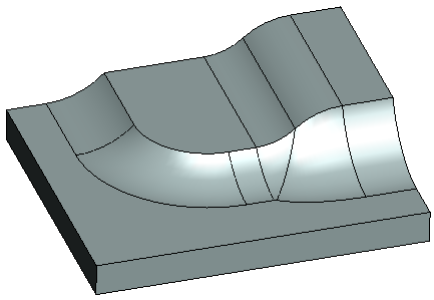
Cliffedge

This is the "sharp edge" overflow, when tangency is not maintained at the bounding edge.

The edge overflown by the blend surface is left unchanged. The blend surface is constructed without distortions in such a way that the cross section spans between the wall and the line of contact of one side and the cliffedge on the other side, without altering the guiding rule of the cross section shape.

To understand this, consider a simple example of a spherical blend with a constant radius. Imagine a ball of the given radius rolling on one wall, keeping contact with the sharp edge on the other wall.





With this constraint, tangency is not provided for the blend surface along the cliffedge, however, the shape of the sections is not distorted and is constructed by the specified rule across the blend surface.

Operation Defining Steps

The operation creating command can be called via one of the following ways:

Icon	Ribbon
	3D Model → Create → Face-Face
Keyboard	Textual Menu
<3DF>	Operation > Blend > Face-Face

The property window and the automenu are used for handling the command and defining necessary components. These work simultaneously and enhance each other. The state of the automenu depends on the operation defining stage and on the purpose and type of the objects being selected. The object selection modes can be manipulated via either the automenu or the property window.

To create the operation, follow the following steps:

- ✓ Select first set of faces (the left wall)
- ✓ Select second set of faces (the right wall)
- ✓ Specify blend type (if necessary)
- ✓ Select the spine (if necessary, depending on the blend type)
- ✓ Specify blending mode (if necessary)
- ✓ Specify the shape of the blend cross section (if necessary)
- ✓ Specify geometrical parameters of the blend cross section
- ✓ Specify boundary conditions and overflow processing (optional)
- ✓ Confirm operation creation

Using Draggers and Decorators

As the face-face blend operation is being defined, the parameters can be specified and various objects selected in the 3D window with the help of auxiliary elements. These elements help quickly review and

modify the configuration of the operation being created. Their quantity, shape and size depend on the defined properties of the operation, on the purpose of the selected objects and on the type and values of the parameters controlled thereby. The elements that control parameters dynamically are called “Draggers”. The elements that merely display some or other operation properties are called “Decorators”.

Draggers are displayed in various colors. By default, the active dragger (the one driven by the mouse at the moment) is drawn red, all the rest - yellow. Decorators are drawn blue by default.

Draggers are used for quick reviewing of the result and correcting it, therefore, the shape follows the actual blend surface with limited accuracy. Dragger does not make an account of all details of the cross section in the case of the surfaces with continuous curvature and is not responding to such settings as “Convexity”, “Softness”, etc.

> Using draggers with numerous active objects on a low-performance computer may cause delays in the system response. To speed up workflow in this situation, the draggers may be turned off. The off switch for the draggers is located in the **Options** section of the property window.

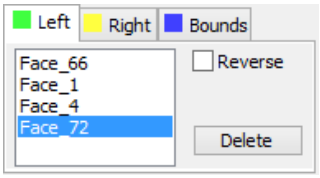
The following topics review all types of manipulators and decorators for each flavor of face-face blend.

Selecting a Set of Faces

Normally, the operation is defined beginning with selecting the sets of faces between which the blend will be constructed. The left and right-wall sets are formed when one of the following automenu options is active:

	<L>	Select left-wall Face
	<R>	Select right-wall Face

Switching between the options is done by pressing an icon button in the automenu.

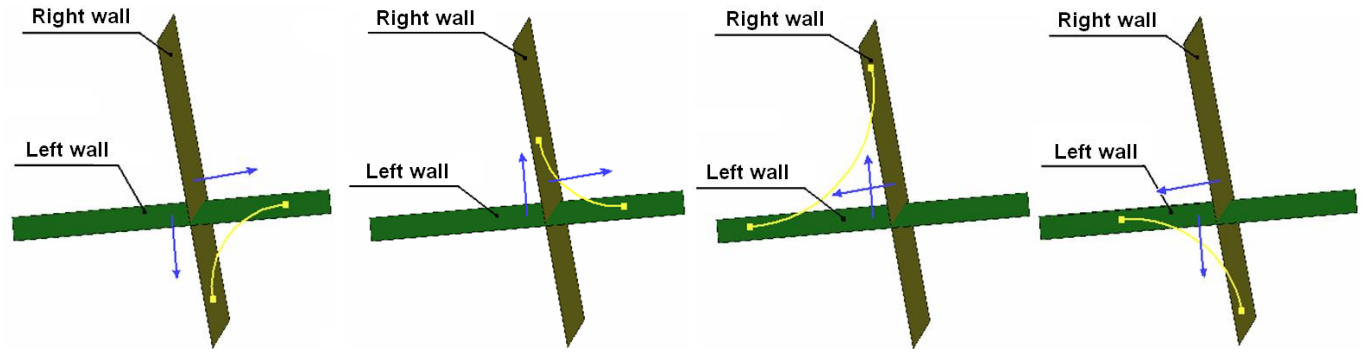


Turning on an option automatically activates the respective tab on the property window. The list of selected faces of the right or left wall is displayed in the provided pane under the tab tag. Switching between the tabs of the property window also activates the appropriate options for face selection.

Face selection is done using the mouse in the 3D view window.

The selected faces of the left wall are highlighted in green by default, the right ones - yellow. Settings of the highlighting colors are done in the system customization dialog box. The current face highlighting color is reflected in a color box on the tab tag for each set.

A decorator is drawn in the middle of each face, shaped as an arrow, marking the side of the face adjacent to the blend surface. In a general case, two walls make four quadrants, each being suitable for constructing a blend surface. The blend is created in the quadrant pointed at by both arrows.



For solid bodies, the direction of an arrow is determined automatically. For sheet bodies, the direction of faces has to be controlled manually by the user. For flipping the system default direction, set or clear the "Reverse" flag.

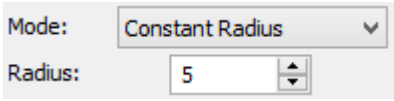
To exclude a face from a wall set, simply select it in the list in the property window on the respective tab and press **[Delete]** button.

Blending Mode Selection

A blending mode can be defined (modified) at any step into the operation setup. The name of the mode is selected in the pull-down menu in the property window. Three modes are available: "Constant radius", "Variable radius" and "Constant width".

"Constant radius" mode settings

In the "Constant radius" mode, the user needs to additionally specify the "radius" of the blend surface. The "radius" input box appears in the property window right after selecting this mode. By default, the system sets the radius equal to 10 units.

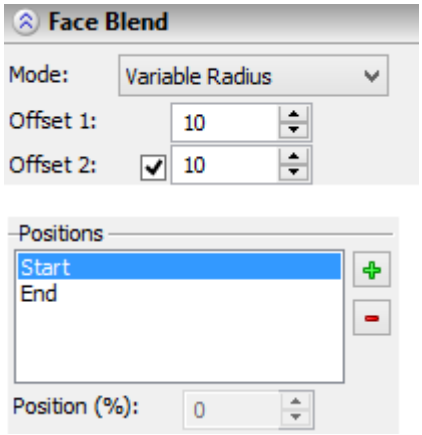


This field is synchronized with the draggers. Depending on the selection of the blend surface shape, the dragger is drawn as an arc or as a line segment (only for chamfer creation). The number of manipulators depends on the number of faces in the right and left wall sets. All manipulators update simultaneously and relate to the same parameter, the "radius".

"Variable radius" mode settings

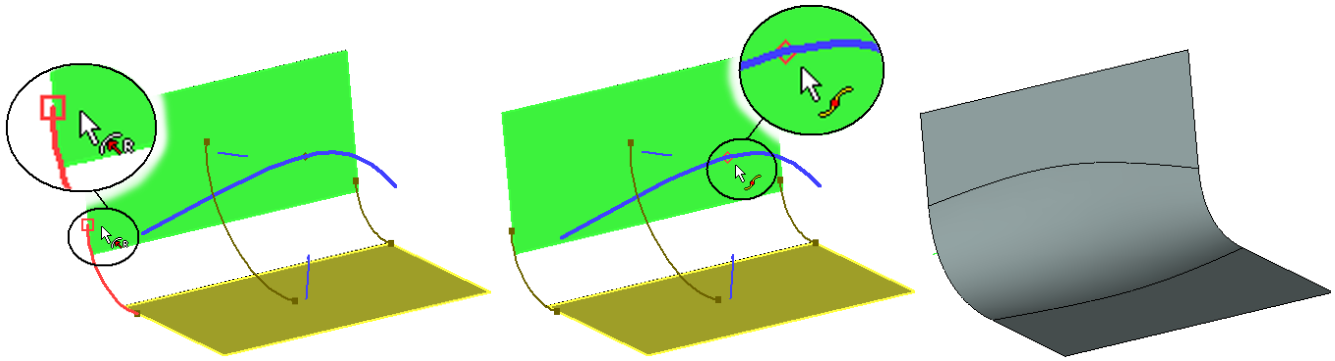
The "Variable radius" mode brings more additional controls in different parts of the property window.

At the bottom of the property window there are standard controls for manipulating intermediate points for variable-radius blend. The special item "Positions" manipulates the list of auxiliary points. The points **Start** and **End** are always included. Other points are named as they are created. Adding and deleting points is done using the buttons **[Add]** and **[Delete]**. A point position is specified in a special input box by percentage of the total length of the blend surface.



The "Variable radius" mode requires setting two offsets for each auxiliary point. Initially, the system sets the second offset equal to the first one, as indicated by the checked box before the second offset in the property window. With this setting, the two offsets change simultaneously. If the setting is cleared, an individual value can be defined for the second offset.

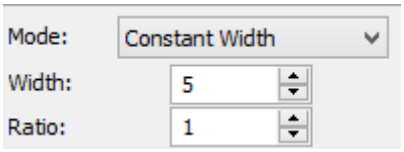
Positions of the auxiliary points and their offset values can be defined by draggers (see the diagram below).



"Constant width" mode settings

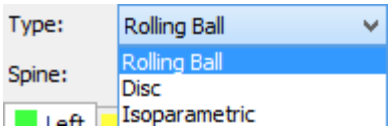
The "Constant width" mode requires two numerical parameters – "Width" and "Ratio". By default, the system presets the width of 10 units and the ratio equal to (-1).

These fields are synchronized with the respective dragger. Depending on the blend shape selection, the dragger is drawn as an arc or as a line segment (for chamfers only). The number of draggers depends on the number of faces in the right and left walls.



Blend Type Selection

The type of the blend operation can be selected from the pull-down list in the property window. One can select any of the three choices: **Rolling Ball**, **Disc**,**Isoparametric**. The system sets the rolling ball type by default.

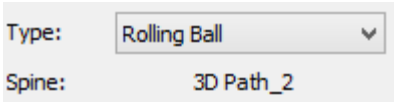


Spine selection

To turn on the spine selection mode, activate the following automenu option,

	<S>	Select 3D path as guide curve

Selection of a 3D path is done in the 3D window or in the model tree by the mouse. The name of the selected 3D path is displayed in the property window.



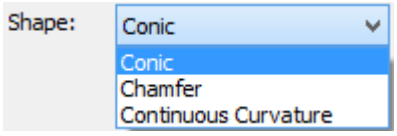
Only one 3D path can be used as a guide. If, say, by a mistake, a wrong path was selected, another one can be selected right away. Selection of a new 3D path cancels the previous, and the last selected path will be used as a guide.

To reject a 3D path selection, use the automenu option:

	<W>	Cancel Selection of 3D Path as a guide curve

Defining the Shape of Cross Section

The shape of the cross section can be selected from the pull-down list in the property window. The list contains three choices: **Conic**, **Chamfer** and **Continuous curvature**. By default the conic cross section shape is defined.

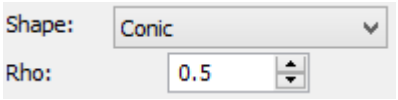


Depending on the blending **mode** and the selected **shape** of the cross section,
additional controls may appear or hide in the property window for controlling the cross section shape and its geometrical parameters.

Conic cross section settings

When using the "Constant radius" blending mode, no additional settings are requires. In this mode, defining a conic shape makes the blend cross sections become arcs.

An additional **Rho** parameter is defined in the "Variable radius" mode for conic cross sections. The parameter value is defined separately for each characteristic point of the variable blend. By default, the value of 0.5 is assigned



to this parameter. In this case, the cross section is a parabola. With smaller values the cross section becomes an ellipse, with greater ones – a hyperbola. This parameter may not be 0 or less and 1 or more.

In the "Constant width" mode the shape of the conic section depends on the definitions of the mode itself, in particular, on the "Offset" parameter. If the offset ratio is defined equal to 1, then the blend cross section will become a circular arc, otherwise it will be a parabola.

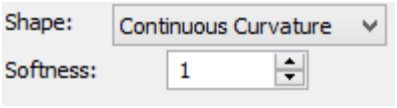
Chamfer settings

No additional settings are required for this shape of the cross section in any blending mode. The points of contact, computed for each cross section according to the chosen mode, are connected by a straight line.

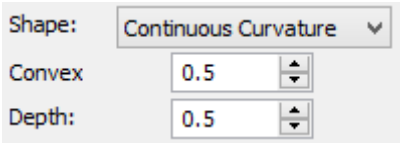
Continuous-curvature cross section settings

The continuous-curvature cross section can only be realized under the rolling-ball blend type. Therefore, selecting this shape for the cross section automatically switches the system into the rolling-ball blend type.

In the "Constant radius" and "Constant width" modes, the shape of the cross section is controlled by the single parameter **Softness**. The softness value must always be greater than zero The system default for this parameter is 1.



In the "Variable radius" mode, the shape of the continuous-curvature cross section depends on two parameters – **Offset** and **Depth**. Both parameters vary in the range from 0 to 1, exclusive. The default value is 0.5. The values of the parameters are defined separately for each characteristic point of the variable blend.

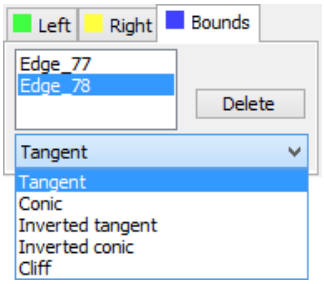


Defining Boundary Conditions

All types of boundary conditions are defined along the specially selected edges. To get the edge selection mode, engage the following automenu icon:

	<C>	Select edge to specify limitation

Turning on this option automatically activates the **Constraint** tab of the property window. The provided pane under the tag displays the list of selected edges. Switching to this tab of the property window from any other mode activates the edge selection mode as well.



The type of boundary conditions can be selected from the pull-down list. Five choices are available: **tangent**, **conic**, **inverted tangent**, **inverted conic** and **cliff**. The default offered by the system is "tangent".

Each constraining edge requires individual setting of the boundary condition type, in a certain order. First, you need to select a constraining edge in the complete list of edges, and then define (if required) the boundary condition type using the pull-down list.

To remove an edge from the list of constraining edges, press **[Delete]** button in the property window on the "Constraint" tab.

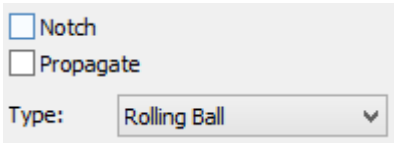
> One should keep in mind that constraints can work only in connection with compatible geometrical parameters of the blend surface that are defined by the blending mode and cross section shape.
>
> Thus, if the cross section parameters prevent the blend surface from reaching a holdline then the tangency condition on one or both walls is impossible. Vise versa, if an "inverted tangent" constraint is imposed, then the appropriate "radius" of the blend surface is such that the blend is not reaching the holdline or the wall itself in the zone of the constraint validity.

It follows from the above that often draggers stop working on a blend surface constrained to certain boundary conditions. For example, when creating an "inverted tangent" blend surface that does not reach the walls initially, the draggers are not displayed. Vise versa, when creating a "tangent to walls" blend surface that does not fit amid the walls then the dragger reflects the largest arc fitting amid the walls.

Overflow Settings

Notch. Tangent propagation

Attributes are provided in the property window for turning on notch handling and tangent propagation. These parameters are accessible for use with all types of blend.



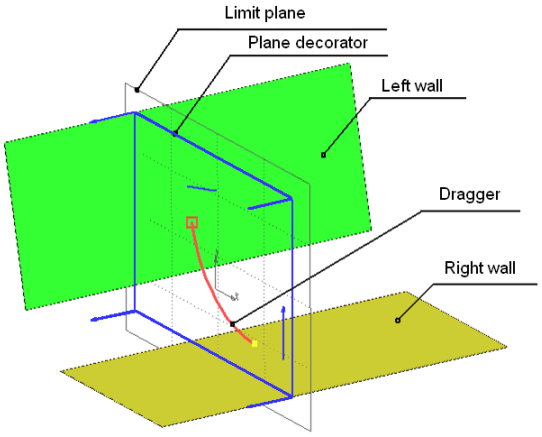
Limit plane setup

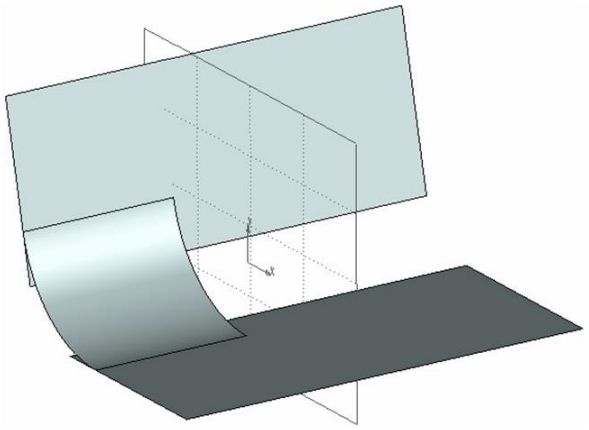
Engage one of the following automenu options in order to select the primary or secondary limit plane:

	<F>	Set first limit plane
	<D>	Set second limit plane

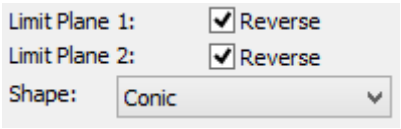
Upon activating the option, workplanes and flat faces will become selectable in the 3D window. Select either in the 3D window or on the model tree using the mouse.

When selecting a limit plane, its decorator is displayed in blue color. The decorator is a rectangle drawn on the limit plane. The size of the rectangle is computed according to the sizes of the right and left walls. Normal arrows are drawn in the corners of the rectangle, pointing toward the side of the blend surface to keep.





After selecting **first** or **second** limit plane, the property window displays the **"Reverse"** checkbox instead of the previous "Not selected" status, for flipping the "keep" side as indicated by the arrows.



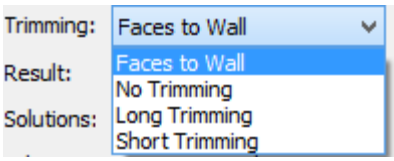
If only one limit plane is to be used then define only the **First** plane. Both the **First** and **Second** limit planes should be defined only when two limit planes are to be used simultaneously.

The following automenu options are provided to reject use of limit planes:

	<Z>	Cancel Selection of First Limit Plane
	<X>	Cancel Selection of Second Limit Plane

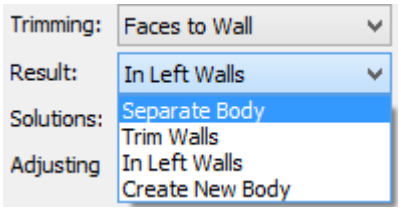
End trimming options

The end trimming options for the blend surface can be selected from the pull-down list in the property window. Four choices are available: **Faces to wall**, **No trimming**, **Long trimming**, **Short trimming**. By default, the "Faces to wall" option is set.



Defining topology of the resulting geometry

A choice of the resulting topology is selected from the pull-down list in the property window. Four options are available: **Separate body**, **Trim walls**, **In Left Walls** and **Create new body**. The system default is **In left walls**.

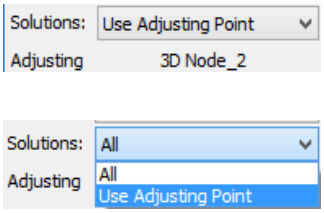


Adjusting solution by a node

If a blend operation results in several sets of blend surfaces, a 3D node can be selected to define the blend set to keep. Selection of the 3D node is done by activating the following automenu option:

	<E>	Select 3D Node, that defines only one nearest surface as result

Upon selection, the name of the node is entered in the blend operation property window. Next, the user needs to explicitly specify whether to use the solution by the 3D node, or use all obtained solutions. The type of the solution is selected in the pull-down list, containing just two items: **All** and **Use adjusting point**. This switch helps quickly alter the solution type without giving up the selection of the adjusting 3D node.



The following automenu option is provided to reject a 3D node selection:

	<N>	Cancel Selection of 3D Node that indicates the nearest result

CREATING A "WIRE-FACE" BLEND

To create a "wire-face" blend you must specify:

- set of faces of solid/sheet body as a left or right wall of the blend. The second wall is not defined;
- set of smoothly conjugate edges of the wire body (3D profile, 3D path) as a boundary condition;
- radius of the blend;
- option for the final result topology (only the "Separate body" or "In left wall").

All other parameters of the operation in this mode are not used.

As in the general case, for selecting faces it is necessary to either activate one of the tabs "Left/Right" in the properties window, or use one of the automenu options:

	<L>	Select left-wall face
	<R>	Select right-wall face

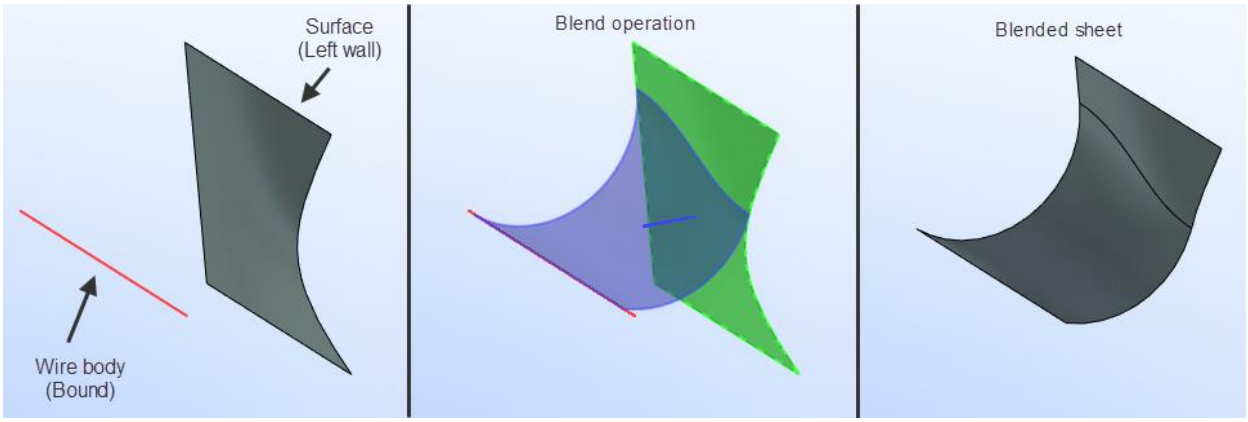
The set of edges is defined as a boundary condition. To select the edges, you must activate the "Bounds" tab in the properties window or automenu option:

	<C>	Select edge to specify limitation

Do not change the type of boundary condition.

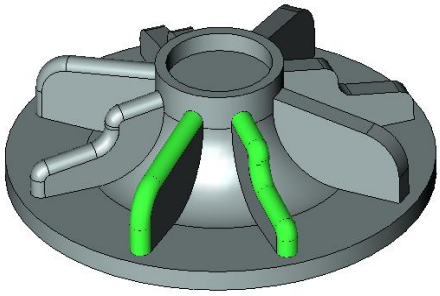
Option for the final result topology is determined by the "Result" drop-down list in the properties window. For this mode of blend, you can use only two options: "Separate body" and "In left wall."

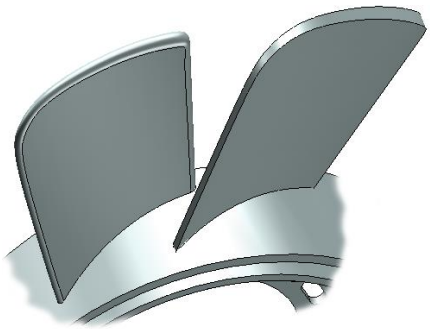
Option "In left wall" can only be set if the original set of faces belongs to a sheet body. In this case, the resulting body is attached to the original sheet body, regardless of which wall has been set in operation - the left or right.



THREE-FACE BLEND

The three-face blending operation creates a blend surface from one set of smoothly connected faces to another such set. The blend surface will be constructed tangent to yet another, third set. A typical situation addressed by this operation is creation of a smooth transitional surface between two opposite faces and along the narrow strip of the side surface of thin-wall parts. The sets of faces for blending are not required to share common edges (intersect).



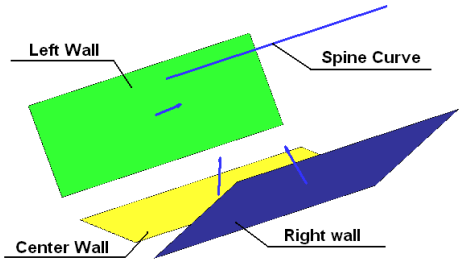


Blending can be performed over solid bodies as well as sheet bodies. This operation can be considered as a special case of the face-face blend, separated into a dedicated command. Since the three-face blend shares many concepts with the face-face blend, first refer to the "Blending faces" section for details on certain terms and concepts.

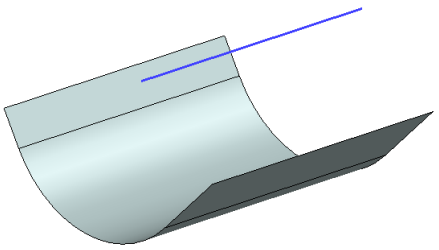
MAIN CONCEPTS. OPERATION CAPABILITIES

General Operation Concepts

Three sets of faces make the basis of the "Three-face blend" operation. These are called **"right"**, **"left"** and **"center"** walls. All faces within each set must meet smoothly. Sharp edges are not allowed. The walls themselves are not required to connect smoothly with each other. All sets of faces can belong either to the same solid body or to one, two or three sheet bodies. Three-face blends can't be created between walls belonging to different solid bodies or mixed sheet and solid bodies.

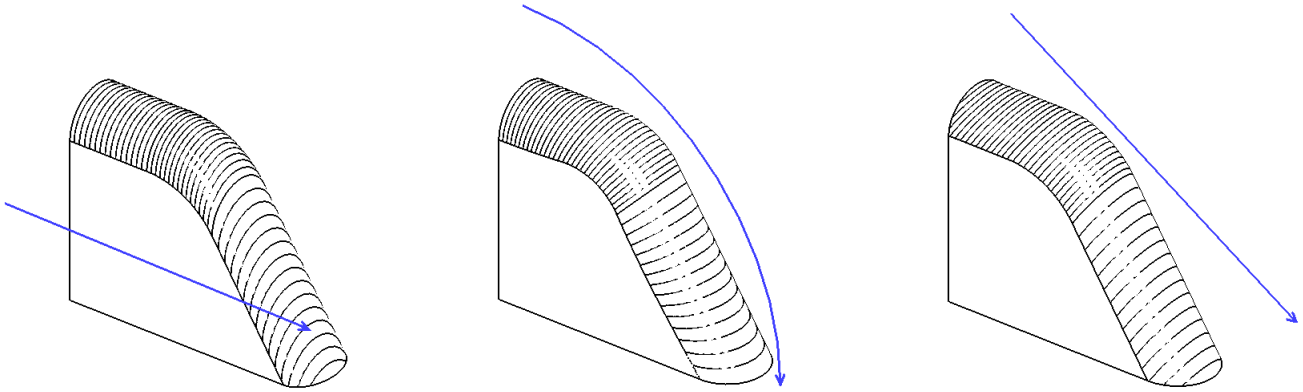


Original configuration



Resulting blend

Three-face blend is always disc-type. The cross section planes are the set of planes orthogonal to the guide spine and passing through the three contact points where the blend surface is tangent to the walls. The blend cross section shape is always represented by a circular arc. The cross-section plane intersects with the three walls. The arc in the cross section is constructed in such a way to be tangent to each wall.



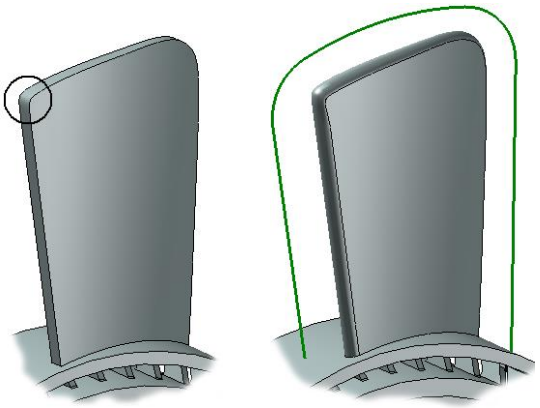
The figures show blend surface geometry dependency on the guide spine

The guide spine is always represented by a 3D path. The spine may not have sharp corners. If the spine is too short for defining all cross sections, it is extended by straight segments tangent at the ends.

> If the left and center walls are adjoining then their common edge (the set of edges) may be used as the guide spine. In this case, selection of the guide spine is optional.

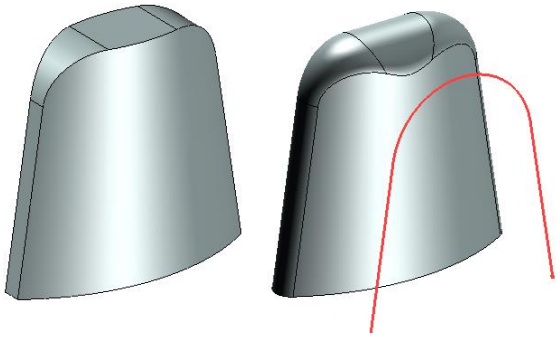
In real situations, the edge between the right and left walls often has complicated shape. In such cases, the blend surface may not be successfully created between three faces. Creating a new guide spine may help.

Make a spine of simpler than the edge geometry, just approximately following the edge shape. Let's review two typical examples:



Example 1. Required is blending of the side strip of a fan blade. In this case, the system may not use the edge between the left and center walls due to the following reason. The edge has a too small radius in the circled area. The blend cross sections would intersect each other if constructed normal to the edge. An additional spine (a 3D path) helps creating a blend in this case that approximately repeats the shape of the edge yet avoids the mentioned traps.

Example 2. In this example, the left and right walls are cone fragments. The part gets wider in the middle. At the first glance, the edge radius is not too small. However, due to the widening part and edge curving in the opposite direction, the system considerably increases the radius of the blend surface that again results in self-intersections. Using a flat spine of the shown shape helps easily create the blend surface.



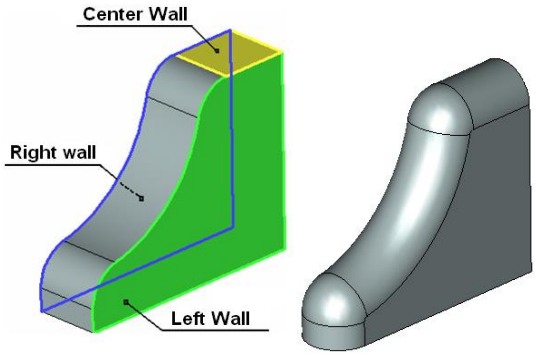
Additional Capabilities

When blending three faces of a solid body, the result will always be a solid body. When blending sheet bodies, the system automatically analyzes results for the possibility of creating one joint body. With the successful creation of one sheet body, the faces of the right and left walls are trimmed with the blend surface. The center wall disappears. The extents of the blend surface are trimmed to the walls. In the case when it is impossible to merge the result into one body, the system returns a set of separate sheet bodies.

Tangent propagation

Just like in other blend commands, this option provides capability of extending the three-face blend surface onto adjoining faces to the selected walls, if these adjoining faces meet smoothly with the walls and each other. This simplifies the operation definition by letting select just one face per wall. The system will attempt to extend the blend surface on all adjoining faces as far as possible.

When using the tangent propagation option, make sure the guide spine provides correct reference for blend construction across the extent of the blending area.



Examples of using tangent propagation functionality.
The edge between the left and center walls is used here as a guide spine

Operation Defining Steps

The operation creating command can be called via one of the following ways:

Icon	Ribbon
	3D Model→ Create → Blend → Three Face
Keyboard	Textual Menu
<3DT>	Operation > Blend > Three Face

The property window and the automenu are used for handling the command and defining necessary components. These work simultaneously and enhance each other. The object selection modes can be manipulated via either the automenu or the property window.

To create the operation, follow the following steps:

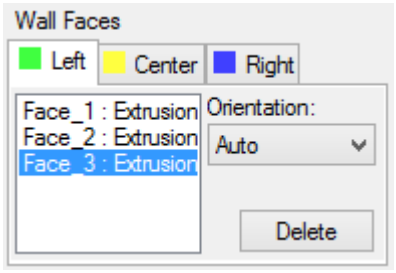
- ✓ Select first set of faces (the left wall)
- ✓ Select second set of faces (the center wall)
- ✓ Select third set of faces (the right wall)
- ✓ Select the spine (if necessary)
- ✓ Specify parameters of the operation (if necessary)
- ✓ Confirm operation creation

Selecting a Set of Faces

Normally, the operation definition begins with selecting the sets of faces between which the blend will be constructed. The left and right-wall sets are formed when one of the following automenu options is active:

	<L>	Select left Face
	<C>	Select center Face
	<R>	Select right Face

Switching between the options is done by pressing an icon button in the automenu.

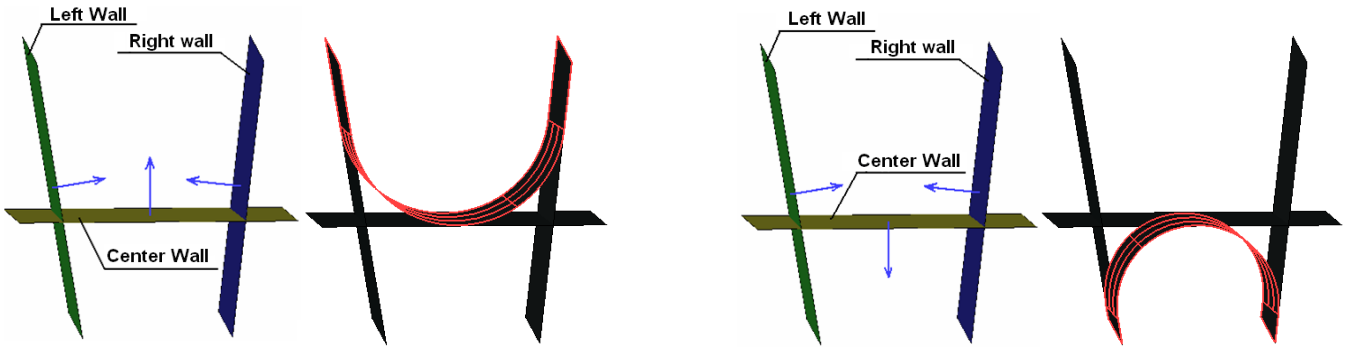


Turning on an option automatically activates the respective tab on the property window. The list of selected faces of the right or left wall is displayed in the provided pane under the tab tag. Switching between the tabs of the property window also activates the appropriate options for face selection.

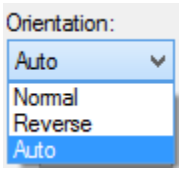
Face selection is done using the mouse in the 3D view window.

The selected faces of the left wall are highlighted in **green** by default, the right ones – **blue**. The set of the center wall is displayed in **yellow**. Settings of the highlighting colors are done in the system customization dialog box. The current face highlighting color is indicated by the colored box on the tab tag for each set.

A decorator is drawn in the middle of each face, shaped as an arrow, marking the side of the face adjacent to the blend surface. The blend is created on the side pointed at by all three arrows.



Control over the arrow directions is done via the provided "Orientation" item right of the face list pane on each tab in the property window. Selection is made in the pull-down list. For solid bodies, the direction of an arrow is determined automatically. In this case, the **"Auto"** option is set for all faces. For sheet bodies, the direction of faces has to be controlled manually by the user.



To change the system default arrow direction, select one of the other two available orientation settings, **Normal** or **Reverse**.

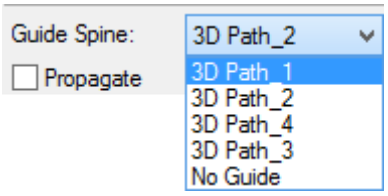
To exclude a face from the wall set, simply select it in the list in the property window on the respective tab and press **[Delete]** button.

Spine Selection

To turn on the spine selection mode, activate the following automenu option,

	<W>	Select guide spine

Selection of a 3D path is done in the 3D window or in the model tree by the mouse. The name of the selected 3D path is displayed in the **Guide Spine** box of the property window. A pull-down list is provided by the box listing the names of all 3D paths present in this model.



This list is convenient when the user knows the exact name of the 3D path intended as the guide spine. To reject a 3D path selection, pick the **No Guide** entry in this list or use the automenu option:

	<K>	Cancel selection

Only one 3D path can be used as the guide spine. If, for example, a wrong path was picked by mistake then another one can be selected right away. Selection of a new 3D path cancels the previous selection, and the last selected path will be used as a guide.

Special Features Setup

Tangent propagation

To turn on the option of tangent propagation, check the respective item in the property window.

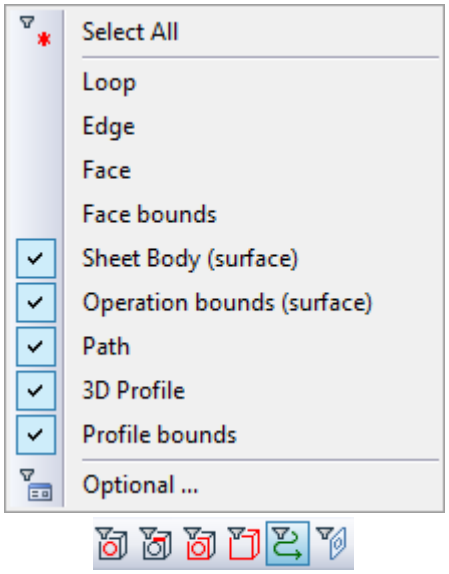
LOFT

This operation creates solid bodies of complicated shapes. The Loft operation can be built on practically any model elements based on one of the three principal geometry types, "point", "wire" and "sheet". Depending on the geometry type of the base elements, the result may be obtained in the form of a solid body or a set of surfaces. The resulting spline surfaces are constructed based on the specified set of sections and guides subject to the imposed boundary conditions. An optimization module is available among the operation settings that forces generation of simple analytical surfaces whenever possible.

MAIN CONCEPTS. OPERATION CAPABILITIES

"Point" in the three-dimensional space can be defined by a 3D node, a 3D vertex, or a "position" parameter on an object. The "wire" geometry elements (the ones with length/perimeter but without area) include all linear objects – edges, 3D paths, loops and some types of 3D profiles. Some times, the system can automatically extract geometry of the required type from the sheet objects, such as faces. "Sheet" objects (with area, but without volume) include all types of surfaces, faces and 3D profiles created from a closed contour. Sometime an operation is accepted as a sheet object, if the operation results in a surface (a set of surfaces).

While selecting elements, always pay attention to the active object selection filters. These are convenient, as, generally, pointing the cursor to a group of elements may not identify the specific desired element. Thus, for example, if we need to select just an edge on the profile rather than the whole profile, the filter should be set up for edge selection.



To turn on the necessary filter, one can use the pull-down list under the currently active option of the automenu. The options possessing such pull-down lists are marked by a black triangle in their lower-right corner. The list of the available filters under a particular option is accessed by pressing and holding . A simplified set of the available filters is simultaneously displayed on the system toolbar in the form of the respective buttons.

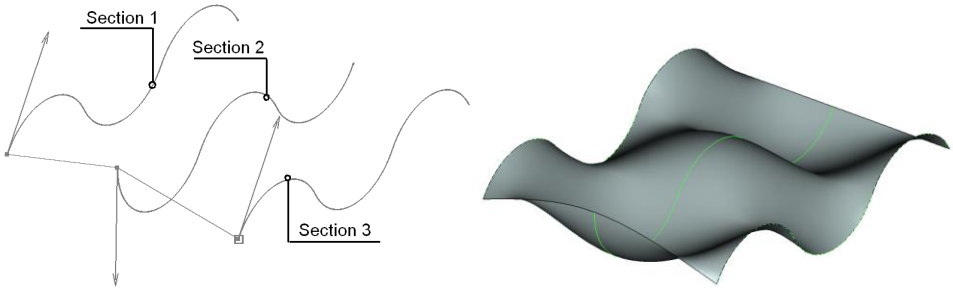
To turn on one filter and simultaneously turn off all the rest, push the filter button with while holding the <Ctrl> key.

Sections

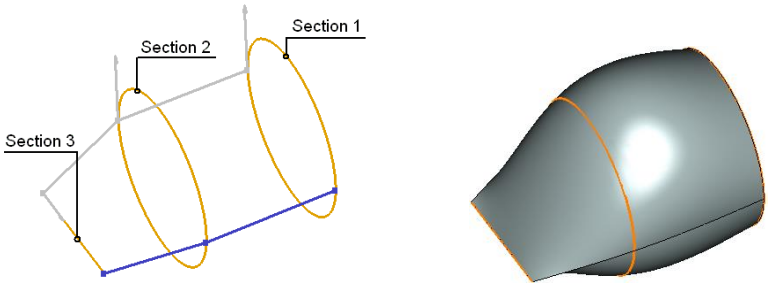
Sections define the surface shape along the first direction. To create a simplest instance of the operation, at least two sections need to be selected.

A section can be represented by either a sheet or a wire object. A wire object may be either closed-loop or open. One section can be composed from several pieces that can be freely added and removed during

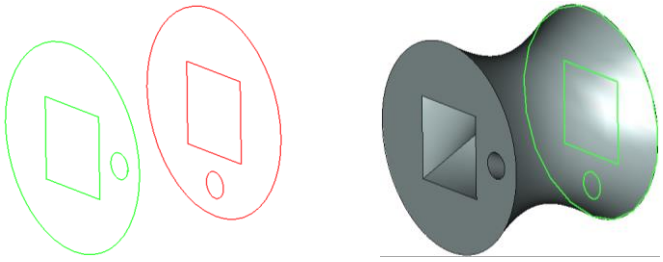
the operation setup. The set of objects making one section must have same-type geometry. For example, one section can be composed of a set of paths and edges, or, alternatively, of a set of adjoining faces.



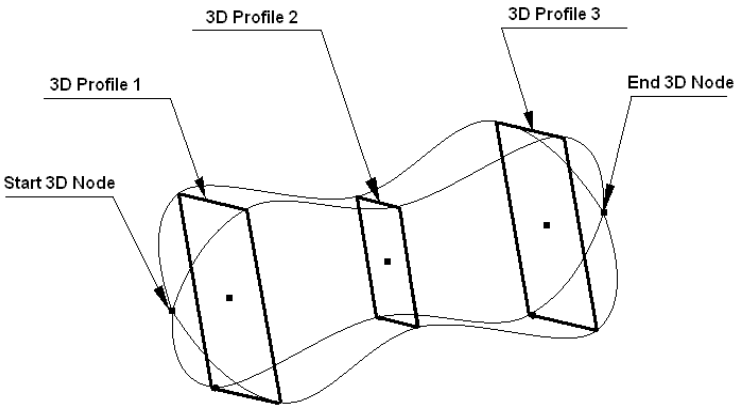
The contours of all sections used for creating the resulting surface must be of the same type – either open or closed. Exception is the case of a "clamped tube" body. In that case, most sections are defined by closed-loop sections, while the first or last section is defined by an open "wire". In order to successfully create such a body, follow precisely the set of conditions listed below.



Multi-contour profiles can be used as sections, if they have same topology. For example, these can be similar profiles or copies of the same multi-contour profile. In this case, no other parameters may be specified (guides, matches, boundary conditions).



A point can be used as the first and last section. In this case, the surface will collapse in the selected end points.



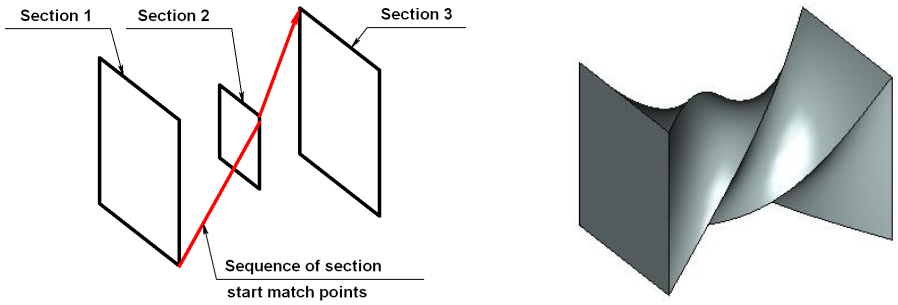
A section possesses such a property as the direction of the contour traversal. When defining the operation, an arrow is drawn on the selected section indicating the contour direction The contour direction is used for computing the surface, it should match for all selected sections. In most cases, the system can automatically determine the contour directions for successful creation. However, if necessary, it can be flipped by the user for each individual section.

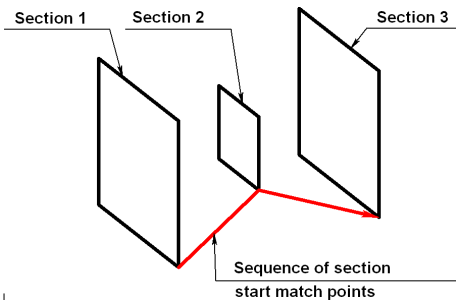
Match Points

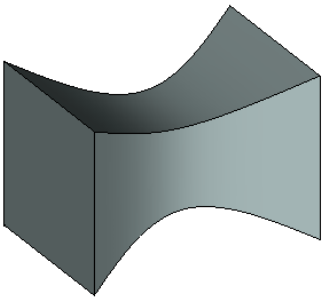
The match points help defining the direction of the surface propagation between the characteristic points of different sections. Always there is at least one match between the points in the sections. Originally, it is set between the start points of the sections. New sequences of match points can be added by the user manually. Any sequence of match points, including the start points, can be edited. The preview of the operation in the 3D window displays each set of match points connected by straight lines as soon as the elements get selected.

In the case of open contours, the start points can only be defined at the start or end of a section.

Each set of matching points includes one point from each section, except for the end sections defined by a single point The match points can be defined by 3D nodes, section vertices or the points defined on a section edge by the “position” parameter.





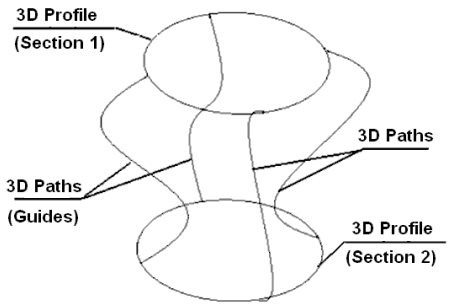


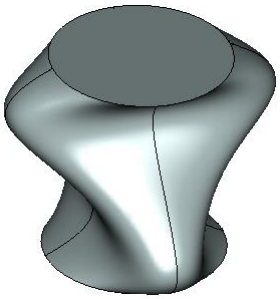
If the end sections are open with the rest closed (as in the "clamped tube" body type), then the following conditions must be met when defining the match points:

- ✓ At least two sequences of match points must be defined passing through the start and end of the first and last open wire.
- ✓ The closed profiles should not have "free" vertices, that is, the vertices not used as match points in any sequence.
- ✓ All match point sequences must meet at the start or end of the first and last open wires.

Guides

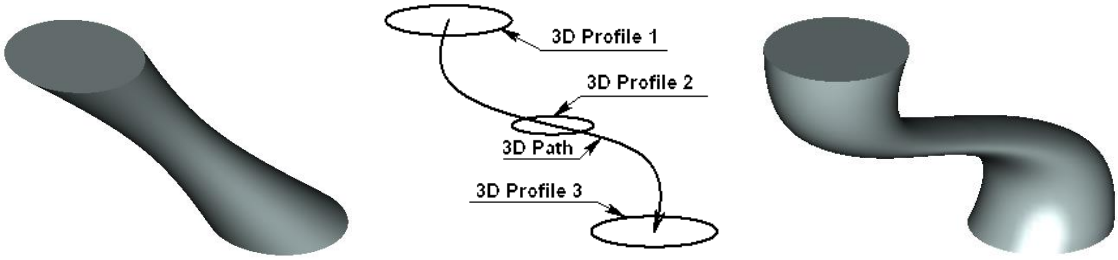
Guides are similar to sections and are used for defining the shape of the resulting surface in the second direction. As opposed to sections, the guides can be made of wire geometry only. If a sheet object is selected, its perimeter will be used instead. All objects must have continuous first derivative, that is, the "wire" should not have any sharp corners. Another condition is the requirement for a guide to intersect with all sections, including the ones defined by a single point. One guide can be composed of several smoothly connected linear objects.





Guides can be closed or open. Closed guides are used for creating closed bodies.

Instead of guides, one 3D path can be used, as if replacing multiple guides passing through all section points. The path defines the boundary conditions for t and he entire resulting surface. Therefore, other boundary conditions are not subject to being defined when the guide is defined. The only exception is the option of specifying the tangent direction with respect to the surface being created as derived from the path at the point where it intersects the section defined by the 3D profile (see below). A path is not required to intersect with the sections. The diagrams following below demonstrate the operation created with neither a path nor guides (left) and the one using a 3D path (right).

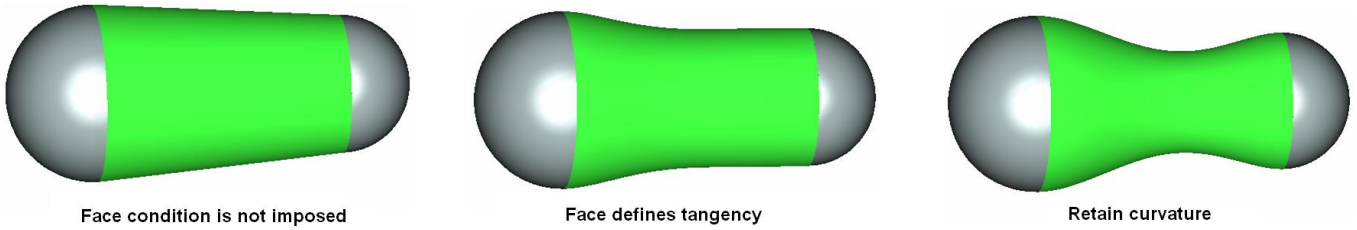


Boundary Conditions

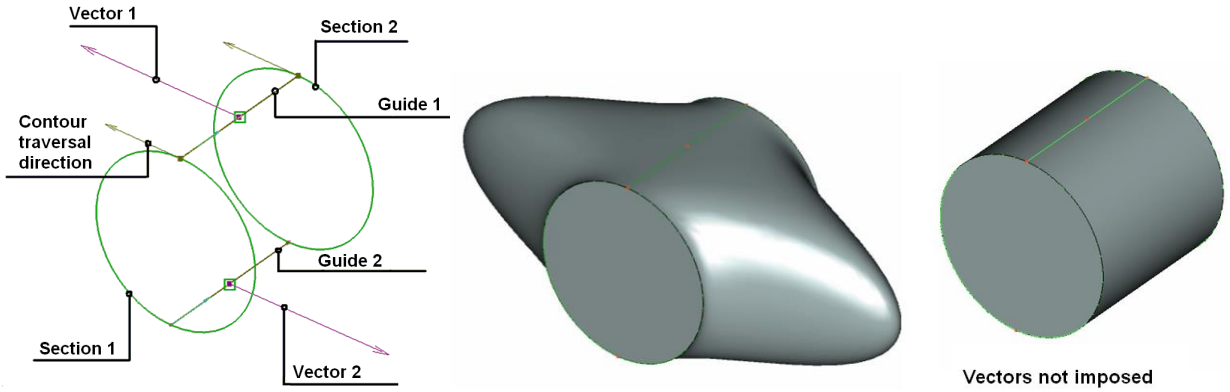
Boundary conditions impose additional constraints on the shape of the resulting surface.

The following boundary conditions can be defined for all sections and guides, except for the case of using a single 3D path as a guide:

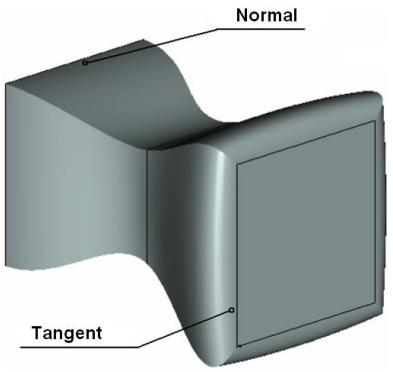
- Faces – these help define the first derivative (the direction) at each point of the section edge and at a segment of a guide, or the second derivative (the curvature) only for a section edge. The direction is derived from the face at the tangency point between the face and the frame edge. The face specified for defining the condition must end at an edge of the section or guide on which the condition is being imposed.



- Vectors – these are used for defining the first derivative (direction) of the resulting surface at an arbitrary point on a section contour or guide.



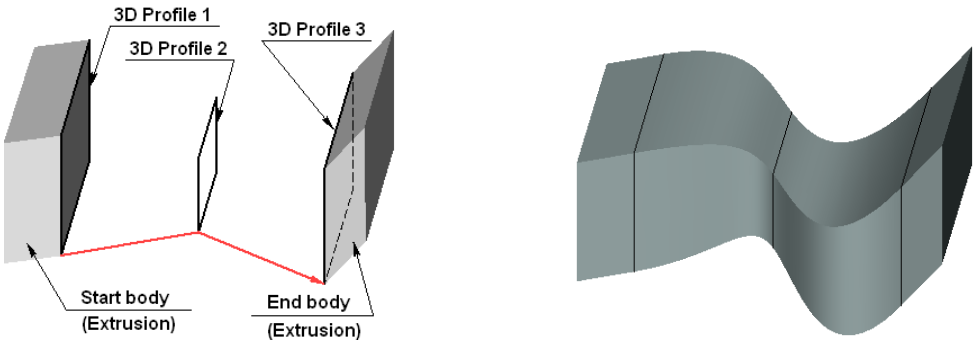
The following boundary conditions are applicable at the start and end sections:



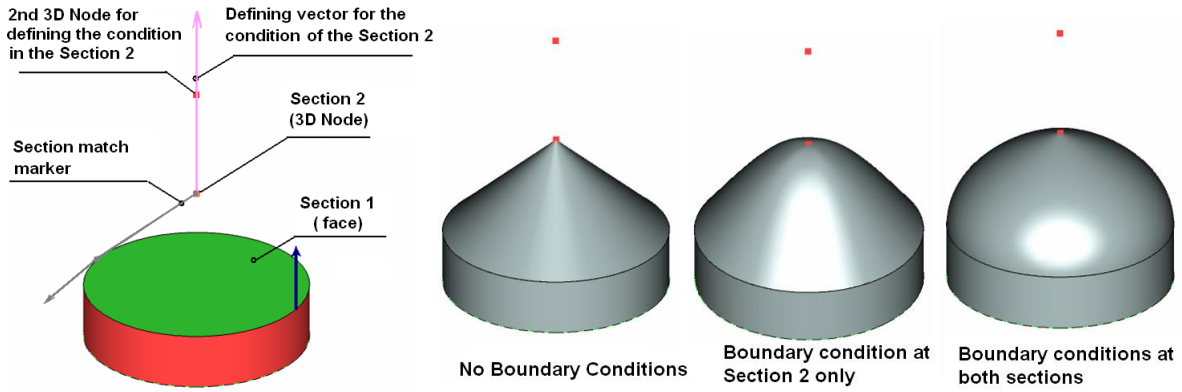
- Normal (outside) – the resulting surface is constructed orthogonal to the section plane. The normal vector is computed automatically and always points outside of the body to be created.
- Normal (to profiles) – same as the above. The normal vector is derived from the profile. This is a legacy functionality of earlier T-FLEX versions.
- Tangent – the resulting surface is constructed tangent to the section plane.

The above three conditions can be defined only for sections that are planar or are actually a part of a plane.

- Body – the body providing the tangency condition. The body must directly adjoin the section by one of the faces, with the face preferably matching the section The tangency condition (the directions) is derived from the side faces of the body.



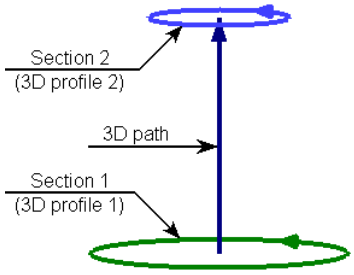
The boundary conditions for the start and end sections – points can be defined as a normal vector of a plane passing through the point. The resulting surface will be made tangent to thus defined plane.



All boundary conditions for sections listed above are mutually exclusive. This means, the same area or point may not be assigned several boundary conditions. No conditions are allowed on the end sections in the case of a closed surface.

Different boundary conditions may conflict. Only one face should correspond to a section edge or guide. If more than one face relates to an edge then the boundary condition is ignored, and the system outputs an appropriate warning. A boundary condition may not be defined by a vector at the intersection of a section and a guide. Two vectors should not be defined in the same point either.

If a single 3D path is defined instead of guides, then only one type of boundary conditions can be specified for the surface being created, which is the 'tangent to path' condition at the point of the path's intersection with the section. The section to which this condition applies must consist of a single 3D profile. The boundary condition is defined by selecting a face of the profile, on which the section definition was based.



Resulting body by sections created without applying any tangency conditions

Resulting body by sections with the tangency condition applied at the sections 1 and 2

Loft Operation Defining Steps

The operation creating command can be called via one of the following ways:

Icon	Ribbon
	3B Model → Create → Loft
Keyboard	Textual Menu
<3SL>	Operation > Loft

The property window and the automenu are used for handling the command and defining necessary components. These work simultaneously and enhance each other. The state of the automenu depends on the operation defining stage and on the purpose and type of the objects being selected. The object selection modes can be manipulated via either the automenu or the property window.

The surface of the operation is computed based on the element skeleton and specified boundary conditions. The skeleton elements are sections and guides.

To create the operation, follow the following steps:

- ✓ Define sections

- ✓ Define guides (optional)
- ✓ Define match points (optional)
- ✓ Define boundary conditions (optional)
- ✓ Confirm operation creation

Section Selection

The operation is always defined beginning with selecting the sections. For successful operation creation, at least two sections are required. Any system object may be used as a section that contains wire or surface geometry.

To start the section selection mode, focus (click) at an empty line in the "**Sections**" pane. This area is on the [**Sections**] tab of the property window. The automenu will then automatically switch to section selection mode.

The following options become active.

	<U>	Select Sections

This option is responsible for section selection mode.

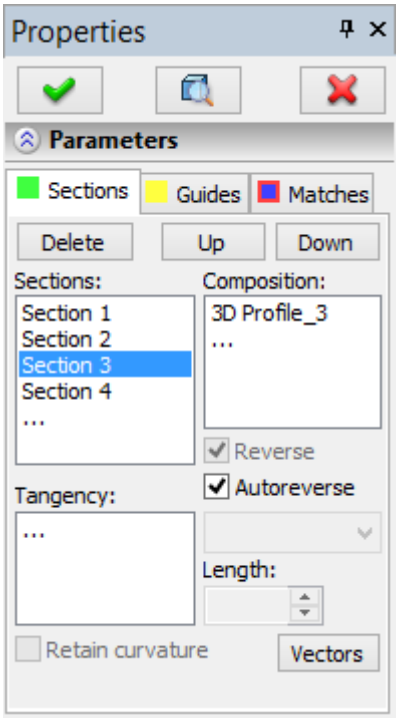
	<R>	Select 3D Profile

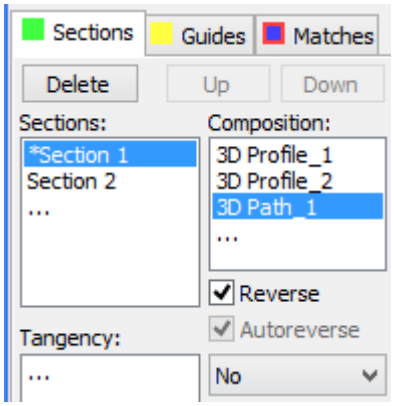
This option provides fine tuning of the filters for selecting objects.

The black triangle in the lower-right corner of the icon indicates availability of the pull-down list.

Then, proceed with selecting objects in the 3D window. Each selected element will be appended in the end of the sections list as another section. Sections can be moved up and down the list with the help of the respective dialog buttons. To remove any object from the list in all three panes of the dialog, highlight the object in the list and press the **[Delete]** button or the <Del> key.

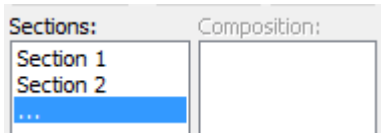
RESTRICTIONS: The start and end sections may not be moved around the list if these are defined by points, or if boundary conditions are defined there specific to the start and end sections.





Each section may be composed of several objects of the same type. The "**Composition**" pane shall be used for defining such complex section. This pane displays the list of names of the selected elements for each particular section. To add to the section contents, focus (click) at the "**Composition**", pane and then select additional objects.

RESTRICTIONS: A section may be composed only of the objects of the same geometry type.



If for some reason the system runs into an error while composing a section contour, this section will be marked by the "*" symbol. This may happen due to various reasons. For example, gaps were produced during model regeneration, or elements disappeared that were part of a section.

By default, the smooth edge chain selection mode is turned on. In this mode, selecting one edge automatically results in selection of smoothly adjoining edges to the selected one, that is, making the selected edge continuation with the continuous first derivative, with no sharp corners. Turning the mode on/off is done via the respective automenu option:

	<G>	Smooth edge chain selection mode

Point objects – 3D nodes and vertices can be selected as the ending sections (start or end). To define such section, activate the following automenu option:

	<E>	Select 3D Node

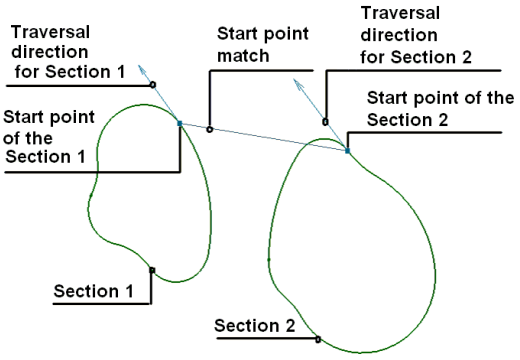
By switching between the selector filters one can select any objects defining a point in the three-dimensional space: a 3D node, 3D vertex, center of an arc, center of a sphere, a coordinate system. The new section – point will always be at an end, in the beginning or the end of the list of sections.

Operation skeleton display when defining sections

The elements selected as sections are highlighted in **green** by default. The active section is highlighted in **red**. Active considered the section that is being currently manipulated. For example, working on a section composition makes it active. Settings of the highlighting colors are done in the system customization dialog box.

The colored square on the [**Sections**] tab is filled with the color of the current highlighting setting of the sections.

A start point is set for each selected section. The set of start points makes the first match sequence. If sections are closed then the start point is set in the nearest to the cursor vertex of the section. For open sections, the start point is set in the nearest end of the "wire". If the system-defined configuration of the section starts is not suitable, it can be changed, or new matches can be added. The detailed description of section matching follows below.

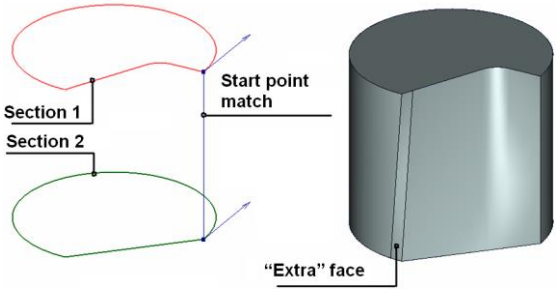


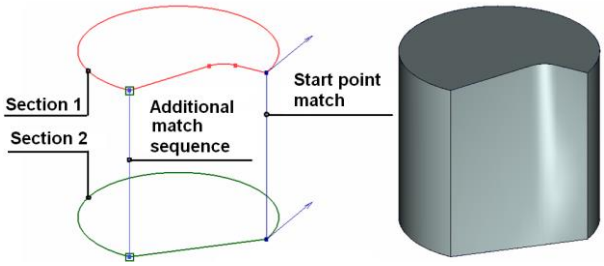
An arrow is displayed at each section start indicating the contour direction. The same direction should be defined in all sections for successful operation creation. A contour direction can be flipped by toggling the **Reverse** flag. Toggling the flag moves the start arrow to the opposite end of the contour.

In the case of closed contours, the system attempts to synchronize the contour directions automatically, using the **Autoreverse** option.

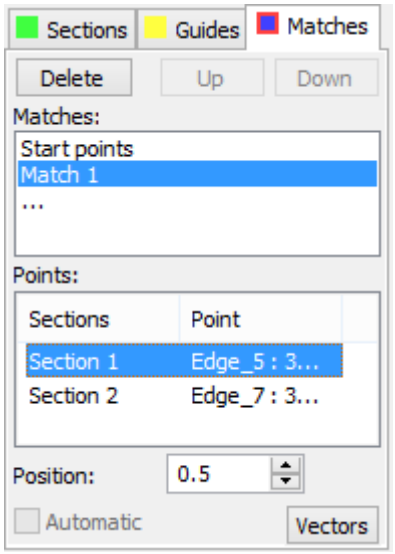
Defining Match Points

Once the sections are defined, the system needs to relate them with each other. One needs to define how to configure the surface. One match sequence is automatically created from the start points once the sections are defined. In complicated cases, however, this may not be enough.





The sections are divided into sectors by the match points. Some of the generatrices of the surface being created pass exactly through the match points. The space between the sectors is divided into an equal number of elements for the primary generatrices to pass through. Should there be at least one vertex within a sector between any two match points on a section, the system subdivides the respective matching sectors in all sections in the same proportion. This is done to achieve an equal number of sectors in all sections. This may lead to creation of additional, sometimes undesired, edges and faces. Therefore, an optimum solution is to define match points in the vertices of the sections. This way of defining the match points should be favored whenever possible.



To switch to match point defining mode, go to the [**Matches**] tab of the operation property window. This tab hosts two related panes with the lists of sequences and the respective match points. Each sequence contains one point per a section, except for the single-point sections. The points are ordered from the start to the end section. The contents of the sequence can be viewed in the second pane, named "Points".

To define a new sequence, place the cursor at an empty line in the "Matches" pane. The system is now ready for defining the points. The following options get activated in the automenu:

	<C>	Define match points

This option sets the mode of match point selection.

By default, all three types of points are selectable in the command. The automenu lets setting up a filter for selecting just one type of object: nodes only, vertices only, or points on edges. The following automenu options are used for such setup:

	<A>	Select All Points
	<N>	Select 3D Nodes
	<X>	Select Vertices
	<K>	Select Points on Edges

The first section is activated and is automatically highlighted in red (by default) in the 3D view window. Once a point is selected on the first section, the second section gets highlighted, and so on, until points are selected in all sections. All vertices of the active section are highlighted as well. The object snapping turns on, helping to bind a point to a section vertex or a 3D node on the section.

If snapping to vertices and nodes is not required then the point can be placed on a section edge. In this case, the point position on the edge will be defined by a parameter varying from 0 (at the edge start) to 1 (at the end of the edge). The edge direction is same as the contour traversal direction.

A point can be selected by pointing the cursor to the desired object and clicking . Object selection occurs per current filter settings.

The **Automatic** flag is used for automatic placement of the match points. The automatic match points can only be defined for flat closed contours under the condition that at most one section is allowed to have the discontinuous first derivative (a sharp corner). As the "Automatic" flag is cleared, the points created automatically are converted to the modifyable objects.

RESTRICTION: Intersecting match point sequences are not allowed.

To edit a sequence or point object, simply select it in the property window, after which redefine it by selecting another object in the 3D window. Thus, to edit a whole sequence (say, the start point one), select the object **Start points** on the **matches** tab, and then pick one by one the desired points on each section. Highlighting and object snapping work just as at the initial sequence definition. To edit a particular point in the sequence, first select this point in the list of the "Points" pane, and then select another object in the 3D window. The respective section is highlighted while editing a point.

Upon finishing editing of the selected object, the system proceeds with the next object in the list – a point or a sequence, or offers to define a new sequence, if the edited object was last in the list.

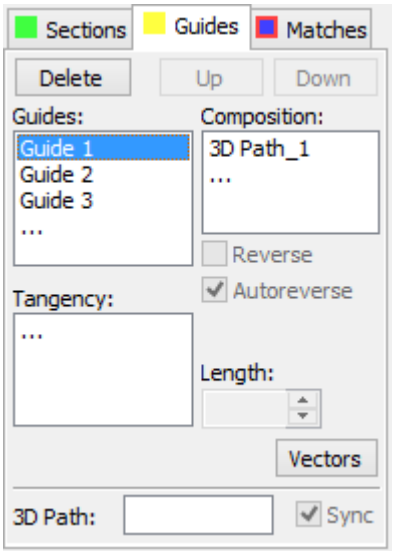
To remove any entry from any pane, select it and press the <Del> key or the **[Delete]** button.

Guide Selection

The guides are side elements of the operation skeleton. Guide definition is mostly similar to the section definition procedure.

To switch to the guide definition mode, go to the **[Guides]** tab of the property window. Just like the Sections tab, this one hosts three panes, **Guides**, **Composition** and **Tangency**. The latter will be used when defining boundary conditions (see below). If an error occurs while composing the contour, then the guide is marked by "*".

To select a new guide, click at the new line of the **Guides** pane. The new line is marked by an ellipsis. Upon switching to the tab, the automenu enters the mode of guide selection. The necessary options become available.



	<V>	Select Guides

This option sets the guide selection mode. For convenience, the necessary object selection filters can be set up. The filter buttons are located on the system toolbar or in the pull-down lists of the appropriate automenu options.

	<R>	Select 3D Profile

This option provides fine tuning of the filters for selecting objects. The black triangle in the lower-right corner of the icon indicates availability of a pull-down list.

By default, the smooth edge chain selection mode is turned on.

	<G>	Smooth edge chain selection mode

The elements can be selected in the 3D window or in the model tree.

> RESTRICTION: A guide must intersect with all sections, including the single-point ones.

Operation skeleton display when defining guides

The elements selected as guides are highlighted in *yellow* by default. The active guide is highlighted in *red*. A guide becomes active as it is selected in the list of guides. Settings of the highlighting colors are done in the system customization dialog box.

> The colored square on the [**Guides**] tab is filled with the color of the current highlighting setting of the sections.

An arrow is displayed on each guide indicating the traversal direction. The direction should be from the start to end section for successful operation creation. The direction is defined automatically for open guides. The traversal direction of a closed contour can be flipped by using the **"Reverse"** flag. When closed contours are used, the system tries synchronizing the directions in the **"Autoreverse"** mode.

3D paths used instead of guides

The direction of the loft surface construction can be defined in each section.

> This functionality is kept primarily for compatibility with older versions. When creating a loft operation anew, guides and boundary conditions should be used for full control over the shape of the surface.

To enter the mode of path selection, click the mouse inside the "3D path" pane. The option will activate in the automenu:

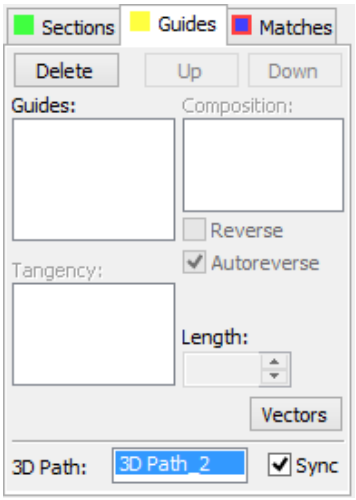
	<W>	Select 3D Path for Sections orientation

Then a 3D path can be selected in the 3D window or in the model tree.

The specified path affects the body shape as follows: Vectors are defined for all match points of all sequences from a match point to the nearest point on the path. The body surface is then computed using the minimum twist algorithm.

To successfully create the operation, it is necessary that the selected path is oriented from the first section to the last one. If the "**Sync**" flag is set in the properties window, the system will automatically change the direction of the selected path as necessary.

To reject a path selection, place the cursor in the "**Path**" pane and press the <Del> key or the [**Delete**] button on the dialog.



The second direction can be defined by either guides or a path, and not both. If a path is used, the guide selection is blocked. Vise versa, if at least one guide has been specified, then the "**Path**" pane becomes inaccessible.

Defining Boundary Conditions

Different boundary conditions can be defined for different operation components. These are defined in order to achieve a surface closely matching the design intent, including various blending features.

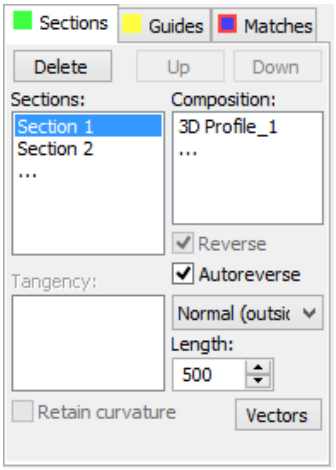
Conditions on ending sections

These boundary conditions are specific to and can only be defined for the start and end sections, if these sections are based on flat contours. The geometrical information contained in a flat contour is sufficient to determine the rules of the surface construction in the area near the given section.

A choice of only two conditions, how the surface meets the section, is available – orthogonal or tangent to the section plane. The degree of surface twisting is defined by a number representing the vector length.

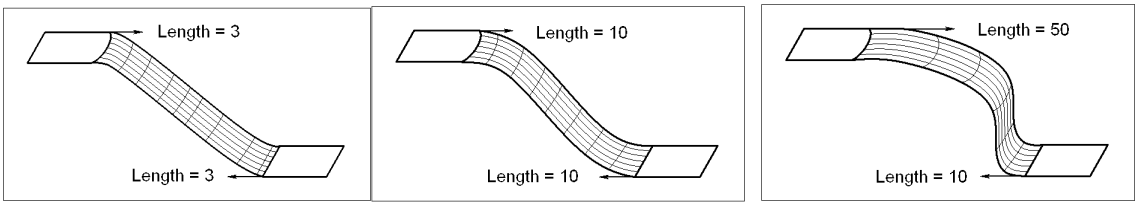
The following steps are to be done in order to define such boundary condition:

- ✓ Select the desired ending section in the full list of sections. The selected section becomes active and is highlighted in red.
- ✓ If the geometry of this section is appropriate then the list will become accessible on this same tab of the dialog containing the following four conditions: **No**, **Normal (outside)**, **Tangent, Normal (to profiles).** Select the desired condition.



✓ Enter the desired numerical value in the **Length** box.

The value is entered in the absolute model units. Therefore, the required value depends on the size of the model. Large models require greater values for achieving the same expected result.



Boundary conditions on the ending sections can be defined by tangency to bodies. The spline surface will be created in this case tangent to the faces of the selected bodies adjoining to the section.

To define such boundary condition, do the following steps:

1. Select the desired ending section in the full list of sections. The selected section becomes active and gets highlighted in red.
2. Click the mouse at the **Tangency** pane. If an ending section was selected then the body selection option activates in the automenu (another available option in the automenu for face selection is described below).
3.

	<B>	Select Solid for setting beginning condition
	<B>	Select Solid for setting ending condition

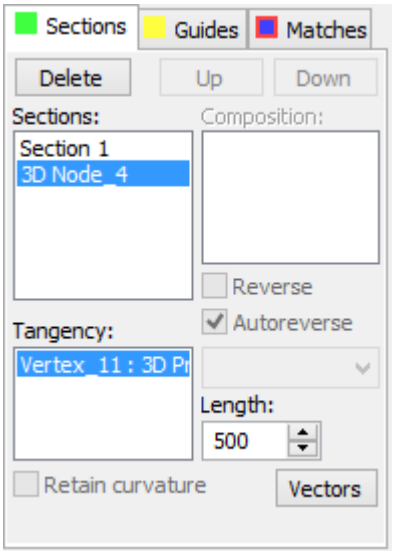
The option appearance depends on whether the start or end section is active.

4. Select a body. The section being processed must match with a face on the body. The surface being constructed will be made tangent to the adjoining faces of the body.

Conditions for sections defined by single 3D point

A plane can be constructed through a single point section. The surface collapsing at an ending single-point section will be made tangent to this plane. To define the plane in space, an additional object is required to define the direction of the plane normal. The surface curvature is defined by the vector length.

The following steps are to be done in order to define this boundary condition:



- ✓ Select the desired ending section in the full list of sections.
- ✓ Click at the **Tangency** pane. If a single-point section is selected then the following options become available in the automenu for selecting new objects :

	<S>	Select first point of pyramid

The first option sets up the filters for selecting points (3D nodes, vertices etc.

The second option is designated for selecting direction-defining objects, such as straight edges, paths, profiles, axes of revolution for rotation operations, coordinate systems, etc.

	<D>	Select Vector Direction for End Point

- ✓ Using one of the options, select a point or direction in the 3D window. Upon selection, the name of the selected object will be displayed in the "**Tangency**" pane.
- ✓ Specify the vector length. Do this by entering a value in the "**Length**" input box.

Conditions for arbitrary sections and guides

The boundary condition can be defined for any sections and guides by faces. Subject to certain restrictions, such condition may define the direction (the first derivative) or the curvature (the second derivative) of the surface being created in the area near the currently active element.

To specify a boundary condition by faces, do the following steps:

- ✓ Select the element subject to the boundary condition being defined. The sections are selected from the **Sections** pane on the [**Sections**] tab, while the guides are from the **Guides** list on the [**Guides**] tab. The selected element becomes active and is highlighted in red.
- ✓ Click the mouse at the **Tangency** pane. The option for selecting tangency surfaces activates in the automenu:

	<F>	Select Tangent Faces

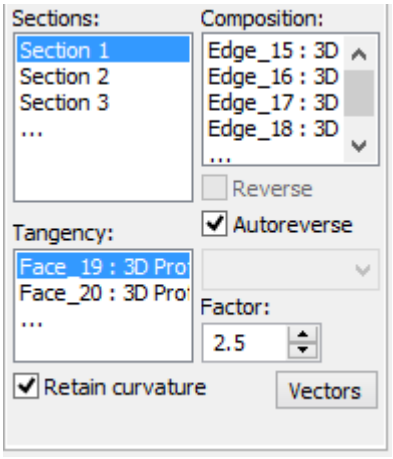
- ✓ Select a face in the 3D window that will define the boundary condition at the section or a guide. If several faces adjoin the skeleton element, select them all. While doing so, keep in mind the following restrictions:
  - The current element (a section, a guide, or a segment thereof) must fully lie on the selected face
  - The selected face must end at the active element

- If two skeleton elements intersect, their respective faces must define the same direction. This means, if guides are defined in the operation then this boundary condition is not acceptable for a whole section. In some special cases, when open sections are used, the system allows using this boundary condition when one guide is present.

In the case of violating these restrictions, the tangency condition is ignored and a respective message is output in the diagnostics window.

It follows from the above that the considered boundary conditions can only be specified at certain segments of a guide. The guide meanwhile must be defined by a set of smoothly connected elements.

The selected faces are highlighted in green (by default).



- ✓ Enter the value in the **"Length"** input box for the selected tangency surfaces. This value determines the degree of the boundary condition influencing the curvature of the surface being created in the area near the current skeleton element.
- ✓ Turn on an additional flag **"Retain curvature"** (optional). This will insure existence of second derivatives in the area where the current skeleton element meets with the selected faces. Without this, only first derivatives are guaranteed.

Please note that when using a single 3D path definition, the face boundary conditions can be specified only for the sections defined by the 3D profile. In this case, you can only select the face of that 3D profile as the face definition.

Boundary condition vectors

A boundary condition can be defined by a vector at any point on the skeleton elements of the Loft operation, such as a point, a section, a guide or a match point.

Vectors are defined in a special dialog box launched be pressing the **[Vectors]** button in the operation dialog. This button is present on all three tabs of the operation dialog. The Vectors dialog box will stay up even when switching between the tabs of the operation dialog. Its contents change depending on the operation mode and on what operation element is currently active.

I. Defining vector on skeleton elements

The following steps need to be done in order to define a vector based on a section or a guide:

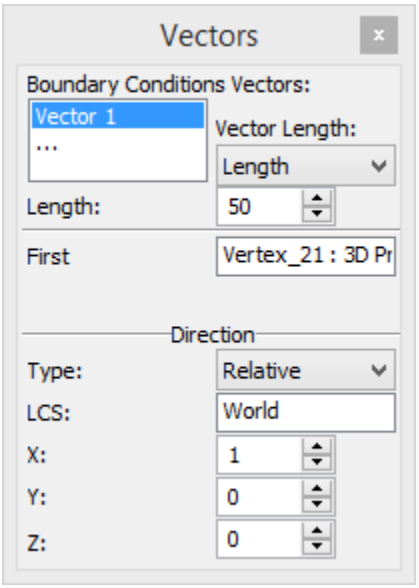
1. Select the skeleton element to base the vector upon. For this, switch to the respective tab of the operation dialog and select the skeleton element from the list. Upon selecting a section or a guide, this skeleton element becomes active and gets highlighted in red.
2. Call the vectors dialog box by pressing the **[Vectors]** button.
3. Click the mouse over an empty line with the ellipsis symbol in the **Boundary Conditions Vectors** pane of the vectors dialog. The system enters the vector-defining mode. All vertices of the active section/guide get highlighted. The following filter tuning options appear in the automenu:

	<A>	Select All Points
	<N>	Select 3D Nodes
	<X>	Select Vertices
	<K>	Select Points on Edges

   The first option is active by default.
4. Define the vector in space. There are several ways of defining a vector. The difference is in the ways of defining the start point, direction and the length of the vector.

The start point can be defined as follows:

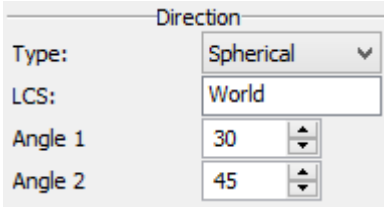
- At a vertex on the section/guide. Pick with the cursor one of the section/guide vertices. The name of the vertex is entered in the "**First Point**" box of the Vectors dialog.
- In a 3D node constructed on an edge of the section or guide. Pick a 3D node with the cursor on the section (guide). The name of the 3D node is entered in the "**First Point**" box of the Vectors dialog.
- At any position on any edge of the section (guide) by defining the parameter of the point position on the edge. The point is created by picking anywhere on the section (guide) edge. The edge name is entered in the "**First Point**" box. The position of the point is defined by the parameter along the edge. The parameter takes values from 0 to 1 and can be modified.



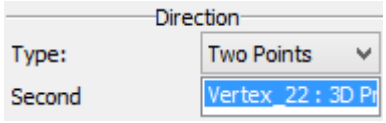
The vector-indicating arrow is drawn from the selected point on the skeleton in the 3D window.

Upon specifying the start point, additional parameters become accessible in the Vectors dialog box for defining the vector direction. The direction of the vector can be defined in one of the following ways:

- In absolute coordinates, assuming that the start point of the direction vector is located at the origin, <0,0,0>. With optional selection of a local coordinate system, the direction is computed in the relative coordinates.
- In spherical coordinates. Two angles must be specified to define the vector direction.

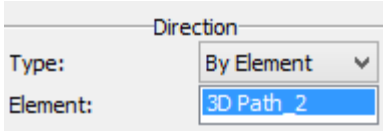


- By selecting the end point. Select an object defining a point in the three-dimensional space – a 3D node, a vertex, an arc center, etc. When switching to the direction defining mode by the second point, the automenu provides the following option with a pull-down list of filters:



	<S>	Select Second Point of Vector

- From a direction-defining element. This can be an edge, an arc, a face, a flat 3D path or 3D profile, a surface of revolution, a coordinate system. The direction is derived from the element itself. It can be, for example, the normal direction to a flat element, or the direction along an axis of revolution.



The following option activates in the automenu when defining the direction from an element:

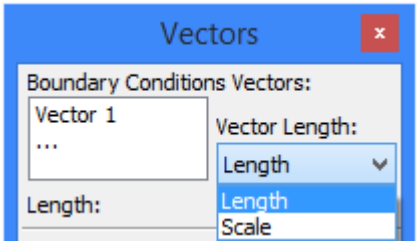
	<D>	Select Vector Direction

To define a local coordinate system, click in the **LCS:** box. The following option will appear in the automenu:

	<L>	Select LCS

Select a local coordinate system in the 3D window or in the model tree. This will let you define coordinates of the direction vectors relative to the selected coordinate system. A separate coordinate system can be defined for each vector.

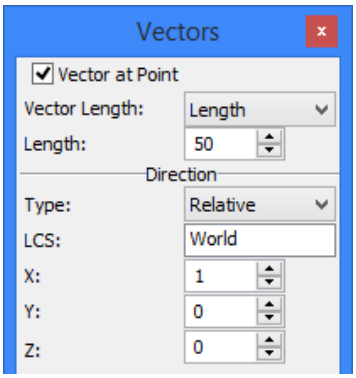
5. A section allows vector conditions of two types. The type of a condition is defined for all vectors constructed on one skeleton element simultaneously. The selection is done from the list in the Vectors dialog box:
   - Vector length definition.
   - Scale factor definition. In this case, the operation automatically computes the absolute value of the derivative along the specified direction and multiplies it by the scale factor.



6. (Optional). More than one vector can be defined for a single skeleton element. To define a new vector on a skeleton element that is already in use, proceed to the item 3.

II. Defining vectors at match points

The vector is defined at a match point according to the same rules as the ones described above for the skeleton elements. The only difference is that the match point selected for defining a vector becomes
automatically the start point of the vector. Do the following steps to define a vector at a match point:



- ✓ Switch to the [**Matches**] tab of the operation dialog.
- ✓ In the **Matches** list, select the desired sequence of match points.
- ✓ In the **Points** list, select the desired match point.
- ✓ Call the Vectors dialog box by pressing the **[Vectors]** button.
- ✓ Set the **Vector at Point** flag. This creates a new vector in the selected match point.
- ✓ Define the vector direction and length (see above).

Using the created loft in a Boolean

To create a Boolean operation, do the following:

Turn on the Boolean creation mode by the automenu option:

	<Ctrl+B>	Select original body for Boolean operation

A Boolean is created when the icon is pushed.

Select the type of the operation from the pull-down list under the above option:

	<Ctrl+'+'>	Addition
	<Ctrl+'-'>	Subtraction

	<Ctrl+'*'>	Intersection
	<Ctrl+I>	Smart mode

Operating principles of the Smart mode:

- If the body being created and the existed selected body have intersection of volumes - the type of Boolean operations: subtraction.
- If the body being created lies entirely within the selected body - the type of Boolean operations: subtraction.
- If the body being created touches and selected body touch each other - the type of Boolean operations: addition.
- If conditions set out above are not met, or a selected body lies inside the created, or error occurred determining the type of penetration, then, Boolean type is not defined - the Boolean operation will not be created.

Select the target body for the Boolean (optional in some cases), using the automenu option:

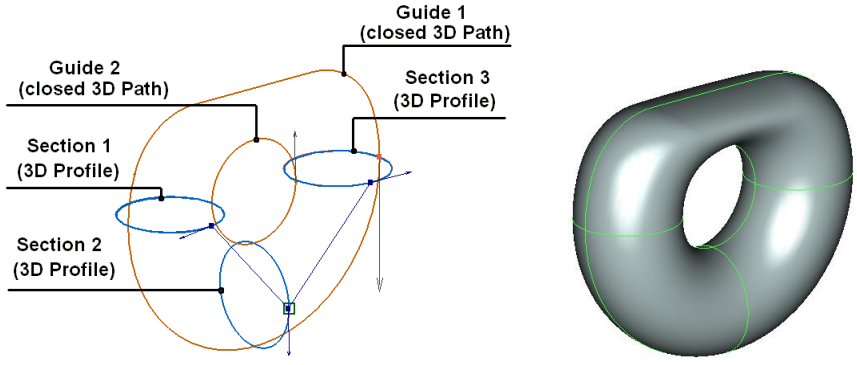
	<Ctrl+T>	Select Target Body for Boolean

If only one body exists in the scene, it is selected automatically. The new body created by the loft operation becomes the tool body of the Boolean.

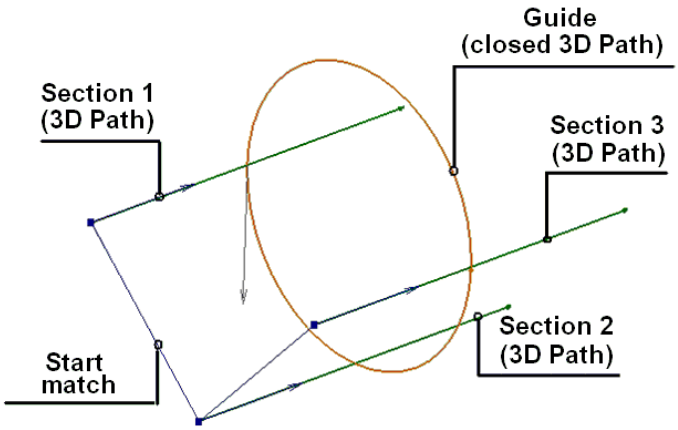
Upon confirming operation creation, first the body is created by the loft operation, and then the specified type Boolean is performed.

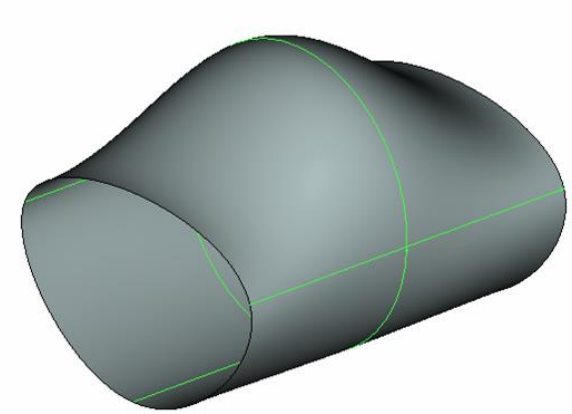
Setting Additional Operation Parameters

The "**Periodic**" flag is used for creating closed bodies or surfaces. If this flag is set, the end section will be connected with the start. This parameter can be set if the start and end sections are not single-point sections and if no special conditions are defined for the start and end profiles.

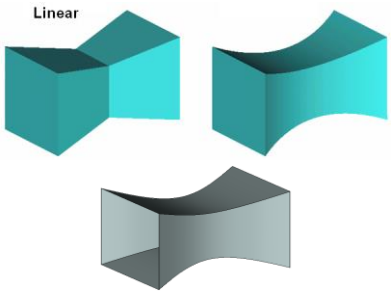


If guides are used together with the "Periodic" attribute then the guides must be closed.



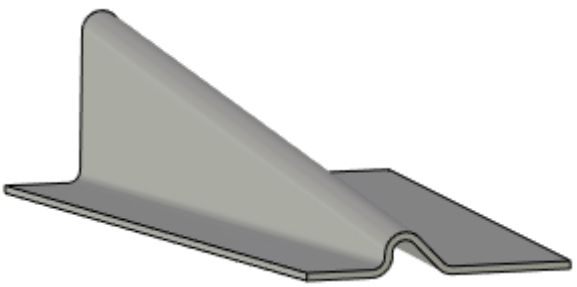


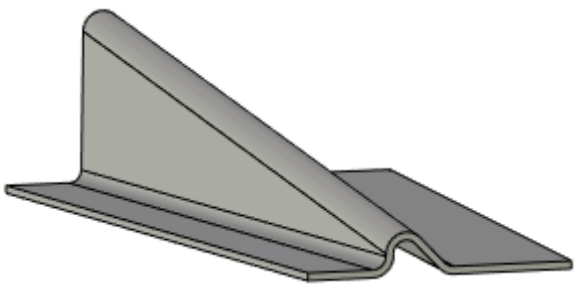
The **Linear** flag – if set, ruled surfaces are created, splines otherwise. A ruled surface is produced by sweeping a straight-line generatrix between two sections.



The flag **Thin-walled**. If sections are defined by elements of sheet geometry then this attribute defines creation of a surface instead of a solid body.

Keep initial splitting. When the option is enabled, boundaries of body surfaces are created based on the initial splitting of sections. The option allows you to create an unfolding of the body without using additional commands.





Optimization Parameters

The **Tolerance** parameter specifies the maximum allowed gap between profiles and guides. Thus, for example, setting rough (large) tolerance in some cases allows handling a situation when the sections and guides do not intersect, but pass near each other. The tolerance defines the distance between them that the system can ignore by assuming that the guides and sections do intersect. Besides, the tolerance affects the minimum size of a profile edge that is defined as ten times the tolerance value. Consequently, the same limit is used as the minimum distance between two distinct match points on a profile, as well as between start points of boundary conditions defining vectors. The lower tolerance limit is 10^{-8}. However, keep in mind that a too small tolerance may slow down the system operation. The system default tolerance is 10^{-5}.

The **Simplify geometry** flag is used for simplifying the geometry and topology of the resulting surface. With this flag on, analytical surfaces are created instead of splines whenever possible, such as a torus, a cylinder, a sphere and a plane.

Blend Parameters

If the resulting body is solid then it can be applied blend and chamfer options. Blending and chamfer creation can be done on the source and target surfaces independently.

Source blend. Setting this attribute defines the mode of edge blending on the original surface, that is the surface on the side of the start section. Blending is done in the way of "Round" or "Chamfer". The parameter of the blend is defined in the next box.

Target blend. Setting this attribute defines the mode of edge blending on the target surface, that is the surface on the side of the end section. Blending is done in the way of "Round" or "Chamfer". The parameter of the blend is defined in the next box.

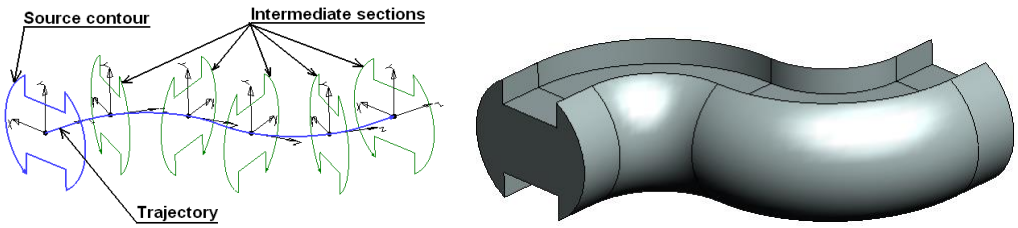
SWEEP

Sweep operation allows creating a solid body by sweeping a contour along a spatial path. This operation belongs to the family of so-called "kinematic" operations, as it results in creating a body swept by the original contour as the latter is moved along a guiding path. Depending on the type of original contour geometry (the contour can have wire or sheet body geometry), the operation will yield a sheet or solid body. As the contour is being swept along the path, its orientation, twisting and scaling can be controlled.

OPERATION MAIN CAPABILITIES

Contour

Any object with wire or sheet body geometry can be used as the original contour. A multi-contour 3D profile can be used as well. Sweeping a contour means placing its multiple instances along the selected path. A contour instance will be referred to as an intermediate section. To define an intermediate section, the system calculates orientation of the coordinate system of that intermediate section. The original coordinate system is calculated as the reference for the original contour orientation. A contour instance creation is done by transforming the original contour to the calculated coordinate system of a given intermediate section.



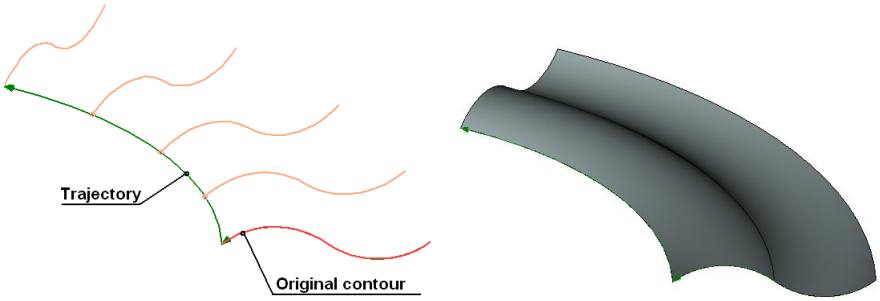
The original coordinate system is calculated at the start of the path. Two of its axes are set perpendicular to the path, and one is tangent to the path at the start point. The original contour position is fixed with respect to the original coordinate system. The contour sweeping is done by placing the original coordinate system together with the contour in other points of the path, in which the coordinate systems are defined for the intermediate sections.

Control Over Contour Orientation

Different geometry can be created, depending on the position of the intermediate sections with respect to the path. One of the three types of the contour orientation can be specified when creating the operation:

- **Perpendicular to the trajectory.** The X-Y plane of the intermediate section coordinate system is always oriented perpendicular to the trajectory. The tangency vector at the point of the section creation defines the Z-axis of the local coordinate system.

This mode is primarily for flat contours. When defining the operation, the plane of a flat contour is always oriented perpendicular to the guiding path in each intermediate point.



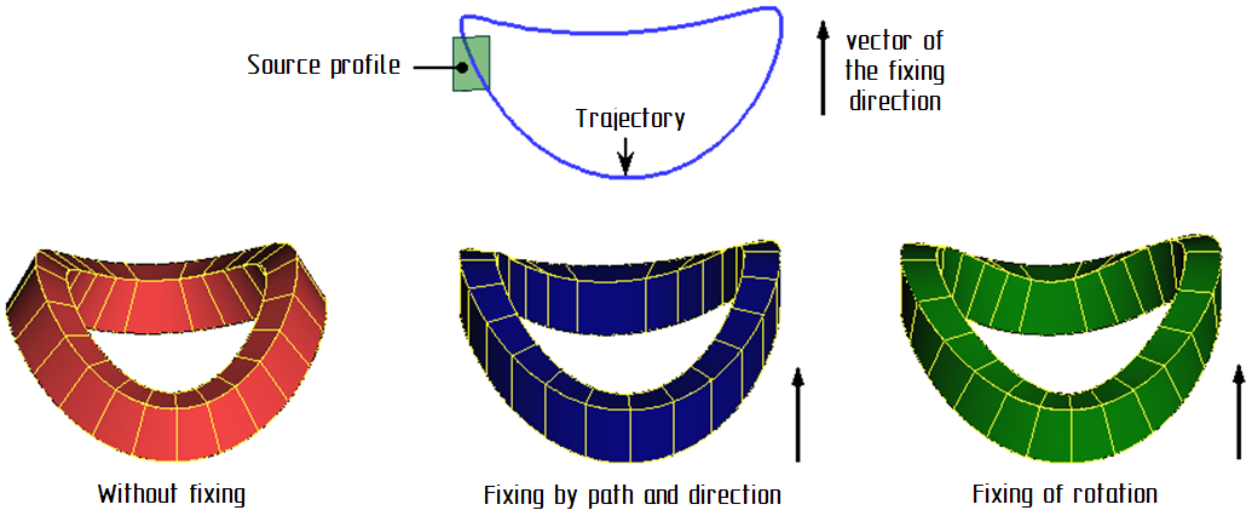
In case of non-planar contour the system automatically calculates some "mean normal" that is used for orienting the contour. We recommend using additional correction of the original contour position (see section "Correction of Contour Original Position") in order to get a predictable result.

When using non-planar trajectory, condition of perpendicularity is not sufficient for obtaining a desired result due to twist of the profile around the trajectory. In these cases it is possible to additionally define *fixing direction*. Specifying fixing direction can change the method of constructing coordinate systems of intermediate sections. The variant of a change depends on the selected method of interpretation of the fixing direction: *fixing by path and direction*, *fixing of rotation*.

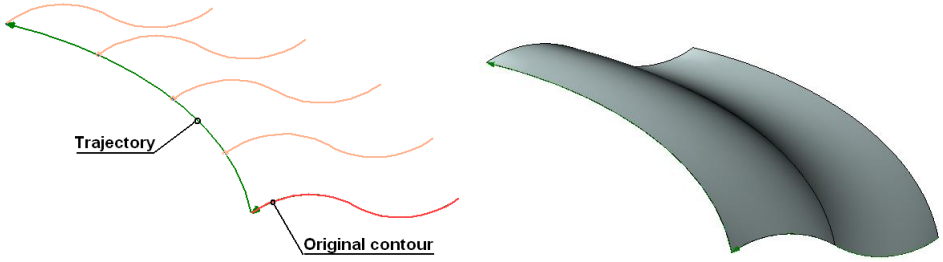
> Vector fixing the direction should not coincide with the tangent to the trajectory at any point of the trajectory.

Fixing by path and direction: tangent to the trajectory at each point of the trajectory is projected onto the plane perpendicular to the fixing direction. The coordinate system of the intermediate section is constructed from obtained projection of the tangent to the trajectory and fixing direction.

Fixing of rotation: the coordinate system of the intermediate section is constructed by the tangent to the trajectory at the given point of the trajectory and by the cross product of the fixing direction vector and the tangent.



- **Parallel to the original profile.** The coordinate axes of each intermediate section are oriented parallel to the axes of the original coordinate system. Sweeping in this way results in the parallel translation of the original contour.



- **By guides.** The operation is created by sweeping a contour along the trajectory while orienting the contour towards the points on the additional guides. At the same time, one of the guides also defines the contour scaling law. In this way, the guides are required to be smooth (no corners). This is a special way of creating the operation that will be discussed in a separate topic (see below). In some particular cases faces can be used for defining only orientation (direction) of the intermediate sections.

Trajectory and Guides

To define this operation, specify the trajectory path and, optionally, one or two guides. The contour is swept along the trajectory path, which must always be defined.

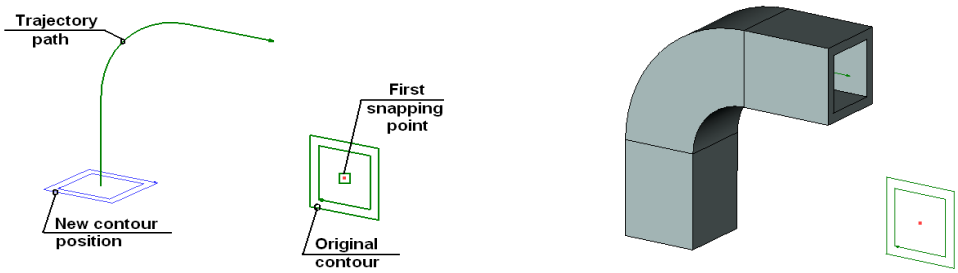
Any object with wire geometry can be used as the trajectory path. Most commonly, the trajectory is defined using a 3D path. However, one can also use 3D profiles, a chain of edges, loops, etc.

Two additional guides are necessary for defining particular twisting and scaling laws for intermediate sections.

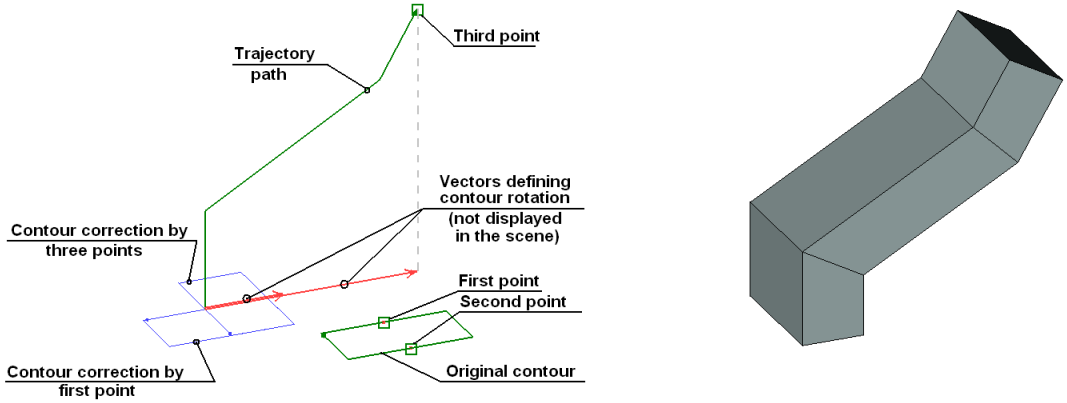
Correction of Contour Original Position

The original contour position can be additionally corrected by selecting one or three additional 3D points.

One point defines the contour snapping to the start of the path. Contour orientation in this case depends on the selected orientation method. If the option “Profile is perpendicular to path” is set, the plane of the flat profile is reoriented orthogonal to the path at the start point. When using non-flat contours, the system automatically calculates some “mean normal” that is used for orienting the contour “plane”. If the option “Profile is parallel to source profile” is set, the contour is moved to the start point of the trajectory path by parallel translation.



In some cases, one point may not be enough for correcting the original contour position since it does not fix the contour rotation about its normal. One can select two additional points in order to specify additional contour rotation about the normal. The second selected point, together with the first one, will determine the original orientation vector (similar to a 2D fragment fixing vector). The third selected point, together with the start of the trajectory, will define the target orientation vector. The projection of this vector on the "plane of the contour" will determine the angle of additional rotation of the contour. If a non-flat contour is used, the "plane of the contour" is determined by the calculated "mean normal".



This drawing is located in the library *"Examples"*, in the folder *"3D Modeling\Sweep\Drawing 3.grb"*.

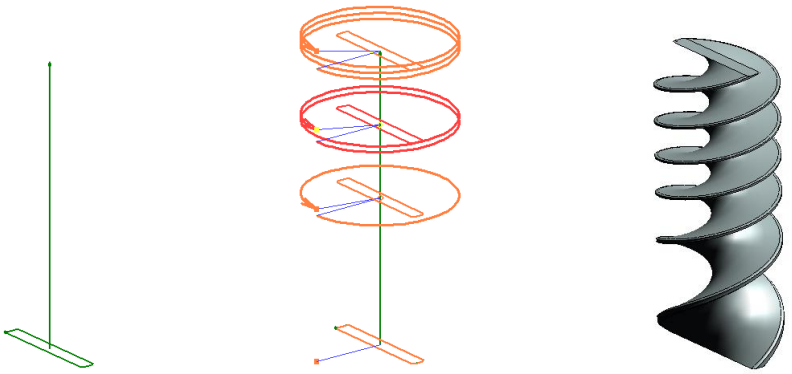
When using non-flat contours, even three-point contour positioning does not guarantee the expected result. In such a case, we recommend setting up the desired contour orientation before starting the operation "Sweep". For example, one can create the contour in the desired position from the beginning or, if that is not feasible, use coordinate system-based copying.

Contour Twist

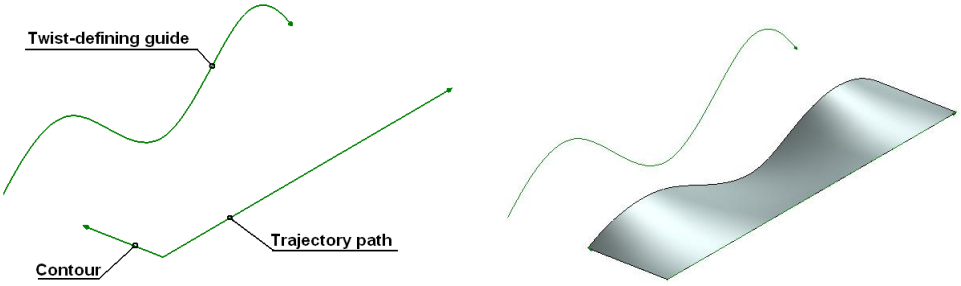
One can define additional contour twist along the sweeping trajectory path. The rotation angle of the intermediate sections can be defined by table or by using additional geometry – a guide path or surface. This approach allows quite different ways of calculating the contour rotation angle.

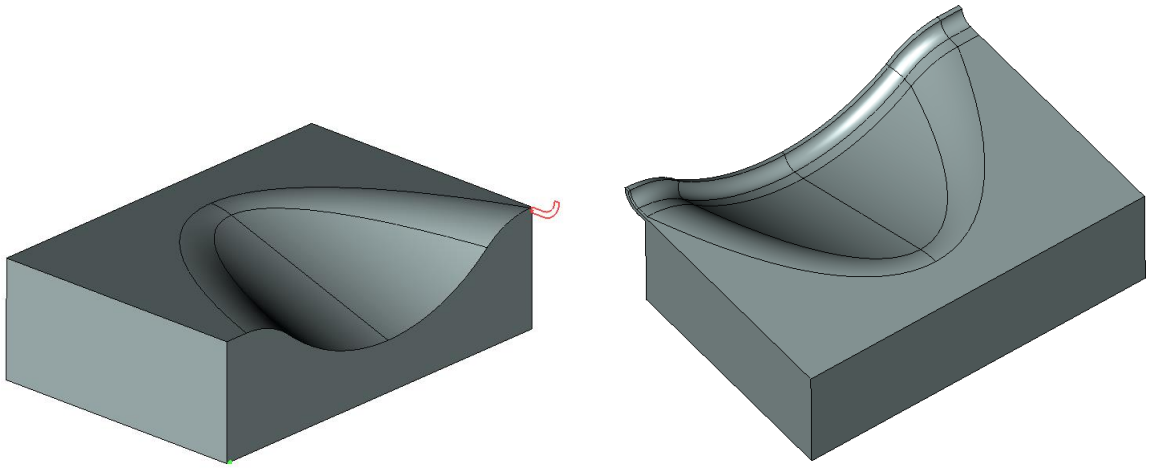
When using a table for defining twist, rotation by the specified angle of each intermediate section is done about the Z-axis of the calculated coordinate system. The table appears as a set of trajectory points, whose position is defined in percent of the trajectory length, with a fixed value of rotation angle entered against each point. When setting up the table of rotation angles, one can use special draggers. This functionality is convenient, for example, for creating sweeps with the surface twisted as a spiral.

Point on trajectory	Position	Angle
Start	0%	0
Position_1	50%	360
Position_2	75%	720
End	100%	1080



The choice of a path or surface is determined by the specifics of a particular task. For example, if the contour is expected to trace a specified curve throughout sweeping, use a guiding path. Alternatively, if one needs to create an auxiliary surface touching another surface, staying tangential to that other one, then a surface can be selected for the swept contour orientation. In this way, the tangency condition will be maintained between any intermediate section and the surface.





When creating bodies using contour twisting (defined by a guide or surface), make sure of the correct original position of the contour. Thus, in the first example the contour must be originally directed towards the start point of the additional guide. In the second example, the contour shall make tangential transition to the adjacent surface in the original position.

The system uses an additional guide or surface for calculating auxiliary vectors defining a rotation angle for each intermediate section. The rotation angle is calculated as the angle between the directions of the auxiliary vector at the start point of the trajectory and at the current point, for each intermediate section. If the calculated auxiliary vector does not lie in the X-Y plane of the coordinate system of an intermediate section, its projection on that plane will be used.

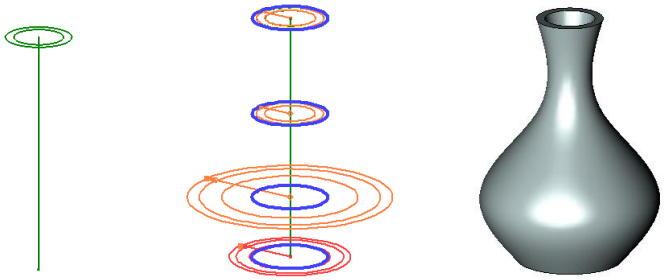
When a surface is used for calculating the auxiliary vector, its direction is calculated in a general case as follows. A point is identified on the surface that is nearest to the origin of the coordinate system of an intermediate section. The sought direction is a tangency direction constructed at this point. Therefore, best results are achieved when the used surface is closest to the trajectory. In this case, the trajectory can be defined by using, for example, edges of the surface.

If a guide is used for calculating rotation angles of intermediate sections, you can select one of the four methods for calculating the auxiliary vector. The methods for calculating auxiliary vectors by additional guides are described in a separate topic (see below).

Contour Scaling

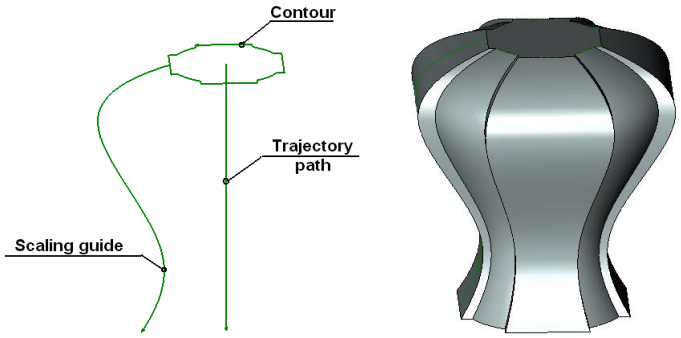
When sweeping the contour, one can define an additional law for scaling the contour with respect to the origin point of a section coordinate system or any other point in the space, related with the section coordinate system and moving together with it. The scale factor can be specified in a table or by using an additional guide.

The table appears as a set of trajectory points, with a fixed value of the scale factor entered against each point. In the rest of the points along the trajectory path, the value of the scale factor is interpolated by the system automatically.



Point on trajectory	Position	Scale
Start	0%	0
Position_1	50%	0
Position_2	75%	3
End	100%	2

When using an additional scaling guide, the scale factor of an intermediate section is calculated as a ratio of the vector length in the current point of the trajectory to that in the first section. The methods for calculating auxiliary vectors by additional guides are described in a separate topic (see below).

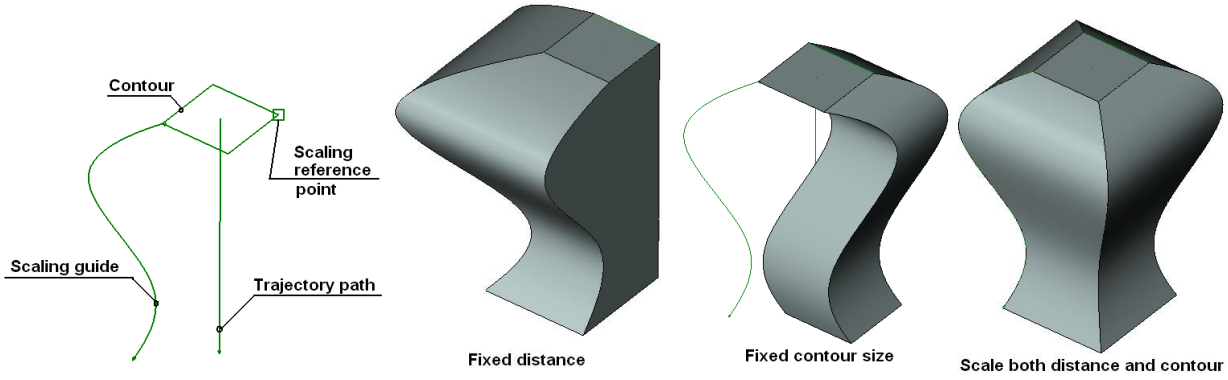


When using a scaling reference point, the way of scaling the contour with respect to that point can be one of the following:

Scale the distance from the origin on the trajectory to the reference point, keep the contour snapped to the reference point, but don't scale the contour;

Scale the contour only; don't change the distance between the reference point in the trajectory;

Scale both the contour and the distance to the reference point.



Sweep Creation by Guides

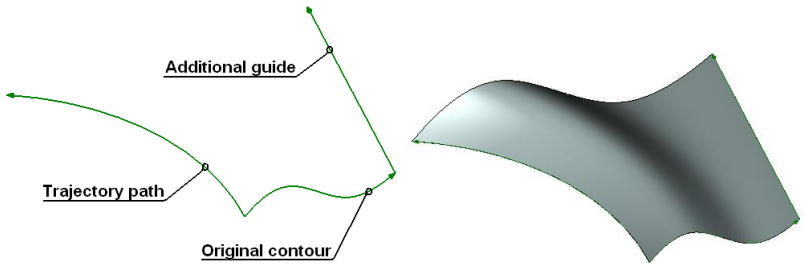
Creating a body by guides requires a contour, a trajectory and at least one additional guide. The first additional guide is used for calculating the first auxiliary vector for each intermediate section. The

intermediate points for defining auxiliary vectors on the trajectory and additional guides are always calculated "by parameter". This vector is simultaneously used for scaling an intermediate section and for orienting it (by calculating the direction of one of the axes of the intermediate section coordinate system). The direction of the second axis of the intermediate section coordinate system can be calculated automatically (as the cross product of the calculated auxiliary vector and the tangent to the trajectory at the origin point of the intermediate section coordinate system).

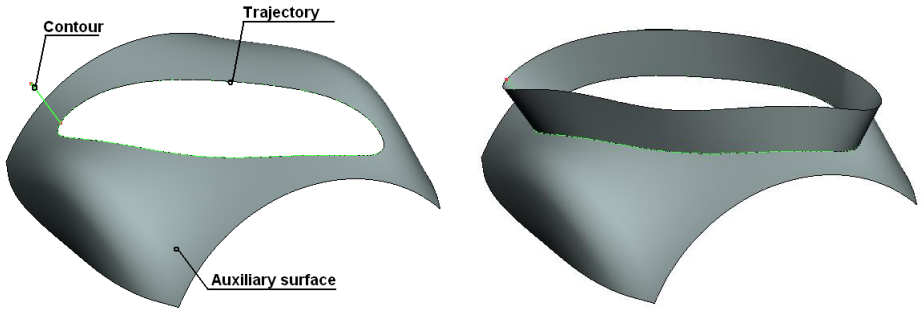
To define an intermediate section orientation other then offered by the system, one could use a second additional guide. That will help the system calculate the second auxiliary vector for each intermediate section. The two auxiliary vectors will define the X-Y plane of the intermediate section coordinate system. The Y-axis of the intermediate section coordinate system will be oriented in the direction closest to that of the second auxiliary vector.

The approach to solid body creation by using an additional guide or guides serves certain specific goals, such as closing a gap between two curves by a surface obtained by sweeping a specified contour. Creating this kind of bodies assumes that the original contour will rest at one of its ends on the start point of the trajectory path, and at the other end - on the start point of the additional guide. In theory, some kind of a result can be obtained by using various orientations of the original contour, however, the result will be rather hard to predict.

If, for example, the original wire contour rests at its ends on the start points of the trajectory and an additional guide, the resulting surface connecting the trajectory and the additional guide will follow the shape of the original contour.

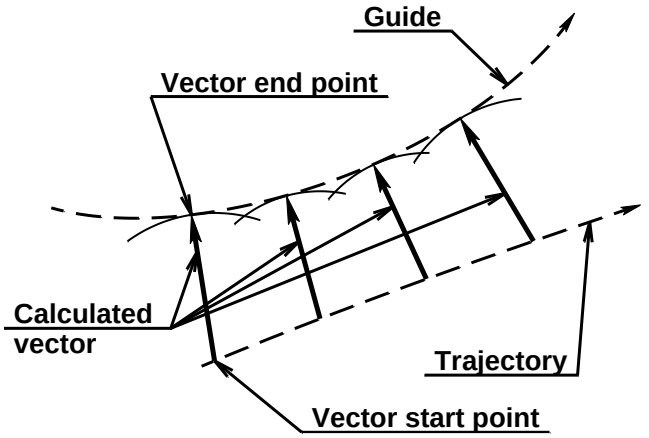


Faces can be used instead of the first and second additional guides. Orientation of the intermediate sections will be defined by auxiliary vectors calculated as perpendiculars to surfaces at points nearest to trajectory points. In practice surfaces adjacent to trajectory are mostly used. The use of surfaces for defining contour twist is applicable when the first two methods of contour orientation ("Perpendicular to the trajectory" and "Parallel to the original profile") cannot be employed. The following example demonstrates sweep operation when the contour-segment is oriented perpendicular to the auxiliary surface in every point of the trajectory.



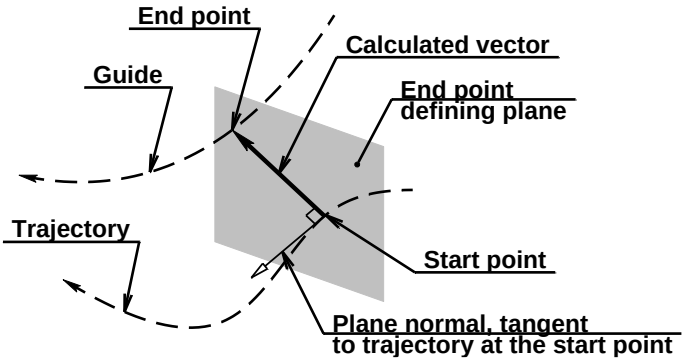
Methods for Calculating Auxiliary Vectors by Guides

Auxiliary vectors are used by the system for orienting coordinate axes of an intermediate section (rotating the section) or for calculating the scale factor for an intermediate section. When using an additional guide, the auxiliary vector calculation is done as follows. The auxiliary vector of each intermediate section starts on the trajectory – at the origin of the coordinate system of the intermediate section. The end point of an auxiliary vector, that defines its direction, is located on the additional guide. Calculation of such end point for defining auxiliary vector can be done by one of the four methods.

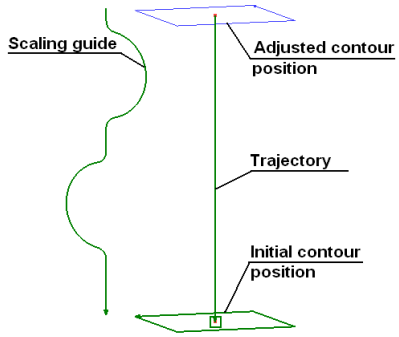


By minimum distance. The end point of the vector will be the point on the guide nearest to the vector start on the trajectory.

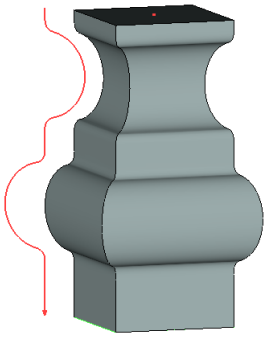
Perpendicular to the path. A plane is constructed through the start point perpendicular to the trajectory. This plane must intersect the guide. The end point is defined as the intersection of the plane and the guide. If no intersection is found between the plane and the guide for some point on the trajectory, an error in operation regeneration is output.



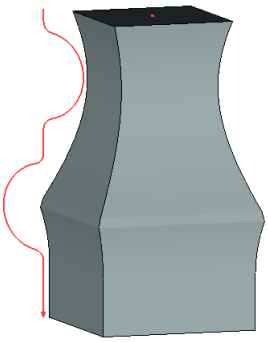
Follows are examples of using options “Perpendicular to the path” and “By minimum distance”:



Defining elements

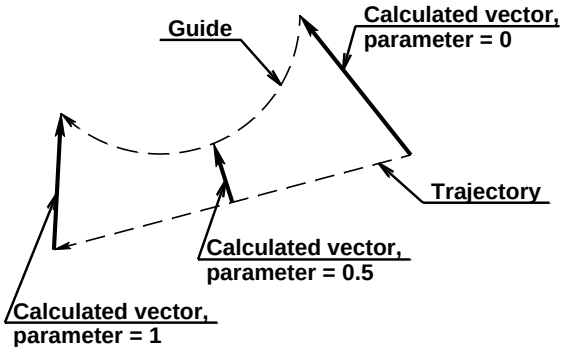


Operation created by the option "Perpendicular to the path"

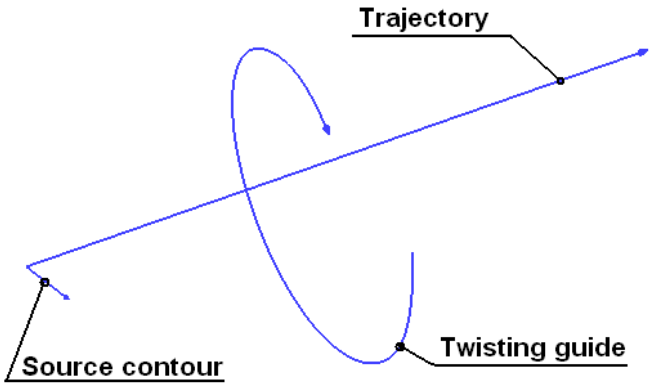


Operation created by the option "By minimum distance"

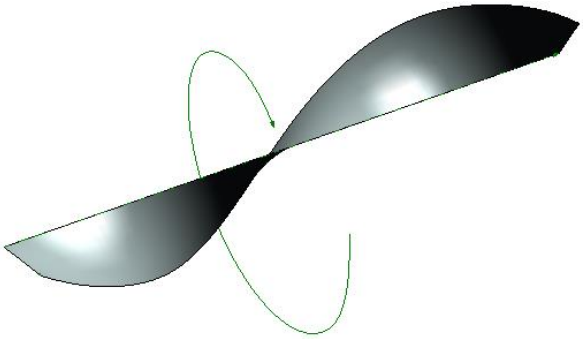
By parameter. Position of any point on a curve can be defined by a parameter – a value defining the point position in percent of the curve length (0 – at the start of the curve, 1 – at the end of the curve). The point on the guide is defined by the same parameter value as the vector start point on the trajectory. This method of calculating an auxiliary vector is least demanding on the orientation and length of the guides.



This option is automatically used when orienting sections "By guides". The user can also turn it on manually when defining the
scaling or twisting law, for example, in situations, when the additional guide is shorter than the trajectory, or other methods are inadmissible. Here is an example of using the option "By parameter":

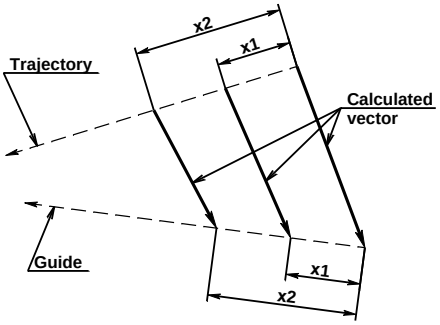


Defining elements



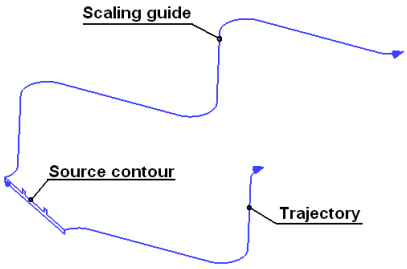
Operation created with option "By parameter"

By offset. First, the length of the trajectory portion is calculated from the start of the curve to the auxiliary vector start. The vector end is then found by the same length on the guide from the origin. In this method, the length of the additional guide must be greater than or equal to the length of the trajectory.



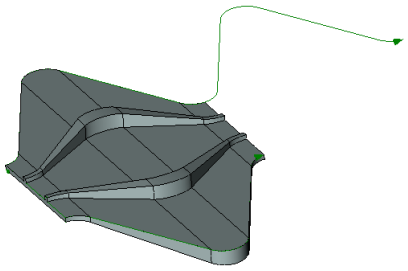
This option can be used when, for example, only a portion of the additional guide shall be used that is equal to the total length of the trajectory.

Follows is an example of using the option "By offset":



Defining elements.

The contour is swept parallel to the original position



Operation created by the option "By offset"

Steps of Operation Creation

To create the operation, use the command **3SW: Create Sweep**:

Icon	Ribbon
	3D Model → Create → Sweep
Keyboard	Textual Menu
<3SW>	Operation > Sweep

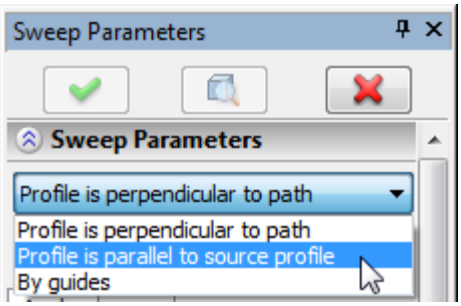
The general algorithm of the operation creation is as follows:

- ✓ Select an option of contour orientation.
- ✓ Select a contour.
- ✓ Select elements defining the path.
- ✓ Define correction of the contour original position. (Optional)
- ✓ Defined additional guiding paths or a surface. (Optional)
- ✓ Define operation parameters. (Optional)
- ✓ Confirm operation creation.

Selecting an Option of Contour Orientation

We recommend defining the contour orientation first, since that will determine the set of options offered by the system in the following steps.

Option selection is done in the property window of the operation, in the provided combo box with a pull-down list. The pull-down list contains all three possible ways of orienting the contour – "Profile is perpendicular to path", "Profile is parallel to source profile", "By guides". The first and, apparently, the most often used setting is the system default. Therefore, if the contour is to be oriented perpendicular to the path, the contour orientation option selection can be skipped.



Selection of the first two options produces an automenu with options for selecting trajectories, defining the correction of the original contour position and selecting twist and scale guides. Selecting the option "By guides" leaves in the automenu the options for defining the correction of the original contour position and selecting first and second guides.

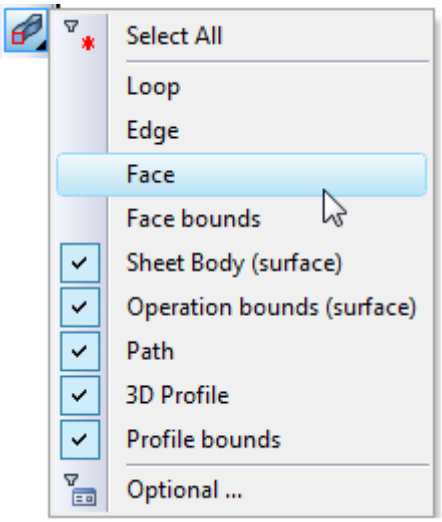
Contour Selection

The first option that turns on automatically in the automenu upon entering the command is the contour selection:

	<R>	Select Contour

The option contains a set of selector filters helping selection of the desired objects. There is a pull-down list behind the icon that appears after prolonged pressing. A contour can be defined by any object with sheet or wire geometry, such as a 3D profile, 3D path, edge, face, surface, etc.

The element selected for defining the contour is highlighted in green (by default).



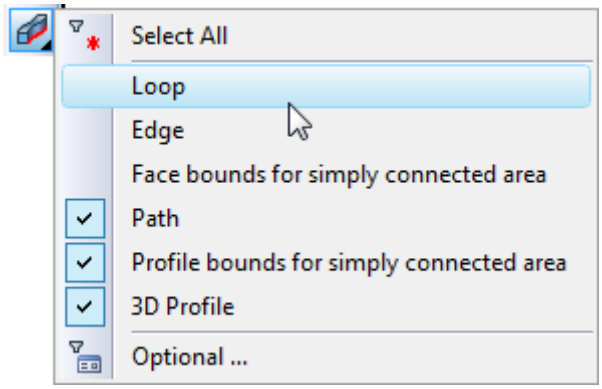
Trajectory Selection

After defining the contour, the system automatically activates the option for the trajectory path selection.

	<T>	Select trajectory

The option also contains a set of selector filters helping selection of the desired objects. In the pull-down list, you can define what objects with wire geometry shall be selectable for defining the trajectory, such as a 3D profile, 3D path, edge, face, surface, etc.

The element selected for defining the trajectory is highlighted in green (by default).



Once a trajectory is selected, the operation confirmation icon becomes accessible. The specified data is already enough to create the operation. The selected contour will be swept from its current position along the trajectory "perpendicular to the path" or "parallel to the source contour".

Defining Correction of the Contour Original Position

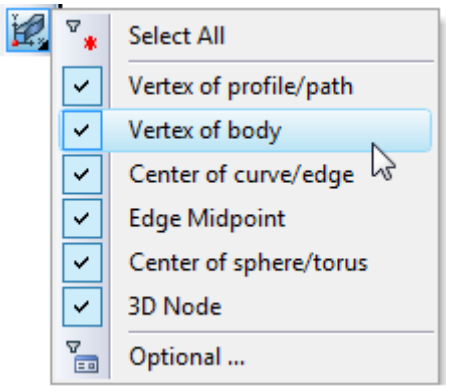
Once the trajectory is selected, the system offers to define correction of the contour original position. If this step is not necessary, you can proceed to further steps by manually selecting the respective automenu options.

The system will first offer selecting the first point of the contour attachment:

	<F>	Select 3D Node as profile starting location

Various suitable objects can be used for defining the point. The option contains a pull-down list for tuning the system for selecting objects of the desired type.

Once the point is selected, the new image of the contour snaps to the start of the trajectory. The image is oriented according to the selected contour orientation option and is snapped to the start of the trajectory at the selected point.



After selecting the first point, proceed to further steps. By default, the system offers more correction options, activating those for defining the second and third points:

	<S>	Select second 3D Node for locating Profile
	<O>	Select 3D Node for locating Profile

After selecting the second point, nothing is updated on the screen. The system waits for the third point selection. After selecting the third point, the image of the corrected contour will be rotated to snap to the third point according to the rules described in "Operation main capabilities".

Specifying Fixing Direction and Contour Orientation Fixing Method

Specifying fixing direction is not a necessary operation and can be omitted.

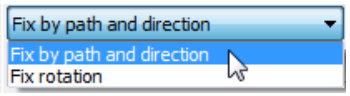
> Fixing direction can be specified only if the contour twist was not defined (in a table form or with the help of guides).

To specify the vector fixing the direction, the following automenu options are used:

	<L>	Select direction
	<1>	Select First Point of Direction
	<2>	Select Second Point of Direction

The option allows the user to define the vector fixing the direction by specifying one 3D element (edge, straight path, etc). Options and are used for specifying the direction vector with help of two 3D points.

The choice of fixing (*fixing by path and direction*, *fixing of rotation*) is selected from the drop-down list in the dialog of the properties window.



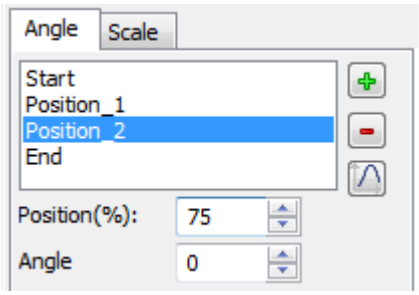
Defining Contour Twist

Contour twist can be defined by a table or by using auxiliary geometry. Let's review all approaches.

Twist definition by table

Twist is defined with the help of the controls on the tab "Angle" of the property window.

To enter new contour twist parameters, press the button . The first time you push it, two special entries are created, “Start” and “End”. Subsequent pushes add new intermediate points along the trajectory. Numerical data for each defining point is entered in the property window. Each point position is defined in percent of the trajectory length. The section rotation angle is entered in the “Angle” box, in degrees. To delete a defining point, select it, and then press the button .



Position of the intermediate points may be specified with the help of a graph. Use button in the properties dialog for this purpose. This button will open the graph editing window. The range of definition by X is 0-100 corresponding to the percentage from the length. The value area is not limited. See more details on operations with graphs in chapter “Graphs”.

To facilitate definition of twist parameters, auxiliary elements are displayed in the scene – draggers and decorations.

A decoration is an image of an intermediate section for each defining point, together with an arrow specifying the direction and the rotation angle. Arrows are positioned along a spiral, showing the full rotation angle. This is convenient when the rotation angle exceeds 360 degrees. Decorations of an inactive point are drawn in the yellow color, while the decorations of the point being currently manipulated – in the red.

The dragger is a pair of blue segments indicating the zero angle and the current angle. There are two handles at the ends of one dragger segment. Dragging those by the mouse with the depressed bottom modifies the value of the rotation angle and the point position on the path. As the pointer approaches a dragger handle, it assumes a specific shape.

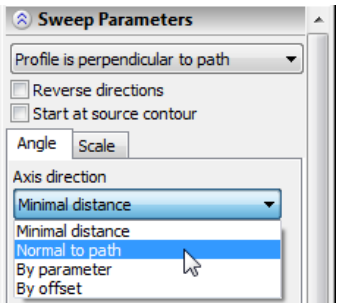
Use of auxiliary geometry for defining twist

A twist-defining guide can be selected by the option:

	<W>	Select guide for defining twist law

Any object with wire geometry can be used as a guide.

After selecting a guide, the dialog in the properties window is automatically switched to the display of the pull-down list from which the method of calculation of rotation vector can be selected. You can choose from four options: “Minimal distance”, “Normal to path”, “By parameter”, “By offset”.



You can select a twist-defining surface by the option:

	<N>	Select surface for twist law

The surface can be defined by a face, a sheet 3D profile or a surface operation. When using auxiliary geometry, draggers and decorations are not available.

Defining Contour Scaling

Contour scaling can be defined by a table or by using auxiliary geometry. Let's review all approaches.

Scale definition by table

Scaling is defined with the help of the controls on the tab “Scale” of the property window. The way of defining parameters is the same as that of defining the twist by table.

The scale law as well as the twist law can be defined with a graph. Use button in the properties dialog for this purpose. The range of definition by X is 0-100 corresponding to the percentage from the length. The function values must be positive only. See more details on operations with graphs in chapter “Graphs”.

Additionally, the property window has a pull-down list for adjusting parameters of scaling with respect to the attachment point.

When defining the scaling law by table, draggers and decorations are also displayed in the scene.

A decoration is an image of an intermediate section for each defining point. Decorations of an inactive point are drawn in the yellow color, while the decorations of the point being currently manipulated – in the red. Additionally, two circles are drawn – a blue one, and a red or yellow one (depending on whether the given point is active). The circles help quickly evaluate the amount of scaling. The size of the blue circle is maintained constant, while the other circle changes according to the scale factor. A provided dragger is a red or yellow arrow along the radius of the second circle. Two handles at the ends of the arrow allow changing the position of the point and the value of the scale factor.

Selection of scaling guide and reference point

A scale-defining guide can be selected by the option:

	<A>	Select guide for defining scale law

Any object with wire geometry can be used as a guide. The system can be tuned for selecting a certain type of objects by using the pull-down list behind the option. The selected object will be highlighted in

green (by default). Once the guide is selected, a combo box in the property window becomes available for selecting a scale vector defining option.

The scaling reference point can be defined by the option:

	<V>	Select scale point

You can select any objects suitable for defining a 3D point.

Once the reference point is selected, a combo box becomes accessible in the property window for selecting an option of scaling with respect to the point. You can choose from the three items – “Maintain size”, “Maintain distance”, “Change size and distance”.

Sweep Definition by Guides

The difference of the “By guides” approach from “Perpendicular to path” and “Parallel to source contour” is that you select a single guide for defining both twisting and scaling. In this way, you can’t select the method of vector calculation or the scaling point. Instead, you define the correction of the original profile orientation by one or three points. The correction makes the system work in the same way as under the option “Perpendicular to path”.

The first guide is selected by the option activated automatically upon choosing “By guides” type:

	<W>	Select guide for defining scale and twist

Any object with wire geometry can be selected. However, the guide must be smooth for successful operation creation. The selected object is highlighted in green (by default).

The guide surface that defines the twist law can be selected with option:

	<Ctrl+W>	Select guide surface for defining twist

The surface can be defined by a face, a sheet 3D profile or a surface operation. When using auxiliary geometry, draggers and decorations are not available.

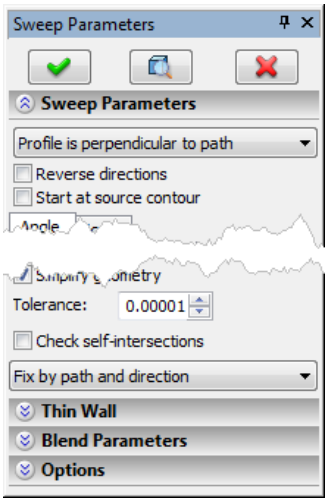
If additional refinement of the intermediate section orientation is necessary, you can select a second guide, using the option:

	<A>	Select guide for defining orientation

Contour orientation by means of the guide surface can be defined with option:

	<Ctrl+A>	Select guide surface for defining orientation

Additional Operation Capabilities



Reverse Directions. When turning on this flag the direction of the trajectory is changed to reverse.

Start at Source Contour. This setting is provided for the cases of using a closed trajectory without a well-defined start point. When the flag is turned on, as a start point (the point from which the body will be constructed along the trajectory), the point of the trajectory closest to the contour is automatically selected.

Simplify Geometry. This setting forces creation of analytical surfaces (planes, cylinders, spheres and tori) wherever possible. If the flag is cleared, exact spline surfaces are constructed.

Tolerance. This parameter sets the tolerance for the surface being created – deviation of the resulting surface from the exact solution. The units are meters.

Check Self-intersections. This parameter allows the user to correct certain cases of self-intersections occurring upon operation creation.

"Sweep" operation belongs to the family of kinematic operations. Therefore, when defining this operation, one can use several additional features supported by majority of kinematic operations:

Thin element creation instead of a solid.

Thickening walls of a body being created by the specified amount.

Blending of the side edges at the start or the end face.

More details on those capabilities can be found in the chapter "Extrusion".

Using the created sweep in a Boolean

To create a Boolean operation, do the following:

Turn on the Boolean creation mode by the automenu option:

	<Ctrl+B>	Select original body for Boolean operation

A Boolean is created when the icon is pushed.

Select the type of the operation from the pull-down list under the above option:

	<Ctrl+'+'>	Addition
	<Ctrl+'-'>	Subtraction
	<Ctrl+'*'>	Intersection
	<Ctrl+I>	Smart mode

Operating principles of the Smart mode:

- If the body being created and the existed selected body have intersection of volumes - the type of Boolean operations: subtraction.
- If the body being created lies entirely within the selected body - the type of Boolean operations: subtraction.
- If the body being created touches and selected body touch each other - the type of Boolean operations: addition.
- If conditions set out above are not met, or a selected body lies inside the created, or error occurred determining the type of penetration, then, Boolean type is not defined - the Boolean operation will not be created.

Select the target body for the Boolean (optional in some cases), using the automenu option:

	<Ctrl+T>	Select Target Body for Boolean

If only one body exists in the scene, it is selected automatically. The new body created by the sweep operation becomes the tool body of the Boolean.

Upon confirming operation creation, first the body is created by the sweep operation, and then the specified type Boolean is performed.

CREATE OPERATION USING FORMING BODY

There is a possibility to use solid body as the source object, which is moving along selected trajectory in the **Sweep** operation. The body spins around the predetermined axis during movement. The resulting "trace" model left by the swept body can be used in further, for example, in Boolean operation.

Thus, it is possible to simulate trace that leaves a milling cutter when machining.

The general algorithm of the operation creation is as follows:

- ✓ Select forming body ,
- ✓ Select trajectory ,
- ✓ Select axis direction of forming body ,
- ✓ Select fixing point of axis ,
- ✓ Select fixing direction , point for starting position and used faces of the body (optional).

Operation Options

Use the following option to select a forming body:

	<M>	Select forming body

You may select only solid body as a forming body. You can't select a sheet metal body or a faceted model.

It does not matter where the forming body is located with respect to the trajectory.

> The operation will not be created if overall sizes of the forming body are too large in compare with the trajectory. It can lead to self-intersections and inability of the operation creation.

After the body selection, you need to select a trajectory, along which the body will move, using option:

	<T>	Select trajectory

You can select a planar path or 3D path as a trajectory.

The trajectory may be closed. You should specify a starting position point on closed vertex to create the operation.

The trajectory should lie in a plane orthogonal to the forming body axis, unless a fixing direction controlling the orientation of the forming body is supplied.

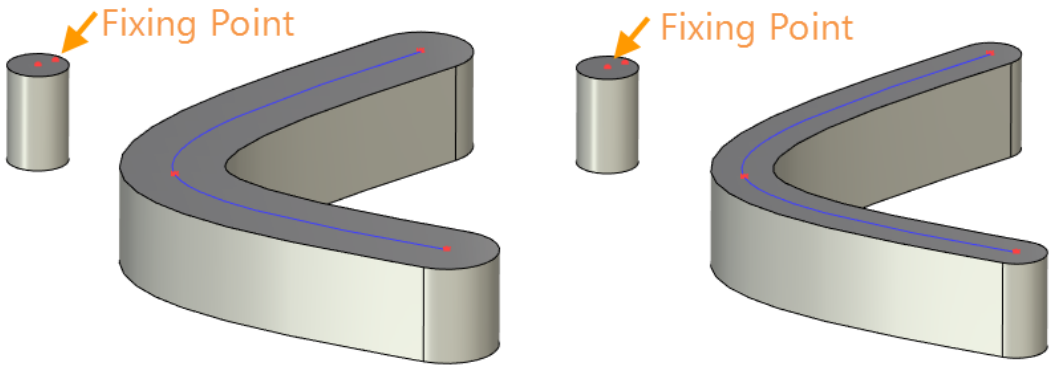
The body will spin while moving along the trajectory. So that you need to select an axis direction using option:

	<A>	Select axis direction of forming body

You should specify an axis fixing point, after selection of the axis.

	<S>	Select fixing point of axis

The fixing point will determine axis location according to the trajectory. The fixing point lies on the trajectory when the body moves along it.



Fixing direction controls the orientation of the forming body according to the trajectory.

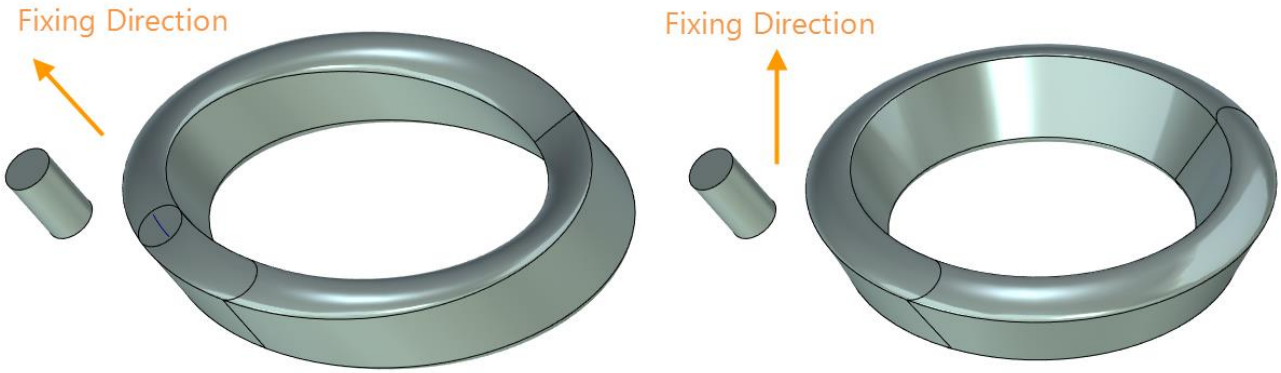
Use the following option to select a fixing direction of the forming body:

	<L>	Select fixing direction

You can also define fixing direction by two points using options:

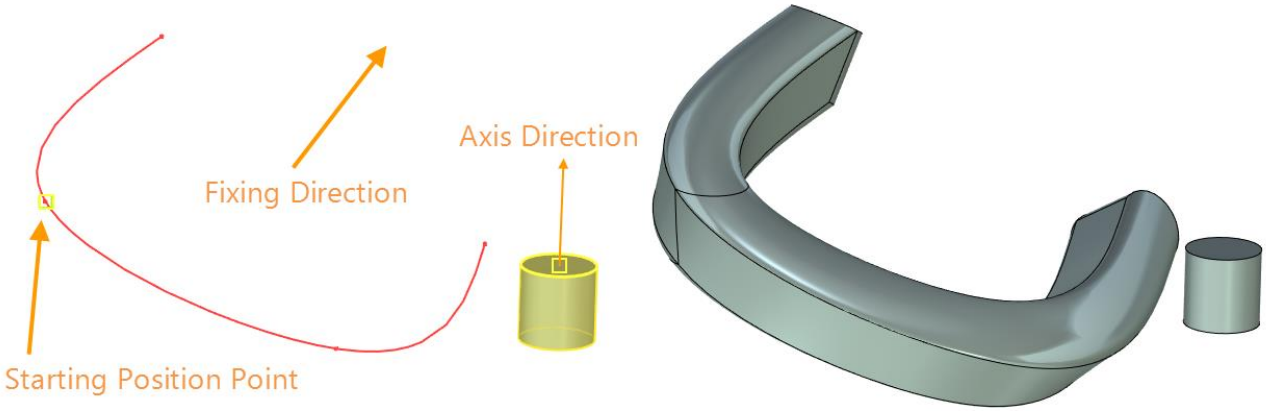
	<1>	Select 1-st point of fixing direction

	<2>	Select 2-nd point of fixing direction



The fixing direction should not have to be in the same direction as the forming body rotation axis.

If the fixing direction is not the same as the direction of forming body axis, you should additionally specify a starting position point.



A 3D trajectory should match the following requirement: all curves of the trajectory should be analytic curves, a C2 non-rational b-curves or a helical curves created by a Parasolid function.

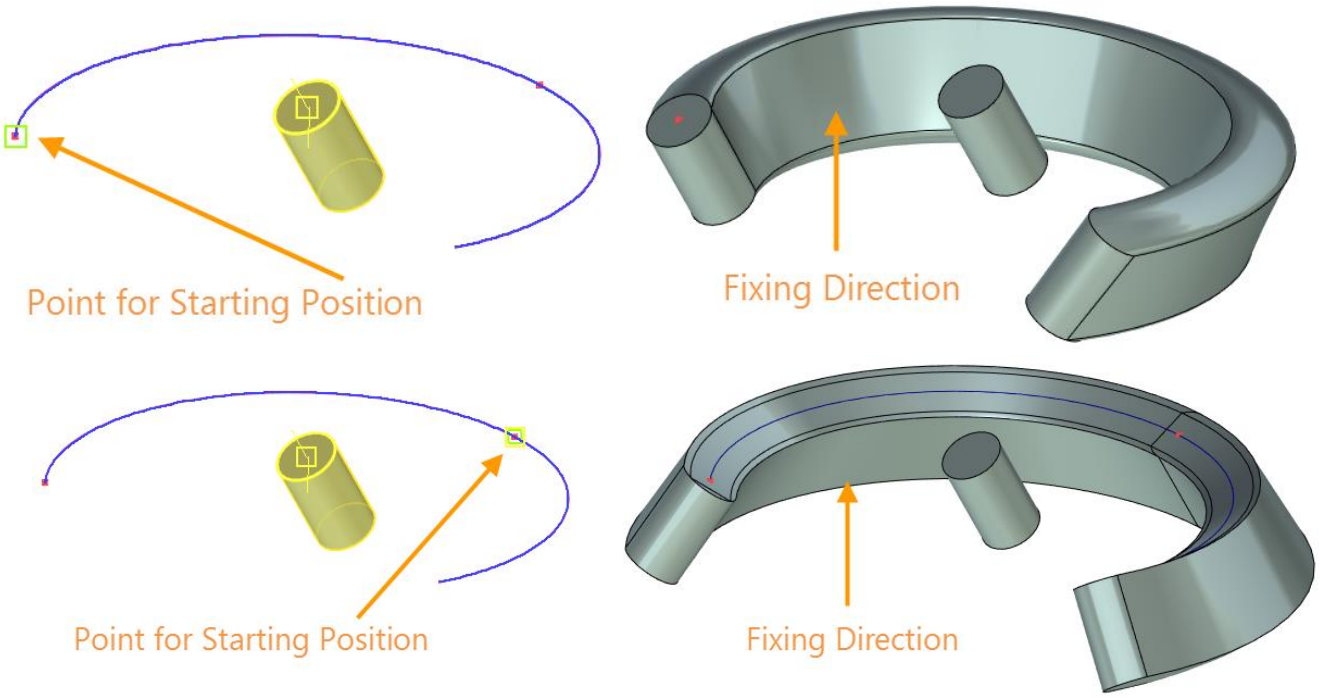
Fixing direction should not coincide at any point with the tangent to the trajectory.

More information about fixing direction can be found in the “Operation main capabilities” section.

To specify a point for starting position of the forming body use option:

	<V>	Select point for starting position

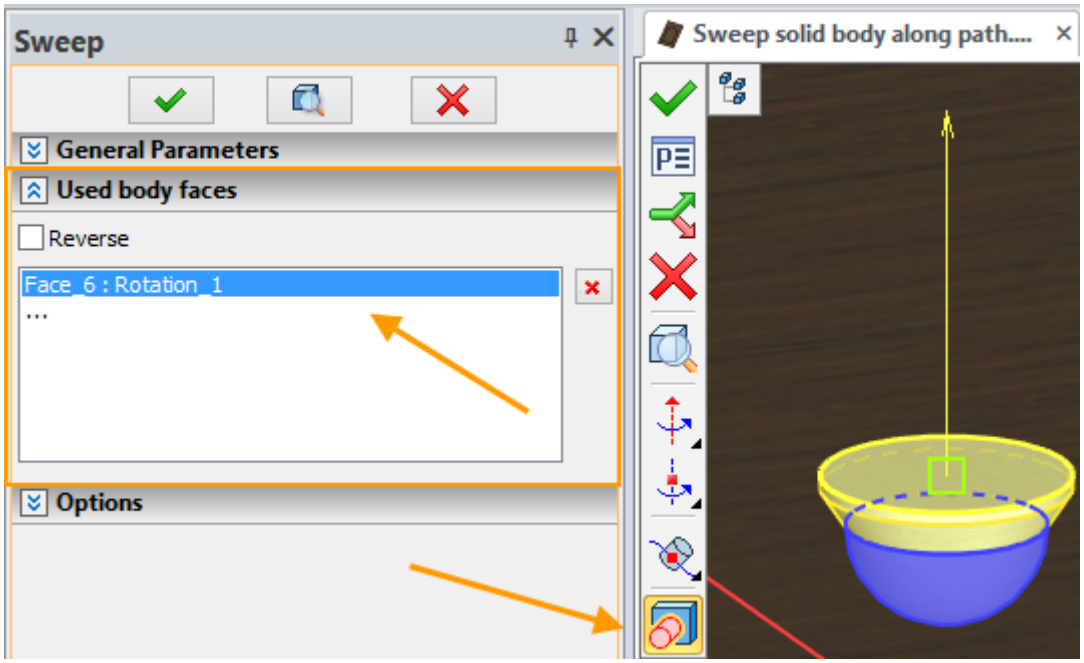
Location of the point on the trajectory affects the overall shape of the resulting body.



You can select body faces of the forming body using option **Select used body faces for simplification**.

	<F>	Select used body faces for simplification

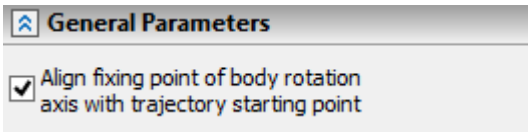
Only selected faces will be used for the operation. If **Reverse** flag is set, the selected faces will not be used for the operation.



The resulting body can be used for Boolean operation.

More information about Boolean operations can be found in "Boolean operation" chapter.

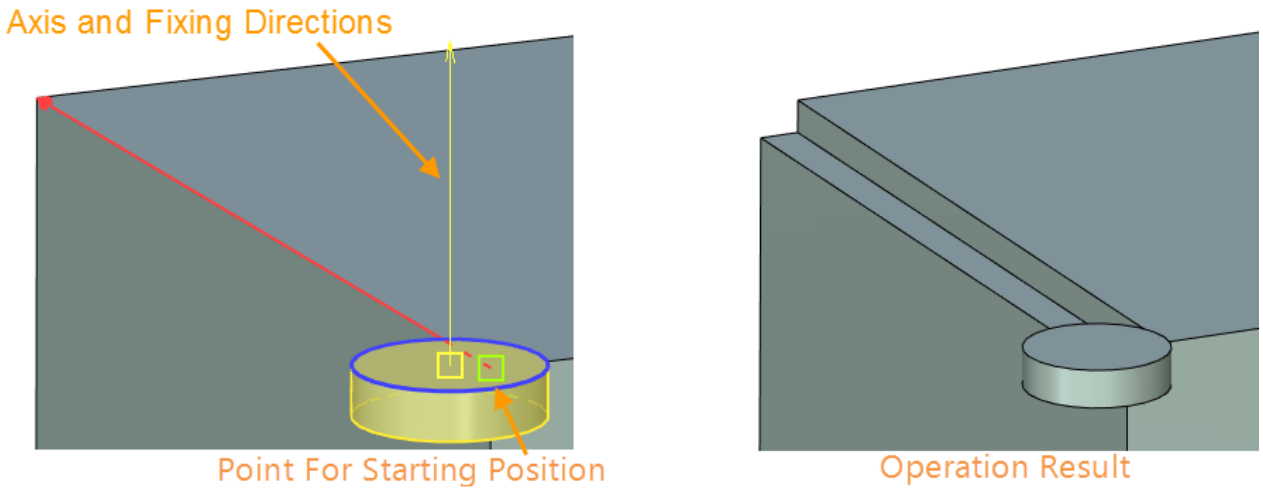
If the forming body should be at some distance from the trajectory, you should disable the **Align fixing point of the body rotation with trajectory starting point** flag.



In this case, after you set all the basic parameters of the operation, it is necessary to choose a fixing direction and a point for starting position.

If the forming body does not lie on the trajectory, the trajectory should be G1 continious.

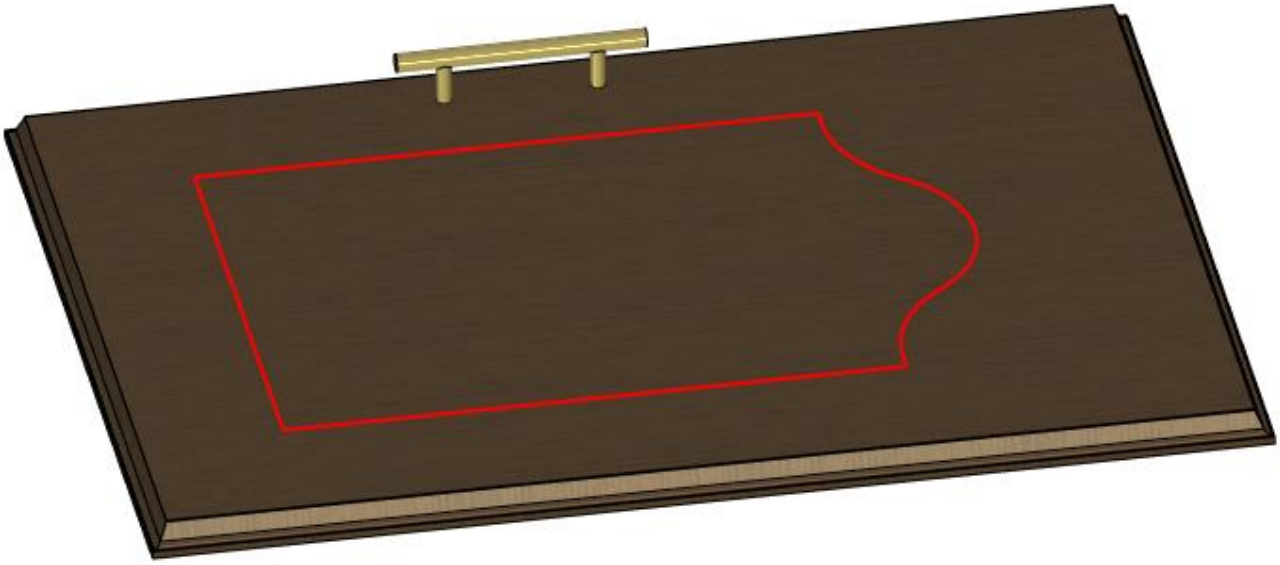
For example, the fixing direction may match the direction of the axis of the forming body rotation and the point for starting position may be chosen at the beginning of the trajectory.



Example of Operation Creation using Solid Body

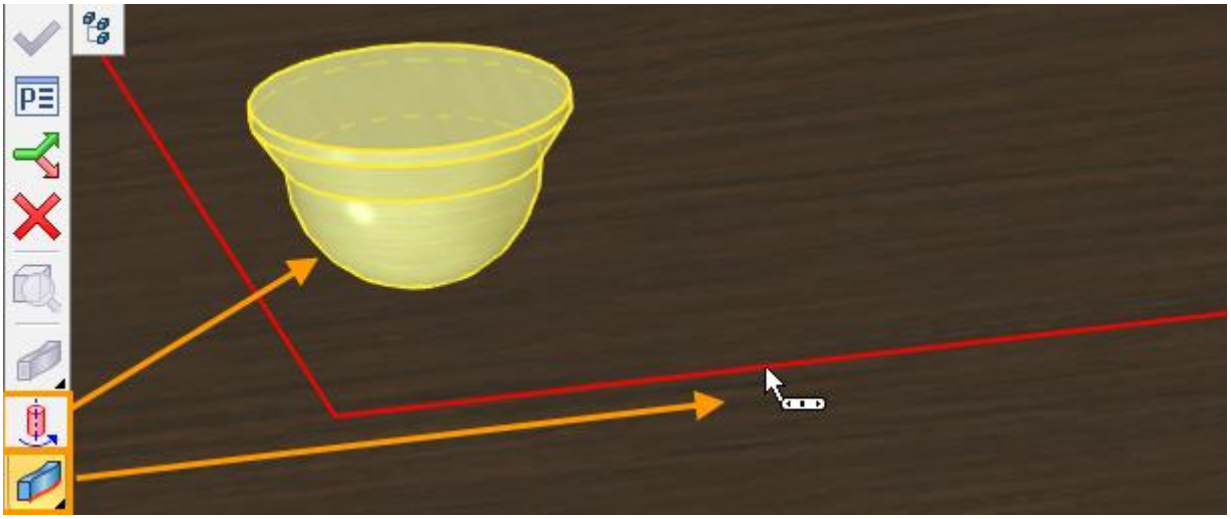
For example, you can model a groove on the door using the operation. For this purpose a trajectory on the treated body and forming body of the desired shape were created:

Trajectory on the treated body:

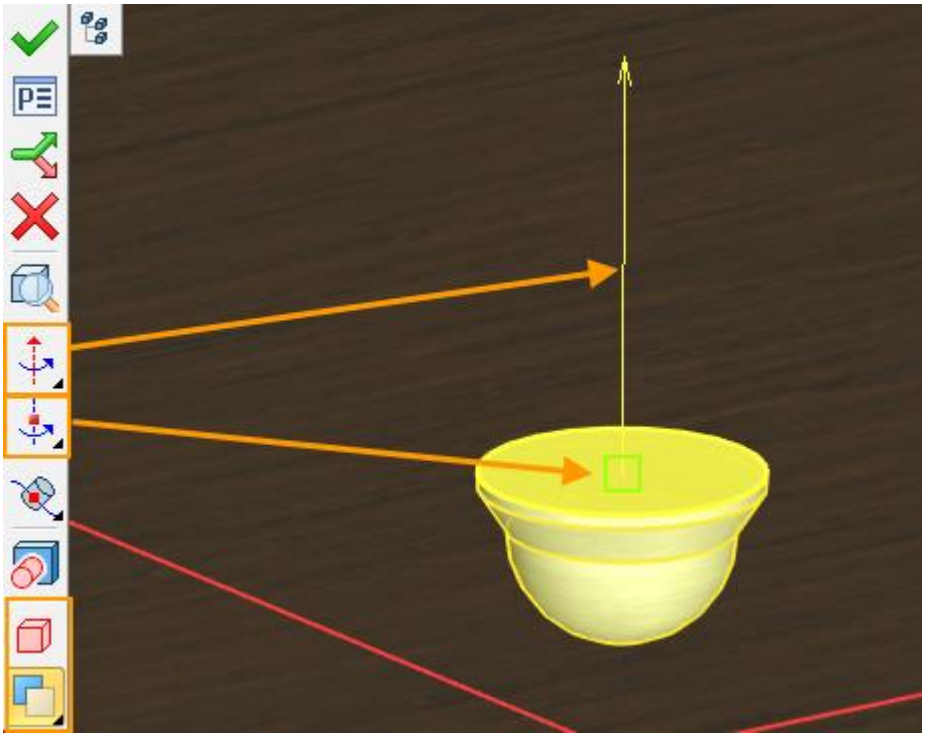


Forming body:

After command activation, you need to select a forming body and a trajectory:



For the forming body you need to define axis of rotation and fixing point of rotation axis .



When the forming body is moved along the predetermined trajectory, it simultaneously rotates around the defined axis, thereby forming the groove. The Boolean operation of **Subtraction** is applied after that. The result is the following:

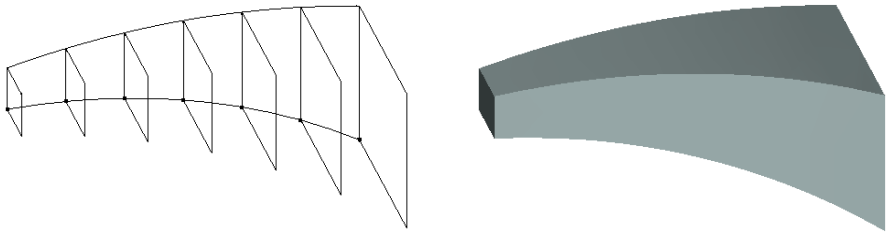
The example can be found in: "*Examples \Release 15 new features\Sweep solid body along path*" file "*Sweep solid body along path.grb*".

PARAMETRIC SWEEP

MAIN CONCEPTS

A parametric sweep is a variation of a common sweep. This operation is used for creating bodies with complex surface geometry. The resulting body is obtained by "sweeping" a profile along an arbitrary or specially defined spatial trajectory. In this case, the system allows defining the variation of the profile parameters as a function of its position.

"Sweeping" of the profile is realized by multiplicating the original profile. The thus created set of profiles shapes up the resulting body. The body can be closed. Depending on the profile geometry type (sheet or wire), the resulting body may be a solid or a sheet.



The driving parameter of the operation is the profile instance number ("copy number"). Numbering of profile instances ("copies") starts from 1 and is automatically incremented by 1 per an instance as the operation creation progresses.

The so-called **current instance coordinate system** is computed for each instance of the profile. Each instance of the original profile is carried over from the original coordinate system to the computed coordinate system of the current instance. The user can select one of the model's local coordinate systems as the original coordinate system. If no local coordinate system is selected, then the global coordinate system is used as the original coordinate system.

The origin and orientation of the coordinate axes of the current instance can be done in two ways. One can specify the rules of varying the coordinate system parameters by an expression. The other way implies the use of a 3D Path or another body surface for positioning the coordinate system of the current instance in space. Each way will be described in details below. The user controls the position and orientation of each profile instance by manipulating the respective coordinate system.

The operation parameters dialog box provides controls over the coordinate system parameters of an instance. The two main types of the operation parameters are - **driven parameters and driving parameters**.

Driven parameters – are those whose values are computed automatically as the operation creation progresses. Independent variables can be used for reading the values of such parameters.

> Independent are those system variables that are assigned a specific value, rather than an expression. Those do not depend on other system variables. On the contrary, some of the

parameters of 2D or 3D constructions may depend on such variables, as well as other driven, or dependent, variables.

A name of an independent variable used with driven parameters is always selected from the list of already existing independent variables in the system.

The parameter **Number of Copy** is always driven. Always use an independent variable to read this parameter for getting the current copy number as the operation creation progresses.

How to use driven parameters?

You can enter the name of an independent model variable in a driven parameter value box to establish a relation between the parameters of positioning an instance in the space and the parameters defining the shape of that profile instance. After computing the position of the coordinate system of the current profile instance, the independent variables assigned to the driven parameters are updated with the respective current values. If an assigned independent variable is somehow involved in defining the profile geometry, then, at each instance creation, the values of the driven parameters will be passed through that variable on to the related model parameters. Thus, the shape of the profile will be recalculated depending on the values of the driven parameters.

The variable that reads the current copy number can be bound to record numbers in a database.

Driving parameters are the fields whose values are defined by the user. An arbitrary expression can be entered in the input box of a driving parameter. Such an expression can include any system variable. For example, you can include the variable that represents the copy number. In this case, you will get a dependency of the parameter on the copy number via that specified expression.

The computations on the way of the operation creation are performed in the following order: the instance counter ("copy number") is incremented; the driving parameters are computed; the driven parameters (if any) are computed; the original profile is recalculated with the current values of the variables; the thus obtained instance is positioned.

Steps of Operation Creation

Follows are the means of calling the command:

Icon	Ribbon
	3D Model → Create → By Parameters
Keyboard	Textual Menu
<3SA>	Operation > By Parameters

The following icons become available in the automenu:

	<R>	Select Contour
	<F>	Cancel contour selection
	<1>	Select Curve defining position of coordinate system

	<2>	Select Curve defining X direction of coordinate system
	<3>	Select Curve defining Y direction of coordinate system
	<4>	Select Surface defining X direction of coordinate system
	<5>	Select Surface defining Y direction of coordinate system
	<6>	Select source LCS
	<7>	Cancel selection of elements defining coordinate system

Depending on the way of defining the coordinate system of the current instance, some parameters may appear as driving or driven. In this case, there are two main approaches to creating a body with parametrically varying profile:

- The approach of free instance orientation
- The approach of orienting instances by paths or surfaces

In each case, different automenu options are used, and a different sequence of steps is involved for defining the necessary elements and operation properties. Follows are the descriptions of each approach presented separately.

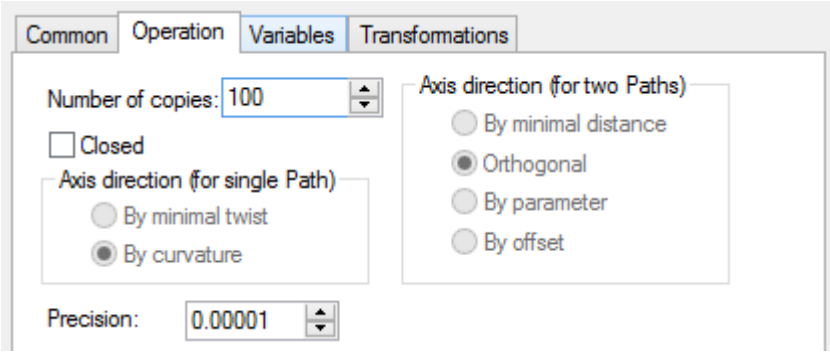
Free Instance Orientation Approach

To create a parametric sweep, simply select a 3D profile, specify the number of instances and assign an independent variable to the parameter “Number of Copy”. Using the variable that follows the instances enumeration, one can create certain dependencies for the driving parameters, as well as for 2D or 3D constructions, which, in turn, define the shape and position of the 3D profile.

One can select the 3D Profile by using the respective option in the automenu:

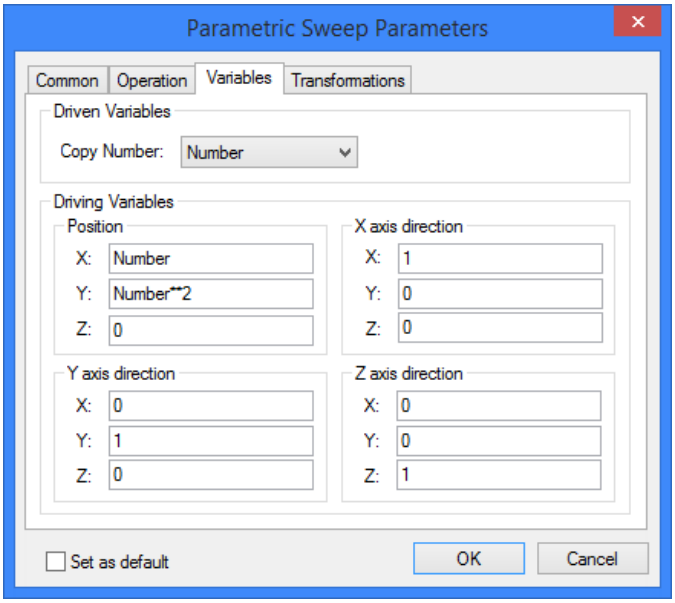
	<R>	Select Contour

The number of profile instances can be specified in the operation parameters dialog box on the tab **Operation**.



Besides the "**Number of Copies**" parameter, this tab also contains an accessible option "**Closing**". With this option on, the body being created from the set of the sections (the set of the instances of the original profile), will be smoothly closed, as if the first section was selected twice.

After defining the number of instances, one can proceed to the tab "**Variables**". This tab has two types of parameters, grouped under the names "**driving parameters**" and "**driven parameters**".



When creating the operation following this approach, the only driven parameter is the copy number, while the driving are the parameters responsible for positioning the current profile instance in space. For convenience, four groups of driving parameters are defined: **Position**, **X axis direction**, **Y axis direction**, **Z axis direction**.

The **Position** group controls the X, Y, Z coordinates of the point defining the origin of the current instance coordinate system.

The rest of parameters define the three vectors used for orienting the coordinate system of the current instance.

You can explicitly describe the position of the current instance coordinate system as a function of the copy number. To do this, define the required dependency in the group of driving parameters (by entering the names of existing variables that directly or indirectly depend on the copy number).

> Each vector defining the direction of a coordinate system axis of an instance is defined by two points. The first point coordinates are <0,0,0>. The coordinates of second points of the direction vectors for the current instance's coordinate system axes are defined by the parameters X, Y, Z from the respective group of driving parameters.

In the case when the vectors defined by the user are not orthogonal or do not make a right-oriented coordinate system, the system refines the definition automatically. Axes definition is done in the order: X, Y, Z. The X-axis direction is left unchanged, the Y-axis is rotated to yield the 90° angle in the XY plane, and then the Z-axis is defined.

In the case when only two vectors are defined (for example, the X and Z-axes), then the order of defining vectors changes, and the third vector (of the Y-axis) is defined last.

Each profile instance is obtained by carrying a copy of the original profile from the global coordinate system to the computed at the given step coordinate system of the current instance.

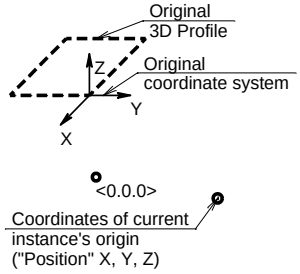
Additionally, one can select a local coordinate system by using the automenu option:

	<6>	Select source LCS

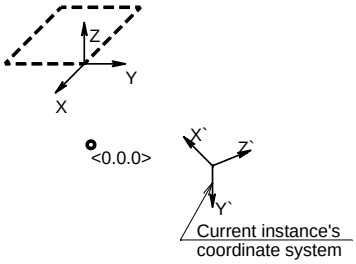
In this case, the selected coordinate system will be used instead of the global one when carrying over the copies of the original profile.

When creating a parametric sweep following the described approach, the system computes the position and orientation of the current instance of the profile in the following order:

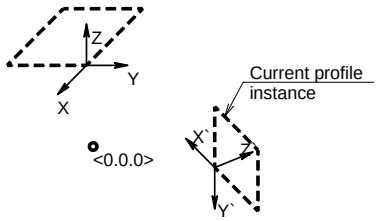
1. The variable reading the copy number is incremented.



2. The absolute coordinates are computed of the origin of the current instance coordinate system. The computation is based on the expressions entered in the driving parameters fields (the group **Position**).



3. The coordinates of the vectors are computed, defining the directions of the current instance's coordinate system axes. The computation is based on the expressions entered in the driving parameters fields (the group **"Direction"**)



4. The coordinate system of the current instance is created in the computed point of the origin, oriented according to the computed direction vectors.

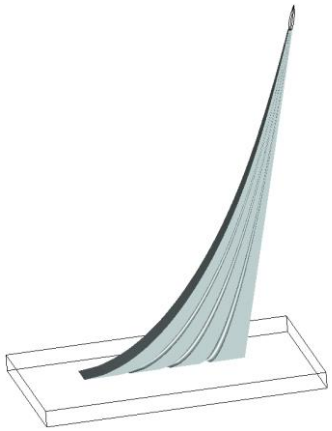
5. If the geometry of the 3D profile depends in some way on the variable that reads the current copy number, then the profile will be recalculated with the current variable value.

6. The original profile's copy is carried over from the original coordinate system to the obtained coordinate system of the current instance. If the original coordinate system was not specifically selected, then the global coordinate system is used as the original when carrying over a copy of the profile.

After computing all instances of the original profile, the result is united in one solid or sheet body using spline surfaces.

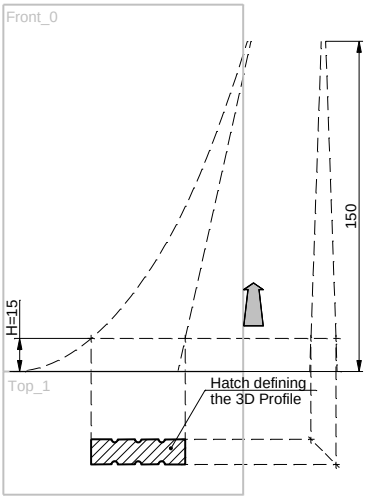
Example 1

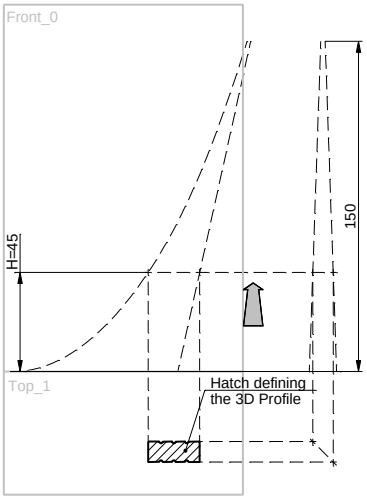
As a first example, let's create a rather simple model (see the diagram on the right hand side). Let's skip the details of creating the auxiliary 2D and 3D construction elements. Instead, let's briefly review the technique itself.



> The file corresponding to this example is located in the library *"Examples"*, the folder *"\3D Modeling\By parameters\ By parameters 3.GRB"*.

This model can be created by sweeping a 3D profile in the space that is indirectly tied to several guides. Let's use four guides in the 2D view, two of which will define the elevation view, and the other two - the left side view. Three guides are straight lines, and one is a spline.





The horizontal construction line is drawn. The position of this line is defined by the expression "H-1". Therefore, it depends on the variable "H".

> Since the **Number of Copy** parameter values sequence begins from one, sometimes a correction (–1) is required in the expressions where the variable **Number of Copy** is entered. In this example, the first instance of the profile should be located at the base, that this at the coordinate Y=0.

Four 2D nodes are constructed at the intersections of the horizontal line and the guides. These nodes can drive the length and width of the profile if attached to vertical lines. A 3D node is created based on one of the 2D nodes in order to fix the profile plane. The grooves on the sides of the profile are constructed in such a way to always divide its side in four equal parts. The grooves radius linearly depends on the length of the profile.

We will need two workplanes – the elevation view and the plan view. The first one is needed for constructing the 3D node, the second - for the profile. The coordinate system origin is located at the model base level (see the diagrams). A straight line is constructed from the origin at the angle of 45° to the horizontal. This line is necessary for establishing the projective relation between the guides on the left side view and the profile hatch on the plan view.

All constructions are interlocked so that varying the variable "H" causes the following chain of modifications: the horizontal line changes its position, that affects the position of the 2D nodes referencing the line. The next link in the chain is modification of the profile hatch geometry and repositioning of the 3D node, to which the 3D profile plane is snapped. Consequently, the 3D profile itself will move in the space.

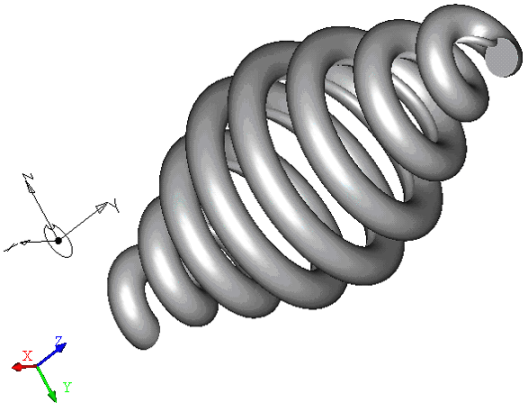
Among the operation parameters, the variable "H" should be assigned the copy number. Specify the number of copies in the **Operation** tab, corresponding to the type of the model, amended as (150+1=151).

It is not necessary to describe the rule for adjusting the instance's coordinate system, since the original profile adjusts its position due to a direct dependency on the copy number. As a result, each instance's coordinate system coincides with the global coordinate system, and, therefore, the current copy coincides with the original profile in the position and geometrical parameters. What matters is how the original profile moves with variations of the variable "H", that is, with respect to the copy number.

The described approach is convenient for using two-dimensional drawing data for constructing a complex model.

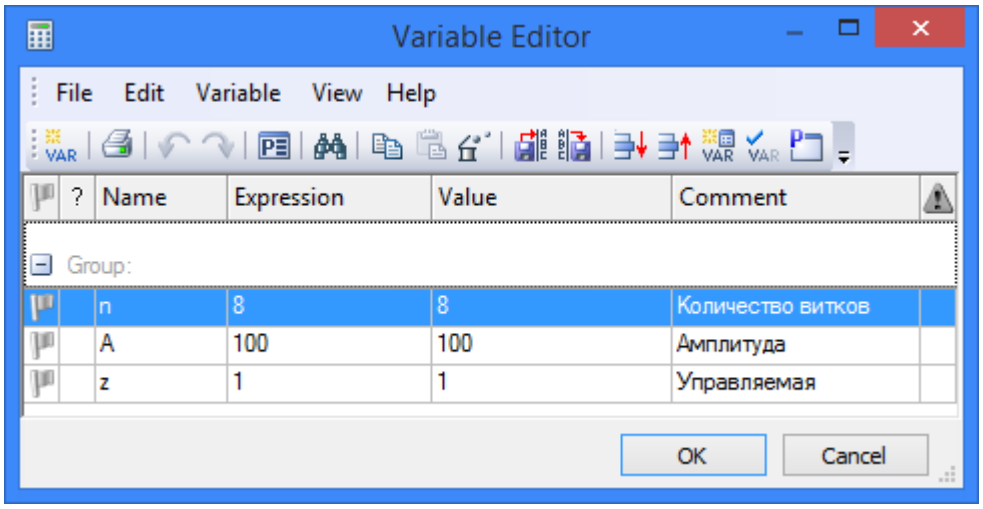
Example 2

The second example demonstrates the approach to the model creation, when the exact rule is defined for the profile movement.

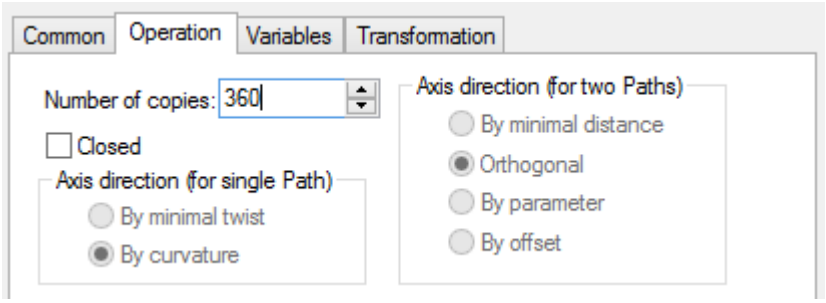


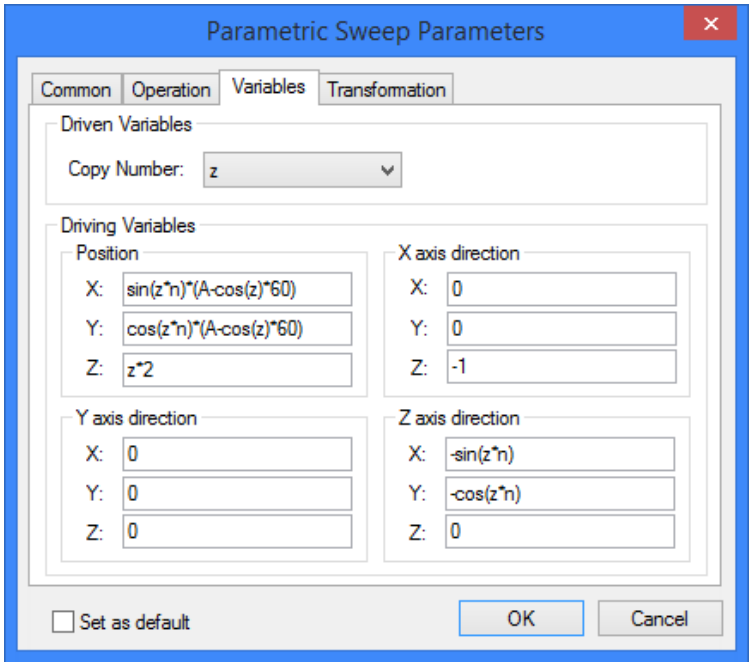
This example is located in the library *"Examples"*, the folder *"\3D Modeling\By parameters\ By parameters 1.GRB"*.

The original data for creating this operation are: a 3D profile; a local coordinate system constructed in the center point of the profile and oriented with respect to the profile as shown in the diagram (the Z-axis lies in the profile plane and is directed towards the groove); several independent variables.



The expressions should be entered in the operation parameters fields, as shown in the following diagrams:





The expressions in the "**Position**" group define the helical trajectory with changing radius. The radius is varied by the cosine function.

The number of copies (profile instances) is equal to 360 due to the following reason: the 360-degree period is used in the expressions of trigonometrical functions.

The diagram shows that the Z-axis of the original profile coordinate system should always be directed towards the center of the spiral (with the groove towards the center), the X-axis – in the opposite direction to the Z-axis of the global coordinate system. The expressions describing these rules are entered in the respective fields of the driving parameters. The Y-axis is defined automatically.

The Approach of Orienting Instances by Paths or Surfaces

Since it is not always possible to describe the rules of the profile instance's parameter modifications by expressions exactly, it is sometimes convenient to use 3D paths or surfaces from other bodies for defining the position and orientation of the current instance's coordinate system.

The origin of the current profile instance's coordinate system is snapped to a 3D path. The position of the origin on the paths can be controlled in two ways.

The X- and Y-axes can be oriented using other 3D paths or surfaces from other bodies.

Selection of a 3D path for snapping the coordinate system origin of the instances is done by the option:

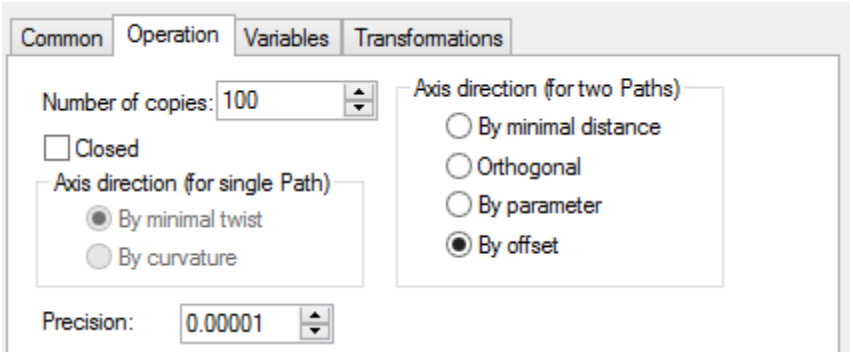
	<1>	Select Curve defining position of coordinate system

Selection of elements for defining the orientation of the X and Y-axes is done by the following automenu options (the Z-axis is defined automatically):

	<1>	Select Curve defining position of coordinate system

	<2>	Select Curve defining X direction of coordinate system
	<3>	Select Curve defining Y direction of coordinate system
	<4>	Select Surface defining X direction of coordinate system
	<5>	Select Surface defining Y direction of coordinate system

When creating the operation using 3D paths, the operation dialog tabs "**Operation**" and "**Variables**" are somewhat different from the described above:



Accessible in this tab are various options for controlling the axes directions. Which group is accessible depends on how many paths were selected altogether: one (for defining the instance's coordinate system origin only) or two (for defining the instance's coordinate system origin and orientation of the X and/or Y-axes).

The group **Axis direction (for single Path)**:

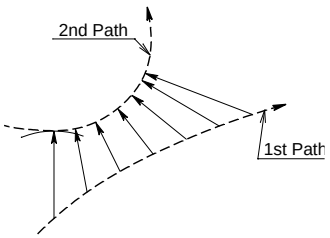
By curvature. The X-axis coincides with the direction of the path's curvature vector. If the 3D path is a straight line, then the axis is oriented along the X-axis of the global coordinate system.

By minimal twist. The X-axis is directed so to provide smooth rotation from the curvature vector direction at the path start to the curvature vector direction at the path end.

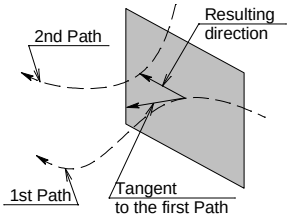
The Z-axis is always directed along the tangent to the path at the point of the coordinate system origin of the current instance.

The Y-axis direction, if not defined, is determined automatically based on the X- and Z-axes.

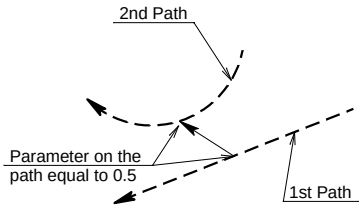
The group **Axis direction (for two Paths)**:



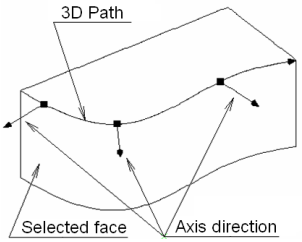
By minimal distance (between the paths at the origin). A vector is constructed from the instance's coordinate system origin on the first path towards the nearest point on the second path.



Orthogonal (to the path). A plane is constructed orthogonal to the first path tangent direction in the coordinate system origin. The axis direction is defined by the intersection point between the plane and the second path. If such point can't be constructed, an error message is output.



By parameter/By offset. An expression entered in one of these boxes is evaluated to define the point on the second path that determines the axis direction.

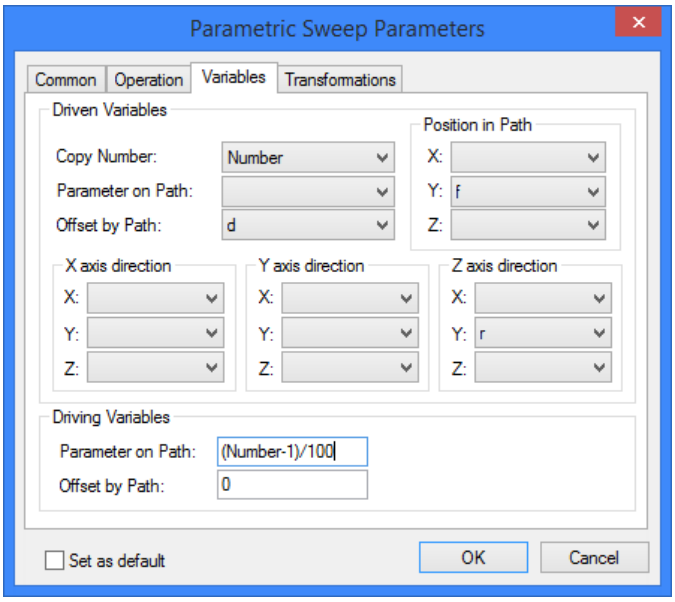


If a face is selected for defining the axis direction, then, in the general case, a point is identified on the face that this nearest to the coordinate system origin located on the path. The normal to the face's surface in this point determines the axis direction. The result will be best when the coordinate system origin lies in the surface. Therefore, use of a face for orienting the axes of the profile instance is recommended in the cases, when the first 3D Path lies on that face.

The thus constructed X- and Y-axes may not be orthogonal (if those are parallel, the system will output an error message). In this case, the Y-axis will be additionally reoriented to become orthogonal to the X-axis in the plane defined by the two axes. The Z-axis will be determined automatically.

If a path or face is specified for the X-axis and not for the Y-axis, then the Z-axis will be directed tangent to the path defining the position of the coordinates origin, while the Y-axis will be determined automatically. If the X and Z-axes are not orthogonal, then the X-axis is additionally reoriented in the way described above.

Let's examine the **Variables** tab:



As shown in the diagram, the group of driven parameters now includes not only the parameter **Number of Copy**, but also the groups **Position in Path**, **X axis direction**, **Y axis direction** and **Z axis direction**. There are now additional parameters **Parameter on Path** and **Offset by Path**.

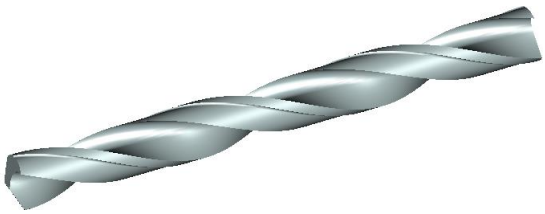
The group of driven parameters **Position in Path** allows reading the absolute coordinates of the current instance's coordinate system origin. The driven parameters from the groups **X/Y/Z axis direction** provide the coordinates of the instance's coordinate system axes. You can use the information about the position and orientation of the current profile instance in the space for defining relations with other model parameters.

The group of driving parameters includes **Parameter on Path** and **Offset by Path**. Parameter on Path and Offset by Path serve the same purpose – to define the point position on a path by various means. Therefore, you can't simultaneously define both the Parameter on Path, and Offset by Path. Parameter on Path varies in the range from 0 (at the start of the path) to 1 (at the end of the path); Offset by Path is measured in the model units and is equal to the length of the portion of the path from the start to the desired point.

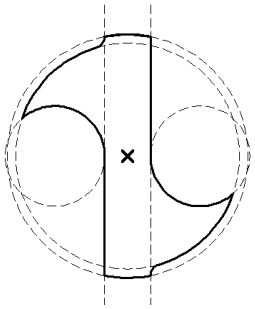
> Once you defined the Parameter on Path driving parameter, you can assign the driven parameter to the Offset by Path only, and vice versa.

Example 3

Let's review a parametric sweep creation using 3D paths on an example of the model of a drill bit's cutting tip. The file with this example is located in the library *"Examples"*, the folder *"\3D Modeling\By Parameters\ By minimal distance.GRB"*.

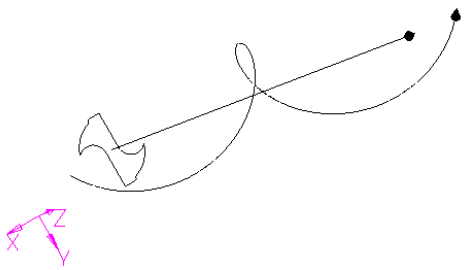


Create a new document using the template with workplanes.



The next step is creation of the 3D profile. For this purpose, activate the workplane "Top view" and draw a contour as shown in the diagram.

For convenience, try to construct the profile in such a way, that its center is located at the global coordinate system origin. In this case, you do not have to construct the local coordinate system for tying instances of the profile to the 3D path during the operation creation.



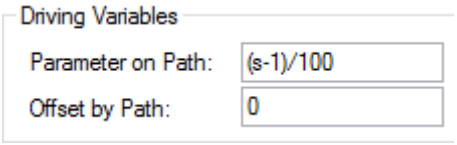
Next, we need to create two 3D paths for positioning the profile in the space when creating the parametric sweep. The path defining the profile position is a straight line coinciding with the axis of the bit. It can be created by two 3D nodes constructed on the axis of the bit. The length of this path defines the length of the cutting portion of the bit. The other path defines the profile orientation in space and is represented by a spiral line. The second path can be easily constructed using a cylindrical worksurface.

In the valuable editor, create an independent variable for storing the value of the copy number. In this example, the created variable was named *s*.

After calling the operation creation command, select the 3D profile, the 3D path defining the profile position and the 3D path defining the direction of the X-axis. On the **Operation** tab of the operation parameters dialog, define the number of copies equal to 101 (with the correction) and select the option **By minimal distance** for the two paths.

On the **Variables** tab, select the variable *s* to assign the copy number. This is the only necessary driven parameter in this example.

Enter the expression in the driving parameter **Parameter on Path** input box, that will define the profile position along the path depending on the copy number. Remember that the Parameter on Path varies in the range from 0 to 1.



ROUTING

GENERAL INFORMATION

T-FLEX CAD provides a set of commands for wire cables, pipelines and ventilation systems design.

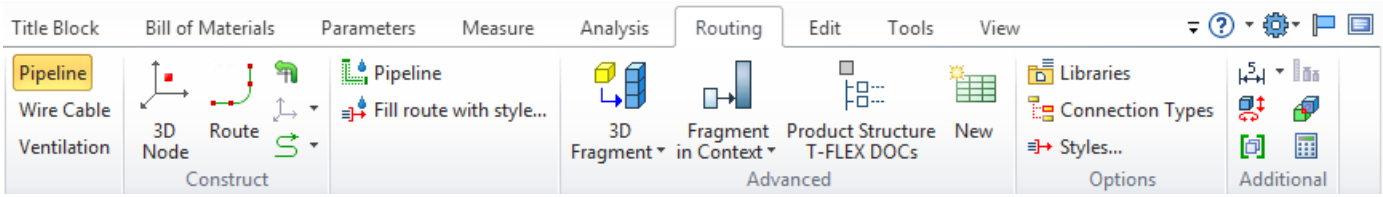
The commands allow to:

- ✓ build space routes manually or automatically;
- ✓ create elements of engineering communications according to certain technical requirements;
- ✓ specify compatibility for routing elements to eliminate design errors;
- ✓ set routing elements styles to specify similar elements connection only.

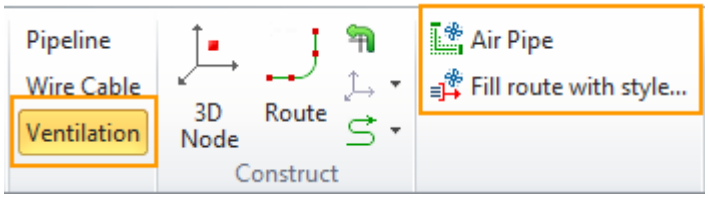
Commands for creating different routing types are divided into the following groups: **Pipeline**, **Electricity** and **Ventilation**. The commands allow to control compatibility of different elements.

You need to activate **Routing** tab to gain access to the commands. Click icon.

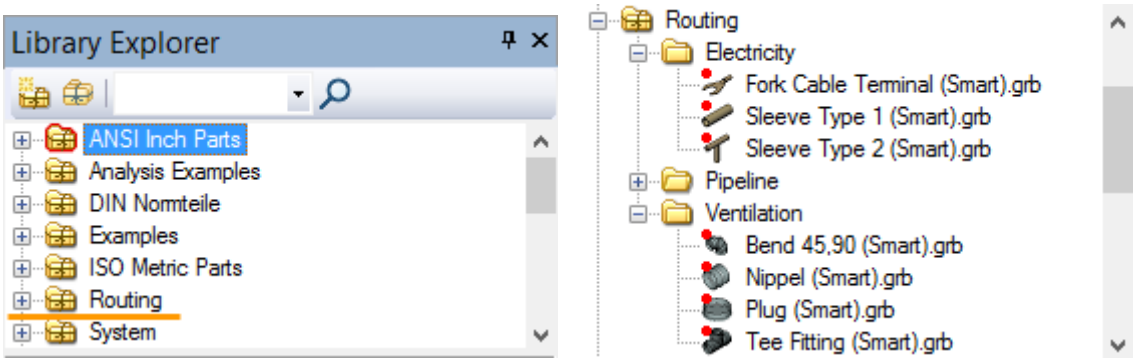
Select **Routing** item in the drop-down list. Here you can find the commands for each routing type and some service commands.



You can select one of the three groups **Pipeline**, **Electricity** or **Ventilation** in the left part. Commands for the selected routing type become available.



Each routing type has its own library of fittings.



Fitting is a specially designed connecting part with the predefined behavior for certain modeling situations such as branching, turns and transitions from one diameter to another. The fitting behavior is specified with parametric tools and macro – API-based small programs built-into model document.

More information about fittings can be found in "Fittings" section.

The routing trajectory is specified in command **3PP: Create Route.**

More information about routings creation can be found in "Routings" section.

Functionality of the Electricity, Pipeline and Ventilation commands of **Routing** group is similar. Only types of allowed connection points, routings and fittings are different for each type.

For example, pipeline parts can be connected only with another pipeline parts. Thus we can avoid mistakes when working with various routings types within one project.

Elements without any specified type can be connected with any routing and fitting type.

More information about creation of routings types can be found in "Connectors types" section.

Use the following command to create routings without any specified type:

Icon	Ribbon
	3D Model → Create → Pipeline
Keyboard	Textual Menu
<3PI>	Operation > Pipeline

The command allows usage of connections and endings from different routings types.

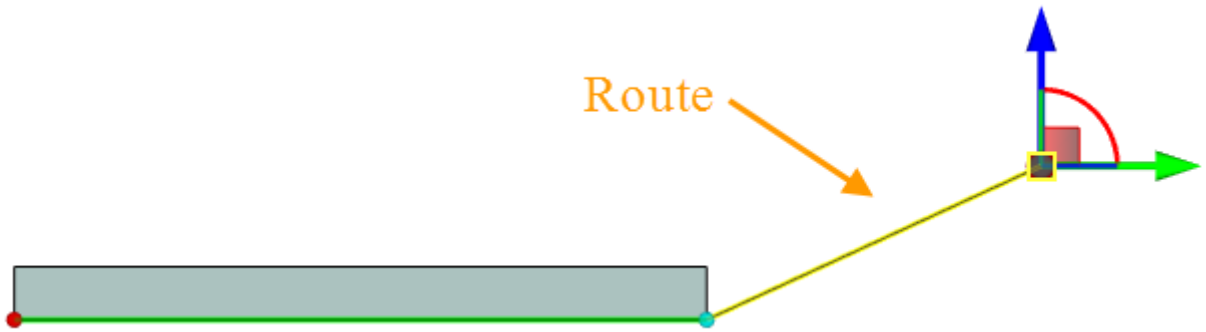
Description of common commands without dividing into different routings types will be given in the following sections.

Pipeline Creation

Using general methods and steps for creating pipeline routing model described below you may create routing designs of other types as well.

To create a pipeline model you may select **Tools** > **Routing** > **Pipeline** > **Pipeline** item from the text menu.

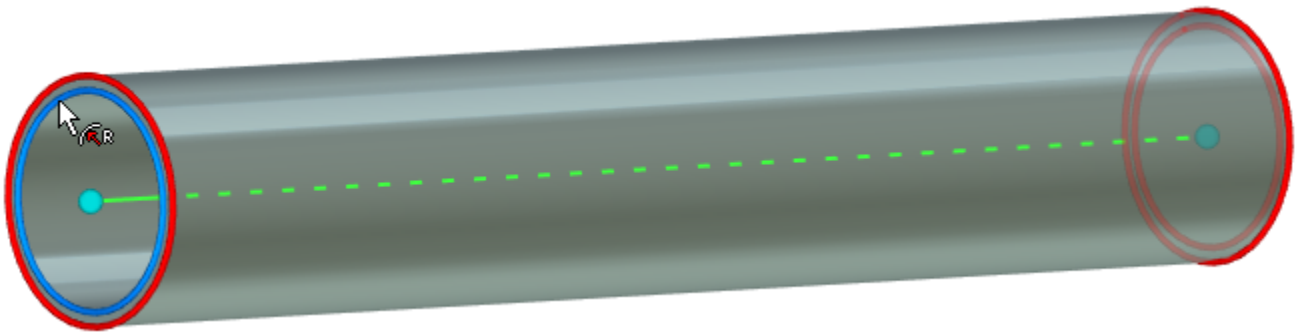
The operation creates a pipe which will be presented as a solid 3D body. The body will be resulted from sweeping of selected contour along a trajectory (route). The route can be created using one of the commands from the **Routing** section: **Laying Route** or **Route**. It can be also specified as a model edge or any other element that determines wire geometry.



The pipe profile shape and dimensions are specified without additional constructions. They are set in the **Properties** window that appears after the command activation.

More information about the pipeline properties can be found in “Pipeline parameters” section.

You can create a pipe along the whole route or its part.



To define the operation, follow the steps:

- ✓ Select 3D curve defining the trajectory.
- ✓ Select starting 3D point of the pipe (optional).
- ✓ Select ending 3D point of the pipe (optional).
- ✓ Define operation parameters (pipe diameter, wall thickness, etc.).
- ✓ Confirm operation creation.

Pipe can be created using one of the two ways:

- In the first approach, the trajectory of the pipe is divided into separate segments – straight lines, circular arcs and splines. A separate body will be constructed for each segment. Depending on the segment type, this can be a cylinder or a torus, or a spline. The resulting operation consists of multiple bodies. If necessary it can be subjected to the **3SD:Divide Solid** operation for getting individual bodies – the pipe segments.
- In the second approach, the trajectory is treated as a single path. The pipe profile will be swept along this trajectory. The result is a single body whose surface consists of a set of spline faces and faces of revolution (cylinder, torus). In some cases, system can simplify the resulting body geometry by replacing (wherever possible) spline surfaces with analytical ones (portions of cylinders or tori).

The first approach is used by default. It is faster than the second and reduces overall regeneration time. However, pipe operation created in this way may behave incorrectly under certain operations (for example, when creating 2D projections or in Boolean operations) because it is made of multiple bodies. If this is your case you may apply the second approach (creating the single body pipe), however, the model regeneration time may increase.

Pipeline Command Automenu

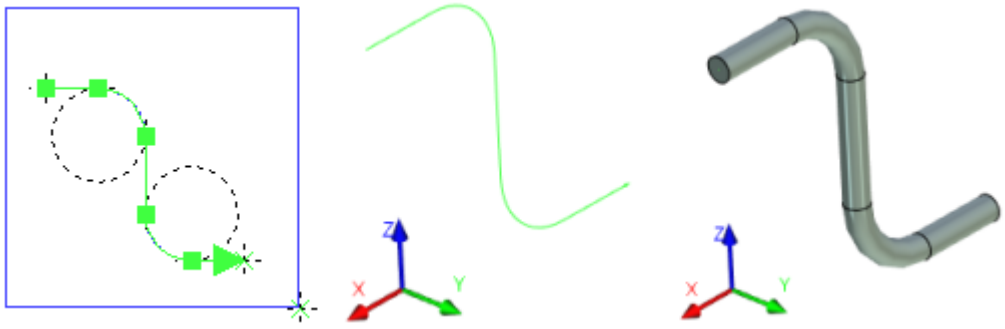
Upon calling the command, the following option is activated in the automenu automatically:

	<T>	Select trajectory

The option allows to define a trajectory (3D curve) for the pipeline being created. When the option is active, you need to select a 3D object that presents wire geometry. The object will define a trajectory for the pipeline.

> If trajectory is presented as a 3D path based on 2D elements, then it can be also selected in the 2D window.

The selected 3D curve and its defining elements will highlight.



You may use a route as a trajectory. If the route does not exist or existing route must be edited, use option:

	<B>	Create/Edit Route

> More information about routes creation can be found in “Route” chapter.

When the route creation is finished press button . You will return to the pipeline creation command.

The pipeline parameters can be specified in the dialog window called via:

	<P>	Set entity parameters

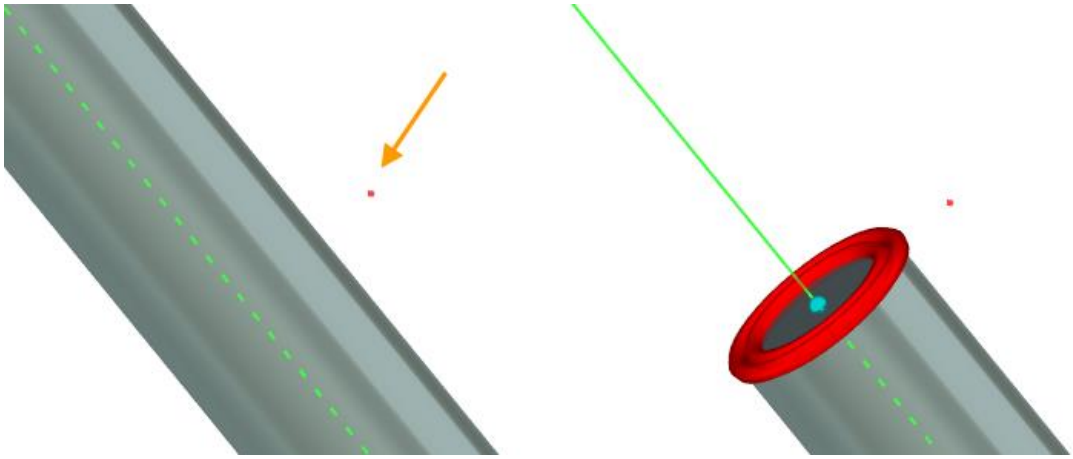
More information about pipeline parameters is given in "Pipeline parameters" section.

You can define additional 3D points that specify pipeline limits – starting and ending points, using options:

	<L>	Select starting point for pipe
	<M>	Select ending point for pipe

The options allow selection of any 3D object that can define a point as a corresponding pipeline end. System creates a "cutting" plane, which is perpendicular to the pipeline routing. This plane becomes a bound of the pipeline.

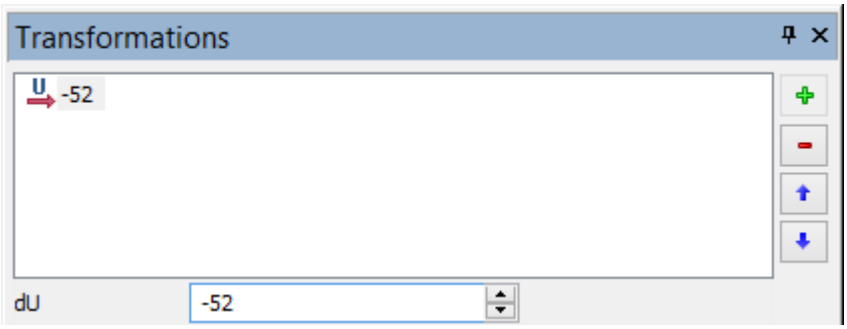
Pipeline ends cannot be located beyond the selected route.



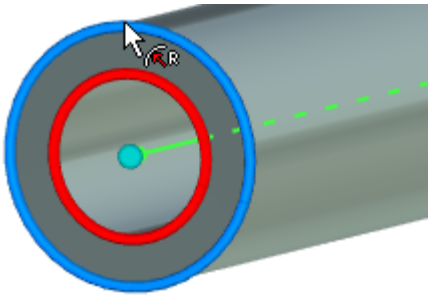
You can edit location of starting and ending points of the pipeline using options:

	<S>	Edit start position
	<E>	Edit end position

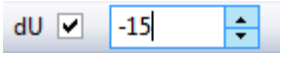
Transformations window appears after selection of one of the options. The current location of bounds is displayed there. The value can be changed manually.



You can also define pipeline limits using manipulators. Point the cursor at the pipeline bound manipulator and move it with pressed . Starting point is specified from the start of the path and has symbol "+". Ending point is specified from the end of the path and has symbol "-".



Current offset value is displayed on the system toolbar.



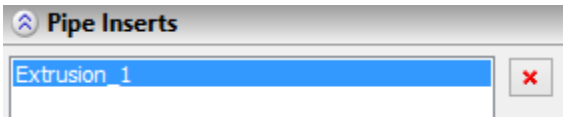
To create a pipe insert, operation that modifies pipe for inserting other parts, use option:

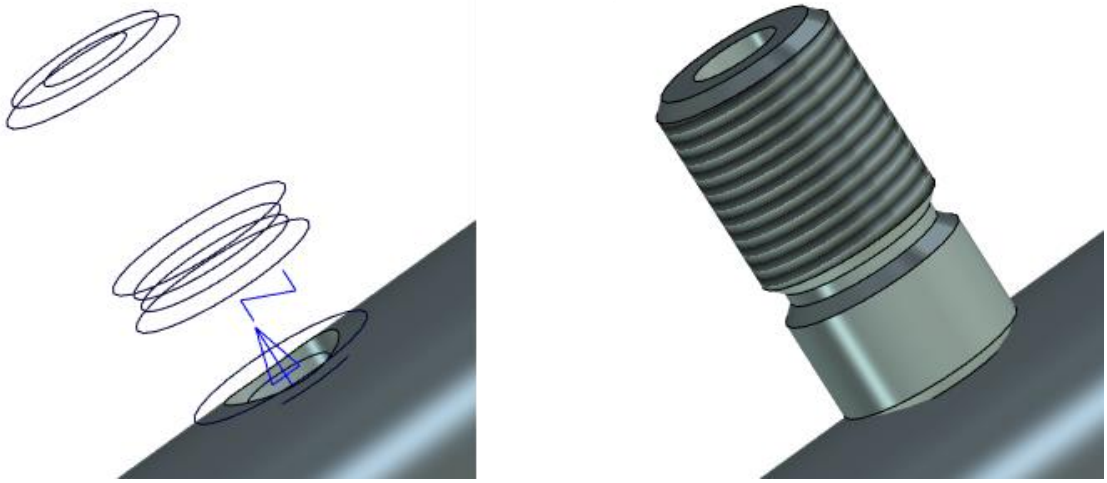
	<O>	Select operation for pipe insert

The option allows you to select body for the pipe insert.

Insert allows system to consider the pipeline as a singular object even if there are fittings that divide it into parts. It is necessary for further pipelines quantity and length correct calculation.

The pipe inserts are displayed in **Pipe inserts** section of the **Properties** window.





You can cancel source path and end points selection via option:

	<K>	Cancel selection of trajectory and end points

You can use the following option for the standards editing:

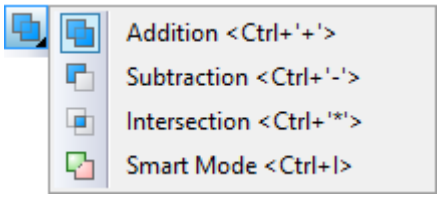
	<G>	Edit standards list

The **Pipeline Standards** window appears after its activation. Parameters of pipeline profile can be specified here according to existing standards.

	<C>	Library Configuration

More information about configurations can be found in "Holes" chapter.

The pipeline creation can be used in combination with the further Boolean operation. For this, you need to select first the type of Boolean operation.



Then select a body using option:

	<Ctrl+T>	Select target body for Boolean

More information about Boolean operations can be found in "Boolean Operation" chapter.

PROPERTIES WINDOW

You can specify the following properties in **Parameters** window:

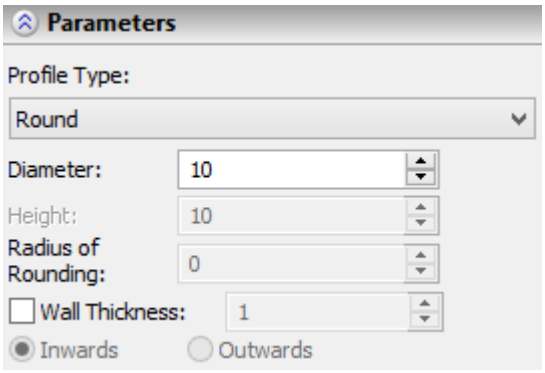
Pipeline Parameters

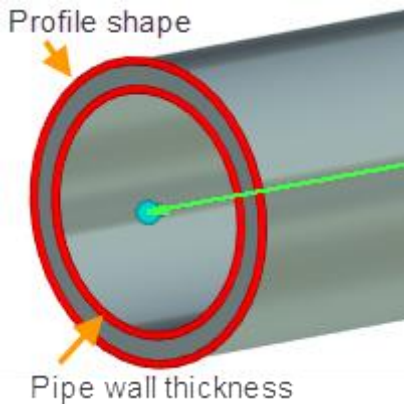
The pipeline profile properties are specified in the **Parameters** section.

Profile Type. You need to select one of the profile types from the drop-down list: **Round**, **Rectangular**, **Select Standard**. According to the selected profile type, you can specify its dimensions.

Round profile

Pipe Diameter. By default, it defines the outer diameter of the pipe. If the switcher is in **Outwards** mode, the parameter defines inner diameter.



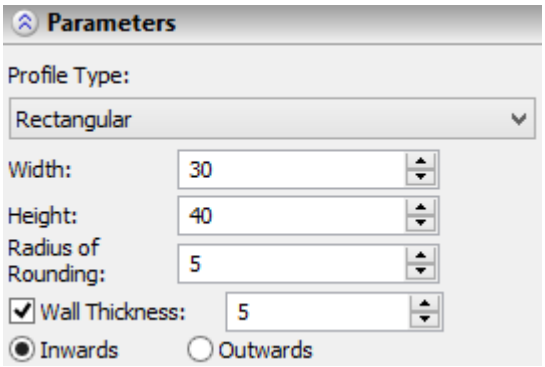


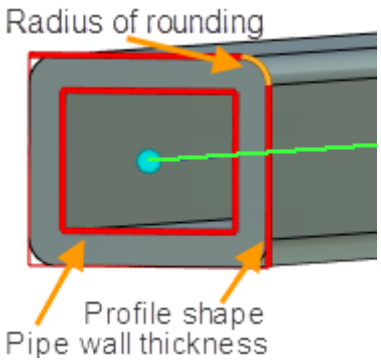
Wall Thickness defines the difference between the outer in the inner diameters of the pipe. By default, the value is put inside the pipe (switcher is in **Inwards** mode).

You can change pipe diameter and thickness using manipulators in the 3D scene. Current value of the parameter can be set on the system toolbar.

> If smooth transition disappears between the two straight pipe segments when creating a pipe, then you must fix correlation between the rounding radius and the pipe diameter.

Rectangular profile





Width, Height, Radius of Rounding specify parameters of outer pipe profile by default. If the switcher is in **Outwards** mode, the parameter defines inner profile.

Permitted values for **Radius of Rounding** are from "0" (without rounding) up to half of the minimum dimensions: length or height.

> If the radius is not within the value or the rounding cannot be created because of other reasons, the system displays the diagnostic message.

Wall thickness. By default, the value is put inside the pipe (switcher is in **Inwards** mode).

> You can change rectangular profile dimensions using manipulators. Current value of edited parameter can be set on the system toolbar.

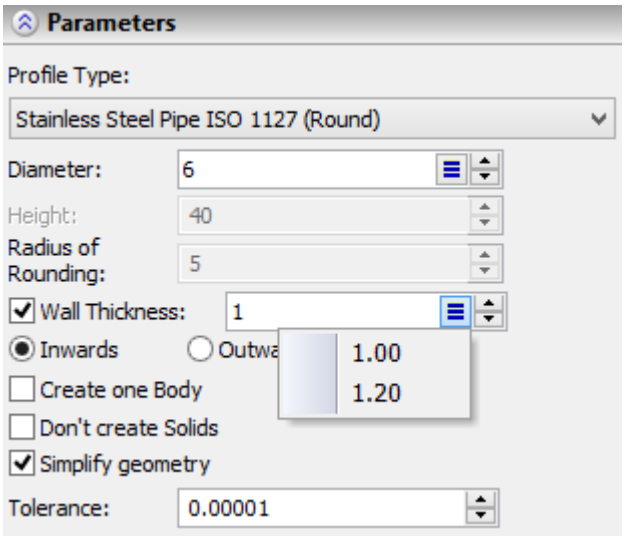
Select standard

If the profile type is specified by a standard, values of parameters are defined from the predefined list. All parameters are divided into control and dependent.

Permitted values of a dependent parameter are defined according to the selected value of the control parameter.

For the round profile type, **diameter** is the control parameter. Dependent parameters are walls thickness, fixation of inside dimension and option for creating one body.

For the rectangular profile type, **height** and **width** are the control parameters. Dependent parameters are walls thickness, fixation of inside dimension and blending.



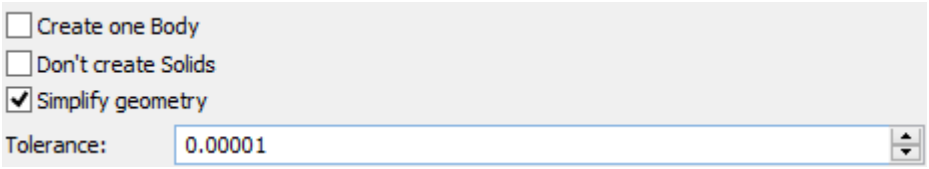
You can manage the standards list using **Standards** window.

More information about the window can be found in “Standards” section.

If the specified value of any parameter does not match the standard, when you edit pipe profile defined by a standard, the system will display a diagnostic message.

Common parameter

The following parameters are available for all profile types:



Create one Body. Parameter specifies the way of pipeline creation. When the flag is not set, the first way (separate segments) is used. The resulting operation consists of multiple bodies. When the flag is set, the second way is used – the resulting operation will present a single body.

Don't create Solids. When the flag is set, the operation will be created but it won't be displayed in the 3D scene. It is useful in models with large number of pipes.

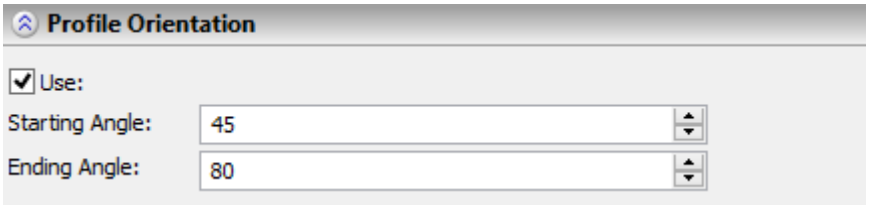
Simplify geometry. With this parameter set, the geometry of the resulting body will be simplified by replacing (wherever possible) spline surfaces with analytical ones (cylinders or tori).

Tolerance. This parameter defines the accuracy of pipe construction, resulting with particular number of intermediate steps when sweeping pipe profile along the spline trajectory to ensure the required accuracy of surface.

Profile Orientation

It is possible to specify rotation of profile while sweeping using flag **Rotate Profile** (available only for rectangular profile).

The profile orientation is the full angle of rotation according to the axis directed along the tangent to the path. It is specified at the pipe ends with two parameters-angles:



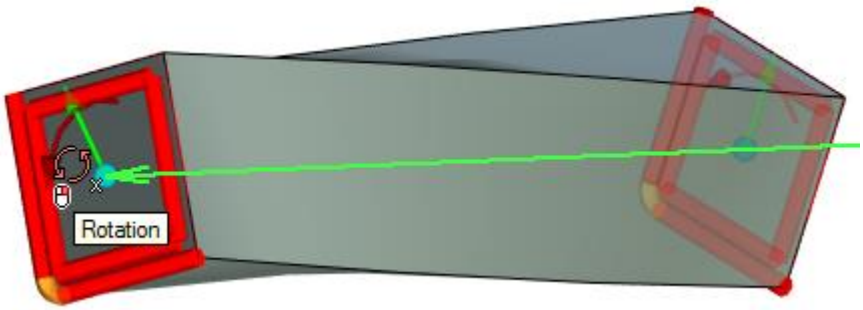
Starting Angle is defined as an angle between Y axes of base LCS and source LCS.

Source LCS - X axis is directed along the tangent to the path, Y axis is defined by the principal normal to the path.

Base LCS – LCS (connector or connection point) which is located on the pipe bound (starting or ending). The X axis is directed along the tangent to the path, the origin merges with the the bound of the pipe (starting or ending).

Ending Angle is specified in the **Properties** window or using manupulator in 3D scene.

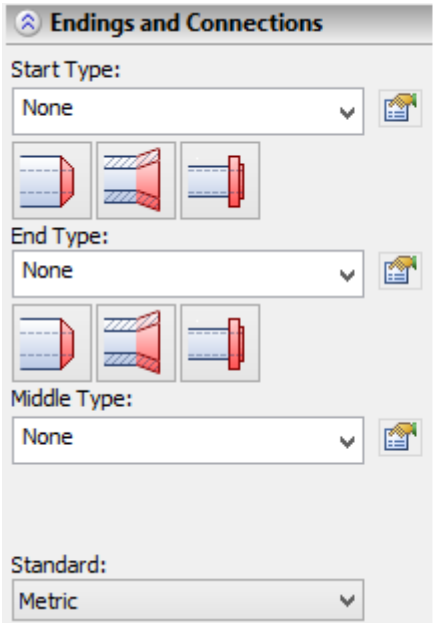
When you use manipulator, the current rotation angle is displayed on the system toolbar.

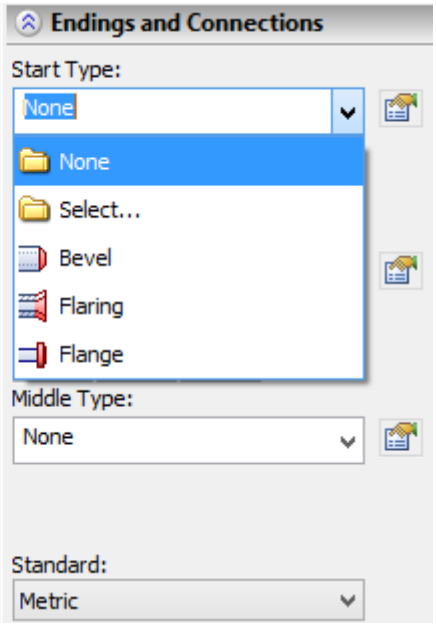


Endings and Connections

There is possibility to set endings and connectors types in the **Pipeline** command by inserting library fragments into the model. Bevels, flanges, threads and so on can be used as such fragments.

In the **3D Model** window such pipeline is displayed as a single object, but physicaly in the model it consists of several fragments (start, middle and end). If you edit any of these fragments the **Edit Pipeline** command activates automatically.





Ending or connection type can be selected either from the quick access drop-down list or by clicking the corresponding icon. If you select type from the list and it is not presented among the icons below, the new icon will replace the last right icon.

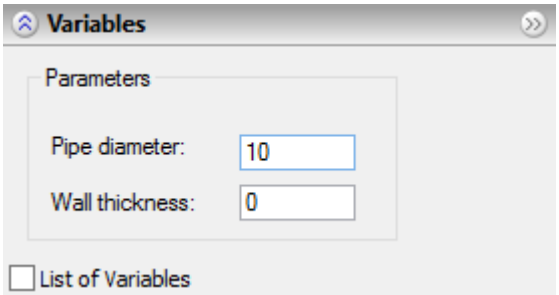
Item “Select” in the drop-down list allows you to select element from custom libraries added to selection list.

The content of the selection list is linked with the value of the **Standard** field and will update accordingly.

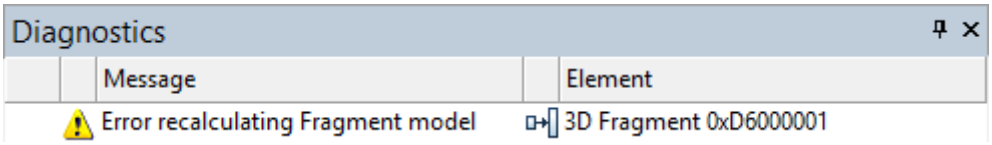
The elements of endings and connection are created as adaptive fragments. They are stored in folder "..\Libraries\Routing". If necessary, library can be supplemented with custom elements.

To define selection from the predefined set of ending and connection elements use **Standard** data field.

Use button **Edit variables** to set external values of fragment variables. The **Variables** section appears after pressing the button.



System checks for correctness of endings insertion and displays message if it is impossible to perform the operation. It can happen if, for example, the ending length is greater than the pipeline length.



You can select ending or connection library using option:

	<B>	Library configuration

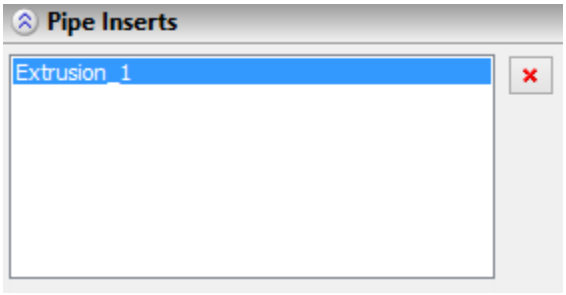
More information about configurations can be found in "Holes" chapter.

Pipe Inserts

This section of Properties window displays elements, used in option **Select operation for pipe insert** and elements, used to define shape of cutting from the pipe.

These elements are not added to the **3D Model** window.

You can delete them using button if necessary.



LIST OF STANDARDS EDITOR

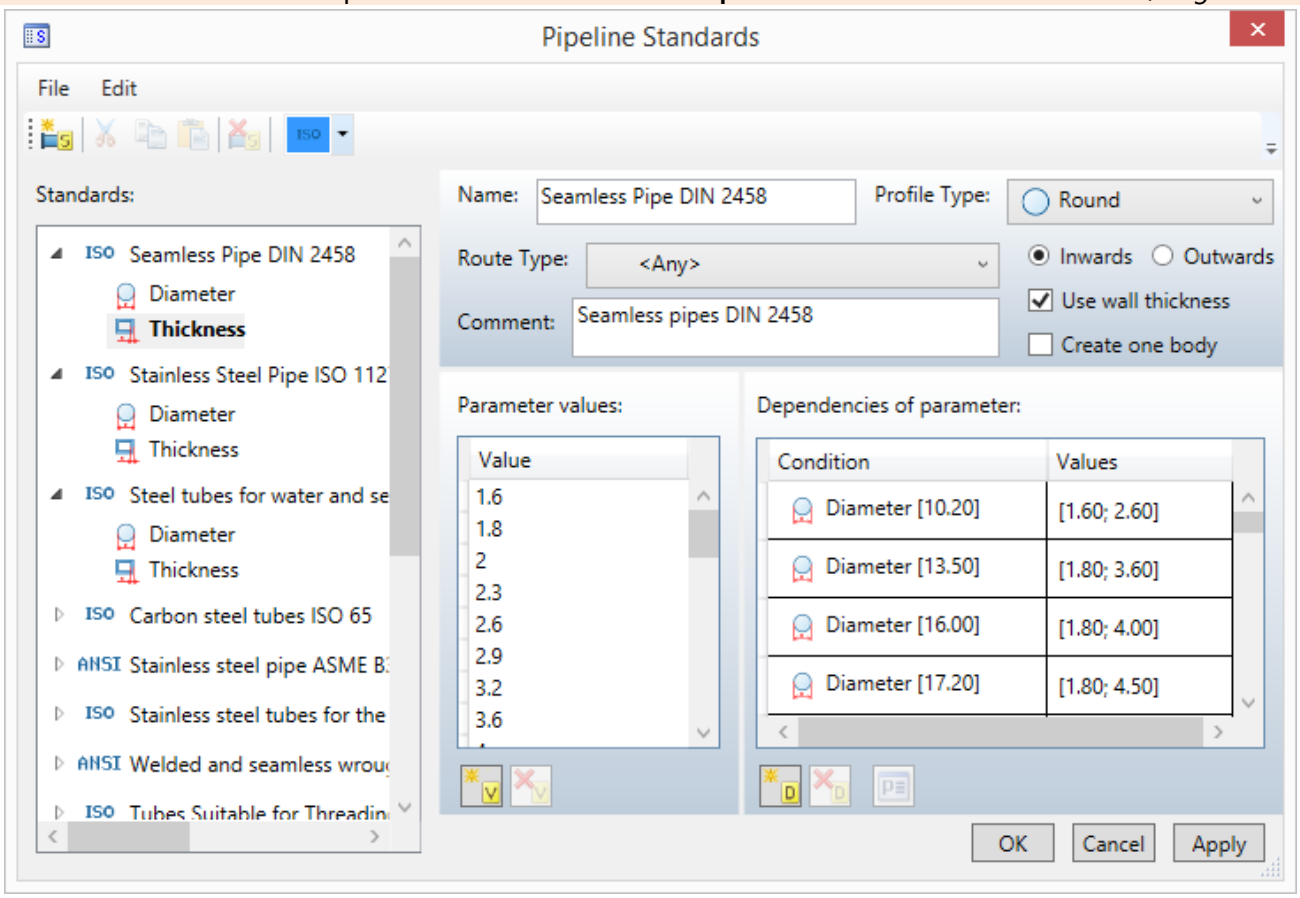
This editor provides functionality for creating list of pipe standards. Here you can set main parameters of a standard as well as define control and dependent parameters.

The editor is called via automenu option:

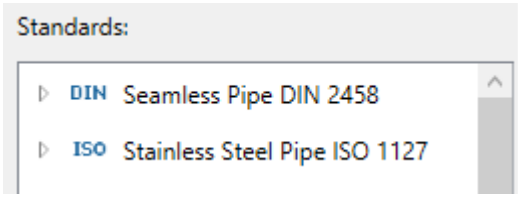
	<G>	Edit list of pipe standards

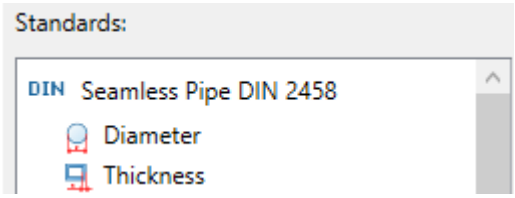
In the left part of the window there is the list of all available standards and pipeline parameters, which are defined by the standards. In the right part you may see the detailed description of the standard.

The list of standards and parameters is stored in file **PipeStandards.xml** inside folder "..*Program*".



You need to specify available parameters values and dependencies between them. Press ▷ to show list of availabale values to the left from the standard name.





After selecting any parameter in the right part of the window you will see the list of available values. The dependent parameter also has the list of dependencies. The dependencies show, which values the control parameter can take according to the particular value of dependent parameter or in the specified interval.

The editor allows to change current standards and add the new ones. To add a standard you need to press button , select standard type (ISO, DIN and so on) from the drop-down list and fill in all necessary fields: **Name**, **Route Type**, **Profile Type**, **Comment**, etc.

Buttons on the toolbar will help you to cut , copy , insert , and delete a standard.

You can specify the following parameters for pipline standard:

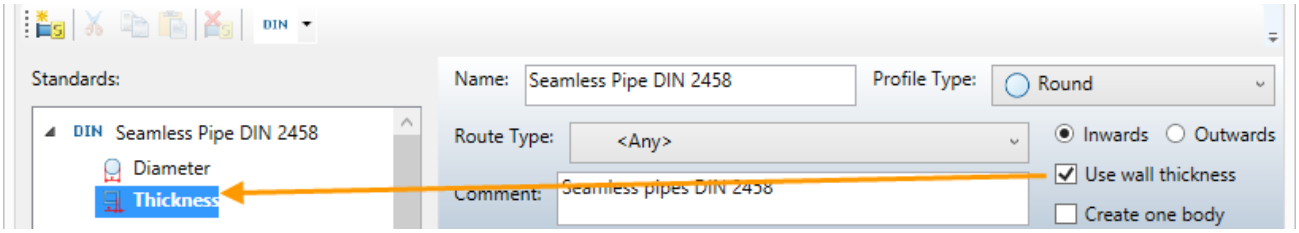
Name – name of standard that will be displayed in the list.

Route Type – defines type of route that may work with this standard.

Profile Type – sets **round** or **rectangular** profile.

More information on profiles types can be found in “Parameters” section.

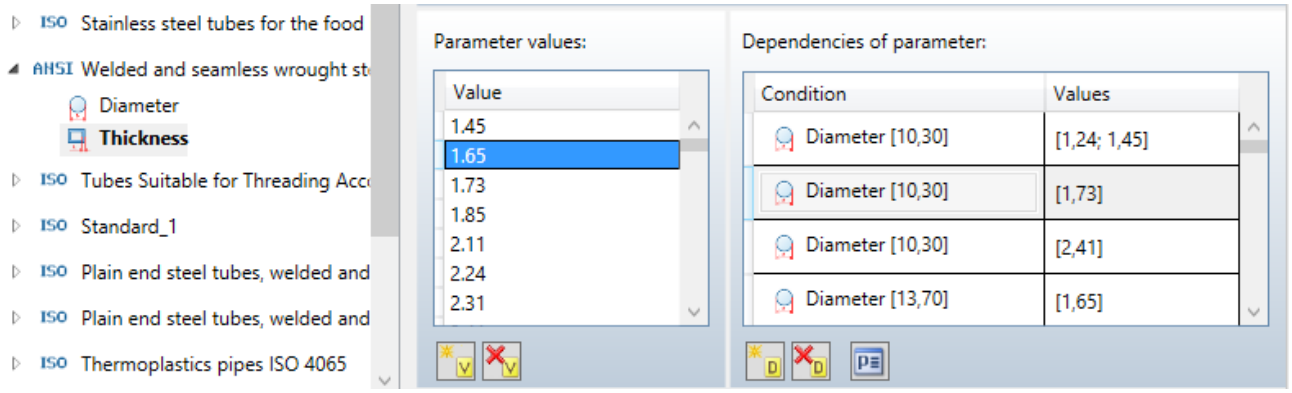
- **Inwards, Outwards**. You can select type of thickness increment Note that in this case changing method of thickness increment (**Inwards/Outwards** switcher in the **Properties** window) will be undesirable. I.e. the way of increment can be changed but it will lead to the diagnostic message.
- **Use wall thickness**. If the flag is set, the **Thickness** appears in the standards list. Without active flag you cannot specify walls thickness.



- **Create one body**. The flag is identical sonamed flag from the pipeline parameters. It defines the way of body creation for the current standard.

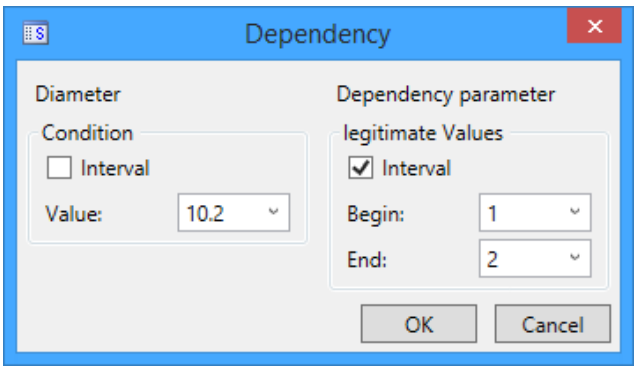
More information about the flag can be found in “Pipeline parameters” section.

After selecting parameter of standard in the left part of window, you can set its values and dependencies in the right part.



The parameter values can be set using button , dependencies are specified using button .

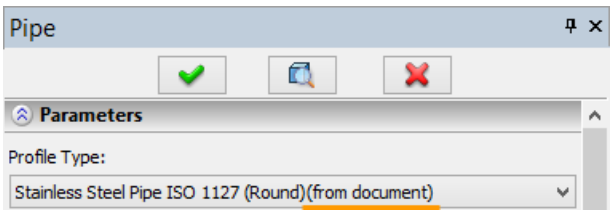
To delete values and dependencies use buttons and or <Delete> button. To edit previously created dependency use button . After pressing the button the following dialog box will appear.



In the **Dependency** window you can set value or interval of the values for control and dependent parameters. To define interval you need to set flag in corresponding field.

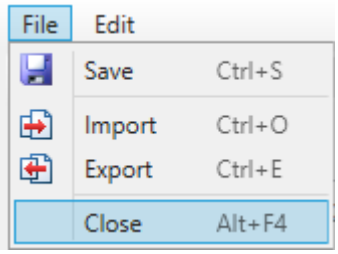
Changes in standard

If the standard for some reason has been changed, it will not affect existing pipeline models based on this standard. But in this case special comment "from document" will appear near the name of the standard in the pipe parameters.



Import/Export of Standards

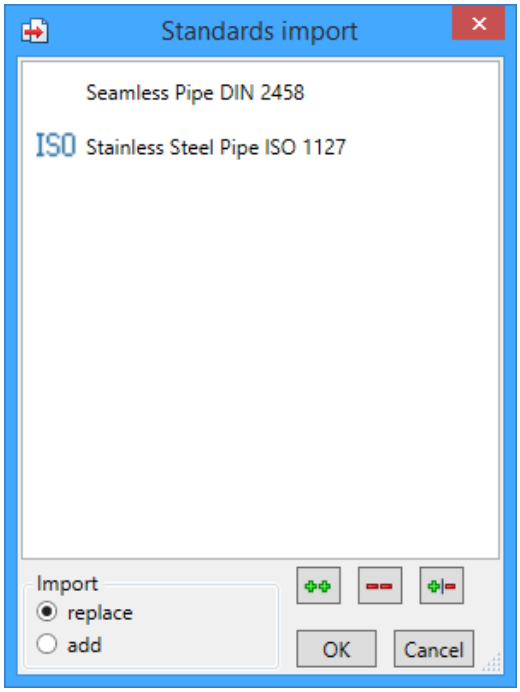
There is possibility to export and import lists of standards from XML-files using commands **File >> Import**, **File > Export** from the menu.

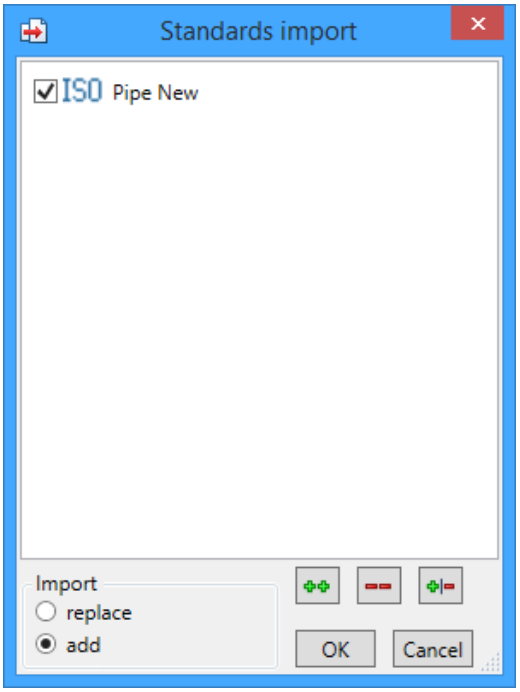


There are two options available for import:

Replace. The current list will be replaced with the list from the external file.

Add. You can select standards that will be added to the list. Selection list will not display standards that already exist in the list.





Icons at the bottom of the window allow you to perform the following actions with the list of standards:

- select all standards.

- cancel selection of all standards.

- invert standards selection.

FITTINGS

Fitting is a connecting part of pipeline, which is usually located at branching places, turns and transitions to another diameter. Fittings are used for sealed overlap of the pipeline and other auxiliary purposes.

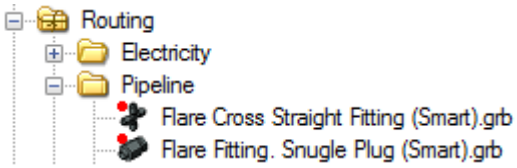
It is convenient to use smart-fragments as fittings. Smart-fragments adapt to objects already existing in the model and affect geometry of connected fragments.

For example, when you insert a cross fitting between two pipes, the crossing pipes will be automatically divided each into two parts. Their ends will be automatically transformed according to the requested connection type.

Special variables, connectors, LCS and macro provide an automatic link with parameters of other routing elements. The macro is included into the fitting document file.

When you need a new fitting you have to create a 3D model file with a special macros that will be used as a 3D fragment. Creation and usage of each fitting type is described below.

Ready to use library fittings files include "Smart" suffix in their names. You can enlarge set of such elements if it is necessary.



Fitting gets part of external variables values from the pipeline parameters. For example fitting can get the pipe diameter. Fitting can also pass parameters values to the pipe, if it is set in the fitting file.

There are following fittings types defined by default: Cap, CornerFracture, CornerRadius, Tee, Crossing, Adapter, Clamp, CornerEnd, Inset.

Fitting Insertion

It is easy to use drag'n'drop mechanism for fitting insertion.

Options for defining fitting location appears right after drag'n'drop of the fitting.

	<P>	Select path
	<T>	Select pipe

When option is selected, the fragment is placed on the selected path point. When you select , system defines appropriate point for fixing on the selected pipe (for example, crossing), if necessary.

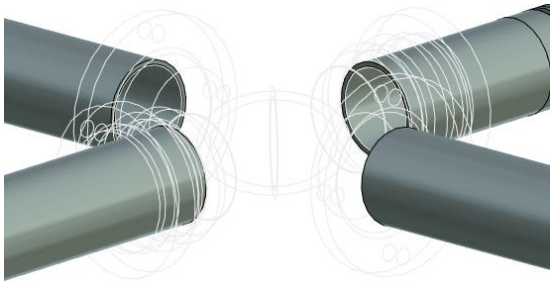
There is a different set of options for each fitting type. It appears after path or pipe selection.

	<R>	Rotate fragment along main axis
	<T>	Rotate fragment along alternative axis
	<V>	Select next fixing point

Options and allow to rotate the fragment around main and alternative directions, changing location of trunks and branches.

Option is available if the fragment is located on the crossing of two pipes/paths.

If the fragment (tee or crossing) is located at the crossing point of paths, you can't rotate or translate it using manipulator.



For CornerFracture, CornerRadius, CornerEnd:

	<R>	Rotate fragment
	<F>	Select next fixing point

The next suitable fixing point for fitting will be automatically found after pressing of or options. The points will be cyclically searched upon further continuous option usage.

The corner can be located on trunks and endings of the pipe.

For Inset, Clamp:

	<D>	Select pipe for Y direction
	<R>	Rotate fragment

When the option is active you need to select pipe, related to which the Y direction will be defined.

For Cap, Adapter:

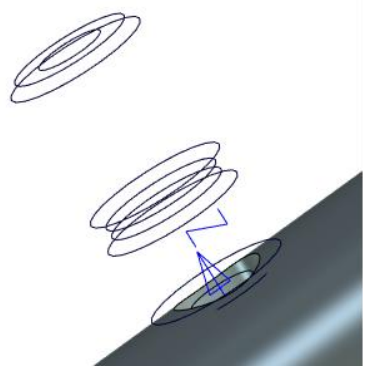
	<R>	Rotate fragment

This type of fittings limits pipe on endings.

For all inserted smart-fragments the following options are available:

	<E>	Wireframe
	<N>	Insert new fragment after confirmation

Wireframe. Allows to switch between wireframe mode and rendering mode of the model.



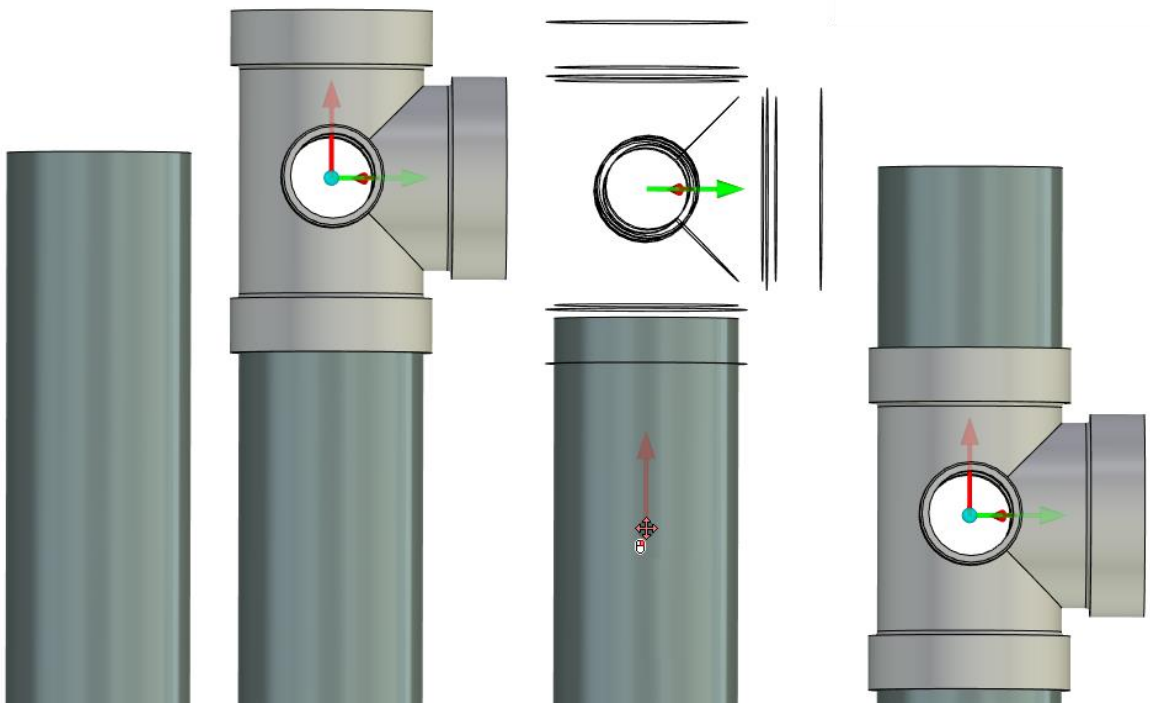
Wireframe mode

Rendering mode

Insert new fragment after confirmation. A new fragment similar to the inserted one will be added after confirmation button pressing.

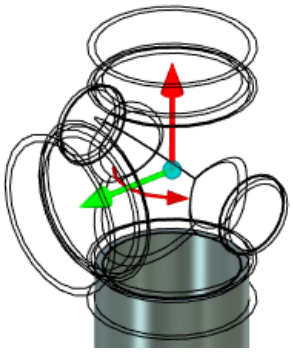
Fittings Basic Highlights

When you insert a fitting at the beginning or end of the pipe, the pipe is not divided. When you change fixing type the pipe will be divided into two parts if possible. The original parameters of pipe endings will be restored.

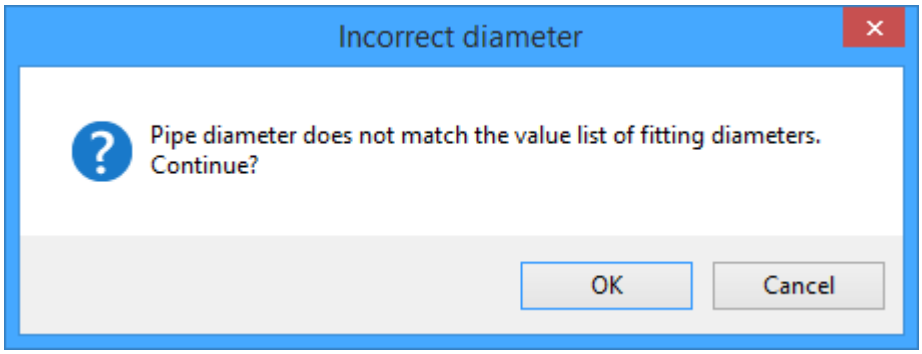


After the fitting insertion or replacement system determines necessity of the pipe dividing. Unused pipe part will be suppressed and removed after exiting the command.

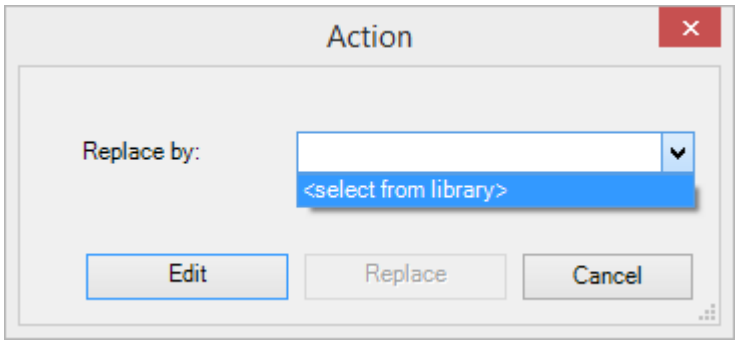
Special manipulator is used to define fitting position. It allows to move fitting and rotate it around the axis.



If fitting does not match the pipe, the corresponding message will appear.

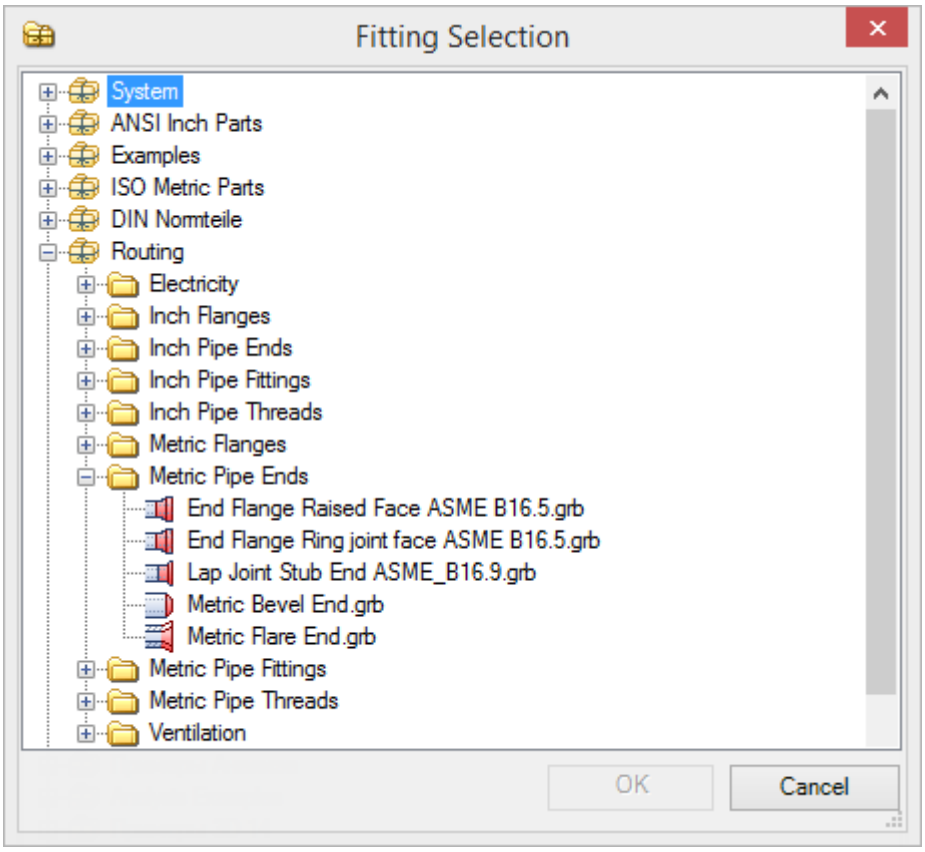


Special window **Action** appears when you edit a fitting. Here you can select option for editing the current fitting, selecting a new fitting to replace the current or cancel editing.

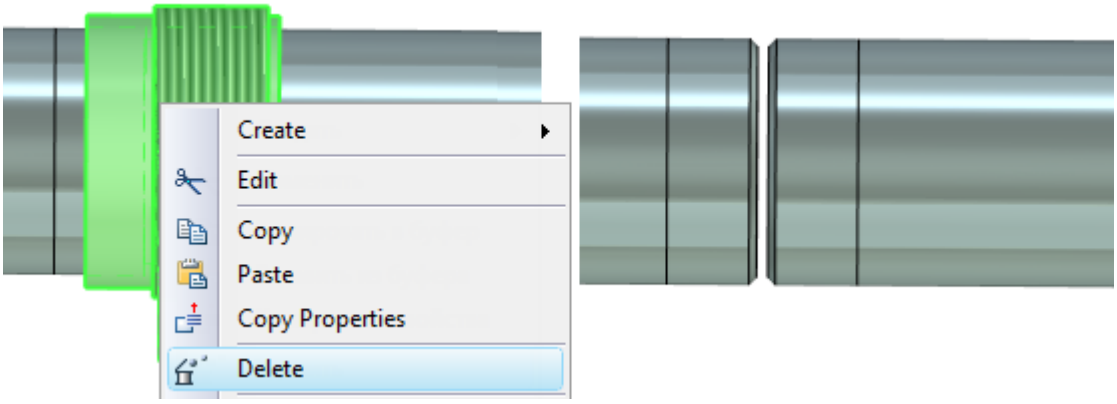


To replace a fitting choose option <Select from library> and select necessary library fragment in **Fitting Selection** window. After selection, you need to press [Replace] button.

After fitting selection the system exit from edition command automatically.



When you delete a fitting, linked pipes won't be deleted.



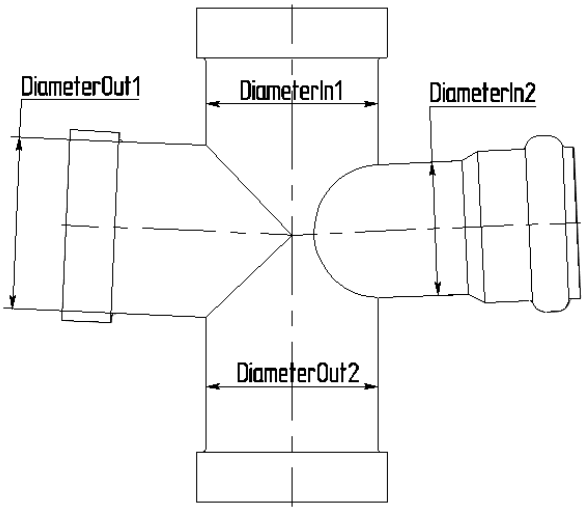
Fittings Parameters Names

Table of external key variables of smart-fragment.

Variable name	Usage	Description
DiameterIn1	Manages input diameter of main direction fitting	Is necessary for Cap, CornerXxx, Tee, Crossing, Adapter, Clamp
DiametrOut1	Manages output diameter of main direction fitting	Is necessary for CornerXxx, Tee, Crossing, Adapter
DiametrIn2	Manages input diameter of alternative direction fitting	Is necessary for Crossing
DiametrOut2	Manages output diameter of alternative direction fitting	Is necessary for Tee, Crossing

External variable Diameter *** is responsible for link between the respective diameters of fitting and pipe. Due to the variables, macro can get diameter of pipe to which fitting is attached and set the appropriate diameter of fitting.

When creating a smart fragment it is possible to use from one (lug, cross with identical diameters of all branches) to four (cross with four different diameters) external variables, depending on the number of input and output fitting diameters.



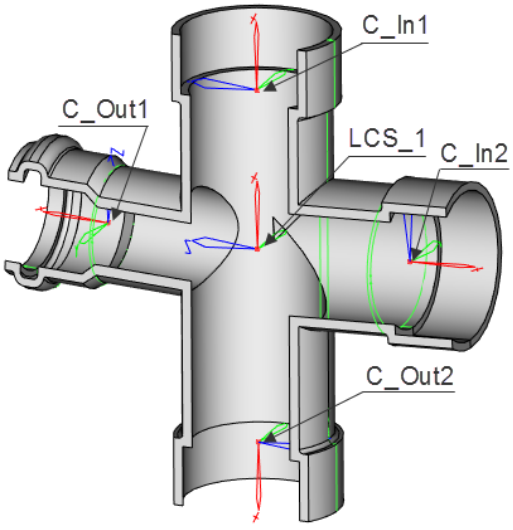


Table of smart-fragment key LCS.

Object	Description
LCS_1	The fragment source LCS must be set as LCS by default in the fragment document parameters.
C_In1, C_Out1, C_In2, C_Out2	Connectors names of input/output main and alternative directions

LCSs listed in the table are responsible for cutting pipe for which fitting is inserted.

LCS_1 responsible for position of fitting on the pipe is the main LCS that must be created for joining fitting with the fragment. The remaining four LCSs are responsible for joining pipes of appropriate diameter and the emergence of the connecting elements (flange and thread) at coupling place of pipe with fitting. These LCSs are considered in macro.

Table of key connector parameters that define smart-fragment insertion logic:

Parameter name	Usage	Description
autoFilePipeEnd	Pipe ending/beginning if connector is located on pipe end. Can be omitted.	Template of the parameter value: <*library*>*file_name*.grb Parameter functions: - creates cosmetic thread on the pipe - cuts the edge for weld - creates flaring
autoLength	The fragments name in the connector direction	Sum of input and output connector lengths is equal to the length of the direction (main or alternative)
autoDiameter	Pipe diameter will be taken from fitting	If missed the original pipe diameter is used.
autoThickness	Pipe thickness will be taken from fitting	If missed the original pipe thickness is used.

Auto – manages the command operating logic when the smart-fragment with this connector is inserted.

The above parameters allow you to specify additional options when you insert the fitting.

Fittings Smart-fragments Behavior in Pipe Systems

Variants of interaction with other fittings

1.	Y-shaped bend	• Creates inner cosmetic thread on pipe • Creates outer cosmetic thread on pipe • Divides pipe segment into two segments • Divides pipeline into more than two segments • Modifies pipe diameter from larger to smaller and back • Cuts edge for weld • Inserted in the middle of pipe segment • Inserted on pipe ends • Inserted on pipe rounds • Inserted on pipe corners • Interaction with a single pipe • Interacts several pipes • Interacts other fittings • Interacts intersecting branches of pipeline
2.	Adapter	• Creates inner cosmetic thread on pipe • Creates outer cosmetic thread on pipe • Divides pipe segment into two segments • Modifies pipe diameter from larger to smaller and back • Inserted on pipe ends • Inserted in the middle of pipe segment • Interaction with a single pipe • Interacts several pipes • Interacts other fittings • Interacts with parallel branches of pipeline • Makes the flaring of pipe/tube/hose
3.	Branch	• Creates a hole in pipe • Cuts edge for weld • Inserted in the middle of pipe segment • Interaction with a single pipe • Interacts with several pipes • Interacts intersecting branches of pipeline

4.	Bushing	• Creates inner thread on pipe • Creates outer cosmetic thread on pipe • Divides pipe segment into two segments • Modifies pipe diameter from larger to smaller and back • Cuts edge for weld • Inserted on pipe ends • Inserted in the middle of pipe segment
5.	Lock nut	• Creates outer cosmetic thread on pipe • Inserted in the middle of pipe segment • Interaction with a single pipe
6.	Set-off pipe bend	• Creates inner thread on pipe • Creates outer cosmetic thread on pipe • Cuts edge for weld • Inserted on pipe ends • Cuts second part of pipe • Modifies pipe diameter from larger to smaller and back • Cuts edge for weld • Inserted in the middle of pipe segment • Interaction with a single pipe • Interacts with several pipes • Interacts with parallel branches of pipeline • Interacts other fittings
7.	Valve	• Creates inner thread on pipe • Creates outer cosmetic thread on pipe • Divides pipe segment into two segments • Cuts second part of pipe • Modifies pipe diameter from larger to smaller and back • Cuts edge for weld • Inserted on pipe ends • Inserted in the middle of pipe segment • Interaction with a single pipe • Interacts with several pipes • Interacts other fittings • Interacts intersecting branches of pipeline

8.	Bend	• Creates inner thread on pipe • Creates outer cosmetic thread on pipe • Divides pipe segment into two segments • Cuts second part of pipe • Modifies pipe diameter from larger to smaller and back • Cuts edge for weld • Inserted on pipe rounds • Inserted on pipe corners • Inserted in the middle of pipe segment • Interaction with a single pipe • Interacts with several pipes • Interacts other fittings • Interacts intersecting branches of pipeline
9.	Crossing	• Creates inner thread on pipe • Creates outer cosmetic thread on pipe • Divides pipe segment into two segments • Cuts second part of pipe • Modifies pipe diameter from larger to smaller and back • Cuts edge for weld • Inserted on pipe ends • Inserted in the middle of pipe segment • Interaction with a single pipe • Interacts with several pipes • Interacts other fittings • Interacts intersecting branches of pipeline
10.	Coupling	• Creates inner thread on pipe • Creates outer cosmetic thread on pipe • Divides pipe segment into two segments • Modifies pipe diameter from larger to smaller and back • Inserted on pipe ends • Inserted in the middle of pipe segment • Interaction with a single pipe • Interacts with several pipes • Interacts other fittings
11.	Cap	• Creates inner thread on pipe • Creates outer cosmetic thread on pipe • Cuts second part of pipe • Inserted on pipe ends • Interaction with a single pipe • Interacts other fittings

12.	Pipe clamp	• Changes geometry of the 3D path • Interacts with several pipes • Interacts intersecting branches of pipeline • Interacts with parallel branches of pipeline • Interacts intersecting branches of pipeline
13.	Reducer	• Creates inner thread on pipe • Creates outer cosmetic thread on pipe • Divides pipe segment into two segments • Modifies pipe diameter from larger to smaller and back • Cuts edge for weld • Inserted on pipe ends • Inserted in the middle of pipe segment • Interaction with a single pipe • Interacts with several pipes
14.	Tee	• Creates inner thread on pipe • Creates outer cosmetic thread on pipe • Divides pipe segment into two segments • Cuts second part of pipe • Modifies pipe diameter from larger to smaller and back • Cuts edge for weld • Inserted on pipe ends • Inserted on pipe rounds • Inserted on pipe corners • Inserted in the middle of pipe segment • Interaction with a single pipe • Interacts with several pipes • Interacts other fittings • Interacts intersecting branches of pipeline
15.	Seal	• Interacts other fittings
16.	Flange	• Divides pipe segment into two segments • Inserted on pipe ends • Inserted in the middle of pipe segment • Interaction with a single pipe • Interacts other fittings

17.	Pipe housing	• Inserted in the middle of pipe segment • Interaction with a single pipe • Inserted on pipe rounds • Inserted on pipe corners • Interacts with several pipes • Interacts with parallel branches of pipeline • Interacts other fittings
18	Compensator	• Divides pipe segment into two sections • Cuts edge for weld • Interaction with a single pipe • Inserted on pipe ends • Inserted in the middle of pipe section • Interacts other fittings

Macros Creation

When you create a new fitting a standard macro text is added into its file. One of the fittings types that were listed above is specified in the macro.

Here is example of typical macro text for the “Crossing” type.

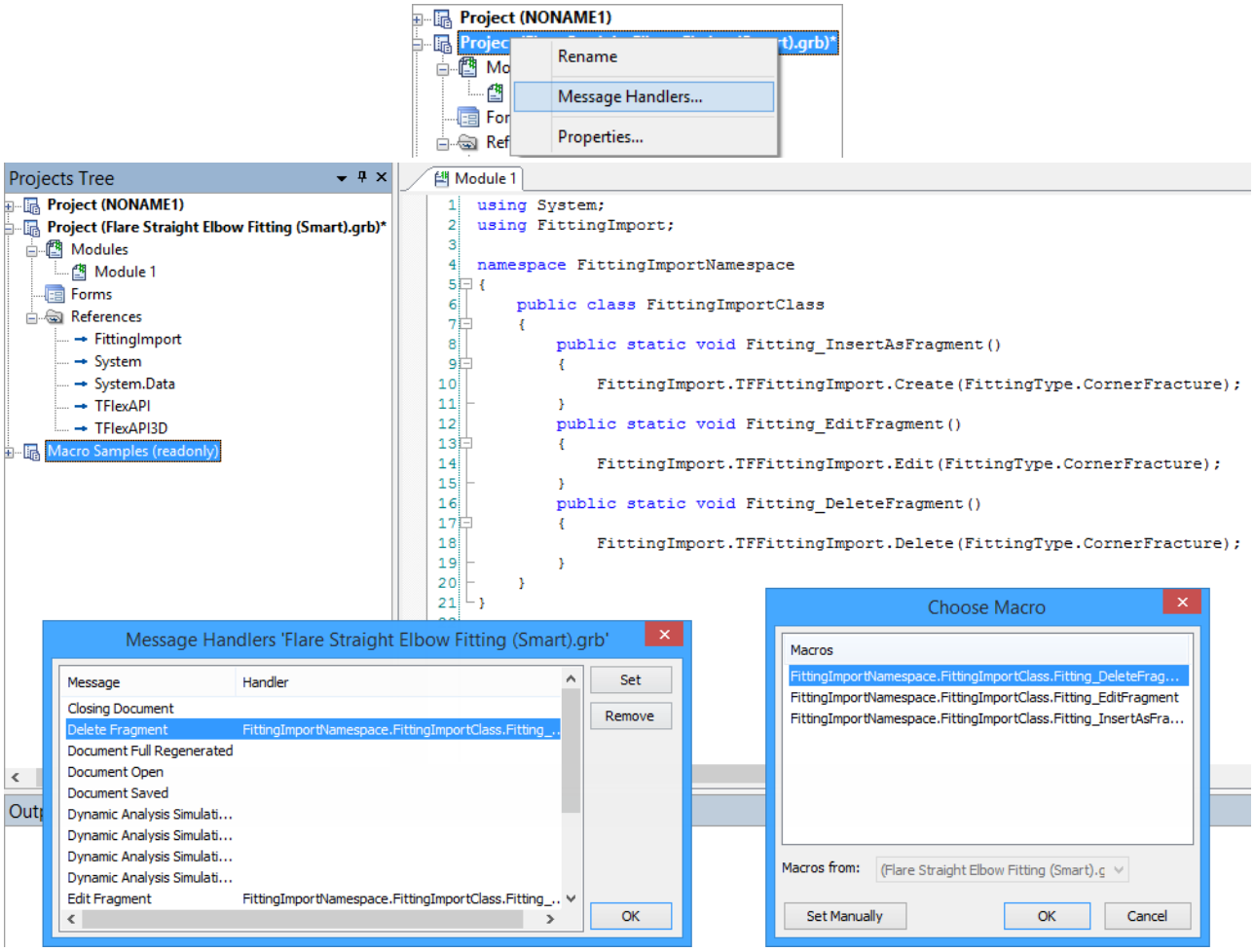
```
using System;
using FittingImport;

namespace FittingImportNamespace
{
    public class FittingImportClass
    {
        public static void Fitting_InsertAsFragment()
        {
            FittingImport.TFFittingImport.Create(FittingType.Crossing);
        }
        public static void Fitting_EditFragment()
        {
            FittingImport.TFFittingImport.Edit(FittingType.Crossing);
        }
        public static void Fitting_DeleteFragment()
        {
            FittingImport.TFFittingImport.Delete(FittingType.Crossing);
```

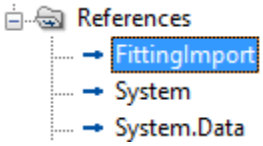
```
            }
        }
}
```

> More information about macro can be found in “Macros” chapter.

Also you need to set three **message handlers** “Fitting_InsertAsFragment”, “Fitting_EditFragment”, “Fitting_DeleteFragment”.



To compile the macro you need to add link to the FittingImport in the project.



Smart-fragment Creation Example

General steps for successful creation of a smart-fragment are listed in this example.

For convenience, the smart-fragment creation is divided into four stages:

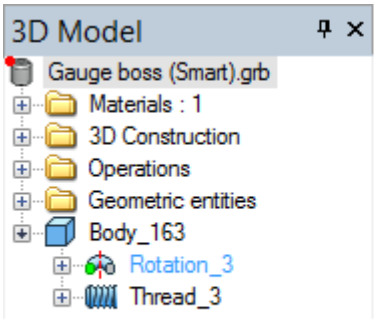
- ✓ Body for fitting creation
- ✓ Body for creation of the cutout shape in pipe
- ✓ Fitting parameters managing
- ✓ Macro creation

Smart-fragment "Examples\ 3D Modeling\Routing\Gauge boss (smart)" was used as an example.

There are special rules for LCS, connectors and variables when you create a fitting. Names of variables and LCSs and their description are listed in "Fittings parameters names" section.

Fitting Body Creation

To begin with, you need to create a fragment body. The **3RO:Rotation** and **3AT:Thread** were used to create the boss.



More information about the rotation and thread operation can be found in "Rotation" and "Thread" chapters.

Fitting Hole Body Creation

The next step is creation of a fitting hole body.

Body for defining an insert in pipe is a fitting element that defines the form and shape of a hole in pipe, in which the fitting is inserted. After insertion, this body disappears.

You need to specify input diameter of main direction fitting "DiameterIn1" and main LCS "LCS_1" for the boss creation.

More information about variables names can be found in "Fitting parameters names" section.

Setting Diameter Variable

The diameter is specified in the variables editor. The variable defines diameter of pipe in which the fitting is inserted and specifies fitting location.

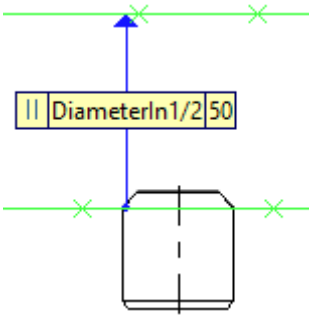
	DiameterIn1	100	100

You can set any value for diameter variable. The macro automatically identifies pipe diameter and add its value to the variables, when the fitting is inserted into an assembly.

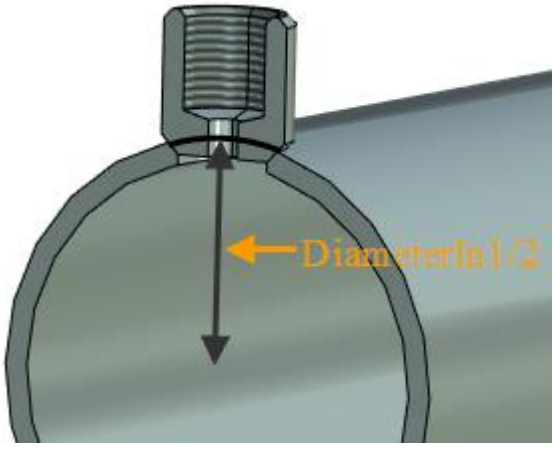
You need to set **External variable** flag for the variable.

LCS Creation

Construction line was created in the example to specify LCS location. The line is at distance "DiameterIn1/2" from the face of the boss cylindrical part. The pipe diameter is half divided that's why the boss will be located on the pipe.



Line that defines LCS location



Boss location on the pipe

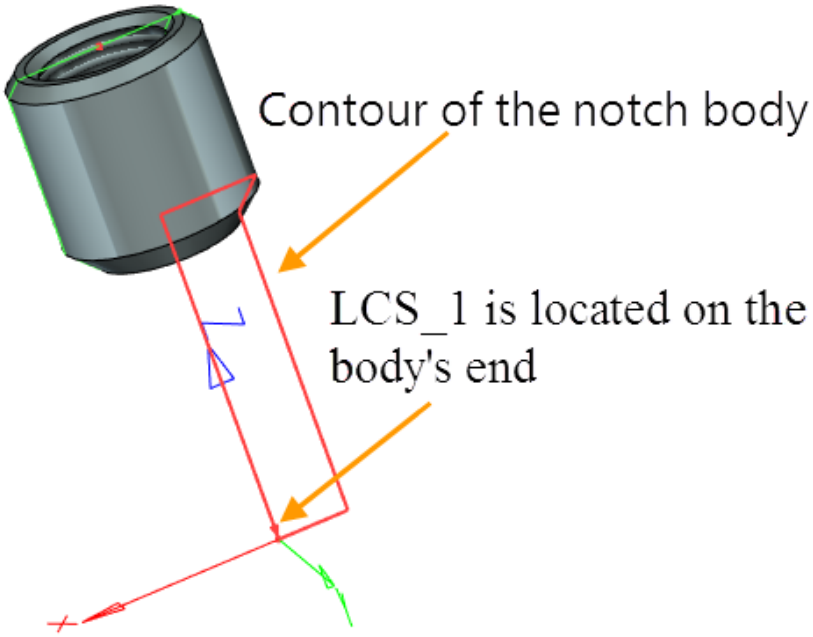
The LCS is renamed to LCS_1. The name was taken from the "Key smart-fragments objects" table.

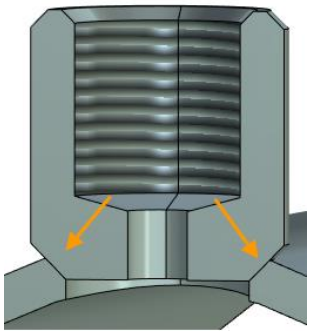
There is no need to create connectors (C_In, C_Out) because there are no fitting elements to connect with the boss.

In this case, the fitting will not join any piping elements or other fittings, so the creation of additional connectors (C_In, C_Out), diameters (DiameterIn2, DiameterOut), or their parameters are not required.

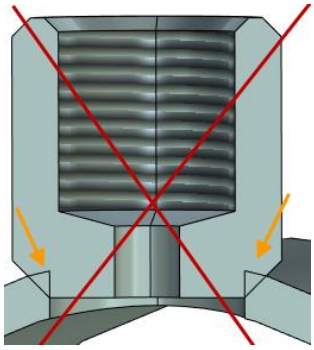
Body Creation

The next step is to create a body to define shape of the cutout in pipe. Contour of the body should follow the shape of the cutout in pipe, so that when you insert fitting there is no intersection of the fitting and the pipe.





Without intersection

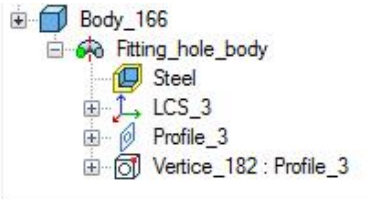


With intersection

Length of body should depend on LCS_1 position. This is done in order to be able to adjust the position of the fitting on the pipe depending on the pipe diameter.

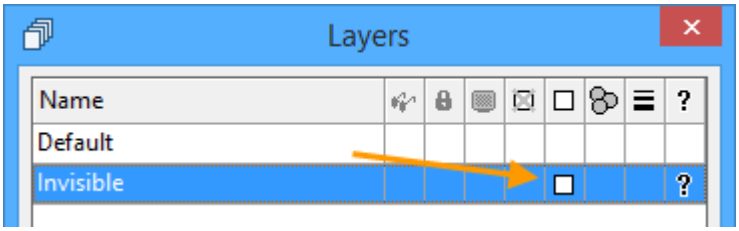
> It is better to have the body with the most simple geometry - a box or cylinder. This will reduce the time of the pipeline regeneration.
>
> If the body is not created, a cutout will have the form of the inserted fragment.

In this case, the body is obtained by rotating profile around axis.

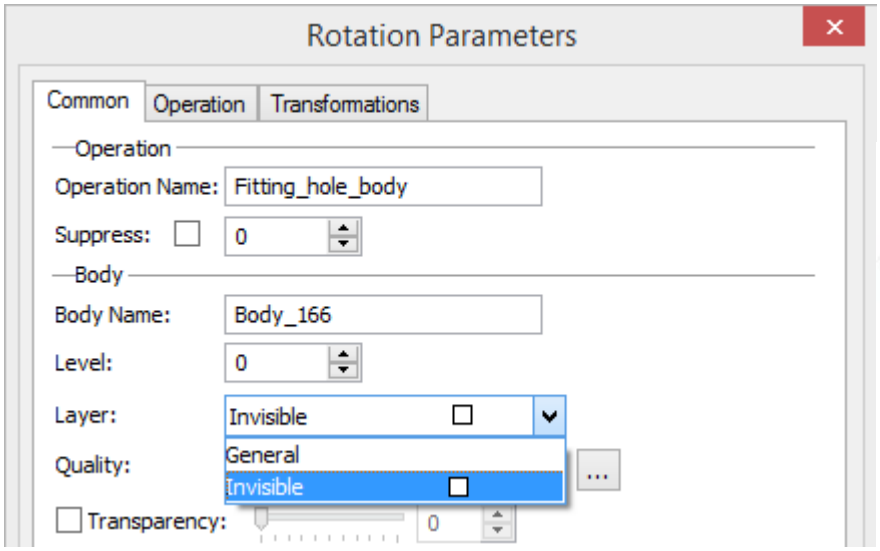


Body to set the shape of cutout in the pipe should be hidden in the 3D scene and on the drawing. In order to hide it you need to do the following.

➔ In the editor of layers, create a new layer and name it - "Invisible" and put mark in the **Visible only when inserted in assembly.**



➔ In the parameters of created body on the **General** tab, you must specify the previously created layer "Invisible", then the body will be hidden in the 3D scene

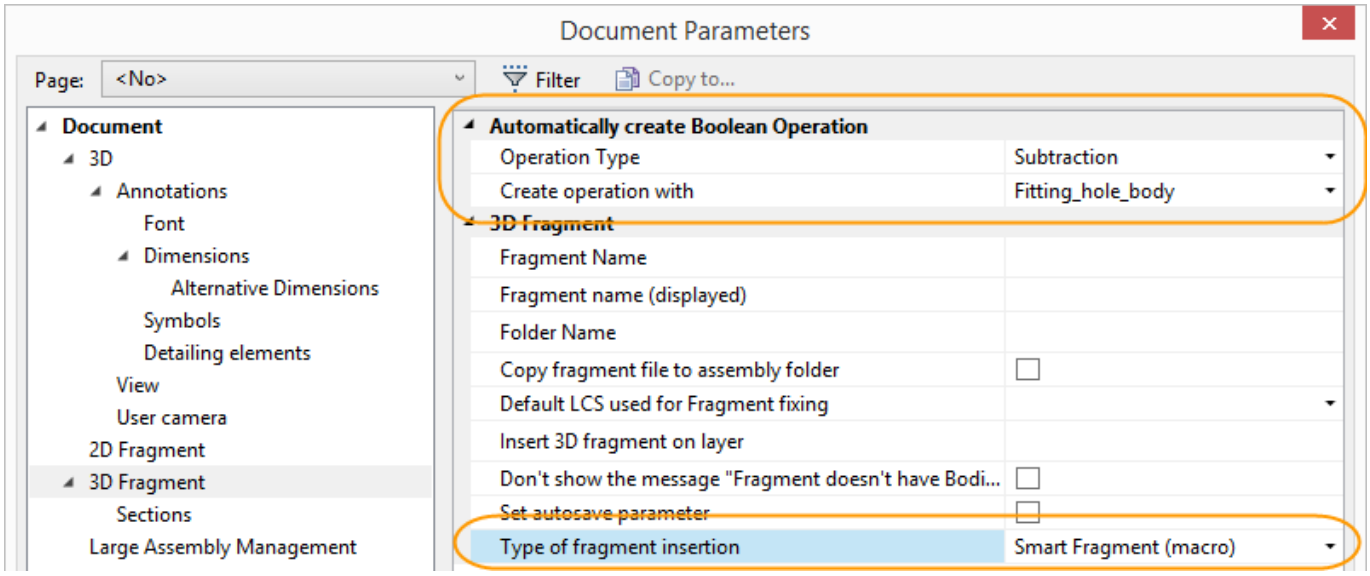


Adjusting Fitting Parameters

After creating the fitting body it is necessary to set the parameters for its insertion into assembly as a smart fragment. To do this, use **ST: Set Document Parameters** command, go to **3D Fragment** tab.

In the window that appears you must:

➔ Set **Type of Fragment Insertion** – **Smart Fragment (macro)**. System will use macro created in Macro Editor when inserting this body.

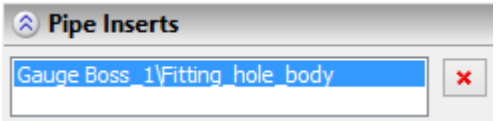


➔ In the **Automatically create Boolean operation** section you should select operation type **Subtraction** from the drop-down list, so that the body create a cutout in the pipe, and in the

drop-down list **Create operation with** select the name of the body that will be used for Boolean operations. In this case - "Fitting_hole_body".

Setting the above parameters allows system to automatically execute macro to create Boolean operation when you insert the smart fragment.

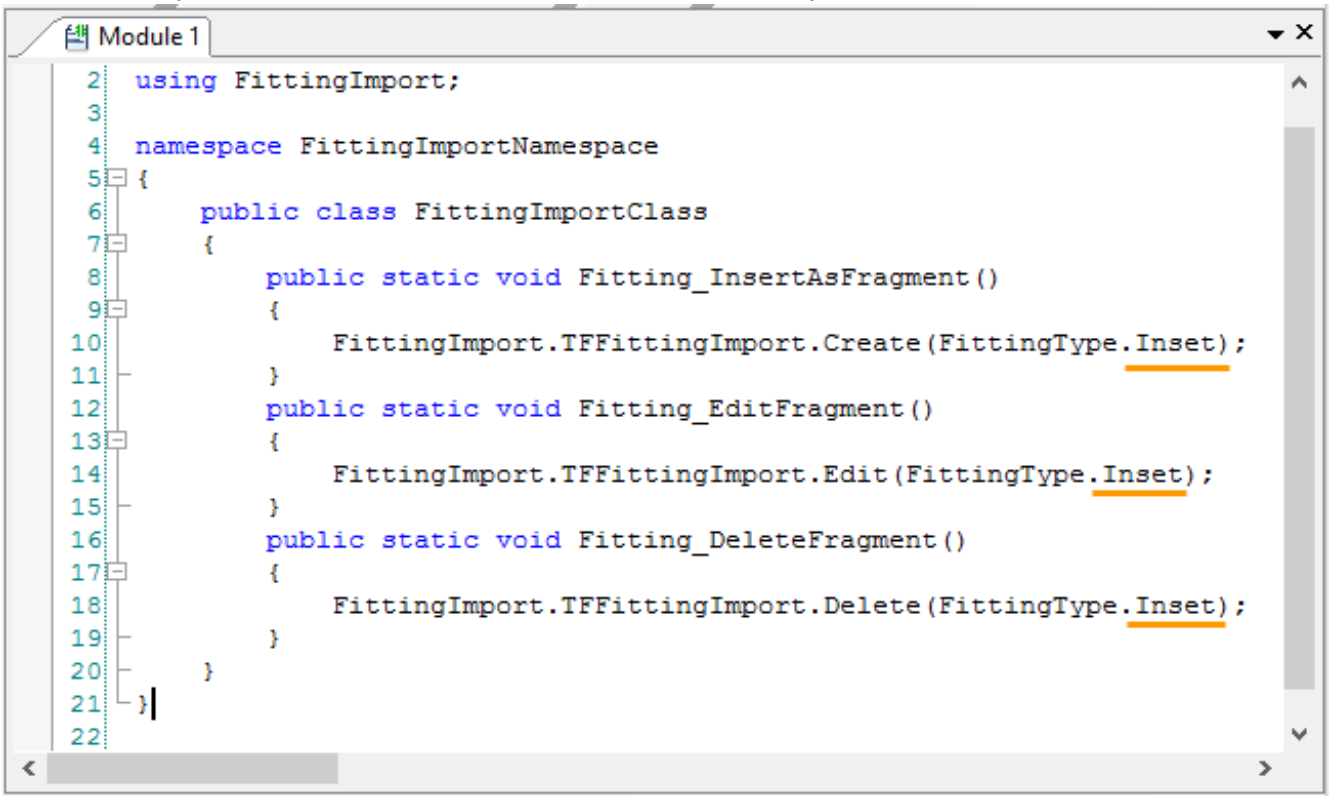
After performing Boolean operation body used as a cutout in the pipe disappears. It will be referenced in the section **Pipe Inserts** in the **Properties** window for the selected element of the pipeline.



Macro Creation

The following is the sequence of actions to create a macro for inserting a smart fragment:

➔ Use macro editor to create a new macro. Macro text for various fittings differs only in the type indicated in parentheses. In this case, this is type "Inset".



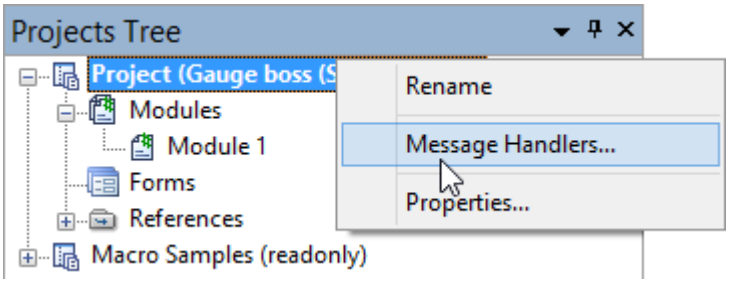

```
using FittingImport;

namespace FittingImportNamespace
{
    public class FittingImportClass
    {
        public static void Fitting_InsertAsFragment()
        {
            FittingImport.TFFittingImport.Create(FittingType.Inset);
        }
        public static void Fitting_EditFragment()
        {
            FittingImport.TFFittingImport.Edit(FittingType.Inset);
        }
        public static void Fitting_DeleteFragment()
        {
            FittingImport.TFFittingImport.Delete(FittingType.Inset);
        }
    }
}
```

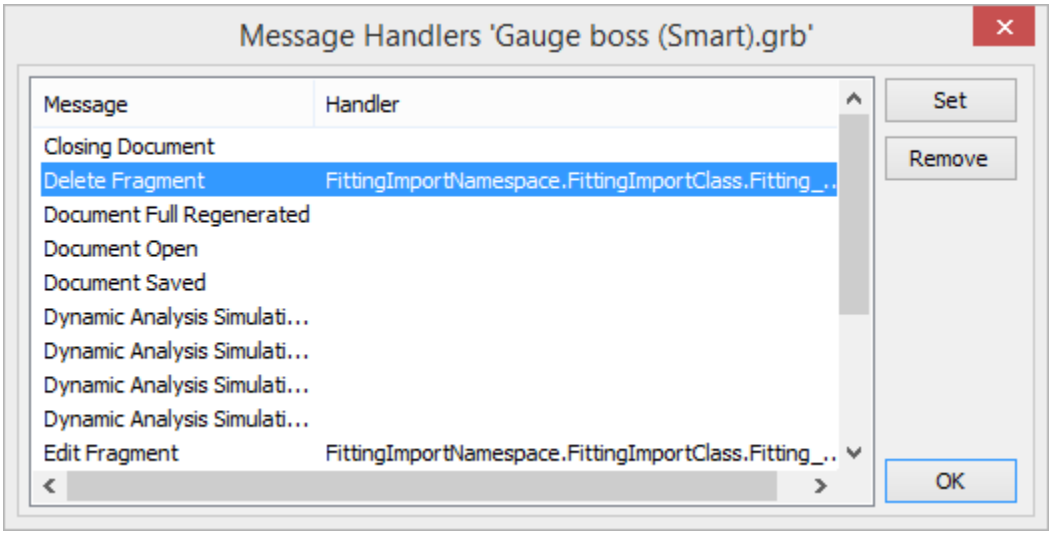
➔ The you need to define three message (event) handlers:

- ✓ Inserting as a fragment (Fitting_InsertAsFragment),
- ✓ Editing (Fitting_EditFragment),
- ✓ Deleting (Fitting_DeleteFragment).

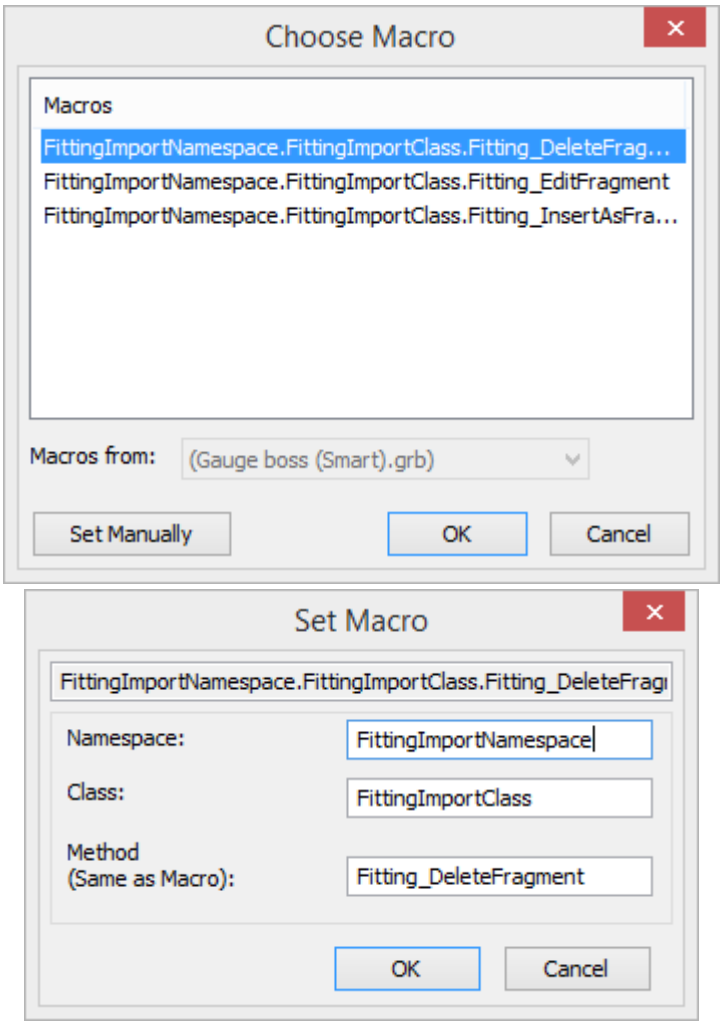
To set the handler you must select the project and from the context menu choose **Message Handlers ....**



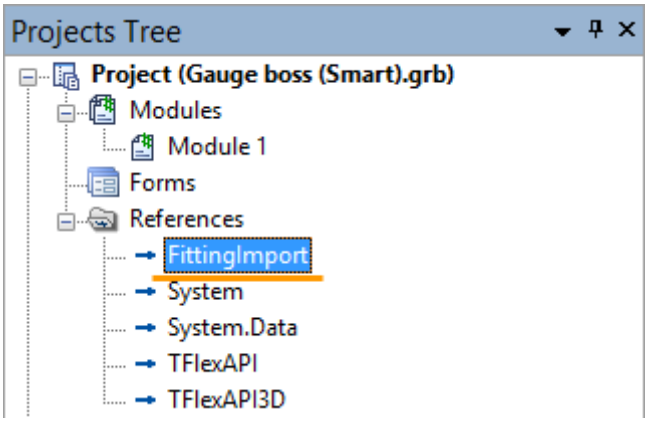
In the window that appears choose one after another three lines "Delete Fragment", Edit Fragment, "Insert as a Fragment" and select them using the macro button [**Set**].



Since the main macro has been previously created in the macro editor, the list will appear from which you can select the desired part of the macro. Macro can be set manually, if necessary, by pressing [**Set Manually**].



➔ In order to successfully compile the macro add link to FittingImport in the project.



After completing all the above items, the fragment file must be saved. Now the created fitting can be used for insertion into assembly using Drag'n'Drop method.

More information about creating macros can be found in the "Macros" chapter.

Example of creating pipe endings and connections

When you create models for pipe middles and endings you should considered a set of rules.

To add new models for pipe middles and endings into the library use command:

	<C>	Library Configuration

More information about configurations can be found in "Holes" chapter.

In the fragment file you must create the following variables:

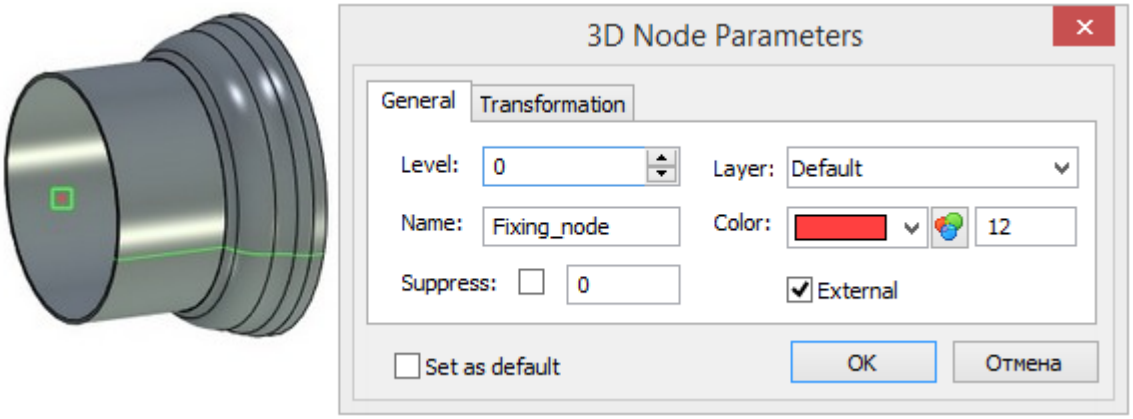
- ✓ **Diameter** or **D** - pipe diameter (permissible values range from 0 to 10000)
- ✓ **Thickness** or **T** - pipe wall thickness (permissible values range from 0 to 10000)-
- ✓ **InnerDiameter** or **InnerD** – flag for usage of internal diameter (possible values 0 or 1)

Note. Variable **InnerDiameter (D)** can be used for the suppression of one of the bodies in the fragment file. One body will be visible when using internal diameter, the second - using external.

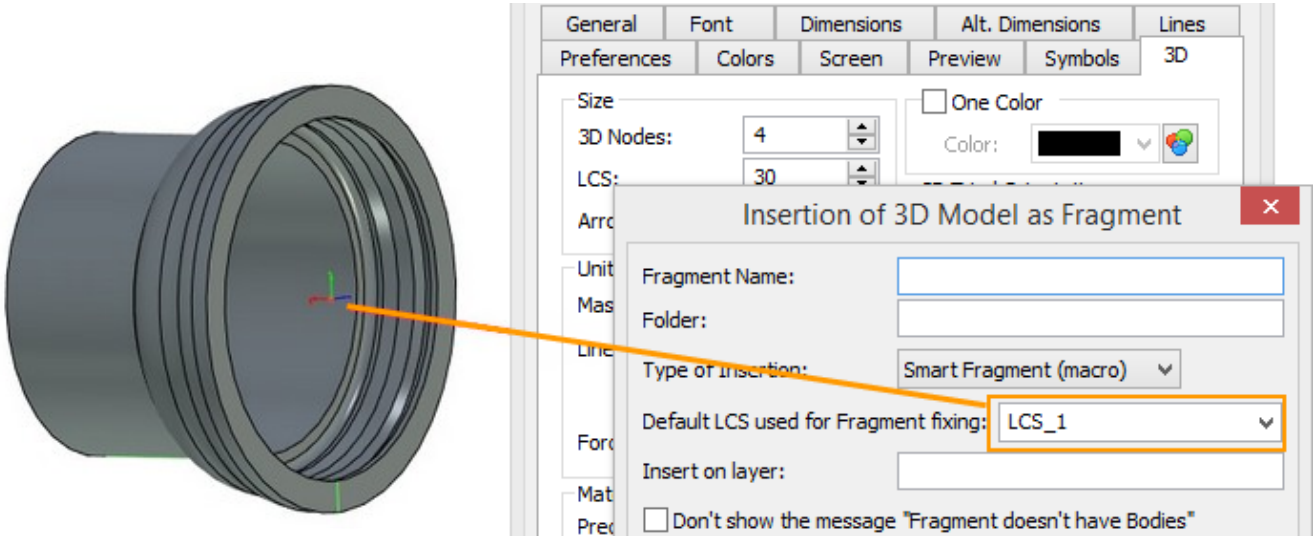
Guidelines for creating fragment of ending and connection

After creating 3D model, you need to:

➔ Create external 3D node (there must be only one). This node will limit the adjoining pipe. It should be placed on the "inner end" of fragment body.



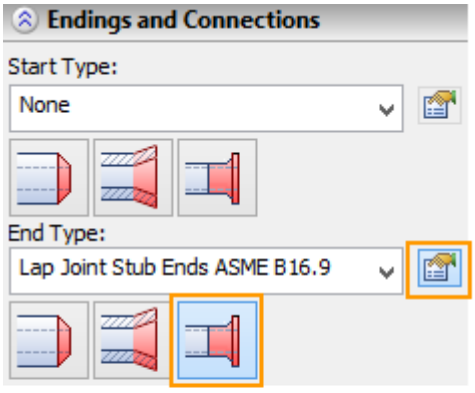
➔ Create external LCS. This LCS should be placed on the "outer end" of fragment body. It will be used for fixing the fragment. It must be assigned as a primary fixing LCS in the document properties command on the 3D tab, button [Fragment].

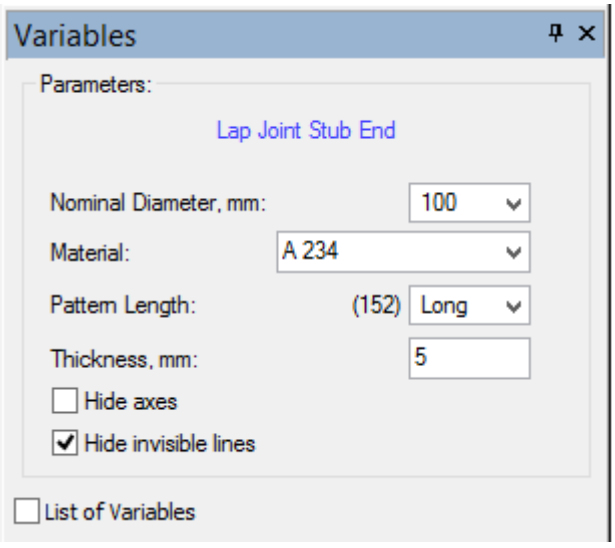


More information on how to create LCS can be found in the chapter "Local Coordinate Systems".

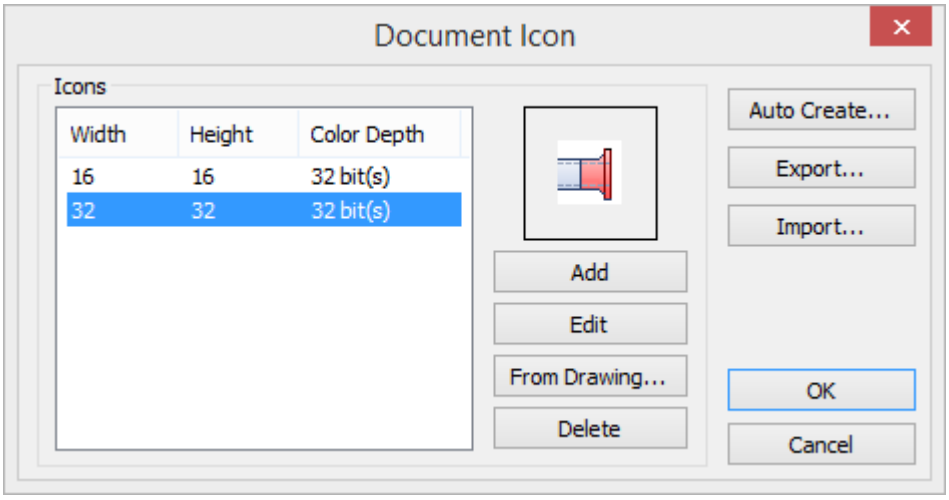
➔ Create custom dialog for entering values of external variables (optional step). This dialog is automatically embedded in the Properties window when you call editing of ending variables in the command of pipeline creation/editing.

Learn how to create a dialog in the chapter "Controls. Creation of Custom Dialogs."





➔ Create icon with the command **Tools** > **Special Data** > **Icon**. (Optional step).



Learn how to create icons in chapter "Document Preview and Icons."

➔ In the variable editor add diagnostics for invalid combinations of variable values.

Read more about diagnostics functions in chapter “Variables”.

Guidelines for creating fragment of the middle part of pipe

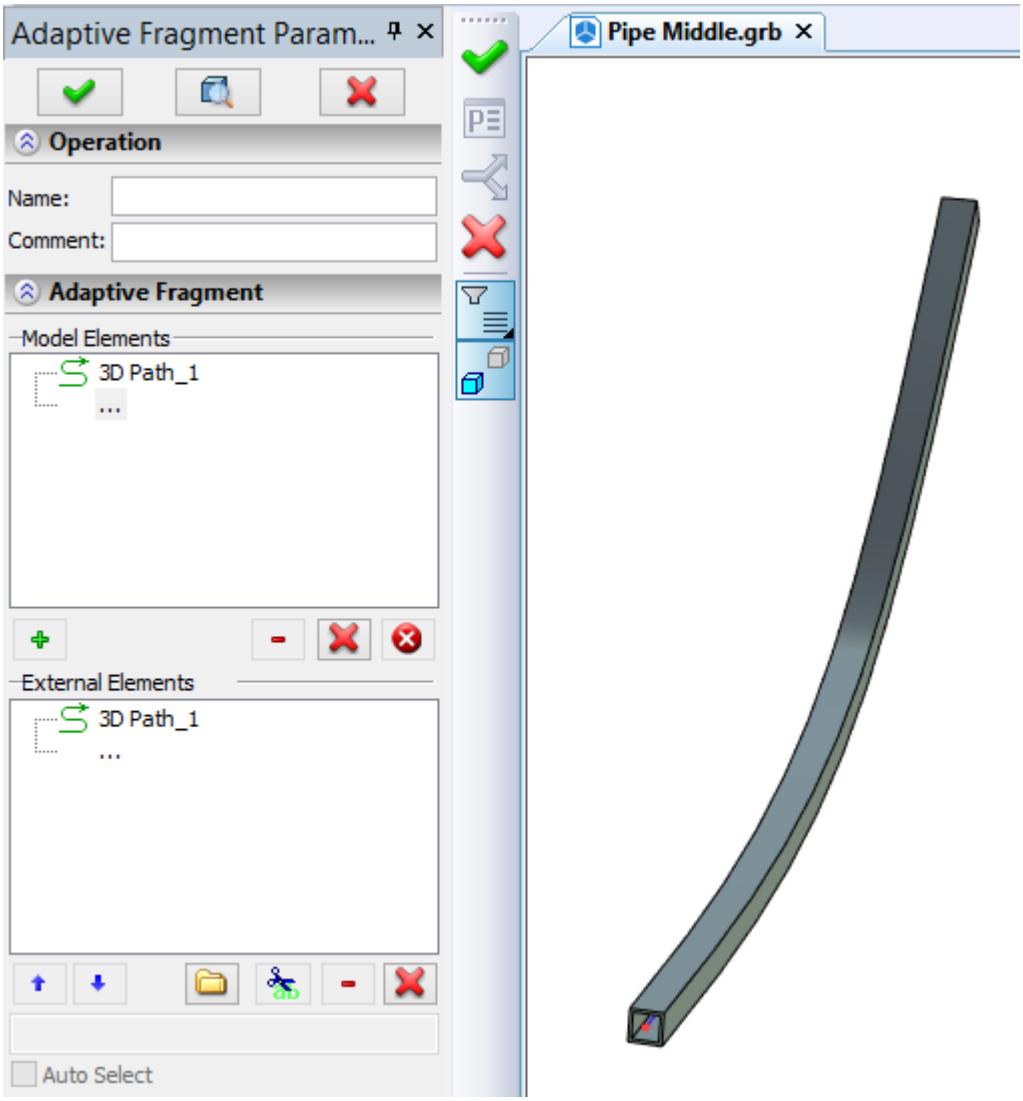
When you need to create some specific pipe geometry like corrugated pipe you may use special mechanism for defining pipe “middle” using fragment.

In the fragment file you must create variables described in the beginning of this section: **Diameter**, **Thickness**, **InnerDiameter**.

To create a 3D model of the middle of the pipe, you can use any convenient operation, such as sweeping along a path. It is important that the body is based on a 3D path.

After creating 3D model, you must set parameters in **Adaptive Fragment** command. This will create a pipeline using any selected route.

Read more about how to create adaptive fragments in chapter "Adaptive 3D Fragments."



Name. Parameter specifies the name of operation, which will be assigned to the pipe using this fragment.

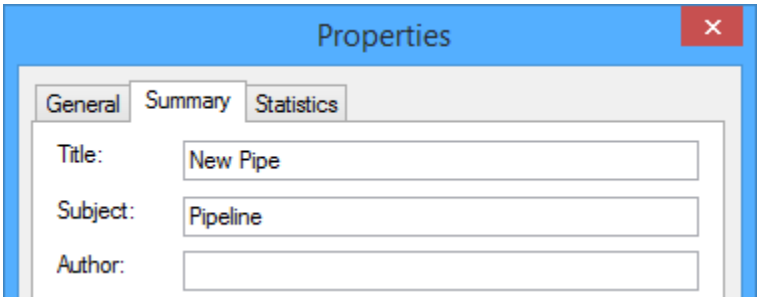
Geometrical parameters:

- **Path** (3D path or route). Element that defines the path of the pipeline.
- **3D pipe start node**
- **3D pipe end node**

> It is important to comply with sequence of geometrical parameters in the list. The first must be path, and then the start and end points.

Creation of icons and dialog of external variables for the fragment is not a mandatory action.

Name that appears in the list of endings and connections in the command of pipeline creation can be set in command **File** >> **Properties** >> **Summary**.



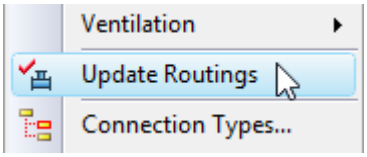
After performing all the above steps, you must save the file.

Updating Routing Models

After changing pipe parameters it is necessary to call the command **Update Routings** because full model regeneration does not apply to the chain of pipelines and fittings.

Keyboard	Textual Menu	Icon
<3PC>	Tools > Routing > Update Routings	

Command "Update Routings" called from the context menu for the selected segment of a pipeline, updates fittings locally, i.e. only at the ends of the selected segment.



Command **3PC: Update Routings** performs updates throughout whole pipeline structure.

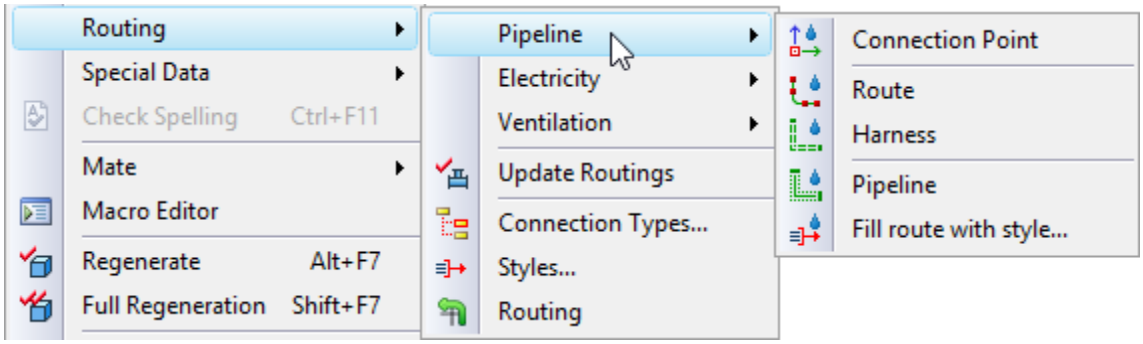
Typing of Routings

In order to automate routings design process, objects can be associated with certain types. Types allow you to exclude mistakes in operations of connecting routing elements to each other.

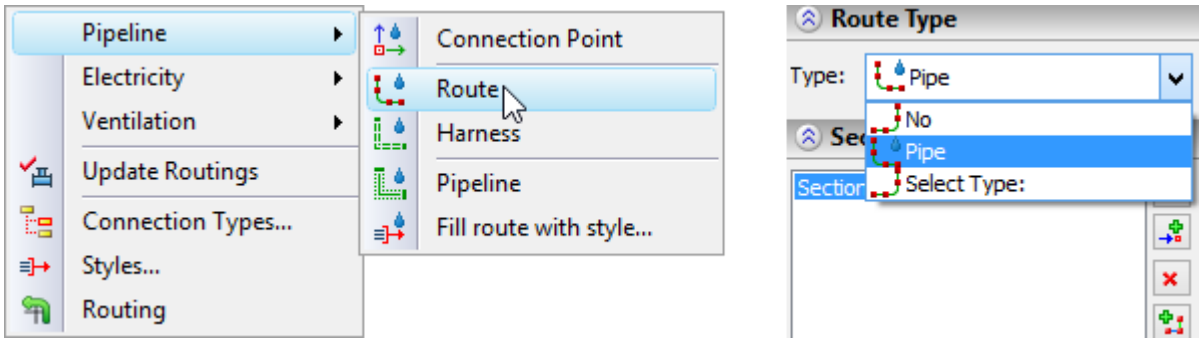
For description of objects compatibility rules and assignment of types use command **TI: Connection Types**. There, you can assign a valid correspondence between the routes and fittings, between the different types of routes.

It is possible to create objects with basic typing. The basic types include Wiring, Ventilation, Pipeline.

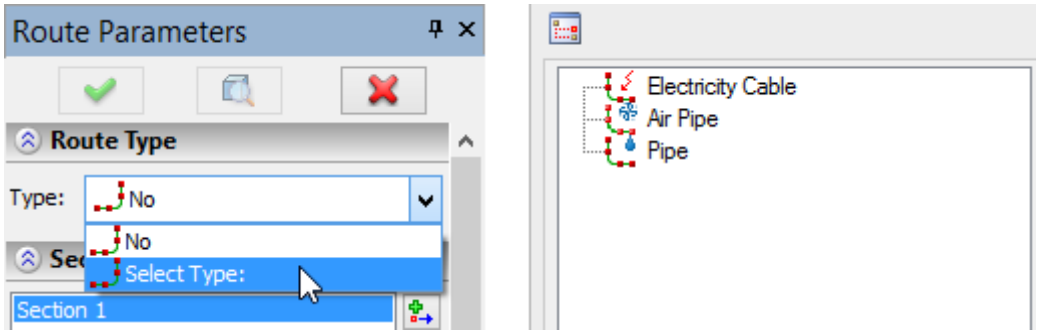
For these basic types menu **Routing** has commands to create connection points, routes, harnesses, routing objects, as well as command for filling route with style.



When you call command **Route** from one of the three groups of **Routing** text menu, system opens dialog of route command with the corresponding value of **Type** parameter (Wiring, Ventilation and Pipeline). To clarify type use option **Select type** in the drop-down menu. It will offer you types, which are the children of the base type.



If **Route** command is called using text menu **Construct** > **Route**, by using "Select type" option you can select type from all types available in the library.



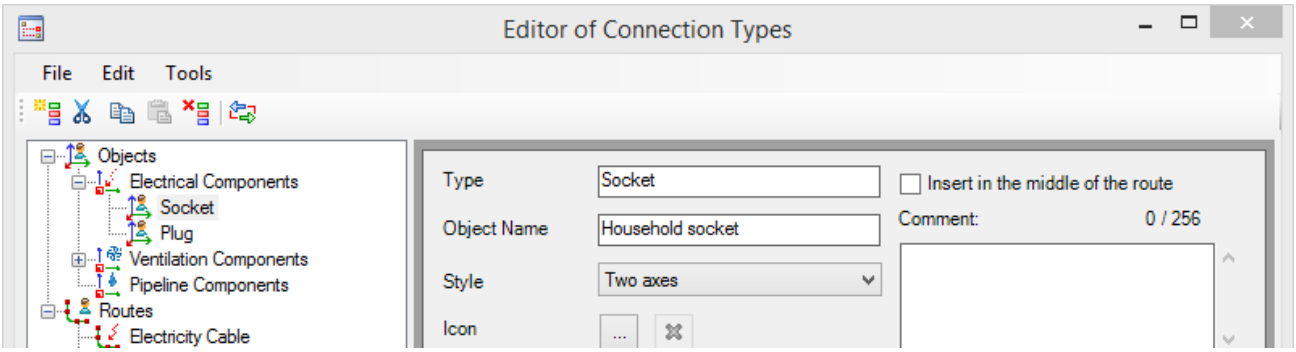
In the drop-down list displays the 10 most recently used types. When you call one of the commands with the base typing you will get types derived only from this base type. When you use the general Route command you will get all used types.

To adjust compatibility of different types you can use Editor of Connection Types.

Editor of Connection Types

With the help of editor of connection types you can set compatibility rules for various types of routings. Compatibility rules help to avoid mistakes when working with various libraries and

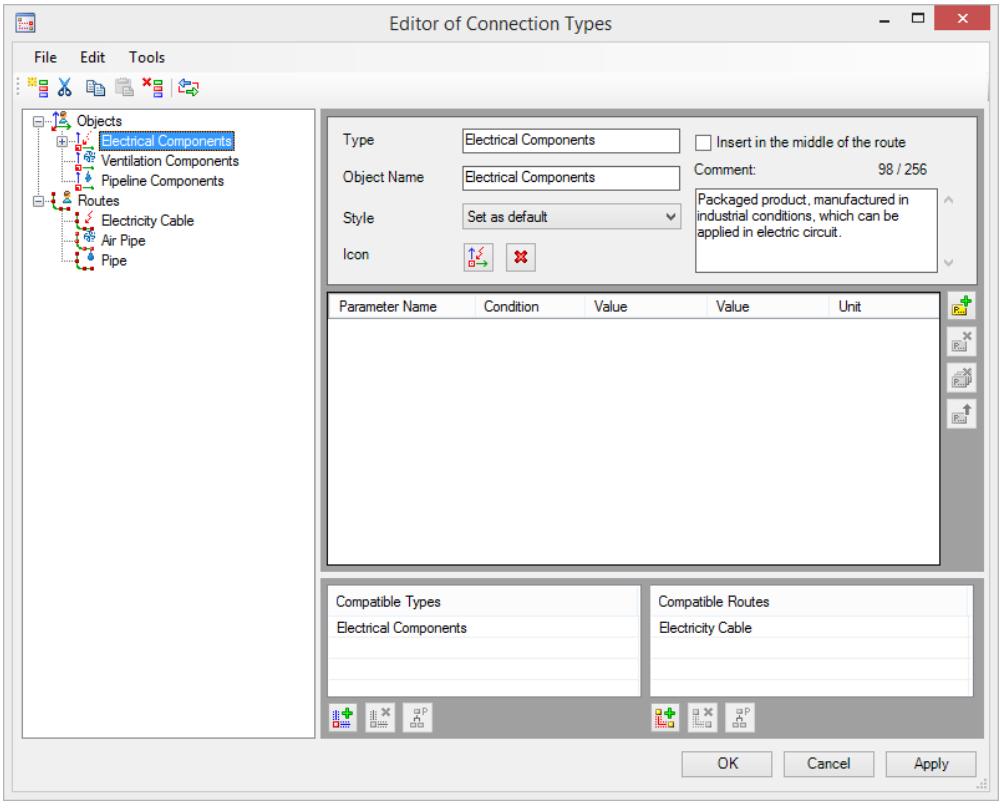
routings within the same project. For example, avoid accidental connection of low-voltage and high-voltage cables in the design process.



Editor can be called using **Connection Types** command of the **Routing** text menu.

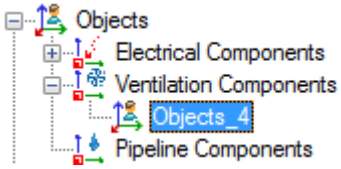
Keyboard	Textual Menu	Icon
<TI>	Tools > Routing > Connection Types ...	

In the left pane there is a tree of types. It consists of two main parts: "Objects" and "Routes".



Above the tree of types there is a toolbar for its editing.

Create. Allows you to create a new type. Object will be added to the selected type from the list. Parameters of a new type are described below.



Cut. The selected object will be cut from tree.

Copy. Allows to copy type

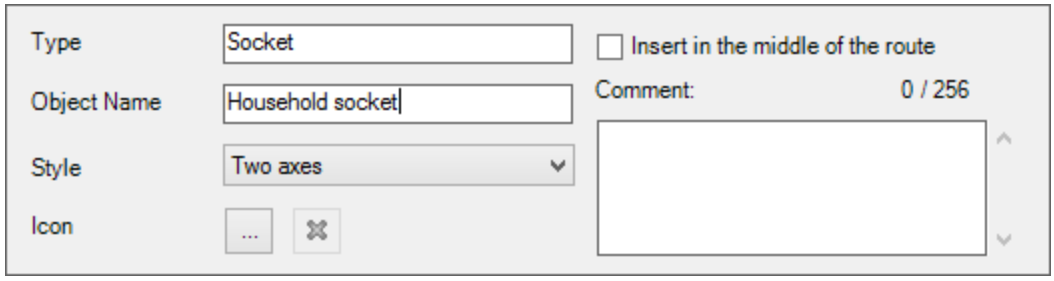
Paste. Allows to insert type. Works with with options **Cut** and **Copy**.

Delete. Allows to delete selected type.

Editor of Physical Parameters. Provides access to the internal directory of the physical parameters. Work with the directory is described below.

Parameters of type

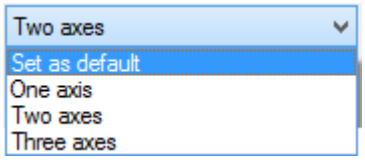
The right pane displays the set of parameters of the selected type.



Type. In this field you can specify unique name of the type to which the object belongs. Type name will appear in the list on the left side of the **Editor of Connection Types** and the drop-down list of types when you create objects (connection points and routes).

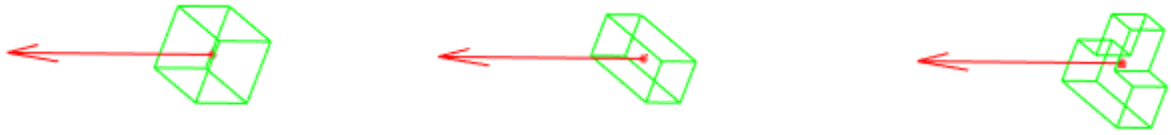
Object Name. Specifies the name of objects for a given type. This name will be displayed in the model tree. Names can be the same for different objects.

Style. Specifies the number of axes required for the proper connection of objects.



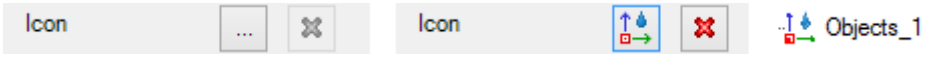
For example, such connection as "plug - socket" requires two axes to match.

Style is taken into account when creating connection points and routes.



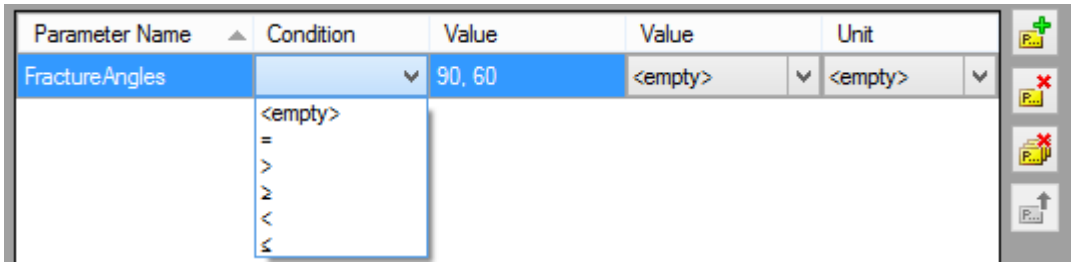
One axis Two axes Three axes

Icon. It allows you to set icon to be displayed to the left of the type. For selecting icon for the file you can use the button ... . In the window that appears you can see the available standard icons. To remove element icon, use the button .



Comment. In this field you can enter a comment to the selected element. The number of characters available for comment is displayed above the field.

For each element you can set a group of parameters.



Parameter Name. If two objects of the same type of parameter names coincide, these parameters are compared. When incompatibility is found, you cannot connect the objects.

Condition. Condition can be selected from the drop-down list.

Value. Is taken into account when comparing parameters.

Physical Parameter. From the drop-down list you can select the physical parameter. Physical parameters are set in the Editor of Physical Parameters.

> For more information about editor of physical parameters refer to section "Physical Parameters" below.

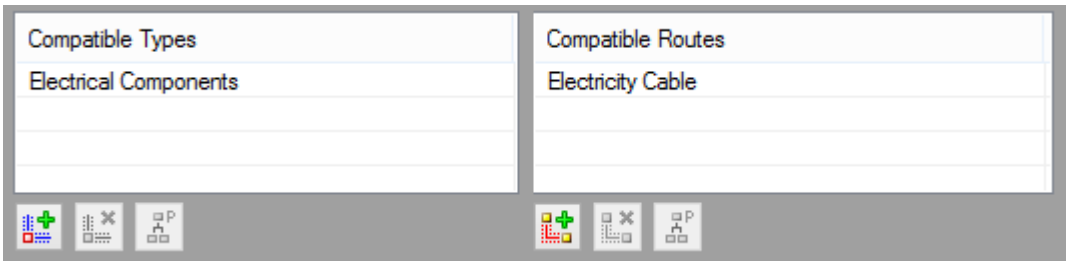
Unit. After selecting the physical parameter you can use set the corresponding unit.

On the right side of the dialog there is a set of buttons to control the inheritance of parameters. By default, objects inherit all parameters of the upper level objects.

Button shows parameters that selected type inherits. To change the inherited parameters use option .

- options allow you to add or delete parameters from the list.

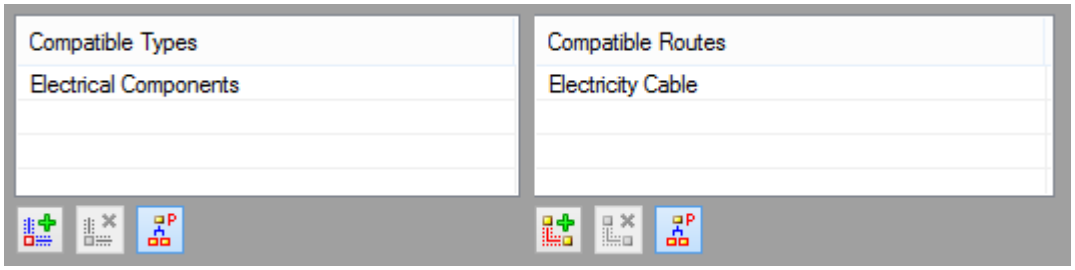
Compatible types can be set in the bottom of the window.



Options and allow you to add compatible types and routes from the list. Options and allow you to delete previously added types and routes.

Option allows you to control inheritance of parameter types. When enabled, all objects within a type inherit all of its compatible elements. After disabling the option, you can delete all but one inherited compatibility.

Only compatible types can be set for the routes.



The tree structure and description of types is contained in ConnectorType.xml file. To manage this file from the "File" menu, use the following commands:

- **Save** (active, if any item in the tree has been changed): saves ConnectorType.xml file in “C:\Users\User_name\AppData\Local\ Top Systems\ T-FLEX CAD 3D...” folder.

- **Import**: replaces the contents of the current ConnectorType.xml file with the contents of the imported file. If ConnectorType.xml file has been modified before importing, system would prompt to save it under another name.

When importing tree of types there are two options:

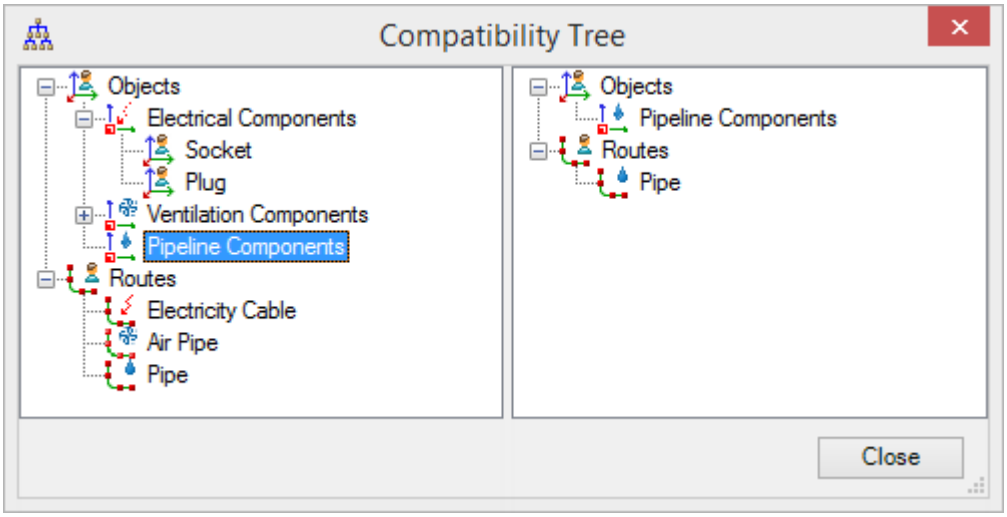
- With replacement: current list will be completely replaced by the list from external file.

- With addition of: the user will be prompted to select the types that need to be put in the current list. The import list will be filtered not displaying the types that already exist in the current list.

- **Export**: saves ConnectorType.xml file to the specified location. If desired, you can specify a different name for the file.

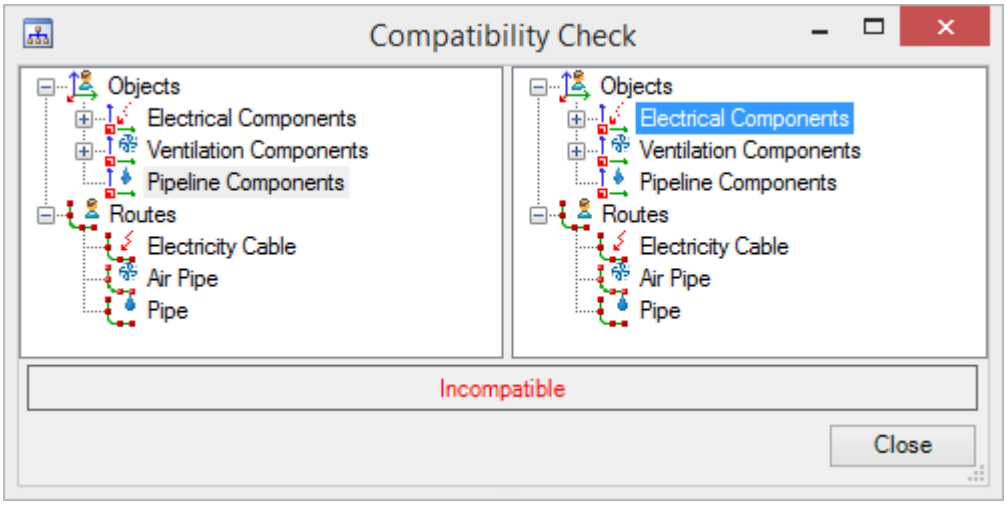
Compatibility Tree

Command "**Tools/Compatibility Tree**" of the "Editor of Connection Types" displays compatibility rules in a separate window. For the element selected in the left part of window systems shows on the right side all compatible objects and routes. Tree allows to see the current structure of compatibility.



Compatibility Check

Command **Tools/Compatibility Check** of the "Editor of Connection Types" allows you to check compatibility of any pair of elements in the list of objects and routes. Select item in the left pane and item in the right pane. Message in the bottom will tell about the presence or absence of compatibility.



Physical Parameters

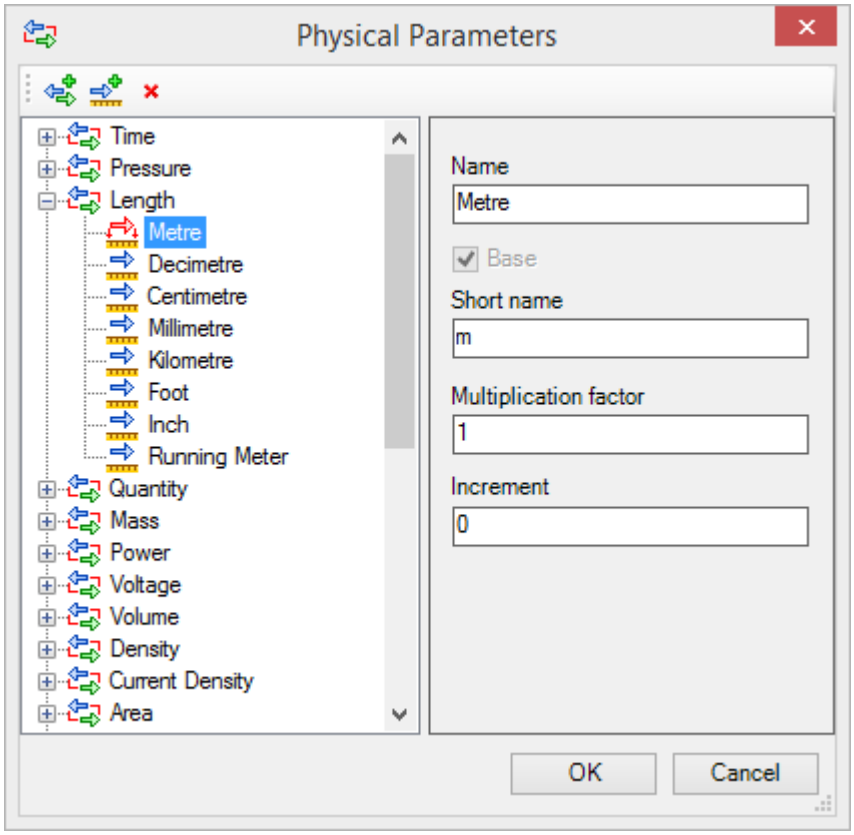
Command **Edit\Editor of Physical Parameters** of the "Editor of Connection Types" displays the internal directory of physical parameters for the editor of connection types. For each parameter there is a list of units with the name and the short name.

Flag **Base** sets the default measurement unit. For other units of measure there is **Multiplication factor** for conversion to the base unit. List of parameters and units of measurement can be edited using the following commands:

- Add parameter

- Add unit

- Delete



HARNESSES

Harness is part of the hierarchical structure of the route, uniting a group of routes or other harnesses. For example, harnesses may be used in automatic laying of routes.

To create a harness it is necessary first to build routes, which will then be combined into harnesses.

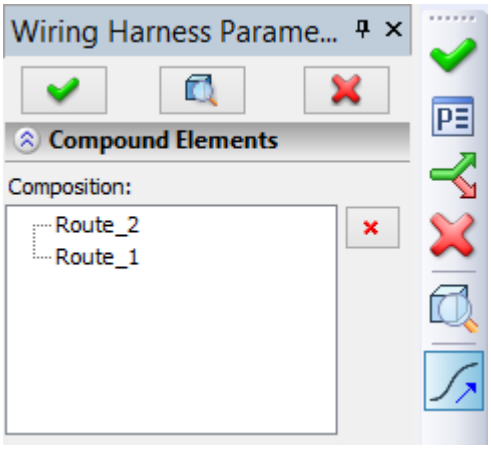
Call to create a harness command is carried out from **Routing** menu. Every new harness is set with the base type.

Icon	Ribbon
	Routing → Construct → Harness
Keyboard	Textual Menu
< >	Tools > Routing > Pipeline > Harness

After activating the command, specify the routes, which will be combined into a harness. Selection of routes is available with option in automenu.

> Obligatory condition of combining two routes into a harness is the presence of at least one point of intersection of two adjacent segments.

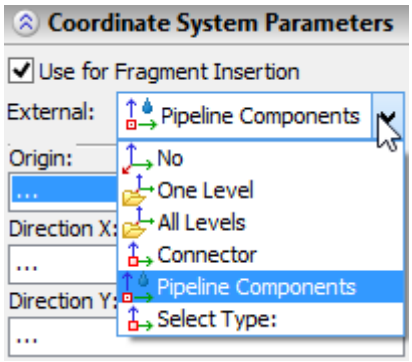
All selected routes are displayed in the **Properties** dialog in **Compound Elements** section.



To complete the creation of harness press .

CONNECTION POINTS

To automate connection of routing elements to each other you may use connection points - connectors. They represent LCS, which have certain types.



If the LCS type does not exist in the used hierarchy of types, next to the LCS there will be "from document" note.

When you create connectors, for their positioning you may use only objects with compatible type, or objects with no type assigned. When typed element is added to a model, system checks whether a given connection point may be used for connecting element, i.e. checks objects for compatibility.

For more information about creating connectors refer to chapter "Local Coordinate Systems".

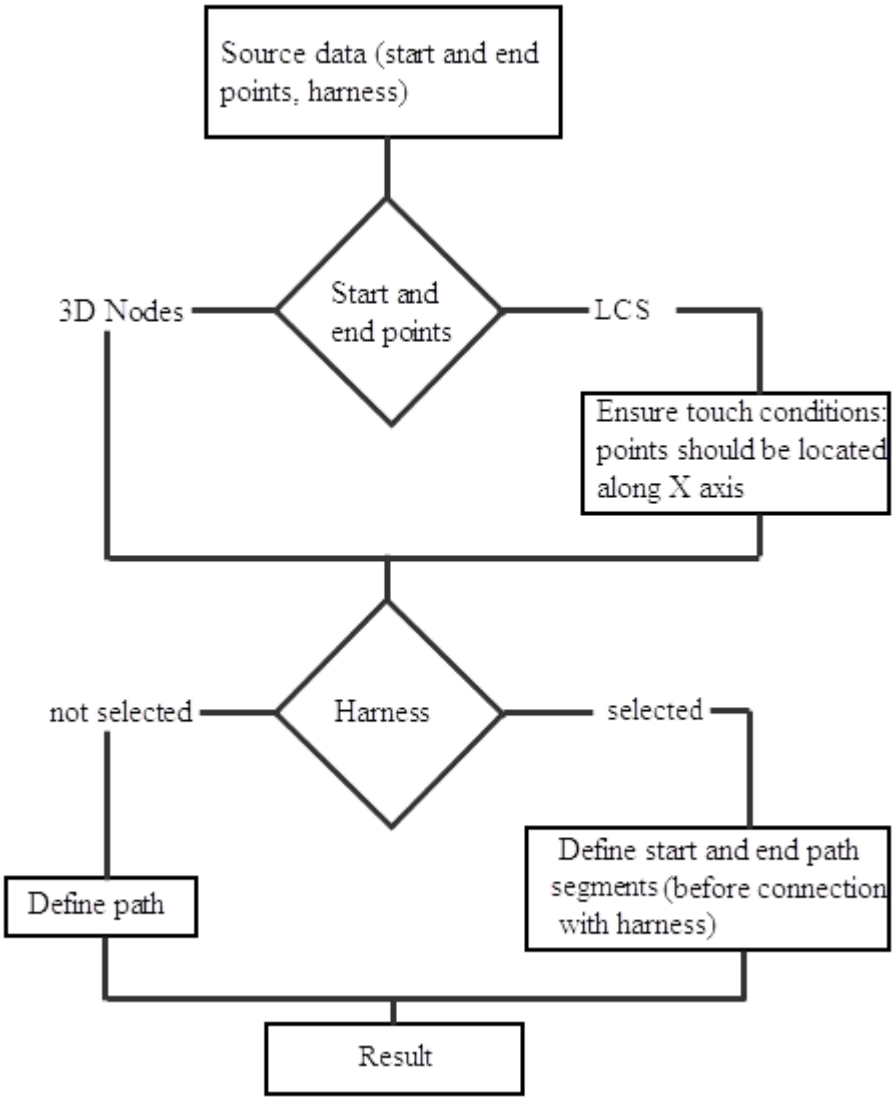
Automatic Laying of Routes

Routing command performs automatic laying of route between selected nodes. To set path of the middle route segment you can use also harnesses.

To access the command:

Icon	Ribbon
	Routing → Construct → Routing
Keyboard	Textual Menu
	Tools > Routing > Routing

The algorithm of working with the command can be represented as follows:



Consider an example.

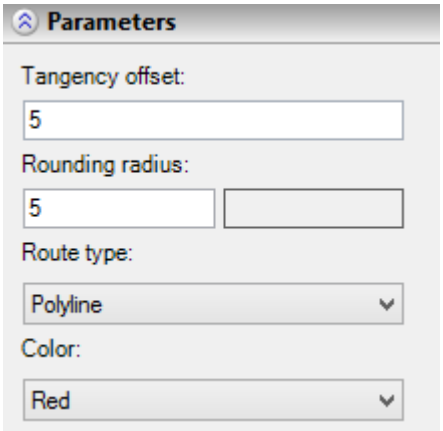
The route will go between the two LCSs. Part of the route will pass through the harness.

After calling the command, the following options are available:

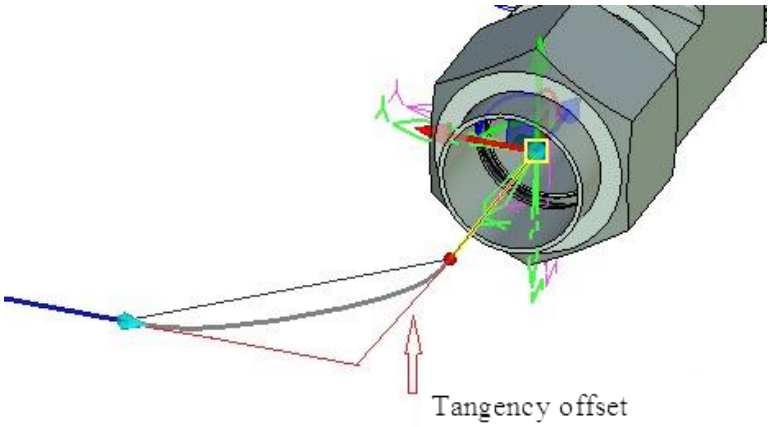
	<F>	Select start point
	<S>	Select end point
	<D>	Cancel start and end points
	<L>	Select harness
	<C>	Cancel harness selection

	<N>	Next
	<P>	Back
	<R>	Resume command operation after applying changes

In **Parameters** dialog you can set parameters of the route being created:



- **Tangency Offset** – specifies the distance by which the segment of the route offsetss from the intersection point with another segment to build tangential coupling between them.



- **Rounding Radius** – sets radius of rounding at the route break points.

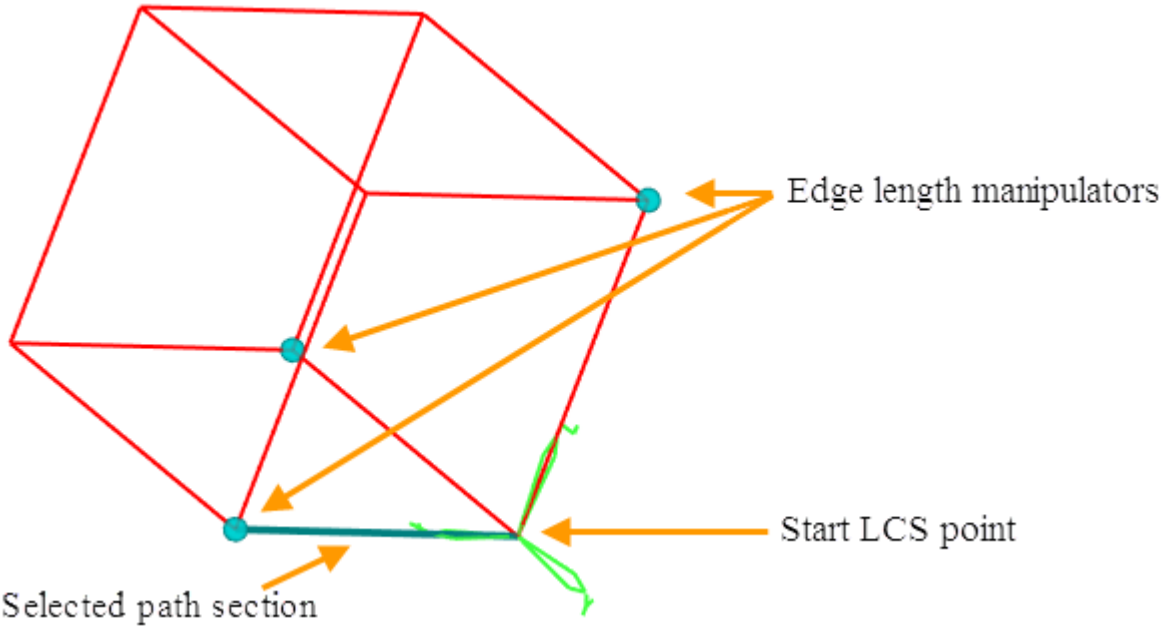
- **Route Type**: polyline, spline by points, spline by polyline - can be set separately for each route segment.

- **Color –** sets route color by default.

Laying the route takes place in several stages. At each stage a segment of the route is constructed. The segment is laid along the edges of the manipulator-box or rectangle. This manipulator appears for each route segment. You can select edges, by which the route will be laid, and set the length of the edges.

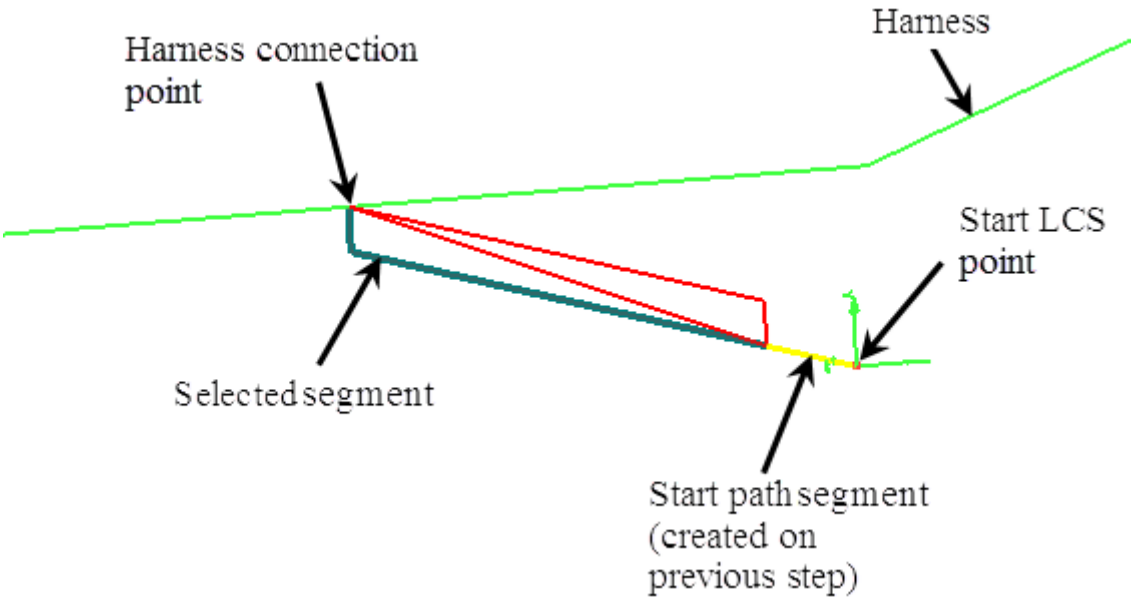
After the initial data for constructing the route - start LCS , end LCS and harness - are set press button **Next**.

After that you will see manipulator-box, with which you need to specify the variant of the route approach to the first point along axis X. To specify the variant, you must select appropriate edges of the manipulator.

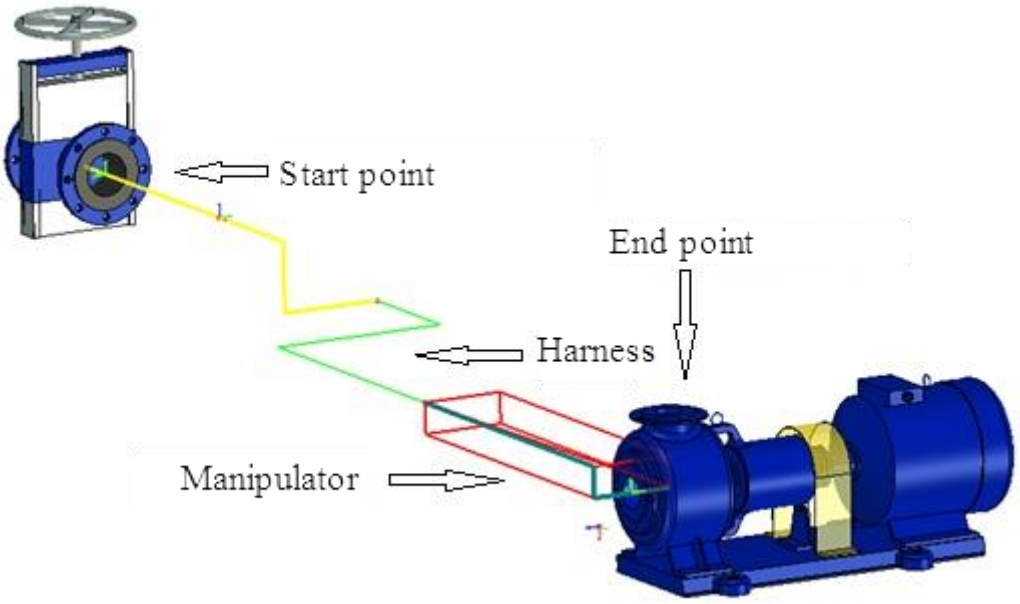


Length of edges can be changed either directly with manipulator or in the Properties window.

After selecting necessary variant, press and repeat the steps for the second point. At the next two steps, system automatically finds the nearest points for connecting with harness and provides a way to define path segments for these points. The action is performed similar to the previous one, by selecting edges of the box. Similarly, you can construct path for the route without using harness.



The final step - a preview of the result. You can confirm the creation of the route by pressing, or with option return to the previous steps to adjust any settings.



After creation the route is available for editing by standard tools of editing routes: you can change the types of segments, add rounding, etc.

Routing Styles

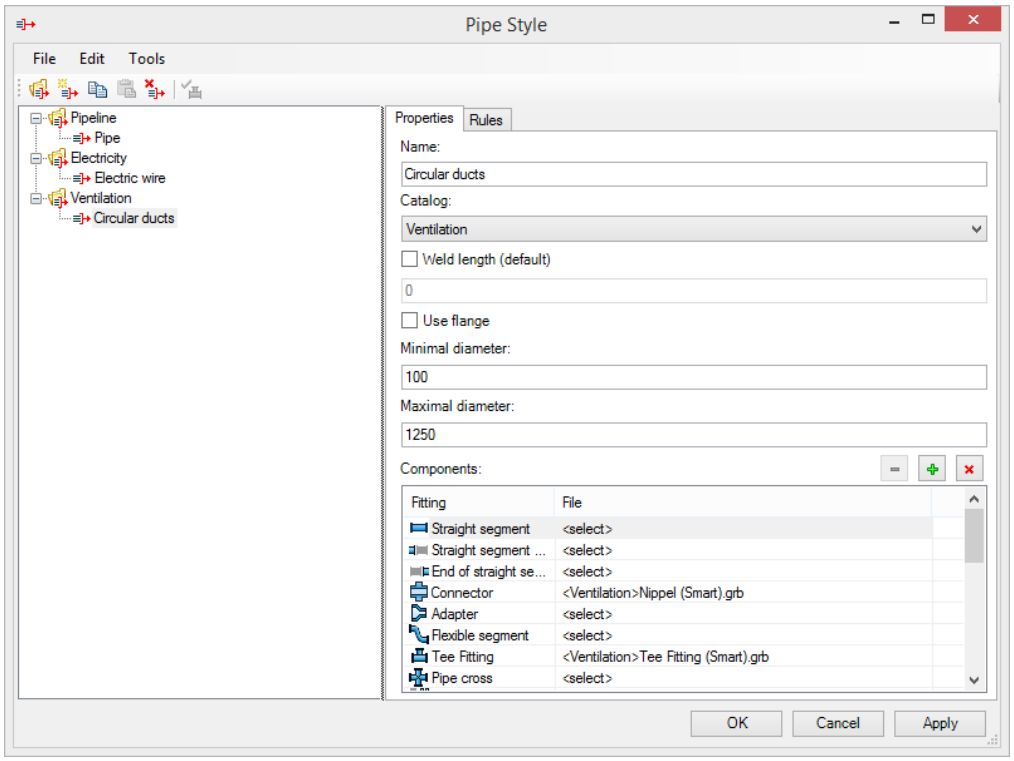
Style is a group of settings designed for automatic filling of routes with the routing objects. Styles usually correspond to standards that are commonly used in the routing design. For example, parts of the pipeline must often comply with certain requirements in length and diameter; in addition, these parts should include fittings of certain materials.

Style definitions are managed in the editor of Styles. With the editor, you can view, add, edit, and delete styles. The definitions of styles can be also imported and exported using file in .xml format. You can change any of the default styles, or use them as the basis for creating new custom styles. In addition, custom styles can be created based on the empty style.

Editor of styles can be called from the **Routing** menu:

Icon	Ribbon
	Routing → Options → Styles
Keyboard	Textual Menu
	Tools > Routing > Styles

After calling the command you will see the dialog of **Routing Styles** editor.



There is a toolbar in the upper part of the dialog:

Create Catalog creates new folder-catalog for styles.

Create Style creates new style in the current catalog.

Copy Style copies selected style.

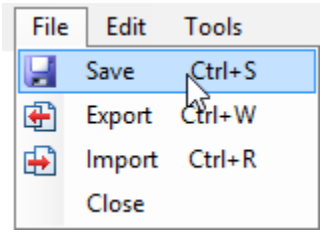
Paste Style paste copied style into selected catalog.

Delete deletes style or catalog.

Update information about fittings – updates information about routing objects (fittings).

On the left there is a list of catalogs and styles availabe for usage.

Created styles can be exported and imported in *.xml format with the appropriate commands in the **File** menu.



Parameters of the selected style are displayed in the right pane.

Properties tab

Name. In this field you can set the name of style that will appear in the list in the left pane.

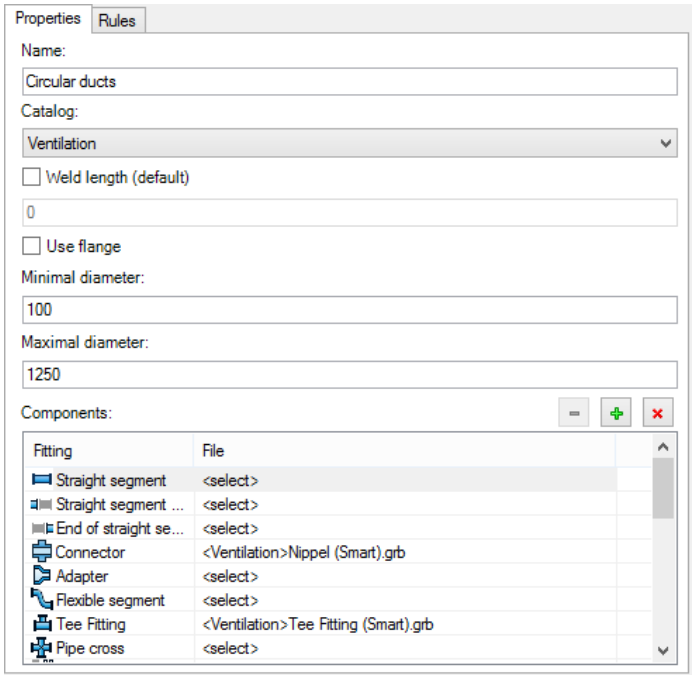
Catalog. The drop-down list contains catalogs that may include the selected style. You can create a new catalog by selecting the appropriate menu item from the drop-down list, if desired.

Weld length (default). When the flag is active, you can set the default weld length.

Use flange. Defines the usage of flanges for style.

Minimal and Maximal diameters define the permissible routing diameters.

Components window displays the list of fitting types and their corresponding files.



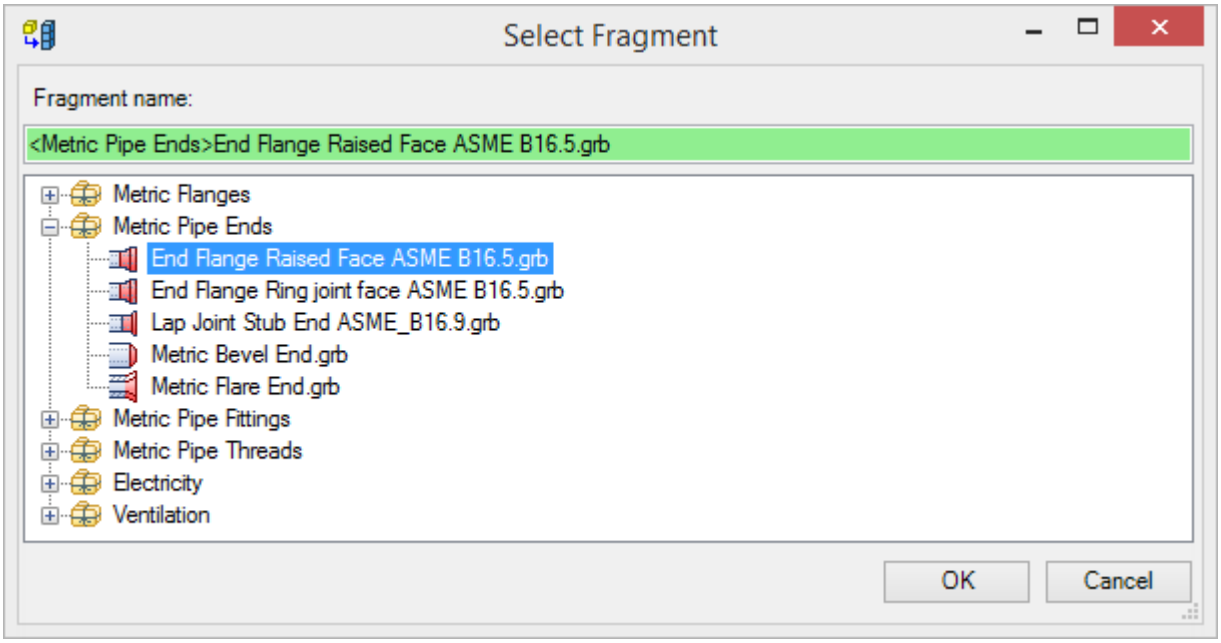
You may use the following button when working with the **Components** dialog:

Delete Component. Deletes selected file from the list.

Select Component. Adds file to the list.

Delete All Components. Deletes all files from the list.

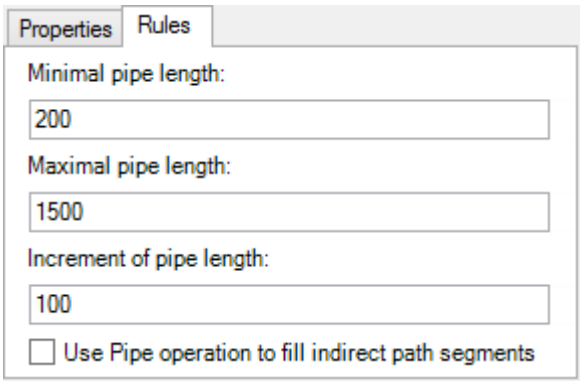
After mouse click in the <select> field or pressing button , the following window appears:



This window displays all library files available for selection.

If the fragment name is highlighted in red, it cannot be selected.

Rules tab.



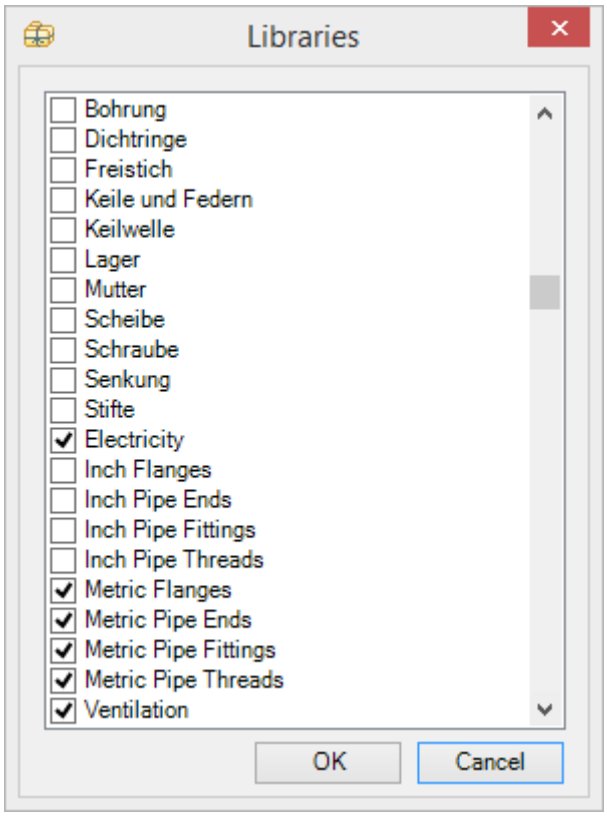
Minimal pipe length, Maximal pipe length - set the minimal and maximal length of the pipeline segment.

Increment of pipe length. You can specify the length of increment. Increment of length is a step with which the system takes into account the above conditions.

Use Pipe operation to fill indirect path segments.

When the flag is set the pipeline will be bent with a given style.

List of libraries for selection consists of libraries marked as available for selection. Use **Tools** > **Libraries** command of Routing Styles dialog menu to mark necessary libraries in the list of all system libraries.



Filling Route with Style

To apply styles, use command route **Fill route with style**, available for all types of routings.

Icon	Ribbon
	Routing → Construct → Fill route with style
Keyboard	Textual Menu
	Tools > Routing > Pipeline > Fill route with style

The following actions are available in automenu after activating the command:

	<P>	Select Route
	<B>	Select start point on route
	<E>	Select end point on route
	<D>	Cancel start and end points
	<A>	Select alternative routes

	<L>	Cancel alternative routes
	<N>	Next
	<M>	Back
	<C>	Clear Route
	<R>	Resume command operation after applying changes

Upon activation, you need to select the main route to create routing.

and allow to set limit for routing. Element will be generated or deleted on the specified segment of the route.

Cancels selection of the start and end points.

Allows you to choose alternative the routes. These routes must be connected with the main route.

Cancels selection of all alternative routes.

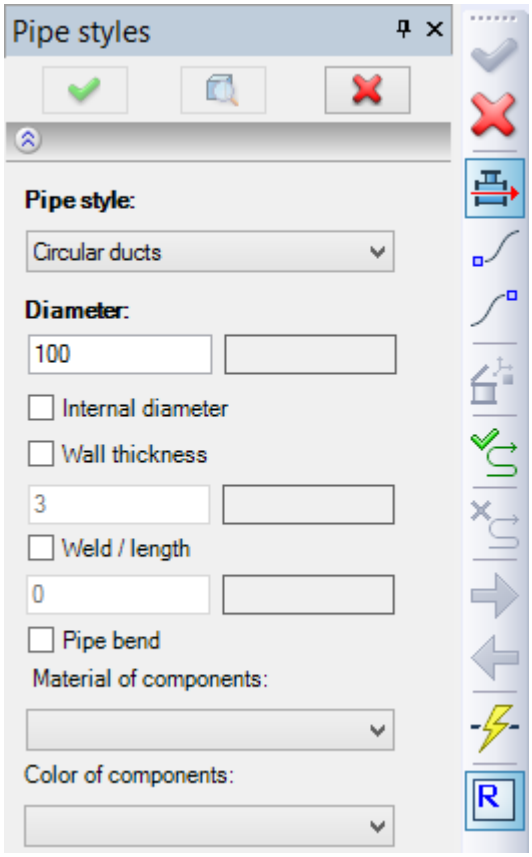
Provides preview of the operation result. After the first press preview is done on the main route , after the next pressing – on alternative .

Returns to the previous preview step.

Deletes all routing components of a route - nodes, LCS, fragments.

Allows you to create a new routing immediately after the creation of the current routing.

In the **Properties** window, you can specify parameters for the created element.



Pipe Style - one of the styles for filling routes can be selected from the drop-down list. The list stores up to ten previously used styles.

Diameter – pipeline diameter is set here. Diameter should be in the range specified in the parameters of the selected style.

Internal Diameter - when checked, the defined diameter is treated as internal.

Wall Thickness - allows you to specify thickness of the pipe wall.

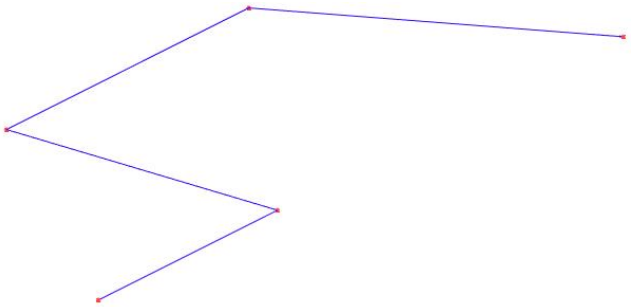
Weld / length – this field defines the gap between the components for welding.

Pipe bend. If library does not have appropriate fitting for rounding, then this option will bend the pipe. When you bend pipe weld option is not taken into account.

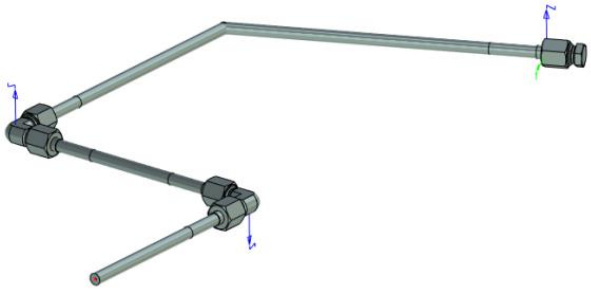
Material of components. This list contains materials used in the current model.

Color of components. Color of shading for pipeline elements.

Green and red colors in the data fields show the possibility of correct usage of the entered value.



Original route



Route filled with style

3D Pictures

Unlike fragments, a 3D picture is a hull of a three-dimensional object, without parametric properties and relations between the picture elements. 3D pictures are used for creating three-dimensional assembly structures.

3D pictures cannot be used as the references for creating other elements, since their faces cannot be selected. You cannot use 3D pictures in Boolean operations either. Their 2D projections cannot be created. The command "Exploded View" cannot be applied to 3D pictures.

You can insert into a T-FLEX CAD document the 3D picture files in the following graphic formats: grb (T-FLEX CAD files), wrl (VRML files), iv (Open Inventor files), x3d, 3ds, ply, obj, stl, dxf, dwg.

To save a 3D picture is part of a T-FLEX CAD document, that already contains a 3D model, set the parameter **Save 3D Picture** on the **Save** tab of the **Document Parameters** dialog box.

To save a 3D picture in a separate file in the iv, wrl, 3d, ply, obj, stl formats, use the command **File > Export**.

> Remember: storing a 3D picture inside a T-FLEX CAD file noticeably increases the file size. The size of the 3D picture itself depends on the mesh density defined in the Document Parameters at the time of the picture creation.

Creating 3D Pictures

To insert a 3D picture into a model, use the command **3I: Insert 3D Picture**:

Icon	Ribbon
	3D Model → Modify → 3D Picture
Keyboard	**Textual Menu**
<3I>	Operation > 3D Picture

Upon calling the command, you get the access to the following actions:

	<Q>	Repeat existing 3D Picture
	<S>	Select or create source LCS
	<T>	Select or create target LCS
	<F>	Select 2D Fragment
	<O>	Select file
	<R>	Repeat previous 3D Picture
	<I>	Multiple iterated insertion of 3D picture

To insert a 3D picture into an assembly, you need to do the following steps:

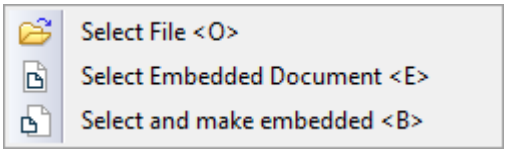
- ✓ **Select a 3D picture**.

  The selection can be done in three ways:

  - ➔ By applying a new 3D picture (the option);
  - ➔ By selecting an already inserted 3D picture (the option).
  - ➔ By selecting an already inserted 2D fragment, using its 3D model (the option).
- ✓ **Select or create the 3D picture's source coordinate system** (the option).
- ✓ **Select or create the target coordinate system** (the option).
- ✓ **Define the 3D picture parameters** (*optional*).
- ✓ **Finish inputting the 3D picture** (in the automenu or in the property window).

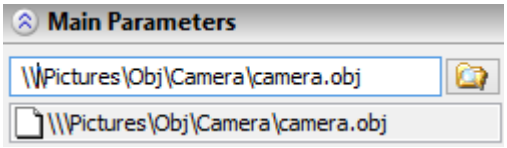
Files can be selected either from the T-FLEX CAD library or from any folder on the disk.

In the drop-down list of option you can choose one of the three sources for the 3D picture.

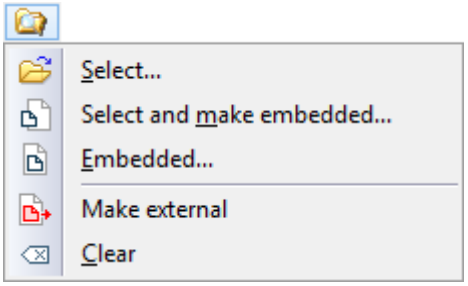


When you select option **Select and make embedded**, inserted 3D picture will be stored in the current document, not in an external file. Option **Select embedded document** appears only for files with embedded documents.

The name of the selected/internal document will be shown in the command's properties window (to the left of the button). Grey (not accessible for editing) field, located a little below is informational and shows the absolute path to the file from which the picture is taken.

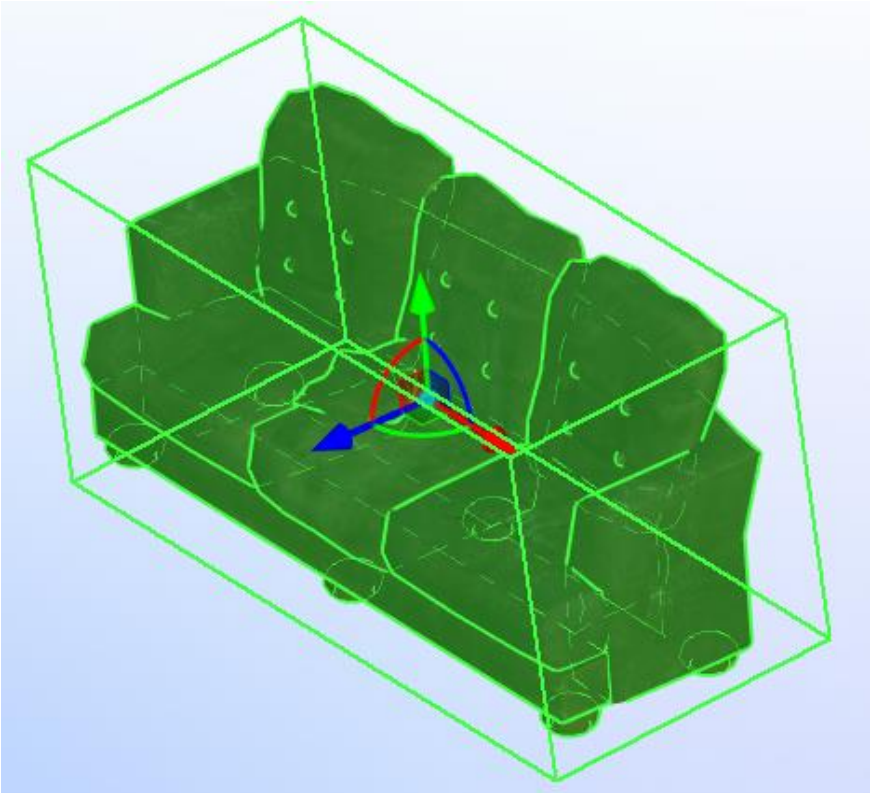


If you press button after selection of the pathname, the drop-down list will appear:



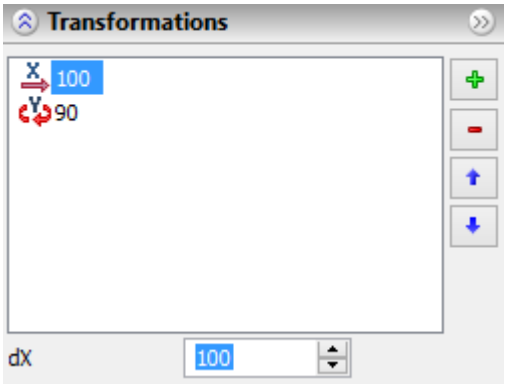
The first three options are the same as for option . Option **Make external** allows uploading of the embedded picture to an external file. Option **Clear** allows quick removing of the pathname.

After adding 3D picture, you can change its position in space with special manipulator that appears in 3D window. By default, the origin of the manipulator is located in the center of the bounding box. If necessary, you can change the position of the manipulator by using option **Select or create source coordinate system**. Unlike the creation of 3D fragment, the original coordinate system of the 3D picture cannot be bound to the picture, because it has no geometric data. In the rest, the process of selection/creation of source and target coordinate systems is similar to the process of inserting 3D fragments. This process is described in the Chapter **Assigning Transformations to 3D Entities**.



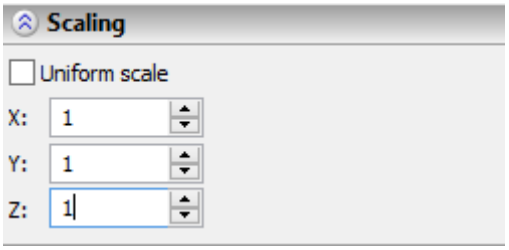
3D Picture Properties

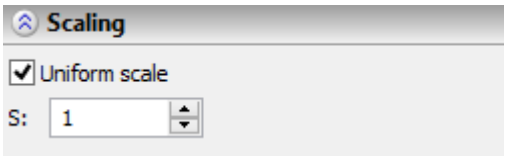
Transformations of 3D picture (translations and rotations) are displayed in the **Transformations**. Optionally section of the properties window. Optionally, you can specify exact values for any transformation in the list.



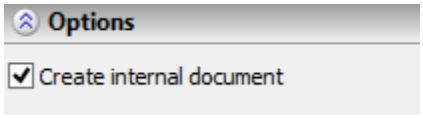
For the inserted 3D picture can be specified the units of measurement. After that, its size will be automatically recalculated. This function is useful for formats, which do not contain data about source measurement units of the picture. In this case, system will use measurement units specified in section **3D** of **Document Parameters** command.

In the properties window, you can specify the scale of 3D picture. The scale may be uniform along all axes or can be set separately for each axis. To specify a non-uniform scale it is necessary to remove the flag **Uniform scale**.





In section **Options** you can set flag **Create internal document**. If the flag is active, the picture file is saved in the current document. This allows you to increase the speed of data loading. Otherwise, when you open a document, the system loads an external picture file.



Option allows to reinsert previously created picture.

Option starts multiple insertion of previously created picture. The insertion is repeated until it is interrupted.

USE OF 3D PICTURES FOR CREATING LAYOUTS

When creating layouts based on objects with fixed dimensions, we recommend using 3D pictures. When rebuilding such 3D assembly models, 3D picture regeneration is much faster than assemblies consisting of 3D fragments.

The layouts using 3D pictures are created in the same way as those based on 3D fragments, with the only difference that the 3D picture is not created automatically when inserting a 2D fragment. The 3D picture is inserted manually by selecting the 2D fragment.

> When inserting a 2D fragment for creating a layout using 3D pictures, make sure that the flag is turned off, **Create 3D Fragments automatically,** on the **Fragments** tab of the command **SO: Set System Options** ("Customize > Options…").

You can get the detailed description of this system capability in the chapter "3D Assemblies Creation" (the section "Assembly creation from 3D fragments. 'Bottom-up' design technique") with the corrections for 3D pictures.

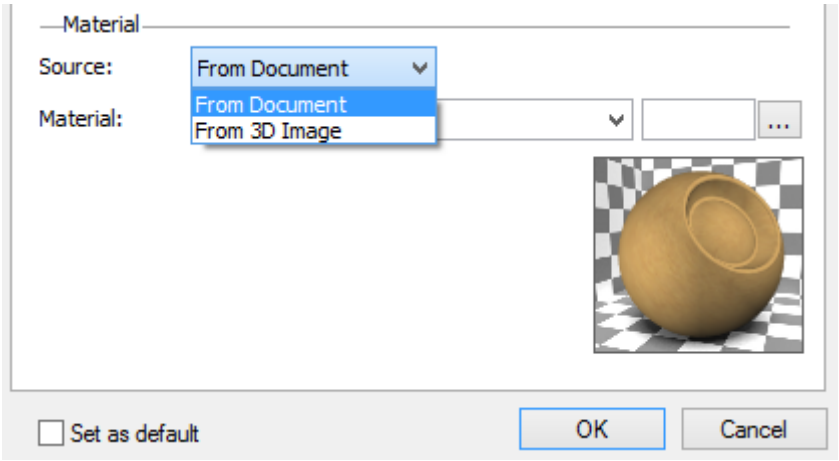
> Only 2D fragments that were saved with enabled option "Save 3D Picture data in model file", can be used for layouts. Note also that only rendered views may be created as 2D projections of assemblies built from 3D pictures.

USAGE OF COATING MATERIALS

You can apply coating material for 3D pictures. There are two ways to do it.

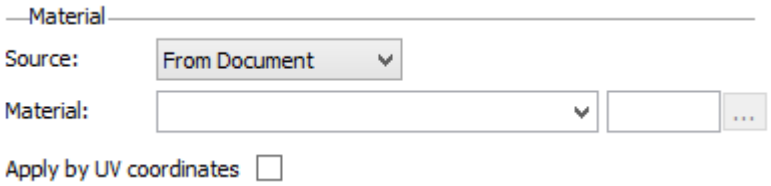
The first way is to apply the coating material to the 3D picture using **drag'n'drop** function. For this purpose, you need to open window **Materials** and to use the coating material from the list of model materials or from the **Coatings** library.

The second method is to choose the coating material in the properties window of 3D picture. To do this, press option in automenu and choose the tab "General". Then, in the drop-down list of field **Source** select option **From document**. After that, the field **Material** becomes available. In this field, you can select the coating material for 3D picture.



It is important that the coating material can be applied only for the entire 3D picture, because it represents a single object.

The option **Apply by UV coordinates** is available in the parameters of 3D Image. The option allows to apply material by UV coordinates that are stored in the 3D image file.



3D Pictures Editing

To edit a 3D picture use command:

Keyboard	Textual Menu	Icon
<3EI>	Edit > Operations > 3D Picture	

More information about this command can be found in the chapter **Editing**.

When editing a 3D picture it is possible to replace it with another one. This new picture can be either loaded from a file or can be defined by selecting other existing picture from the scene. For the last case, you need first to use option:

	<F4>	Execute edit command

Next you need to select option in automenu and click on any existing 3D picture in the scene. Current 3D picture will be replaced with it.

External Model

This command is intended for inserting 3D models in the Parasolid, STEP, ACIS , IGES, Siemens NX (Unigraphics), Creo/ProE, SolidWorks, Catia V5, Catia V4, Autodesk Inventor, Solid Edge, Rhino, I-Deas, JT, PRC, VDA-FS formats. The model inserted in this way becomes a normal operation of T-FLEX CAD. Remember, however, that an external model that is made of several operations becomes a single object upon inserting in T-FLEX CAD. To do further manipulations over a separate body of such model, you first need to use the operation **Divide Solid**.

Inserting an external model resembles creation of a 3D fragment, however, with limited capabilities. Thus, for example, you cannot set the values of external variables. Neither can you select the original attachment coordinate system (the model global coordinates are used instead).

Modes of Working with External Model

When inserting an external model, the current document keeps the reference to its file and its original geometry. In the future, the user can select one of the two modes of working with the external model:

In the first mode, the system tracks all changes to the external model file. If the file modifies, the external model adjusts in the current document.

In the second mode, the changes to the external model file are ignored. When working with the model, the geometry is used that was stored in the current document upon the last exchange with the model file. Nevertheless, the reference of the model file is saved in the current document anyway, which allows reverting to the first mode. Use of this mode provides fast handling of the current document with less memory required, however, it increases the size of the document when saved on the disk. This mode supports working with the document with an external model inserted in it, even when the external model file is absent.

The second mode is turned on automatically, if the system fails to find the external model file.

External Model Insertion

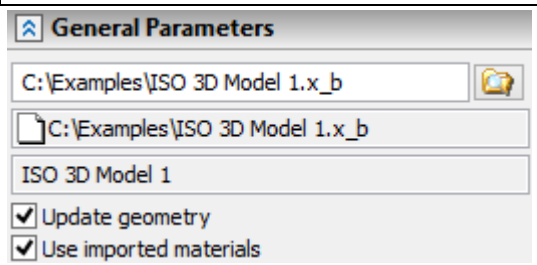
To insert a Parasolid-format model, use the command **3MO: Insert External Model**:

Icon	Ribbon
	3D Model → Modify → External model
Keyboard	Textual Menu
<3MO>	Operation > External Model

To insert an external model, you need to do the following steps:

1. **Select external model file**. The selection of the external model file is done in the command property window or by the option:

	<1>	Select External Model



There are two additional information fields in the external model properties dialog below the file name input field. One of them displays the full path to the selected external model file, the other one - the external model source file (that is, the file based on which the model was created).

The source file data is displayed on the condition that the source file contains the respective information.

The **Open Source File** command will be available in the context menu of the created external model. This command opens the source file if it exists in the path specified, by using the application matching the given file type.

2. **Select or create source coordinate system**.

   Selection or creation of source coordinate system can be done by option:

	<S>	Select or create source LCS

   Coordinate system of external model is used by default.

3. **Select or create target coordinate system**.

   Selection or creation of target coordinate system can be done by option:

	<T>	Select or create target LCS

   This option is set automatically upon selecting the external model. Selection of the target coordinate system amounts to selecting an LCS at which the external model's coordinate system should be placed. If no coordinate system is selected, the global coordinate system is used by default.

   Selection of the target coordinate system can be undone by the option:

	<3>	Reset target LCS

   The process of selection/creation of source and target coordinate systems is described in the Chapter «**Assigning Transformations to 3D Entities**».

4. **Define the external model parameters** (*optional*).

   The following parameters can be defined in the operation property window:

Update geometry. This parameter defines the mode of working with the external model. When the flag is set, the first mode is used (tracking all changes in the model file). Unsetting the flag turns on the second mode.

Use imported materials. This parameter supports use of the original materials of the external model.

The external model parameters can be defined in the common operation parameters dialog box called by the option:

	<P>	Set entity parameters

5. **Complete operation creation** (in the property window or in the automenu).

While creating an external model, the system runs the consistency check on the imported model. If there are errors in the model, then a window is displayed with the error message and suggestions for possible actions.

An external model can also be created by executing the command **IM: Import Drawing Or Model** (see the chapter "Exporting and importing documents" in the two-dimensional design and drafting user manual).

External Model Editing

To edit an external model use command:

Keyboard	Textual Menu	Icon
<3EI>	"Edit > Operations > External Model"	

You can change position of chosen external model or replace it with another one, loaded from a file with the help of this command.

More information about this command can be found in the chapter **Editing**.

ARRAYS

An array is an operation in T-FLEX CAD for creating copies of already existing 3D objects.

The placement of created copies (array elements) depends on the array class: linear, circular, array by points, array by path, parametric array. Array elements can be positioned along the direction vector, at the specified step (linear array), on a circle about the specified axis (circular array), along a path (array by path), at the specified 3D points (array by points) or following the prescribed parametric law (parametric array).

The source objects for creating an array (copy objects) can be selected among 3D construction elements, operations, Bodies and faces. The set of objects replicated in the array can contain several elements, however, of the same type.

The arrays are divided into the following types, depending on the type of the objects to copy: arrays of construction elements, arrays of operations, arrays of Bodies and arrays of faces. All arrays of the same class share in creation and editing techniques, regardless of the array type.

The copies resulting from any array except array of Bodies can be used in subsequent operations. The result produced by an array of Bodies can only be used for creating another array.

ARRAY TYPES. SPECIFICS OF EACH TYPE

All arrays are distinguished by the types of the copy elements as follows:

Arrays of construction elements;

Arrays of operations;

Arrays of Bodies;

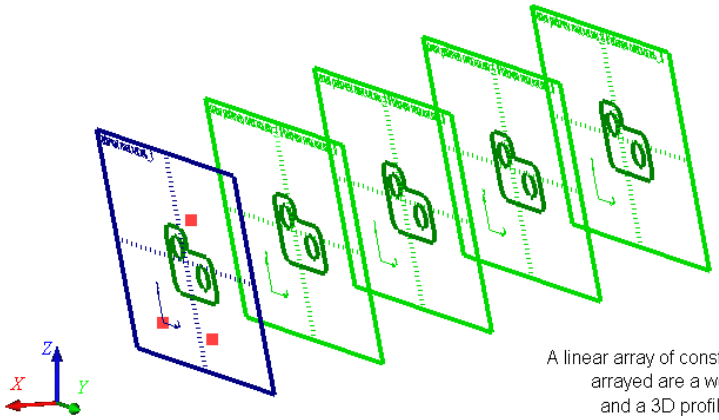
Arrays of faces.

Creation of any array begins from selecting its type.

Array of Construction Elements

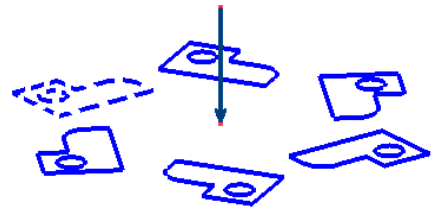
An array of construction elements allows creating a special type of 3D elements – *an array of 3D constructions*. Arbitrary 3D construction objects can be selected as the objects to be arrayed, except sections, light sources and cameras. Those could be 3D nodes, 3D paths, 3D profiles, workplanes, etc. The same array can combine construction objects of different types, such as, for example, 3D profiles and 3D paths.

> When creating an array of workplanes of any class, 2D drawing pages of the type “Workplanes” are automatically created for each array element.

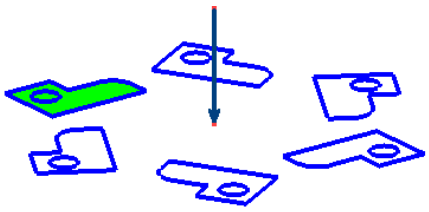


A linear array of construction elements:
arrayed are a workplane
and a 3D profile on it

The source construction elements are not included in the array. A “blank” is created in place of the source element (elements) in the array. However, an additional copy can be included in the array, coinciding with the position of the source element.



Circular array of 3D profiles, without an additional copy

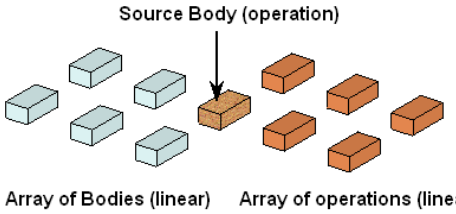


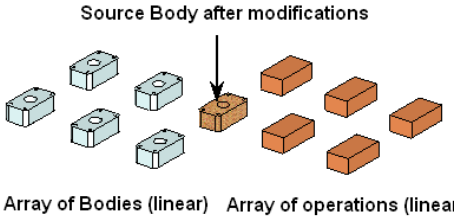
Circular array of 3D profiles containing an additional copy
in the position of the source element
(the additional copy is filled with color)

Array of Bodies and Array of Operations

Arrays of Bodies and arrays of operations create copies of bodies already existing in the scene. The source object for arrays of these types is an operation.

An array of operations copies results of a particular operation only. If another operation is created on top of that one, such as, for example, blending an edge, the array will not be affected. On the other hand, an array of Bodies replicates the whole Body, created by using the selected operation. If this Body is modified in the future, the array will be regenerated, accounting for new operations included in the Body history.



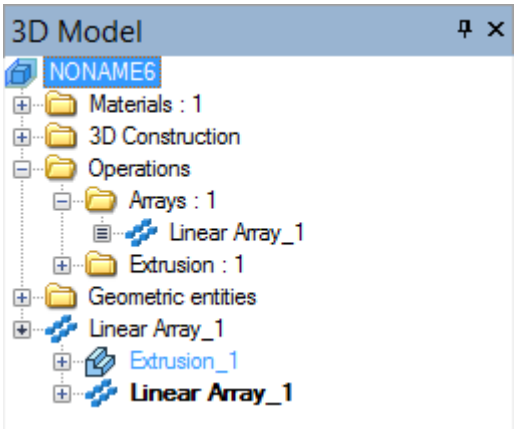


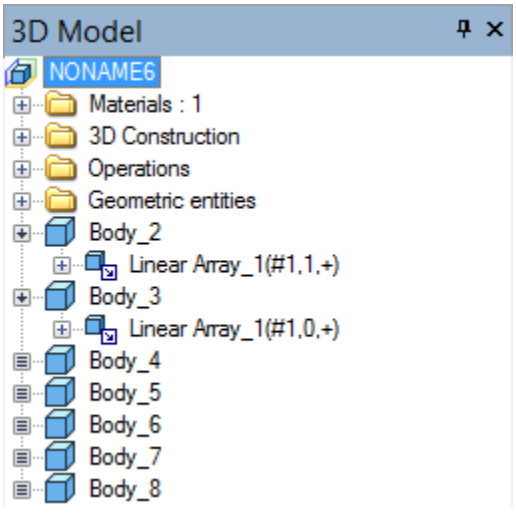
When an array of Bodies is built, a new 3D Body is created which is actually the array. The source Body (Bodies) is not included in the array (its place in the array is filled by a “blank”).

An array of operations requires a specific setting regarding whether or not to include the source operation in the array composition. If the source operation is included in the array, it will be subjected to all parameters defined for this array, such as the material, system-wide parameters (level, layer, color), transformation parameters, etc. Besides, if the operation selected for array creation is the last operation in the history of a Body, then this Body disappears upon the array creation, being replaced by the array.
In the cases, when the source operation is not included in the composition of the array, then the operation itself and the Body that it creates will remain independent objects of the 3D model. The array in this case will have a "blank" in the position corresponding to the source operation location (just like in the case of creating a construction array).

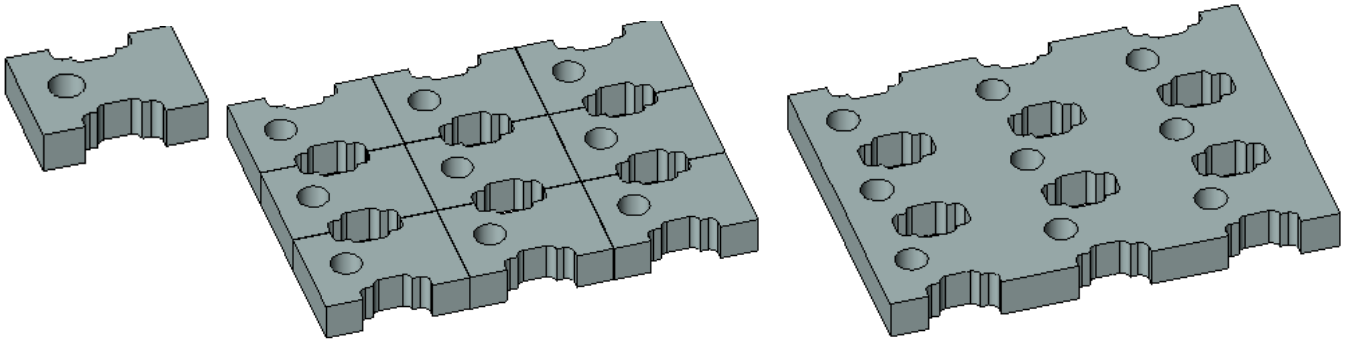
The following additional manipulations can be applied when creating any Body arrays and operation arrays:

Separate to Solids – the created array is automatically divided into separate Body copies;





Combine Result to one Solid – all array elements are merged together by the Boolean addition.



Source operation

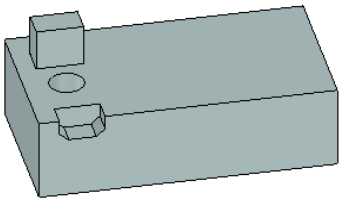
Resulting array without combining copies
(side edges of separate copies are visible)

Resulting array with combining copies

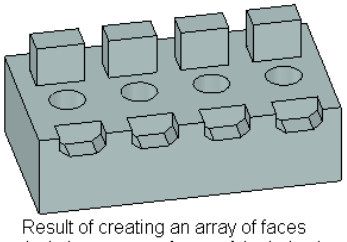
Array of Faces

The main purpose of any array of faces is to add holes and protrusions to the existing Bodies of the 3D model, repeating design features already present in the given Body. Therefore, an array of faces is always created based on one particular Body in the model: all copied faces must belong to that Body. The set of copy faces resulting from the array creation will also be included into the source Body.

No array of faces creates a new 3D Body; rather, the existing one is altered by the array.



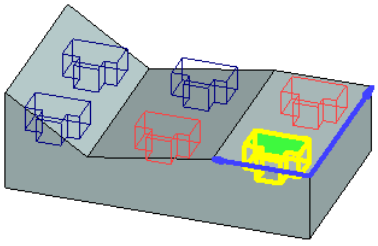
Source body



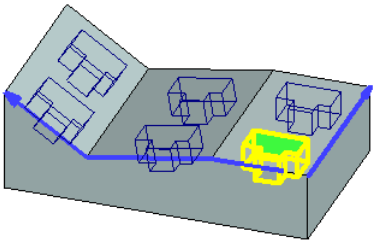
Result of creating an array of faces (copied elements are faces of the hole, the slot and the protrusion)

An arbitrary set of faces can be selected for creating an array, except that it has to fully define the geometry of the hole, slot or protrusion of the part being copied. The copies must be created within the bounds of the source body.

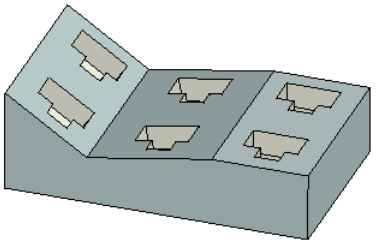
Creation of an array of faces may not affect the topology of the source body. Therefore, arrays should be created in such a way, that the copies of the faces intersected with the faces of the source body in the same way as the faces of the source set.



In this linear array of faces, copies intersect with the source body differently than the original set: the array will not be created



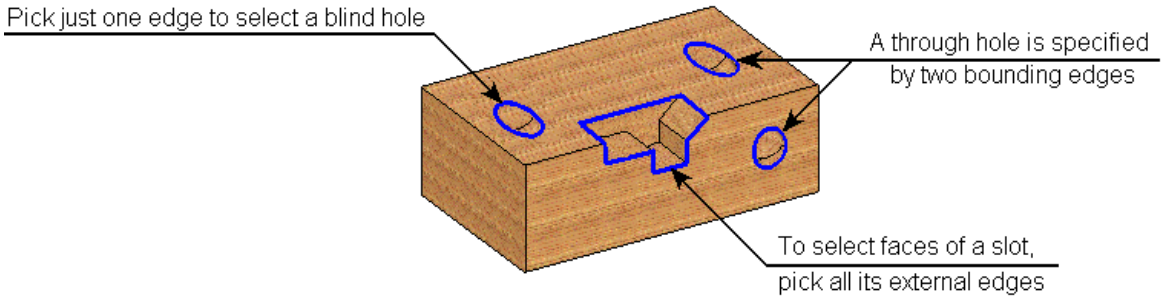
Array of faces by path insures similar intersection of the created copies with the body faces, as the originals



Result of creating an array of faces by path

When specifying the set to array, you can select:

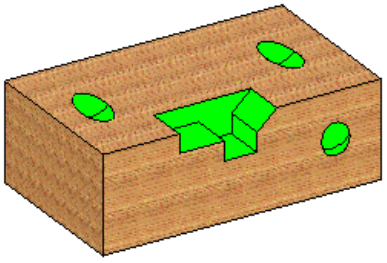
Edges surrounding a set of faces. A set of edges defines two possible sets of faces (we will call them the "inner" and "outer" sets with respect to the specified chain of edges). The system can automatically select the set to copy (judging by the least area) or let the user choose.



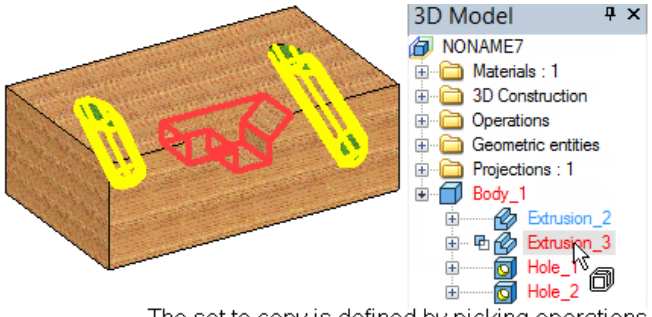
The selected edges must make one or several close sequences (loops).

All faces to be included in the source set of the array;

The operation that created the desired set of faces (for example, the operation “Hole”). In this case, the array will keep a link to the selected operation. If in the future the geometry of such operation will change, the 3D array will regenerate accounting for the changes.



The set to copy is defined by picking all faces



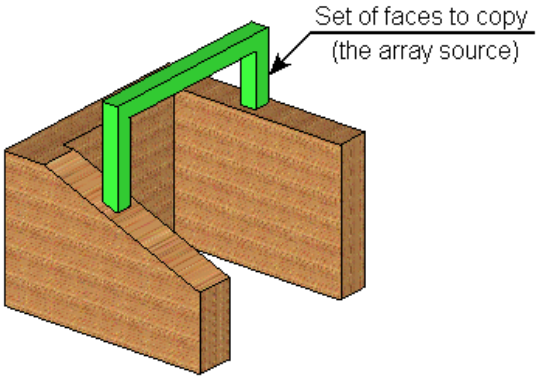
The set to copy is defined by picking operations

The following optional modes can affect the resulting array of faces:

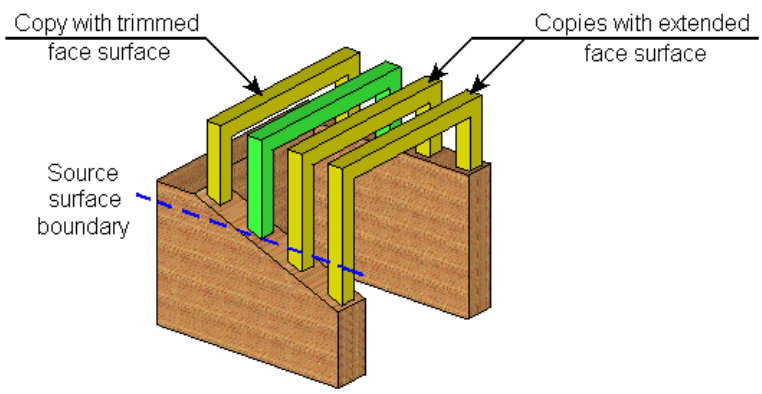
Copy to one face. Use of this mode considerably reduces the array regeneration time. However, this mode can only be used when all elements of the array being created lie on the same face as the source set;

Extend Faces (used only in combination with the “Copy to one face” mode). This mode allows lifting up the requirement of "similar intersection" of the source set of faces and all arrayed copies with the faces of the source body. The system will be automatically extending or trimming the copied faces in order to insure their correct connection to the body.

Use of this mode is possible only in the case when all copies are located within the range of the same body faces, as the source set.

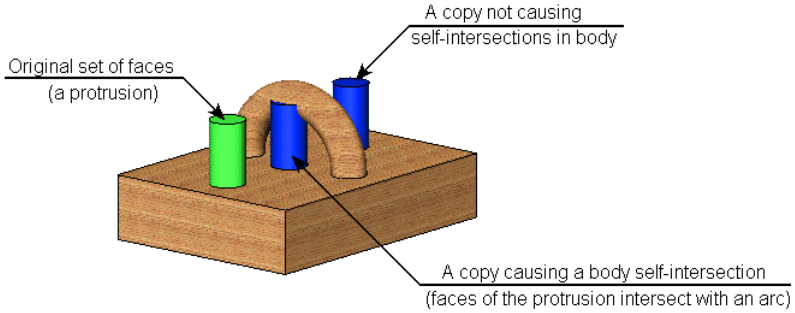


Source body



Array of faces created with the option "Extend Faces"

Check Intersection with other Faces (used to avoid body self-intersections). When using this mode, the system checks face copies for intersections with already existing body faces. If such an intersection is encountered, then array creation is aborted.



Shown array (replication of a body protrusion) will not be created in the "Check intersection with other faces" mode due to an intersection of one all the copies with an arc feature of the source body

- *Do not check body's boundaries* (enabled by default). When using this mode correctness of the resulting array is checked once after creation of all copies. While doing so the system checks the resulting copies only according to the following criterion: they must lie within the limits of at least one of the body's faces. This check does not exclude entirely appearance of errors in cases when the faces-copies lie on the faces boundaries but nevertheless reduces by several times the construction time of a large array.

  When the "Do not check body's boundaries" mode is disabled the system before the start of copying checks whether the result of copying is situated outside the boundaries of all body's faces. If so, these faces-copies are just not created. The second stage of checking is carried out after copying – the created faces-copies are checked for intersection with the boundaries of that face of a body onto which they were copied. If this intersection for a copy is found, this copy is removed from the array. This work mode provides for larger reliability of the array construction mechanism but significantly increases its construction time.

ARRAY CLASSES. SPECIFICS OF EACH CLASS OF ARRAYS

All 3D arrays created in T-FLEX CAD can be distinguished by the *way of copying* into the following classes:

Linear;

Circular;

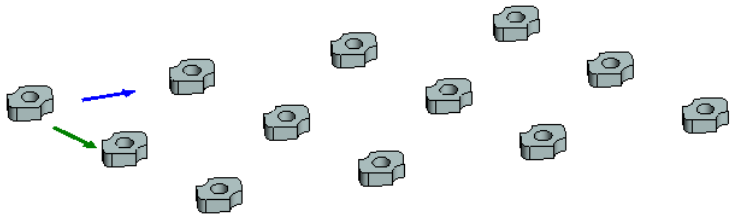
By Points;

By Path;

Parametric.

Linear Array

The linear array is created by placing copies of the source objects along one or two direction vectors. Depending on the number of the direction vectors, it will be either one-dimensional or two-dimensional array. All elements of the linear array lie in one plane.



The direction vectors of a linear array are defined either by two 3D points (in this way, the vector will be directed from the first selected point to the second one), or by a single 3D object suitable for defining a vector in the space.

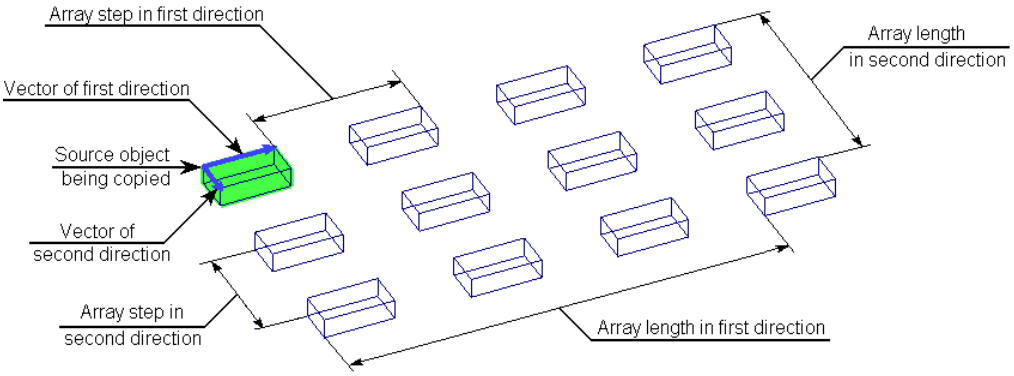
The array size along each direction is characterized by the number of copies, the placement step and the total length of the array in the given direction. To define an array, you need to specify just two of the three parameters. Depending on the mode used, it can be:

Number of copies and step;

Total length and step;

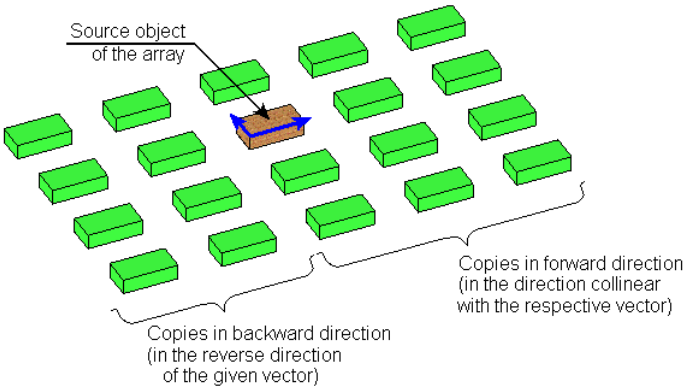
Number of copies and total length.

In any of the modes, one of the specified parameters (the array step or the total length in the given direction) can be automatically calculated by the system based on the length of the direction vector and the value of the other parameter.



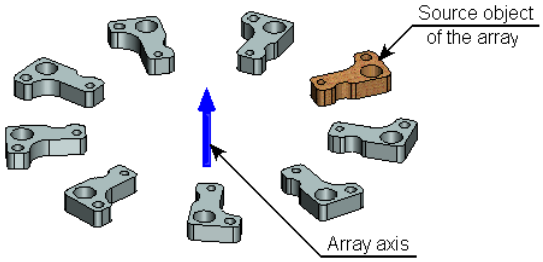
The number of copies along each direction vector always includes the source object being copied (even if it is not included in the composition of the array). For example, the array shown on the above diagram contains 4 copies in the first direction and 3 copies in the second direction.

A linear array allows creating copies not only forward, but also backward along each direction vector. The backward copies are created with the same step as the forward-going copies along the given vector. The number of copies in the backward direction can either be the same as in the forward direction along this vector, or specified by a separate numerical value.



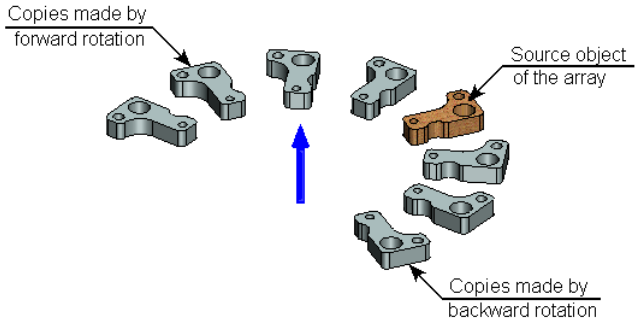
Circular Array

In the circular array, the copies are placed on a circle about the array axis. The axis of revolution of a circular array is defined, just as the direction vectors in a linear array, by selecting two 3D points or one 3D element suitable for defining a vector in the space.



The position of array elements about the axis is determined by the number of copies, the rotational step per copy and the total angle of elements rotation about the axis. The copy rotational step determines, by what angle to rotate the source body about the axis to place the first copy. The second copy is created by rotating the first one by the same angle, and so on. The rotation is done in the clockwise direction, as watched along the array axis.

Backward rotation is possible, including simultaneous creation of copies in two directions – forward and backward. The backward rotation is done with the same step as the forward one, while the number of copies may differ.



To define an array, you need to specify two defining parameters out of the three, for example:

Number of copies and step;

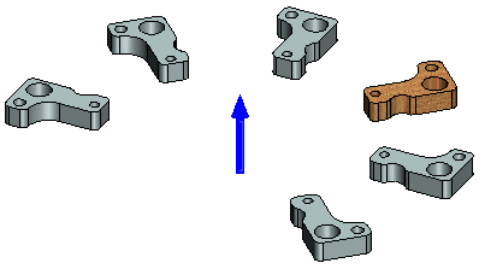
Total angle and step;

Number of copies and total angle.

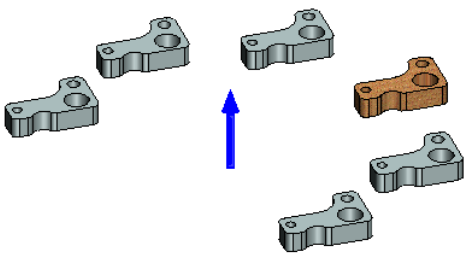
The third parameter is calculated by the system automatically.

The rotation step or the total angle of the arrayed elements can be defined by two additional 3D points. The sought angle will be equal to the angle between the planes passing through the rotation axis and the respective 3D point.

When copies are created, there are two possibilities for a copy's orientation: *rotated* and *parallel-translated*. In the first case, a copy is created by actually rotating the source body about the array axis. In the second case, the source body is copied without rotation about the axis (meaning that orientation of each copy in the space is the same as the source body orientation).



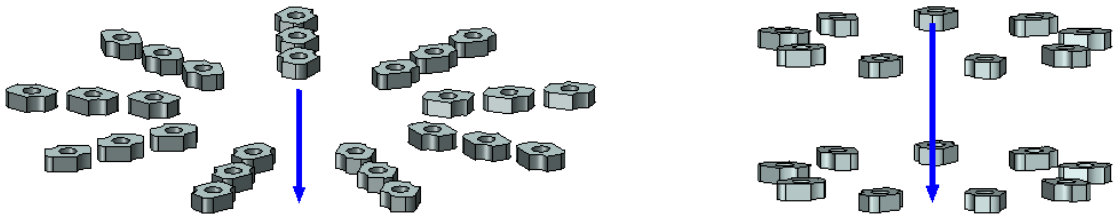
Circular array with copies rotated about the axis



Circular array with copies translated parallel to source

The circular array also allows creating copies in the second direction: in the radial direction of the array. In this case, array elements will be located in one plane, making up a pattern of concentric circles.

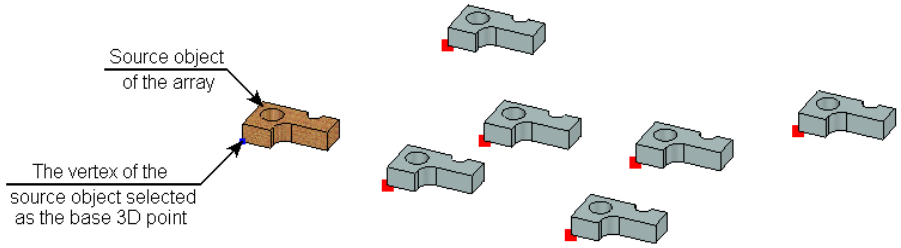
Alternatively, the second direction can be chosen along the axis, with the array elements put on a cylinder.



The number of copies along the second direction is specified in the same way as in the linear array.

Array by Points

To create a 3D array by points, select the base 3D point on the array's source object, and an arbitrary number of target 3D points (into which the source object will be copied). The arrayed elements will be placed in the specified 3D points.



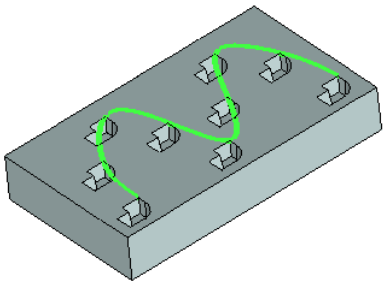
Array by points (target 3D points are defined by 3D nodes)

The target points are defined with successive selection of single 3D points (3D nodes, vertices, etc.) or arrays of 3D nodes. The list of target points of an array may contain both 3D points and arrays of 3D nodes.

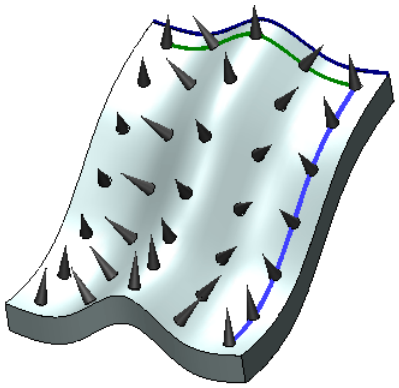
Array by Path

In the array by path, the placement of the array elements is defined by one or two 3D curves. Such an array can be considered analogous to the linear array, however, with the copies placed along one or two spatial curves rather than straight lines. Those curves can be represented by arbitrary 3D curves (3D paths, 3D profiles, curvilinear edges, etc.). If a closed 3D curve is selected, you need to additionally specify a 3D point, whose projection on the selected curve would determine the start of the array curve.

Use of a single guide curve results in a one-dimensional array, two guide curves – in a two-dimensional array by path. For simplicity, let's first review creation of an array by one guide curve.



Array by path with one guide curve



Array by path with two guide curves

The placement of copies (array elements) along the guide curve depends on the following array parameters: number of copies, step and total length of the array along the given curve, and, additionally, the copies' placement option.

Number of copies, step and total array length along the guide curve are interrelated parameters. To define the array, you need to specify just two parameters out of the three. The omitted parameter will be calculated by the system automatically. The total array length can be evaluated by the guide curve (the system will determine the curve length, and then take the resulting value as the length of the array in the given direction).

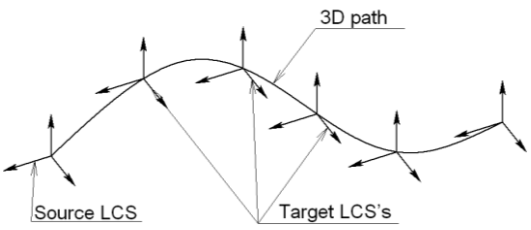
The total array length may not exceed the length of the guide curve.

The placement option determines:

- The position of the copies' placement points (array elements) along the guide curve. Depending on the used placement option, the step of the array by path determines either the distance along the path between the fixing points of the copies or the path's chord length.
- The orientation of copies in the space. The position of copies in the space is determined by bringing the target coordinate system of each copy over the source coordinate system of the object being copied (that case, the source object of the array). The source coordinate system is created at the start of the guide curve, while the target – at the fixing point of the particular copy. The axis orientation of either coordinate system is defined by the copy placement option.

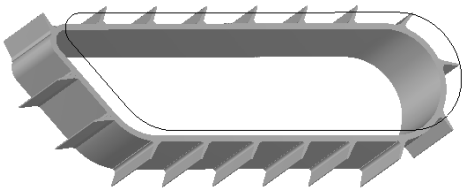
There are the following copy placement options:

With offset by path. The target coordinate systems of all copies are placed along the path according to the specified step and "number of copies" parameters. The axis orientation of those coordinate systems is fixed and does not depend on the 3D path geometry.



This placement option can be used, if you need to put copies of the source body along the guide curve, preserving the spatial orientation of each copy as that of the source body.

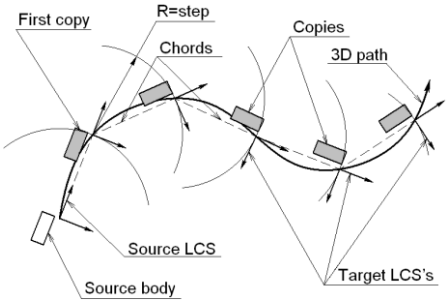
With minimum twist by path. The X-axis of the target coordinate system of each copy is determined as to provide smooth rotation of the curvature vector from the start to the end of the path. The Z-axis always points in the direction tangent to the path at the origin of the current copy's coordinate system.



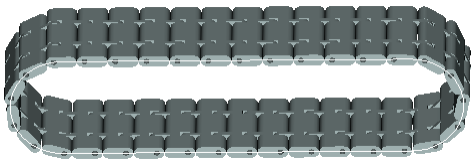
An example using an array by path with minimum twist by path as a model of a conveyor belt.

This example can be found in the library *"Examples"*, in the folder *"\3D Modeling\Arrays\Conveyor.grb"*.

By span. To find the position of the first copy, a sphere is created with the center at the path start and the radius equal to the array step. The copy will be placed at the intersection point of the sphere and the 3D path. Next, another sphere is constructed with the center at this point, and so on. This is how attachment points of all copies are determined. The target coordinate systems of the copies are placed in those points. The X-axis of a target coordinate system is oriented by the incoming chord of the 3D path. A path chord is defined by the attachment points of two neighboring copies.



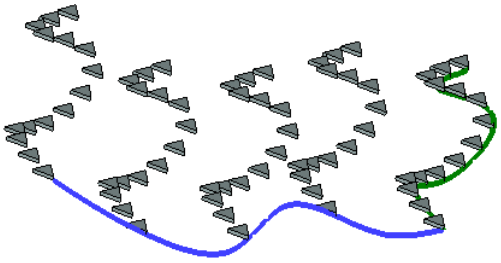
The source coordinate system is located at the start of the path. Its X-axis points toward the first copy's position. With this copy placement option, the array step is the chord length of the 3D path.



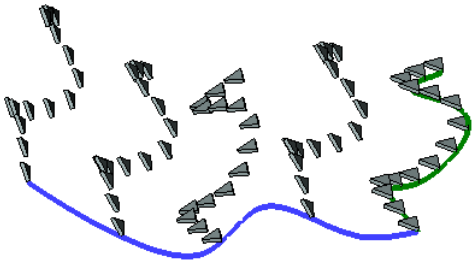
This placement option can be successfully implemented, for example, in caterpillar belt and chain drive design.

> A caterpillar belt design example is provided in the library *"Examples"*, in the folder *"\3D Modeling\Arrays\Track.grb""*.

The two-dimensional array by path is created similarly. The set of parameters is defined for the second guide curve in the same way, including the step and total length of the array in that direction and the copy placement option. Note that the source object in the second direction will be the complete set created by the one-dimensional array along the first guide curve.



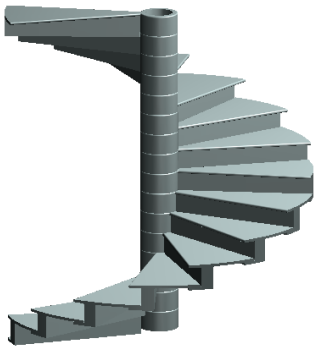
Two-dimensional array by path with the "With offset by path" placement option in the second direction



Two-dimensional array by path with the "With minimum twist by path" placement option in the second direction

Parametric Array

The parametric array uses a universal approach that allows creating an array by parametric copying of the source element. The same approach is used in the "By Parameters" operation and in one of the 3D path creation modes. The algorithm of face approach is described in details in the "Parametric sweep" chapter. Therefore, the current topic just briefly touches upon the algorithm description and the typical examples.



The steps of this staircase are elements of a parametric array of bodies (3D fragments)

> A parametric array can be an array of bodies, operations or construction elements only. Parametric arrays of faces cannot be created.

To create a parametric array, you need to define the law of calculating the so-called **coordinate system of the current copy**. The copying of the array source object is done in the direction from the source coordinate system to the calculated target coordinate system of the current copy.

The user can select one of the model's local coordinate systems as the source coordinates. If no local coordinate system was selected, then the global coordinate system is used as the source coordinates.

The coordinate system of the current copy (particularly, its origin and axis orientation) can be defined in one of the two ways:

- *Free copy orientation*. Modifications of the coordinate system parameters of the current copy are described by expressions.
- *Copy orientation by 3D curves or surfaces*. To define the spatial position of the current copy's coordinate system, use the appropriate 3D curves or surfaces of other bodies. By managing the copy's coordinate system, you control the position of each copy.

The position of a copy's coordinate system is controlled by the array parameters. All parameters are divided into two types - **driven parameters** and **driving parameters**.

Driven parameters are the parameters whose values are calculated automatically during element creation. One can introduce independent variables to read the values of such parameters.

Driving parameters are the entries whose values are specified by the user. The entries for the driving variables permit arbitrary expressions including any system variable. For example, including a variable keeping the copy ID number, in such an expression, will make the given parameter depend on the copy ID.

Some array parameters can be sometimes driving, and sometimes driven. The leading parameter "**Number of copy**" is always driven. It definitely requires assignment of an independent variable that will be tracking the current copy ID number during the operation processing. The copy number is automatically incremented through the array calculation, starting from 1, up to the specified quantity, with the step equal to 1.

The calculation is done in the following order: the copy number is assigned the next ordinal number, then the driving parameters are calculated, then the driven parameters are calculated (if any), and, finally, copying is done.

When using the free copy orientation option, just select the array source object, specify the number of copies and assign independent variable to the "Number of copy" parameter. By using the variable assuming the current copy number, one can introduce certain relations in the entries of driving parameters, as well as in 2D or 3D constructions, and make the geometry of the original body, in turn, depend on those relations.

An example of a parametric array with free copy orientation is a model of a spiral staircase, as shown on the diagram at the beginning of this topic.

The copy orientation option by path or by surface is convenient in the cases when defining the expression for the exact law of the copy parameters modification is not feasible. In such cases, it is convenient to use various 3D curves or surfaces for specifying the position and orientation of the current copy's coordinate system.

LIMITATIONS AND EXCLUSIONS

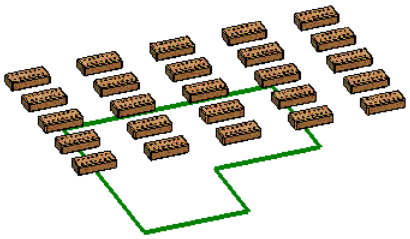
Limitations and *exclusions* are used to exclude some of the created copies from an array.

Limitations

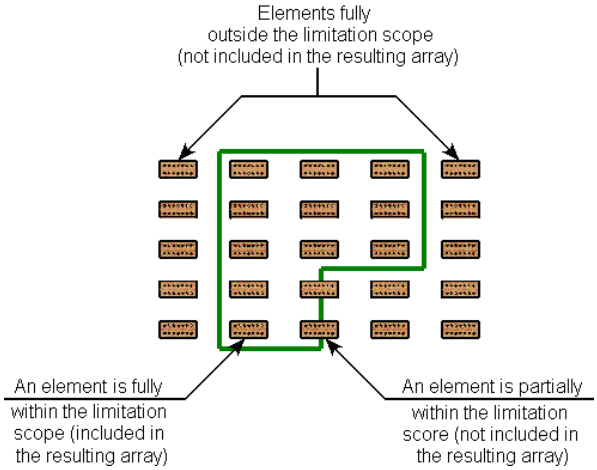
Setting a limitation means defining a subset of the 3D space to which copies creation is limited – *limitation scope*. When in this mode, all copies that are created per the main array parameters are checked by the system for falling in the limitation scope. If a copy fully falls within the scope, it will be included in the resulting array. If a copy is located outside the limitation scope or fits in but partially, it is not included in the resulting array.

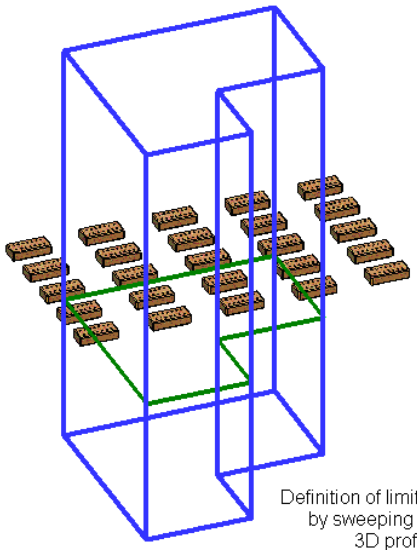
> Limitations may not be available with all classes of construction element arrays, and are not available with any parametric arrays.

The limitation scope is defined by a 3D object with sheet geometry – a face, 3D profile or a sheet body. The selected object is swept from both sides along the limitation axis for the infinite distance. The resulting swept area will be the limitation scope of the array.

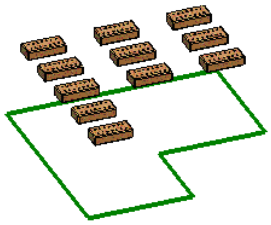


Source array and 3D profile defining array limitation scope



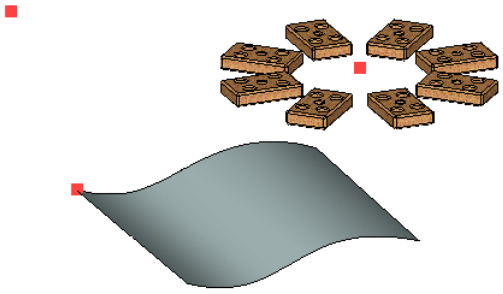


Definition of limitation scope
by sweeping selected
3D profile

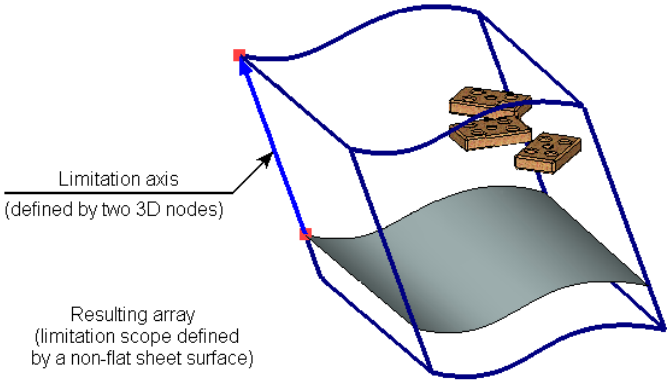


Resulting array

If the limitation scope is defined by a flat 3D object, then the limitation axis coincides, by default, with the normal of the selected object's plane. If necessary, you can specify an arbitrary direction of the limitation axis by defining the axis directions by two 3D nodes or one 3D object suitable for defining a straight line. In the cases when the array limitation scope is described by a non-flat sheet 3D object, the limitation axis must be explicitly specified.

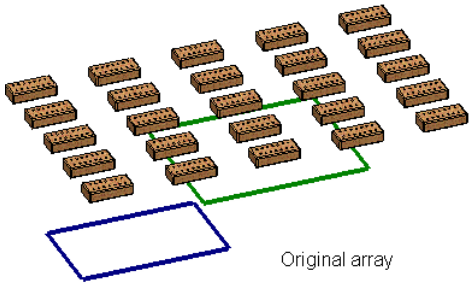


Original circular array

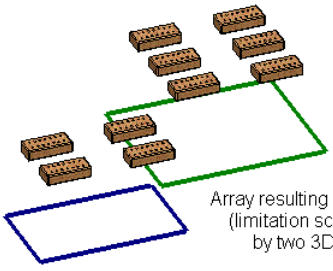


Resulting array
(limitation scope defined
by a non-flat sheet surface)

The limitation scope can also be defined by multiple sheet 3D objects. All objects are swept in the same direction. The limitation scope will be the sum of the scopes swept by each of the selected object.



Original array



Array resulting from limitations
(limitation scope defined
by two 3D profiles)

Exclusions

Exclusions, or excluded elements are the array elements that shall not be included in the resulting array. Any array elements can be excluded, except the source object.

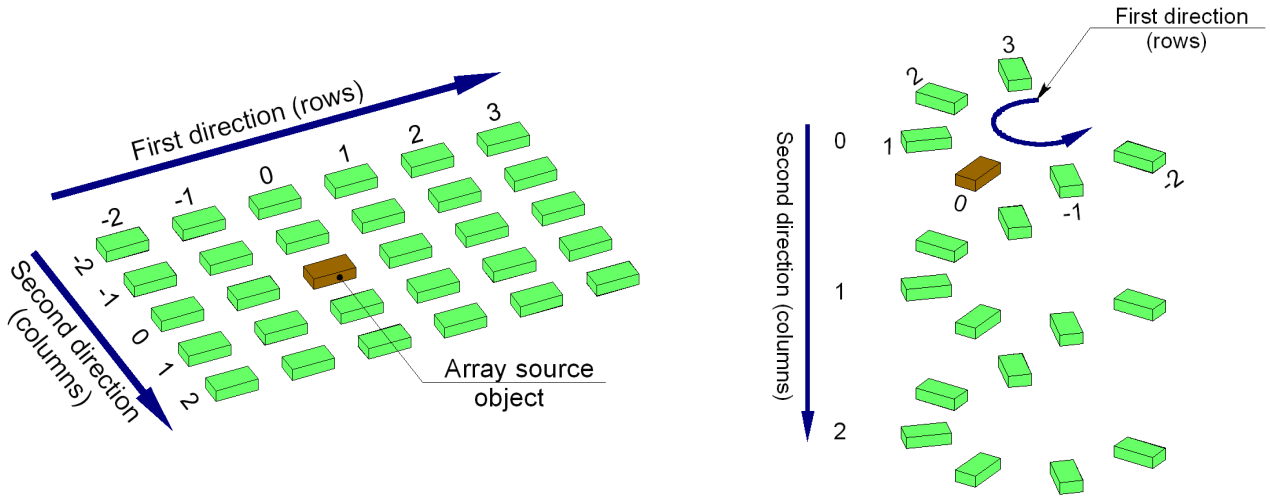
> Exclusions can be defined for the following array classes: linear, circular and array by path.

An exclusion is specified by its position in the original array, that is, the column and the row in the array, at whose intersection this element is located (in the cases of the circular array and array by path – generalized "row" and "column"). The dimension of the array in the first direction defines the number of rows, in the second the number of the array columns.

Rows and columns in arrays are numbered as follows:

- 0, 1, 2, …, N-1 – in the forward direction (N is the number of copies in a row/column in the forward direction)
- -1, -2, …, -M+1 – in the backward direction (M is the number of copies in a row/column in the backward direction).

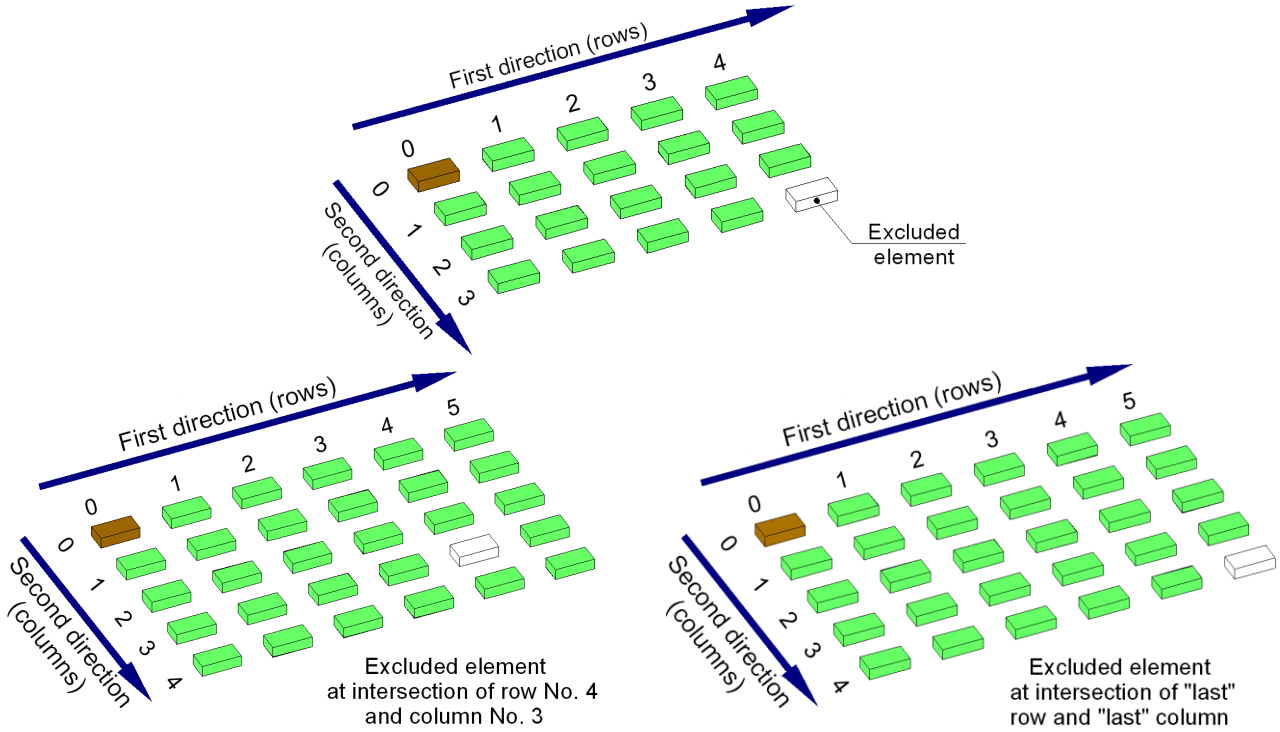
> Note that the column and the row numbers, in which the source object of the array is located, are 0. This is important when defining exclusions.
>
> You cannot exclude the array element with the coordinates (0,0) from the array of Bodies.



Instead of the number of particular rows/columns, one can use descriptive characteristics defining their position in the original array:

- *Starting* row/column is the row/column in which the source object of the array is located (the number 0);
- *Last* row/column is the row/column, which is the last in the forward direction of array creation;
- *Last in Opposite Direction* row/column is the row/column, which is the last in the backward direction of array creation;

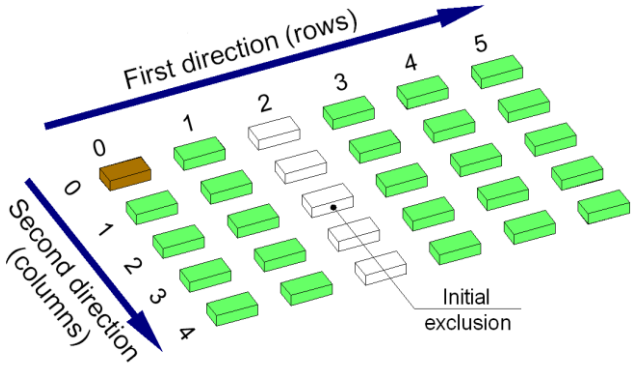
This way of specifying exclusion locations is convenient for removing elements of the outer rows/columns from the arrays of variable size, especially when the array dimensions are defined by the total length and the step, or the total length and the number of elements. Suppose, for example, that you need to exclude an element in a linear array of size 5x4, that is located at the intersection of the last row and the last column of the array (assuming that the copies are created in the forward direction only along both vectors). One can simply specify the element at the intersection of the row number 4 and column number 3 as the element to exclude. However, should the array grow (for example, to 6 rows and 5 columns), the excluded element will have the same exact coordinates (the last row will be now number 5, and the column – number 4). Instead, to have specifically the last element of the array always excluded, the exclusion should specify the element at the intersection of "the last" row and "the last" column.



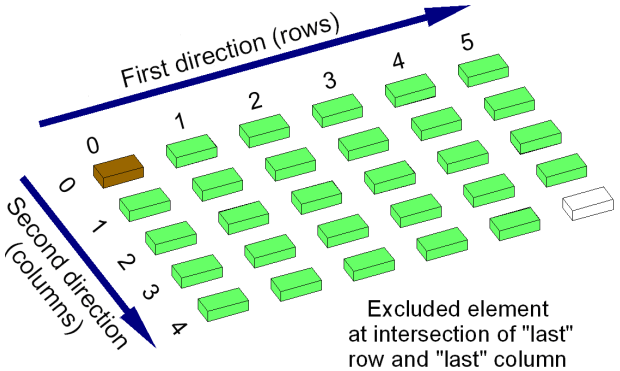
Each specified exclusion can be subjected to the following additional options:

Whole row exclusion. Use of this option causes exclusion of the whole row with the specified element from the array;

Whole column exclusion. With this option, the column with the specified element is excluded from the array;

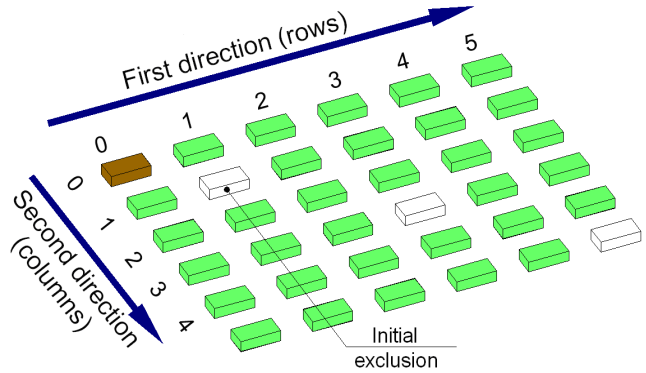


Result of exclusion in whole row exclusion mode

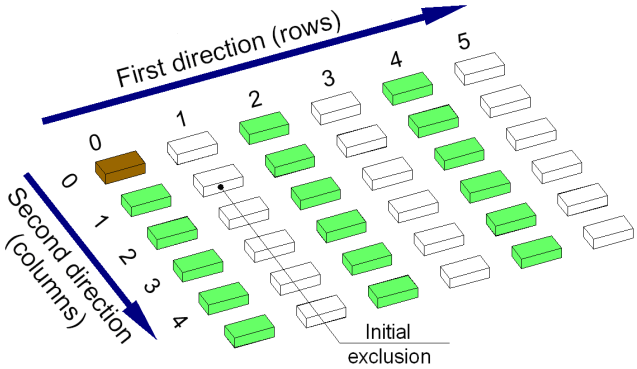


Excluded element at intersection of "last" row and "last" column

Cyclical Iteration. The result of applying this option depends on whether the previous options are used. If those are not used, then use of this option will cause deletion of not only the original exclusion, but also the elements following it after the same number of rows and columns from the start of the array. If this option is used together with the previous ones, the whole cyclically repeating rows or columns will be excluded.



Result of cyclical repetition of the exclusion



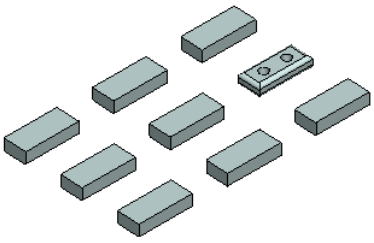
Result of cyclical repetition of the exclusion in whole row exclusion mode

Modifying Number of Copies in Array. Referencing Array Elements

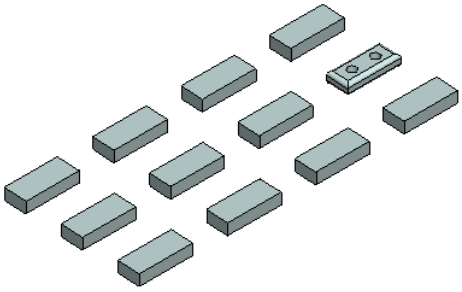
When the number of copies of any array changes, then additions or deletions of elements occur immediately before the last array copy in the given direction. The IDs of geometrical components in the elements of the outmost array rows and columns do not change through such additions or deletions (outmost are the first and the last elements in any direction).

Therefore, any operation created on an element of the first or the last row (the first or the last column) of the array stays intact through any changes in the array dimensions. For example, an edge blend created on an element of the last row in an array of Bodies or operations, will regenerate correctly, regardless of increases or decreases in the number of rows in the array.

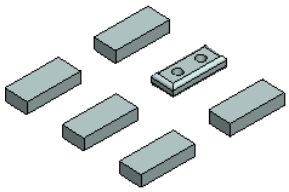
Operations created on "inner" copies of the array could run into regeneration errors, as the array dimension is decreased.



Edge blending and hole operations are valid on an element of the last row in a linear array of bodies



Result of increasing number of rows in array



Result of decreasing the number of rows in array

CREATING 3D ARRAYS

The following commands are used for creating an array:

Keyboard	Textual Menu	Icon
<3AL>	Operation > Array > Linear	
<3AR>	Operation > Array > Circular	
<3AN>	Operation > Array > By Points	
<3AP>	Operation > Array > By Path	
<3AA>	Operation > Array > Parametric	

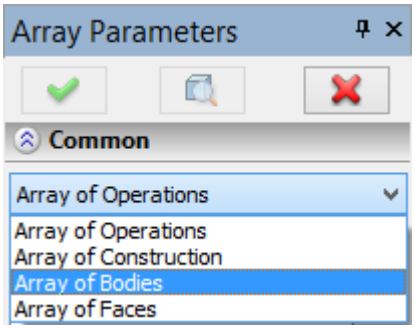
The user interface of the above commands is similar. Therefore, the further description applies to all arrays, unless specifically mentioned.

To create any array, upon calling the respective command, do the following steps:

- ✓ Select the type of the array to be created (array of construction elements, array of operations, array of Bodies, array of faces);
- ✓ Select source objects of the array;
- ✓ Select the guiding elements and define the main array parameters (the actions performed at this step depend on the type of the array being created);
- ✓ Define advanced array parameters. The set of available modes controlled by those parameters depends on the array class – see the section "Array classes. Specifics of each class of arrays";
- ✓ Define limitations and exclusions (if necessary and possible);
- ✓ Confirm array creation (in the automenu or in the properties window).

Selecting Array Type and Array Source Objects

The array type selection is done in the properties window of the respective command. The section "Common" of the properties window begins for all arrays with a combo box for selecting the required type.



> The item "Array of faces" is not available in the combo box of the parametric array.

Depending on the selected type, one of the following options for selecting the array source objects will be provided in the automenu:

	<N>	Select construction
	<N>	Select source 3D operation (for arrays of operations)
	<N>	Select source 3D operation (for arrays of Bodies)
	<N>	Select faces to copy or their boundary edges or loops

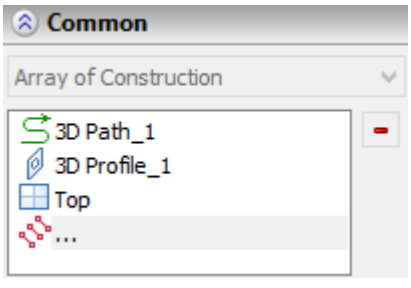
The selected objects will be put into the list of the array source objects in the operation properties window.

> To resume selection of source objects at any stage of array creation, simply focus on the pane of the source elements list. The selection option will turn on automatically.

To undo selection of some of the selected objects, click it in the list and press the button .

> Note that once the first object to be arrayed is selected, the entry defining the array type becomes inaccessible. To change the array type, you need to remove all 3D objects from the list of the array source objects. Only after that, you will be able to select another array type.

3D construction elements and operations are selected by simply picking those either directly in the 3D scene (by) or in the model tree.



To select a 3D Body, you can pick *any* of the operations contributing to this Body (again, either in the 3D scene, if that is possible, or in the 3D model tree). The *Last* operation of the given Body will be automatically put in the list of the selected objects.

Selection of source objects for an array of faces requires more close consideration.

Selecting source set of faces for array of faces

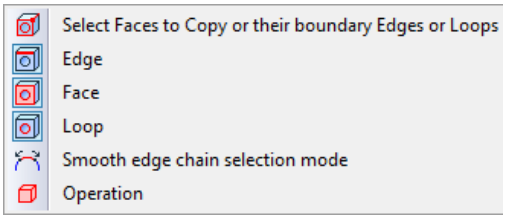
The pulldown list of the option for selecting a set of faces to copy allows choosing the way of selecting: direct selection of faces, selection of their bounding edges, selection of the operation (the same filters are duplicated by the system toolbar).

The first item of the pulldown list turns on all filters simultaneously.

To select the operation that created the desired set of faces, activate the filter. This filter has higher priority than other filters in the list. Therefore, if any other filter is turned on simultaneously with the operation selection filter, such as, for instance, the face selection filter, then the operation will be put in the list of the source objects for arraying, regardless of what was selected by the user – an operation or a face. That means, if a face is picked in the 3D scene, then the whole operation that created that face is put into the list of objects to array. This property of the filter allows quick selection of operations by a pick on an edge or face.

To enable selection of individual faces for the set to array, the option should be turned on, while the option – off.

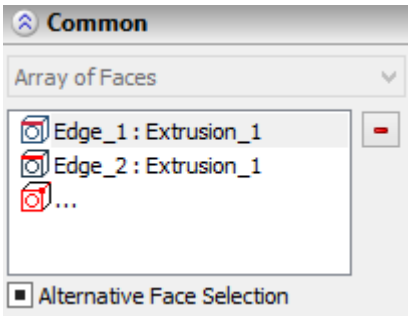
To select edges (or a chain of edges at once) surrounding the desired set of faces, turn on the filters (for individual edge selection) and/or , (for selecting a whole chain of edges). Meanwhile, the option must be turned off.



Which set of faces from the two possible sets, defined by the specified chain of edges, will be selected as the source object for the array, is determined by an additional flag **Alternative Face Selection** in the properties window.

The flag has three states: on by default ▣, on ☑ and off ☐.

To let the system choose the set of faces automatically (by the least area condition), set the flag in the default state ▣. If the further manipulations with the 3D model will alter the balance of the areas, the system will automatically switch to the other set of faces as the source object of the array.



In the cases, when you want to specify one of the two possible sets of faces explicitly, use the two other states of the flag (on ☑ and off ☐). Each of those will correspond with a particular set of faces. Which set

of faces corresponds to which particular state of the flag is arbitrarily defined by the system upon specifying the set of edges. In this case, no future changes in the model will affect the set of faces being arrayed.

Defining the Guiding Elements and Main Array Parameters

Linear array

To define the direction vector of a linear array in the first direction, use the following automenu options:

	<D>	Select first direction

This option allows defining the direction vector by one 3D element (a straight edge, a face, etc.).

The options below allow defining the direction vector by two 3D points:

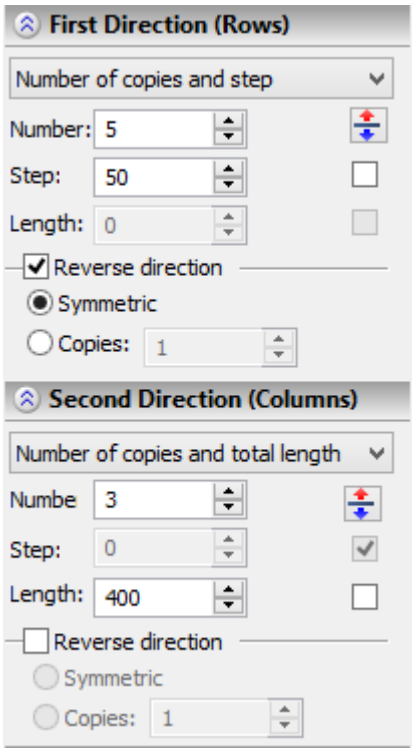
	<F>	Select first direction starting point
	<S>	Select first direction ending point

Undoing selection in either case is done by the option:

	<K>	Reset first direction selection

Similar options are provided in the automenu for defining the second direction vector:

	<J>	Select second direction
	<T>	Select second direction starting point
	<O>	Select second direction ending point
	<L>	Reset second direction selection



The array size is defined in the operation properties window similarly for both direction vectors. The array parameters in the first direction are defined in the section **First Direction (Rows)**, in the second direction – in the section **Second Direction (Columns)**. The buttons in each section help quickly flip the direction of the respective vector without redefining it.

The first entry in each section is a combo box for selecting the parameters to define: **Number of copies and step**, **Total length and step** and **Number of copies and total length**. Depending on the choice you have made, the following entries for defining the respective parameters will be provided below: **Number** – allows defining the number of copies in in the given direction, **Step** and **Length** – define, respectively, the step and the array length in the given direction.

To have the step or the array length be automatically calculated from the vector length, set the flag next to the respective parameter. In this case, the input box becomes inaccessible.

To create array copies in the backward direction along the direction vector, set the flag **Reverse direction** in the properties window section corresponding to the desired vector. In this case, the switch becomes accessible for defining the number of copies in the backward direction. To have as many copies as in the forward direction, choose **Symmetric**. To define a different number of copies in the backward direction, switch to **Number** and enter the desired number of copies.

Circular array

To define the axis of a circular array, you can use the following automenu option:

	<A>	Select axis of rotation

This option allows defining the array axis by selecting a single 3D element.

The following options allow defining the axis by two 3D points:

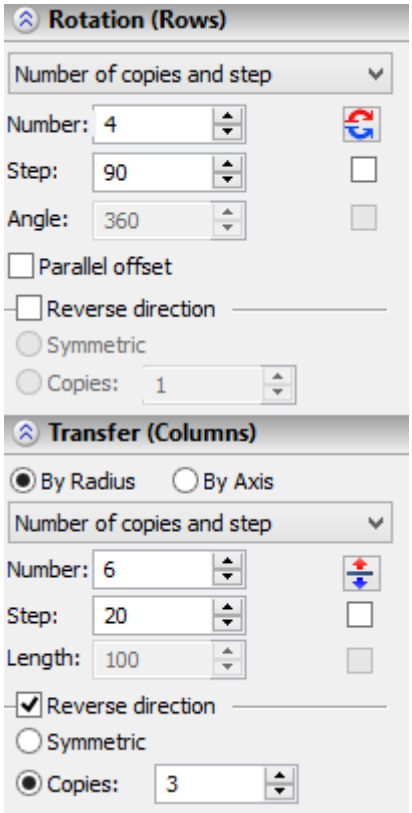
	<F>	Select 1st Point of axis
	<S>	Select 2nd Point of axis

To cancel selection in either case, use the option:

	<K>	Cancel axis of rotation selection

The array size is defined in the operation properties window. The array parameters in the first direction (rotation about the axis) are defined in the section **Rotation (Rows)**, in the second direction – in the section **Transfer (Columns)**.

Parameters of the circular array in either direction are defined in the same way as the respective parameters of the linear array. The only difference is the additional flag **Parallel offset**. When the flag is cleared, the array is created by rotating copies about the axis. To create an array with copies parallel-translated, turn on this flag.



To define the rotation step or the total angle of the array by two additional 3D points, use the following automenu options:

	<T>	Select starting point of Rotation
	<O>	Select ending point of Rotation

Those options become accessible after specifying the array axis. Once the points are defined, the respective parameter input boxes become inaccessible (the step or the total array angle).

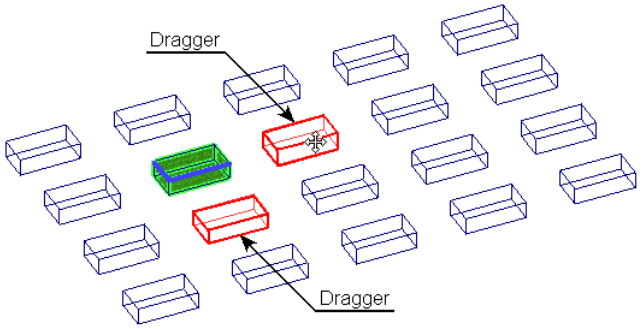
Selection of the additional points can be canceled by the option:

	<L>	Cancel selection of points for Rotation

Using draggers in linear and circular arrays

You can use draggers to define array parameters (number of copies, step and total length of the array in each direction) in both the linear and the circular arrays. Those allow, depending on the way of defining array parameters, setting either the number of copies, or the step, or else the total length (total angle) of the array.

The draggers appear as array elements painted *Red* (by default). As the pointer approaches a dragger's image, it changes to or (depending on the parameter controlled by the dragger). If at this moment you start dragging the mouse with the button depressed, then the wireframe preview of the operation being created will start rubberbanding, while the value of the respective parameter will be adjusting in the properties window.



Array by points

The base point on the source object of the array is specified by the automenu option:

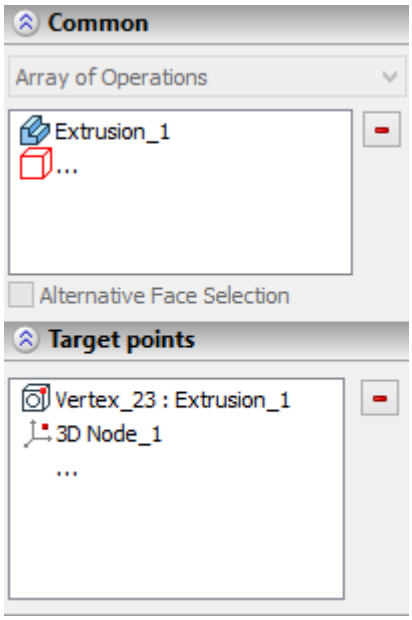
	<S>	Select point of source position

This option automatically activates upon selecting the source object of the array.

Once the base point is selected, the option for defining the target points becomes active:

	<T>	Select point of target position

The option allows sequentially picking an arbitrary number of target points. The selected points are put in the **Target points** list in the operation properties window.



Array of 3D nodes can be selected with option:

	<A>	Select Array of 3D Nodes as Target Points

Selection of points can be undone in two ways. All selected points can be rejected by the command automenu option:

	<K>	Cancel points selection

Any point can be removed by clicking it in the list of target points in the properties window and pressing the button .

Array by path

The first guide curve of an array by path is selected by the option:

	<T>	Select first guide curve

The starting point on a closed curve is specified by the following automenu option:

	<S>	Select start point on closed path

To undo a point selection, you can simply select another point or use the option:

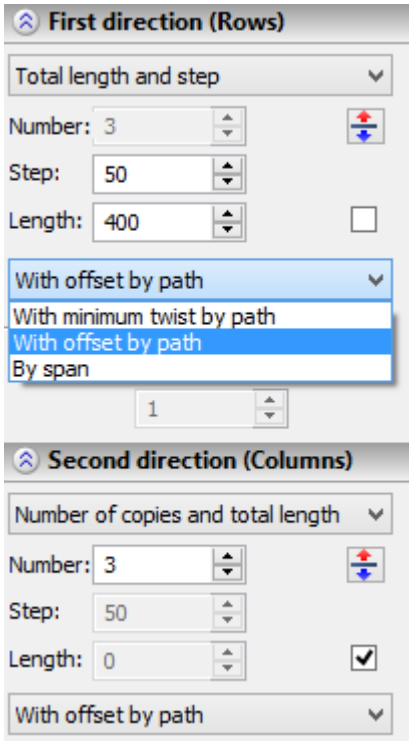
	<K>	Cancel start point on closed path

The similar set of options is used for defining the second guide curve:

	<O>	Select second guide curve
	<F>	Cancel selection of second guide curve

	<B>	Select start point on closed path
	<R>	Cancel start point on closed path

The array size is defined in the operation properties window. The array parameters in the first direction (along the first guide curve) are defined in the section **First Direction (Rows)**, in the second direction – in the section **Second Direction (Columns)**.



The array by path parameters in either direction are defined similar to those in the linear array.

Placement option in each direction is defined by selecting the desired choice from the combo box located at the bottom of each section.

When using the orienting method **By span**, the array size in the given direction can be defined only by the option **Number of copies and step**.

Parametric array

The following options are provided in the automenu for creating the parametric array (those become accessible after selecting the source object of the array):

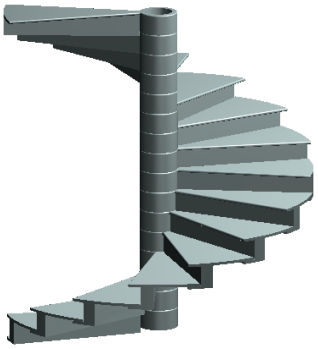
	<1>	Select curve defining position of coordinate system
	<2>	Select curve defining X direction of coordinate system

	<3>	Select curve defining Y direction of coordinate system
	<4>	Select surface defining X direction of coordinate system
	<5>	Select surface defining Y direction of coordinate system
	<6>	Select source LCS
	<7>	Cancel selection of elements defining coordinate system

As was mentioned earlier, creation of a parametric array is similar to creating a parametric sweep. This technique was described in details in the chapter "Parametric sweep". Here, we will review the array creation on the example of a spiral staircase.

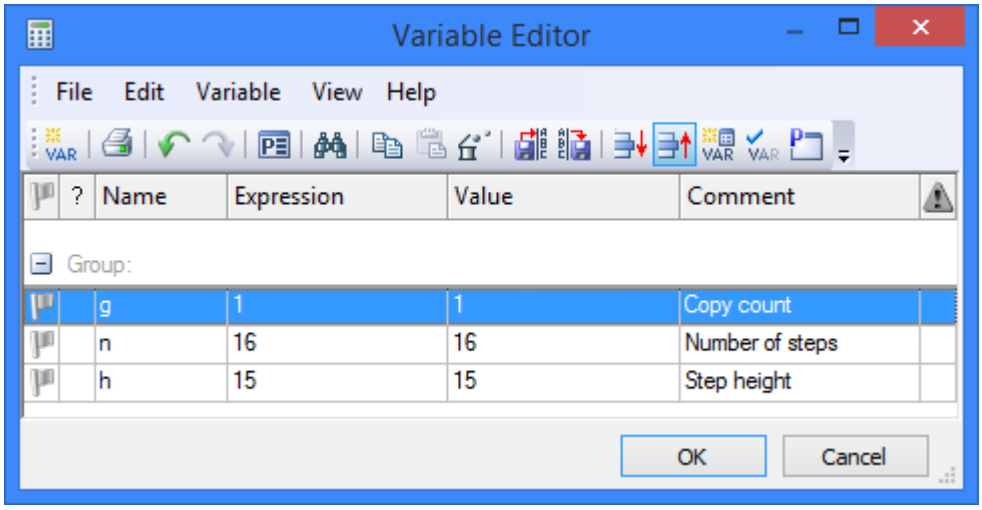
> The example files are located in the library *"Examples"*, in the folder *"\3D Modeling\Array\Winding Stairs.grb"*.

The model is built as a parametric array with free orientation of copies based on the 3D fragment of one step of the staircase.



When creating an array in this way, just select a body, specify the number of copies and assign an independent variable to the parameter "Number of copy". By using the variable assuming a copy's ordinal number, you can define certain relations in the driving parameters input boxes, as well as in the 2D or 3D constructions defining the position and geometry of the source body.

In this example, a step of the staircase is copied with a change in the height and rotation angle about the staircase axis. For convenience of handling the model, the following variables are created in the variable editor:

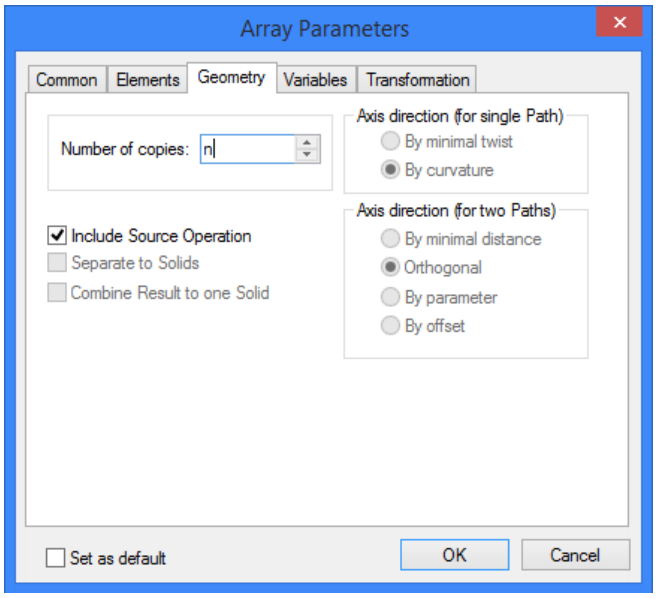


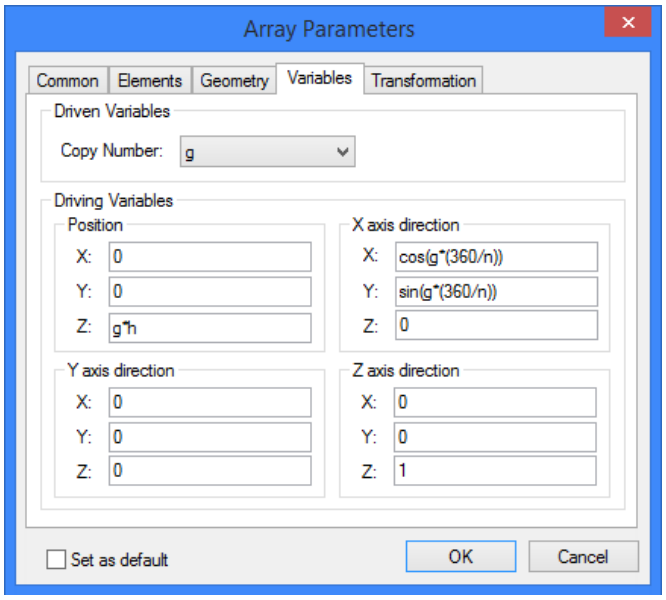
The step of the spiral staircase is inserted in the document as a 3D fragment. This fragment is inserted in such a way, that the axis of the staircase coincides with the global Z-axis. If the fragment were located arbitrarily, then one would have to select a local coordinate system, using the option:

	<6>	Select source coordinate system

Its Z-axis must have been directed along the staircase axis.

In this example, the expression defining the position of each copy depending on the copy count number can be entered in the input boxes of the driving parameters. The array parameters can be defined in the properties window or in the operation parameters dialog box.



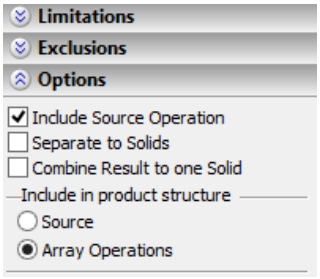


Positions of copies will be calculated by the linear law along the global Z-axis with the step defined by the variable h. The X-axis of the current copy's coordinate system will be rotated about the axis of the staircase. This moment of the X-axis is defined by the **X-axis position** parameter group. The Z-axis direction of a copy's coordinate system coincides with the staircase axis and with the global Z-axis. The Y-axis direction of a copy's coordinate system should be calculated automatically, therefore all driving parameters in this group are set to zero.

Defining Optional Array Parameters

Optional parameters of all arrays are defined in the section **Options** of the respective operation's properties window. What parameters are available in the section depends on the array type.

The following flags are available for arrays of construction elements, operations and bodies:



Include source operation (available for arrays of construction elements and operations only). If this parameter is set, then the source objects are included in the array. The parameters assigned to the array (color, material) also affect the array source objects.

Separate to Solids (available for arrays of Bodies and operations). Setting this flag turns on the mode of dividing the array into separate bodies (each array copy is automatically turned into a new Body).

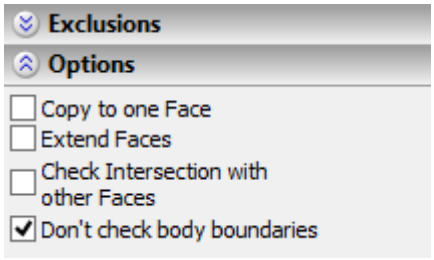
Combine Result to one Solid (available for arrays of Bodies and operations). With this flag set, the array copies are merged together by the Boolean addition operation.

Additional parameter **Include in product structure** is relevant only for arrays based on 3D fragments. It determines what will be put into product structure:

Always Source. Only source fragment will be included into product structure regardless of array parameters. Results of array creation are not taken into account in product structure;

Array results. The results of array creation are included into product structure. Depending on the status of the flag **Source**, the result can be either all copies of array (when the flag is activated) or copies of array and the source object (flag is turned off).

The optional parameters of the array of faces are different from the parameters of other arrays:



Copy to one face. Set this flag to turn on the "Copy to one face" mode.

Extend Faces. Turns on the mode of extending faces. This is always used together with the previous flag.

Check Intersection with other Faces. This flag turns on the mode of checking for intersections of the created array copies with the existing body faces.

Don't check body boundaries. This flag turns on the identically-named mode. By default the flag is enabled.

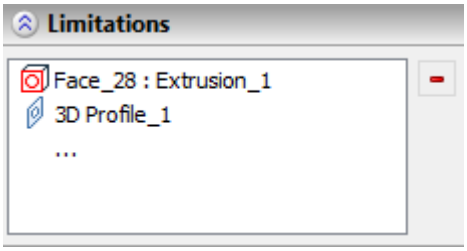
Defining Limitations

To specify a 3D element defining the limitation scope, activate the following option in the command automenu:

	<Alt+C>	Select limiting elements

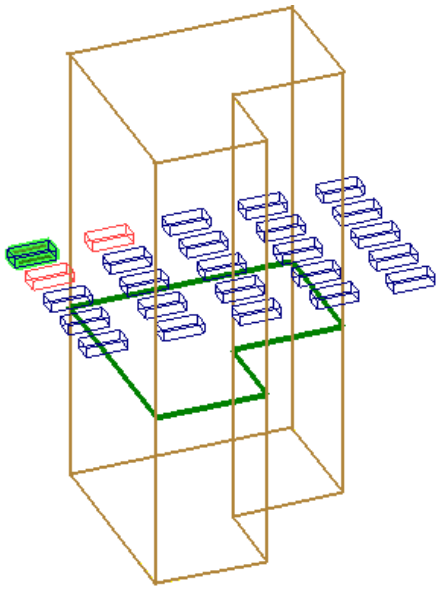
3D elements are selected in the 3D scene by clicking . 3D profiles and operations can also be selected in the 3D model tree. The selected objects are put in the list of limiting objects in the section "Limitations" of the operation properties window.

To cancel selection of some of the specified objects, click those in the list of the limiting objects and press the button .



To start defining limitations, you can simply focus on the respective entry of the operation properties window. The option for specifying the limiting objects will be activated automatically.

If a flat 3D object was selected, the limitation scope will be hinted in the 3D scene as a sweep along the normal direction of the selected object. If non-flat the limiting objects are specified, the limitation scope will be created only upon specifying the limitation axis.



To define the limitation axis by a single 3D object, use the option:

	<Alt+D>	Select auxiliary direction of limits

To define the axis by the start and end points, use the options:

	<Alt+F>	Select first point of auxiliary direction of limits
	<Alt+S>	Select second point of auxiliary direction of limits

To reject selection of the limitation axis, use the option:

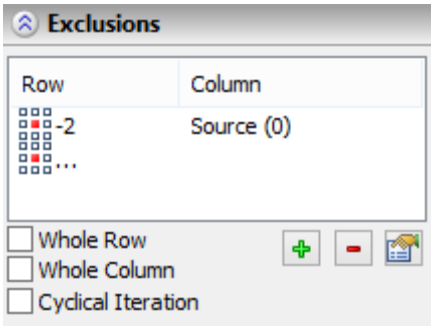
	<Alt+K>	Cancel selection of auxiliary direction of limits

Defining Exclusions

To specify the array elements to be excluded, activate the following automenu option:

	<ALT+X>	Select elements to Exclude

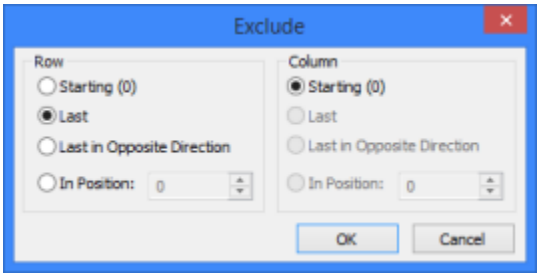
To select an element to be excluded, just click its preview image in the 3D scene by the . The selected element will be highlighted *Gray* in the 3D scene and added to the list of elements to exclude in the "Exclusions" section of the properties window.



The list of excluded elements is made into a table that displays the numbers of the rows and columns of the selected exclusions. To remove an exclusion from the list, pick it in the list and press the button .

The position of the elements specified by selecting in the 3D scene, is entered in the table using numerical notations for the row and the column. The button allows defining an exclusion by using descriptive characteristics of its position. Upon pressing the button, a dialog box appears for specifying an element to exclude.

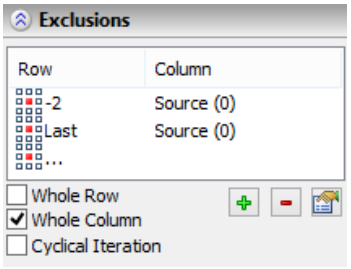
In this dialog, specify, at intersection of which row and column lies the element to exclude, by setting the **Row** and **Column** switches in the desired states. Besides the descriptive characteristics, you can also use here they numerical notations of rows and columns. To do this, select the **In Position** choice and enter the number of the row/column.



Upon closing the dialog box by pressing the **[OK]** button, the specified exclusion will appear in the list of the elements to exclude.

To use the mode of excluding a row/column in the mode of cyclical repetitions for one of the specified exclusions, select it in the list of the elements to be excluded in the properties window and set a flag for it, representing the desired mode:

- **Whole row** – sets the row exclusion;
- **Whole column** – sets the whole column exclusion;
- **Cyclical Iteration** – sets cyclical repetitions of the exclusion.



3D COPYING

3D copy – operation of the T-FLEX CAD system which allows a user to copy existing operations, Bodies, 3D construction elements and even collections of faces.

Parameters of copying are determined by specifying two LCS: source coordinate system of copying and target one. The transformation is performed by moving the source coordinate system along with connected with it objects to the location occupied by the target coordinate system.

The operation of creating 3D copy allows a user upon copying to symmetrically reflect selected elements with respect to one of the planes of the target coordinate system.

When creating 3D copy of operations, it is possible to automatically combine the created copy with the source operation, and also, use the move mode (when the source bodies are removed upon copying).

MAIN PRINCIPLES

Objects of Copy

The collection of copied objects of a single operation can contain several elements of the same type (only operations, only Bodies, only faces, etc.). Depending on the type of the 3D copy (i.e. on the type of the selected objects of copy), parameters and capabilities of the operation will be changing (see section “Types of 3D copies”).

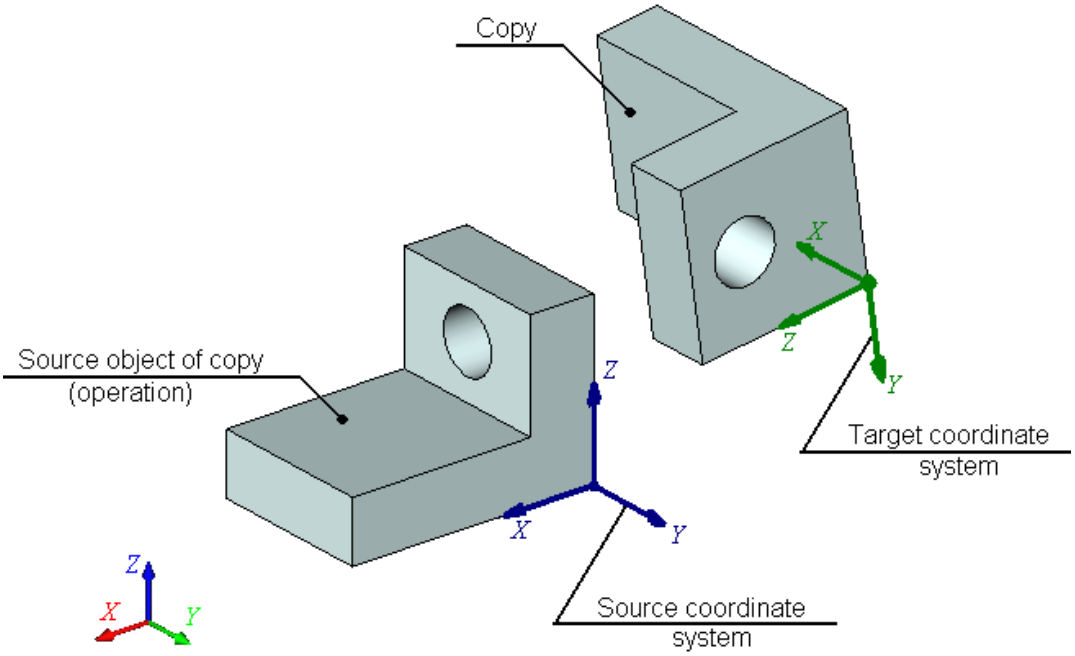
Source and Target Coordinate Systems

Any existing LCS can be selected as the source and the target coordinate systems.

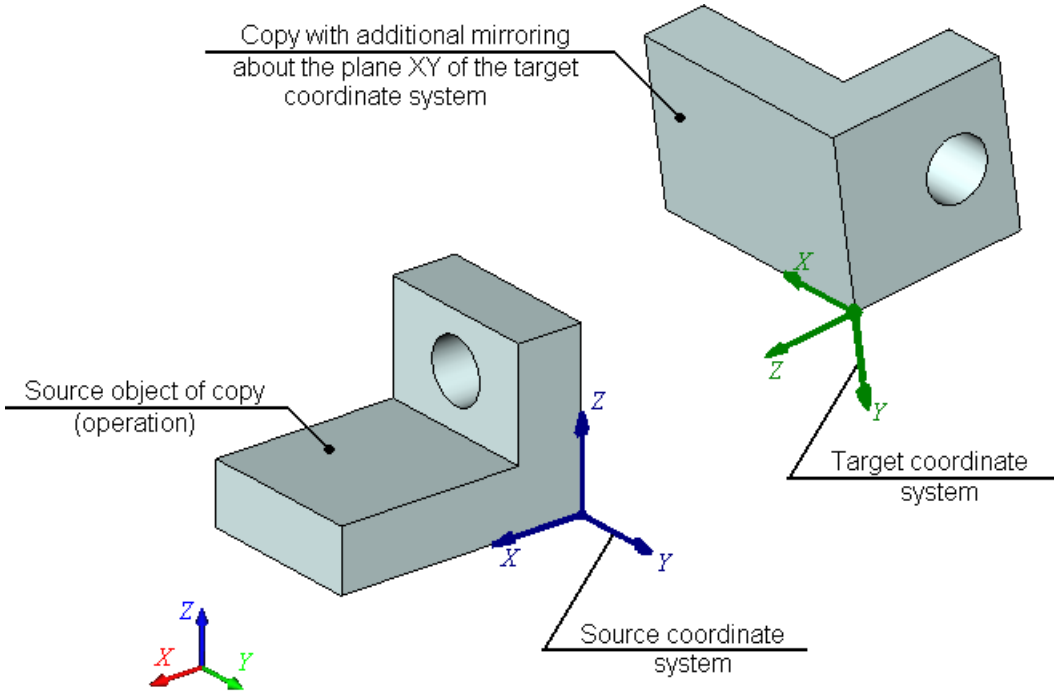
If the required LCS (source or target) is not found in the 3D model, it can be created automatically when defining the 3D copy. To do this, just select a 3D point as the origin for the LCS being created. A new 3D element – LCS will be created at this point. The axes of the newly created LCS will be directed along the axes of the world coordinates.

General Principles of Copying

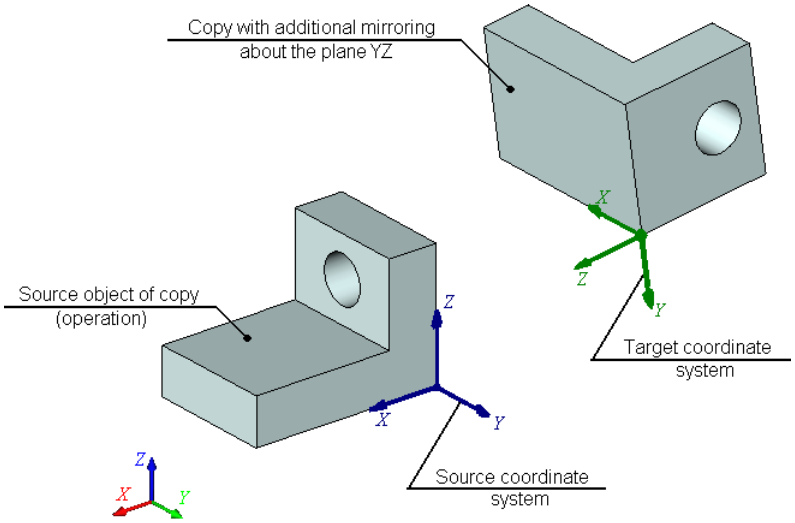
The position the copy being created is defined in a general case by carrying the source coordinate system over the target one.



The copy being created can be additionally subjected to symmetry (mirroring) about one of the principal planes of the target coordinate system.



When creating a copy with additional mirroring, one can specify just the source LCS. In this case, the system assumes that the source and the target LCS coincide.



The additional mirroring of the copy produces the result similar to that of the 3D Symmetry operation. Such copies can be used, for example, for creating symmetrical parts. You create the half-part first, then do copy with mirroring, and, finally, the Boolean addition of the two halves.

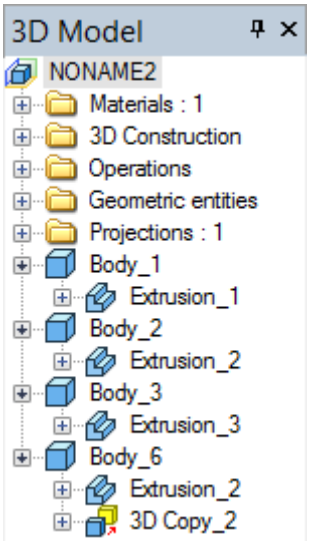
Types of 3D Copies

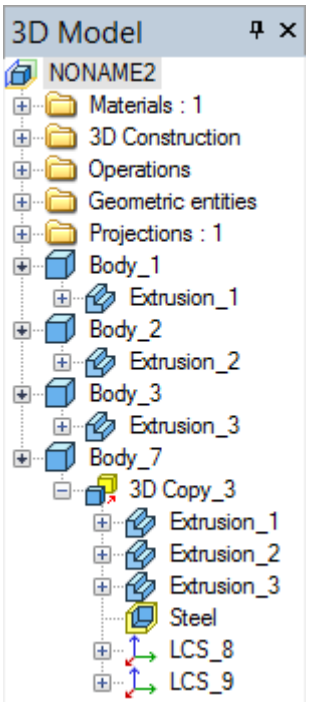
Copy of Operations

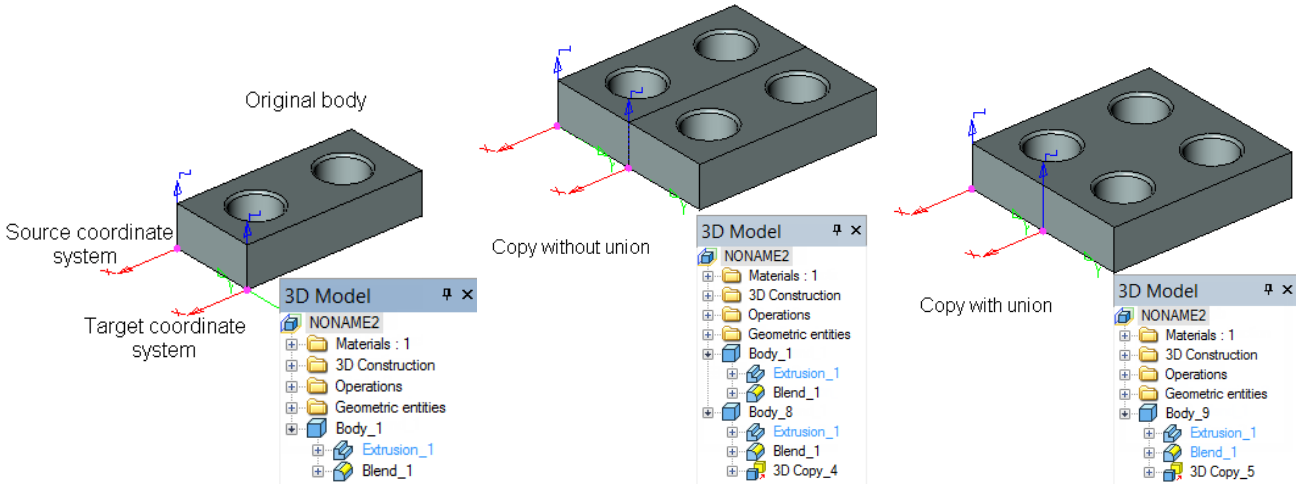
3D copy of operations allows a user to copy existing operations of the model. As a result of creating the copy operation, new Bodies appear – copies of the source operations.

In the structure of the model the result of 3D copy of operations is shown in the following way: if for creating the copy only one operation was selected, then the last operation of the created Body-copy will be the 3D copy operation itself. On the other hand, if the 3D copy was created for several objects simultaneously, then the references to the copy operation will serve as the basic operations of the new Bodies.

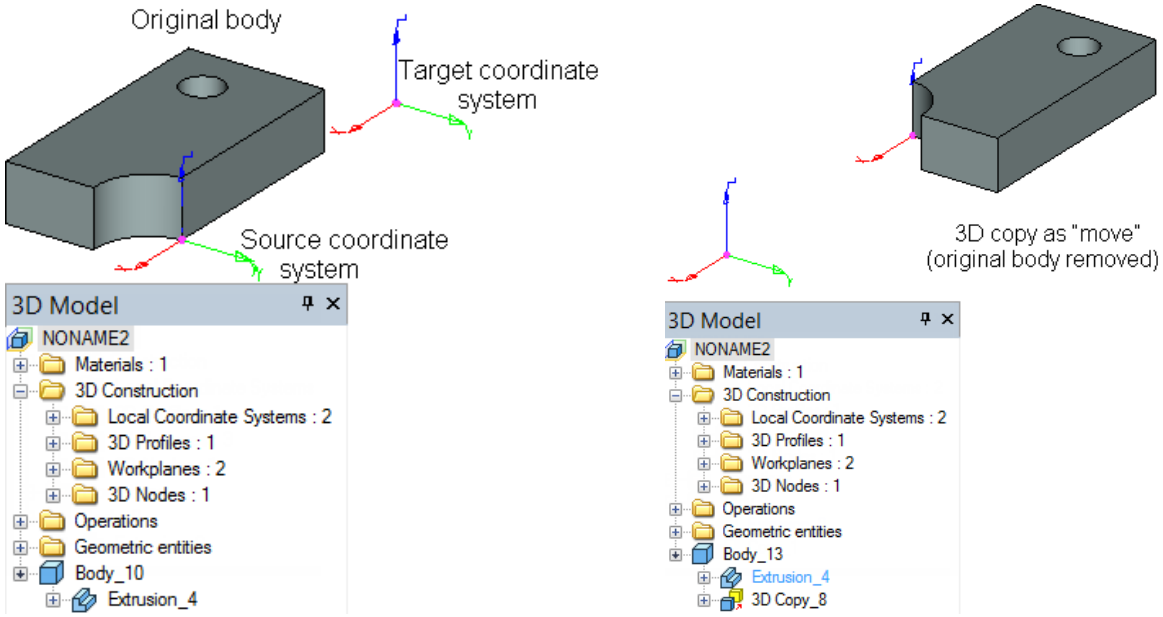
Optional mode of *copying with union* automatically combines the created copy with the body of the source operation. If the collection of the objects of copy contains more than one operation, then each created copy is combined with the corresponding to it source operation.







3D copy of operations can work in the move mode. In this case, the body of the source operation is removed, and only created copy remains in the 3D scene and in the tree of the 3D model.



Copy of Bodies

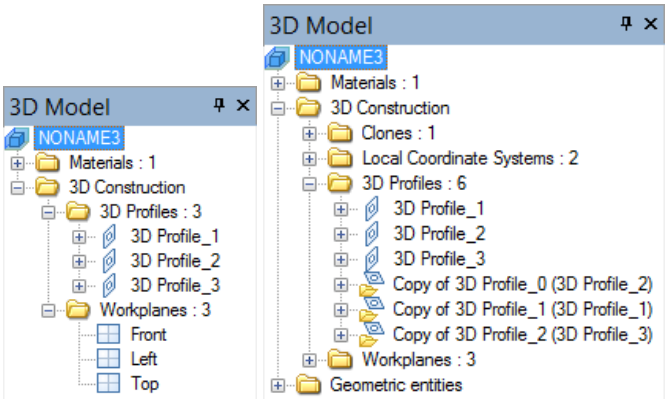
3D copy of Bodies is created in the same way as the copy of operations. But while the 3D copy of operations copies only results of specific operations, the copy of Bodies copies the whole Body obtained by using the selected operation. If this Body is modified in the future, the 3D copy will be regenerated, accounting for new operations included in the Body history (similar to arrays of Bodies).

Copy of 3D Construction

The copy operation allows a user to create copies of any 3D construction elements.

In the tree of the 3D model, the created operation is put into the special folder "3D Construction\Copies". The elements-copies, obtained as a result of using this operation, are also put into the corresponding folders of the tree. For example, when creating the copy of a 3D profile, the resulting profile-copy will be put into the folder "3D Construction/3D Profiles".

When copying workplanes and LCS, the additional transformation of symmetry can not be applied to the created copies.

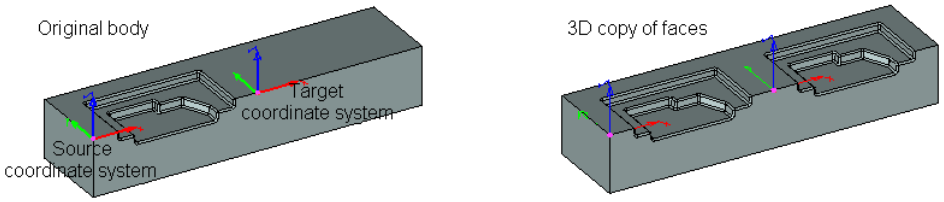


The tree of the 3D model before and after creating 3D copy of construction elements

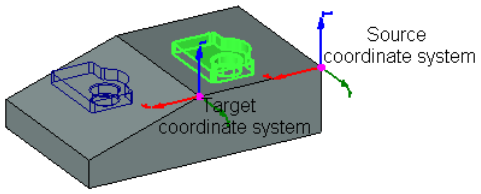
Copy of Faces

Copy of faces allows a user to add to the bodies present in the 3D model the holes and raised portions which duplicate already exiting structural elements.

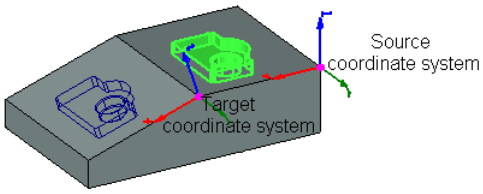
3D copy of faces is always created on the basis of a single body of the model: all faces being copied have to belong to the same body. The outcome of creating the copy operation – collection of faces-copies – is also included into the source body.



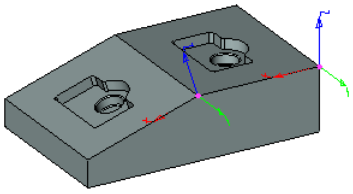
The collection of faces selected for creating 3D copy can be arbitrary, but it has to fully describe the geometry of the copied structural element of the part. Faces-copies have to lie within the bounds of the source body and intersect with its other faces in the same way as the faces of the original collection.



Intersection between faces of the copy and faces of the original body differs from the intersection of the original collection of faces (copy of faces will not be created)

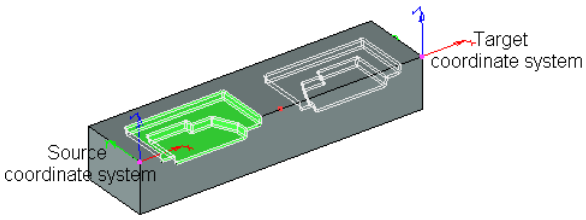


Copy intersects the original body in the same way as the original collection of faces (3D copy will be created successfully)

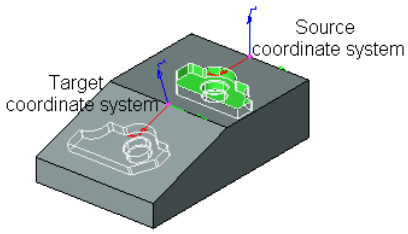


3D copy of faces

When creating the copy with the symmetry, one more additional condition is imposed on the source collection of faces: all edges bounding the collection of faces being copied have to be belong to the same “base” (not included into the list of copied faces) face of the body.



3D copy with symmetry relative to YZ plane will not be created, because edges bounding the collection of faces belong to more than one face used as "base"



3D copy with symmetry relative to XZ will be successfully created, because all edges bounding the collection of faces belong to one "base" face

Specifying the initial collection of faces is carried out in the same way as in the arrays of faces by:

Selection of closed sequence of edges bounding the desired collection of faces.

Direct selection of all desired faces;

Selection of the operation that created the desired collection of faces.

The outcome of creating the face copy operation can be also affected by the following additional modes:

Copy to one Face. Use of this mode considerably reduces the copy regeneration time;

Extend Faces (used only in combination with the “Copy to one face” mode). The system will be automatically extending or trimming the copied faces in order to insure their correct connection to the body;

Check Intersection with other Faces (used to avoid body self-intersections).

> Mode detailed description of these modes can be found in the chapter “Arrays”.

Limitations of Copy Operation

When creating and editing copy operations the following limitations exist:

- If at least one of the source objects of 3D copy was used for creating the body (including 3D weldment), then the mode “Move” will be locked;
- If at least one of the objects-results of the operation was used for defining any other object of the TFLEX CAD model, then upon editing 3D copy, the option of modifying the collection of the source objects will be locked as well as the option of turning on/off the mode of copying with union (for 3D copy of operations);

These limitations are relevant to the 3D copies of all types except the copy of faces.

Moreover, when copying the workplanes or LCS, the additional transformation of symmetry cannot be applied to the created copies.

Copy Creation

To create a 3D copy use the command **3CP: Create Copy**:

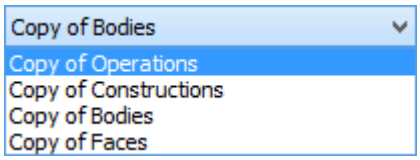
Icon	Ribbon
	3D Model → Create → Copy
Keyboard	Textual Menu
<3CP>	Operation > Copy

To perform the copy operation, do the following steps:

- ✓ Select the source objects of copy;
- ✓ Select the source coordinate system;
- ✓ Select the target coordinate system;
- ✓ Specify parameters of copy;
- ✓ Finish the operation creation.

Selection of Copy Type

The type of the operation being created is selected in the command's properties window with the help of drop down list: **Copy of Operations**, **Copy of Construction**, **Copy of Bodies**, **Copy of Faces**.



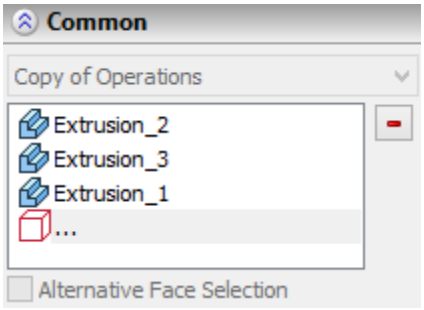
Depending on the type of copy selected in the list, the set of options in the command's automenu and the set of parameters in the properties window will be changing.

Selection of Source Objects of Copy

For selection of the source objects of copy, one of the following options will be available in the automenu (depending on the type of operation selected):

	<N>	Select source 3D operation
	<N>	Select source 3D operation
	<N>	Select Construction
	<N>	Select Faces to Copy or their boundary Edges or Loops

Selection of Source Objects for Copying 3D Operations, Bodies, 3D Construction



The objects can be selected directly in the 3D scene or in the model tree. The selected objects will be put into the list of the copy source objects in the operation properties window.

To select a 3D Body, you can pick *any* of the operations contributing to this Body (again, either in the 3D scene, if that is possible, or in the 3D model tree). The *Last* operation of the given Body will be automatically put in the list of the selected objects.

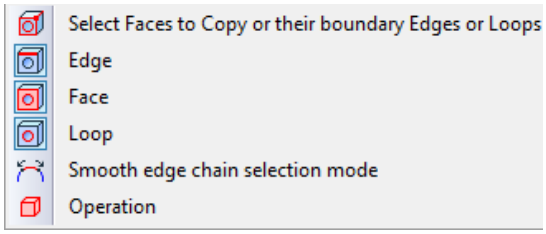
Note that after selection of the first object (operation, construction element), the option / / will be automatically turned off, and the system will move to specifying the source coordinate system. For selecting several objects of copy, a user has to independently reactivate the option for selection of the objects of copy in the command's automenu or place the cursor in the row "..." in the list of the selected objects (in the command's properties window).

To deselect any of the previously selected objects, indicate it in the list and press the button .

> Note that once the first object to be copied is selected, the entry defining the copy type becomes inaccessible. To change the copy type, you need to remove all 3D objects from the list of the copy source objects. Only after that, you will be able to select another copy type.

Selecting source collection of faces for copy of faces

Selection of the source collection for copying faces is done in the same way as selection of the source faces for the array of faces, described in the chapter "Arrays".

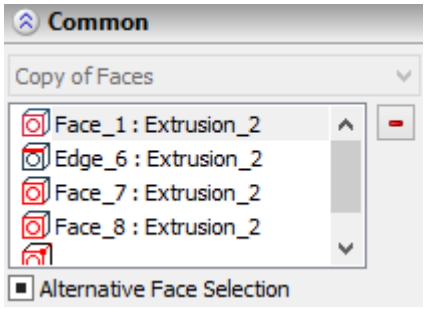


The pull-down list of the option for selecting a set of faces to copy allows choosing the way of selecting: direct selection of faces, selection of their bounding edges, selection of the operation (the same filters are duplicated by the system toolbar).

> Details on the work with filters from the pull-down list of the option are given in the chapter "Arrays", when describing creation of the array of faces.

Selection of operations, bounding edges or the faces themselves is done in the 3D window. Operations can be also selected from the tree of the 3D model.

When specifying the collection of faces with the help of bounding edges, an additional flag **Alternative Face Selection** shows which of the two collections of faces defined by the specified collection of edges will be chosen as the source object of copy. The flag has three states:



On by default ■. The system selects the collection of faces automatically (by the least area condition). If the further manipulations with the 3D model will alter the balance of the areas, the system will automatically switch to the other set of faces as the source object of the copy;

On ☑/Off ☐. Each of these two states will correspond to a specific collection of faces (correspondence is determined by the system itself). With any further modifications of the model, the collection of faces being copied does not change.

When creating the copy of faces, the option remains active until a user independently activates the option for selection of the source/target coordinate system.

Selection of Source Coordinate System

Selection of an existing LCS as a source coordinate system is done by using the option:

	<S>	Select source LCS

For automatic creation of an LCS, use the option:

	<F>	Select Point for creating Source LCS

This option allows selecting an arbitrary 3D element suitable for defining a 3D point that will be the new LCS origin.

Selection of Target Coordinate System

Selection of the target coordinate system is done similar to selecting the source coordinates. An existing LCS is selected by the option:

	<T>	Select target LCS

A new LCS can be created at a 3D point by using the option:

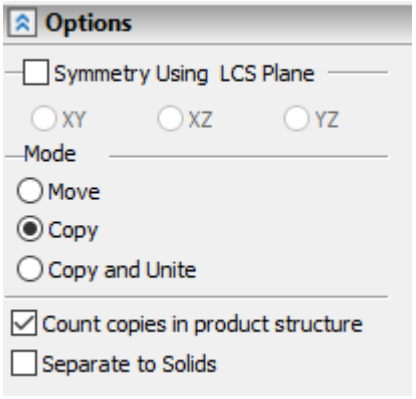
	<W>	Select Pont for creating Target LCS

To refuse the selection of LCS, use the option:

	<C>	Cancel selection of target coordinate system

Specifying Parameters of Copy

The necessity of applying the symmetry transformation to the copies being created is specified in the command's properties window with the help of the flag **Symmetry Using LCS Plane**. The plane of the target coordinate system, with respect to which the symmetry will be performed, is selected with the help of the toggles: **XY**, **XZ**, **YZ**.



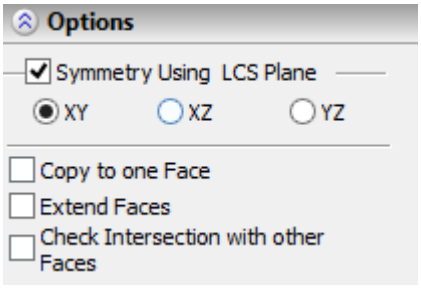
In the dialog of the operation's parameters invoked by the option , it is also possible to specify parameters for transformation of the copy. Displacements and angles of rotation specified there will be applied with respect to the target coordinate system of the copy.

Additional flag **Count Copies in product structure** is used only when copying 3D fragments included in the BOM. When checking this flag, the copies of the fragments will be also taken into account in the product structure. When this flag is turned off, only the original fragment is included in the product structure.

Flag **Separate to solids** is used when copying several operations at once. A separate Body-copy is created for each of them.

When creating 3D copy of operations, the copy mode is selected with the help of a toggle **Mode** in the group **Options** in the properties window of the command: **Move**, **Copy**, **Copying and Unite**.

The optional parameters of the copy of faces are different from the parameters of other copies:



Copy to one face. Set this flag to turn on the "Copy to one face" mode.

Extend Faces. Turns on the mode of extending faces. This is always used together with the previous flag.

Check Intersection with other Faces. This flag turns on the mode of checking for intersections of the created array copies with the existing body faces.

3D SYMMETRY

3D symmetry – operation of the T-FLEX CAD system which allows a user to symmetrically reflect selected 3D objects with respect to a given plane. 3D construction elements, operations, bodies, faces can be selected as source objects for creating the symmetry (objects of symmetry).

When creating the symmetry of operations, it is possible to automatically combine the created copy with the source operation as well as use the move mode (when the source bodies are removed upon copying).

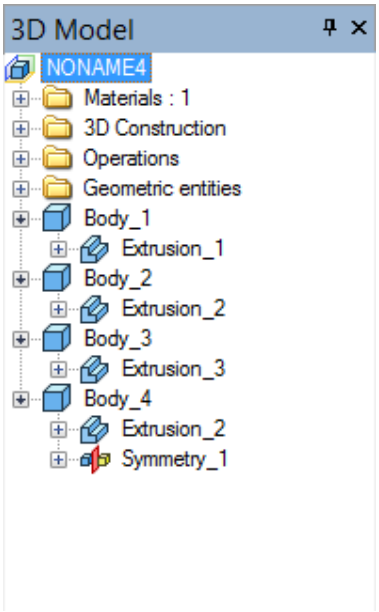
SYMMETRY TYPES. GENERAL INFORMATION

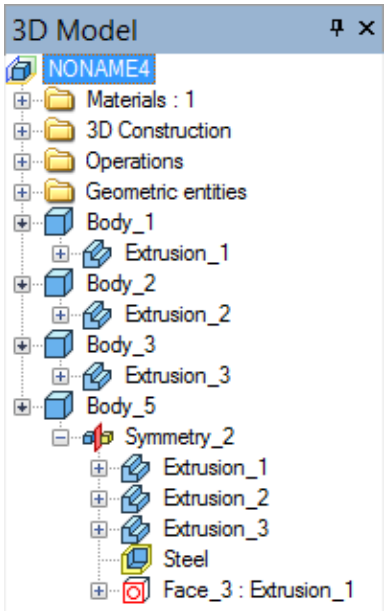
The collection of copied objects of a single operation of symmetry can contain several elements of necessarily the same time (only operations, only Bodies, only faces, etc.). The result of execution of the operation and its parameters depend on the type of the initial data of symmetry.

Symmetry of Operations

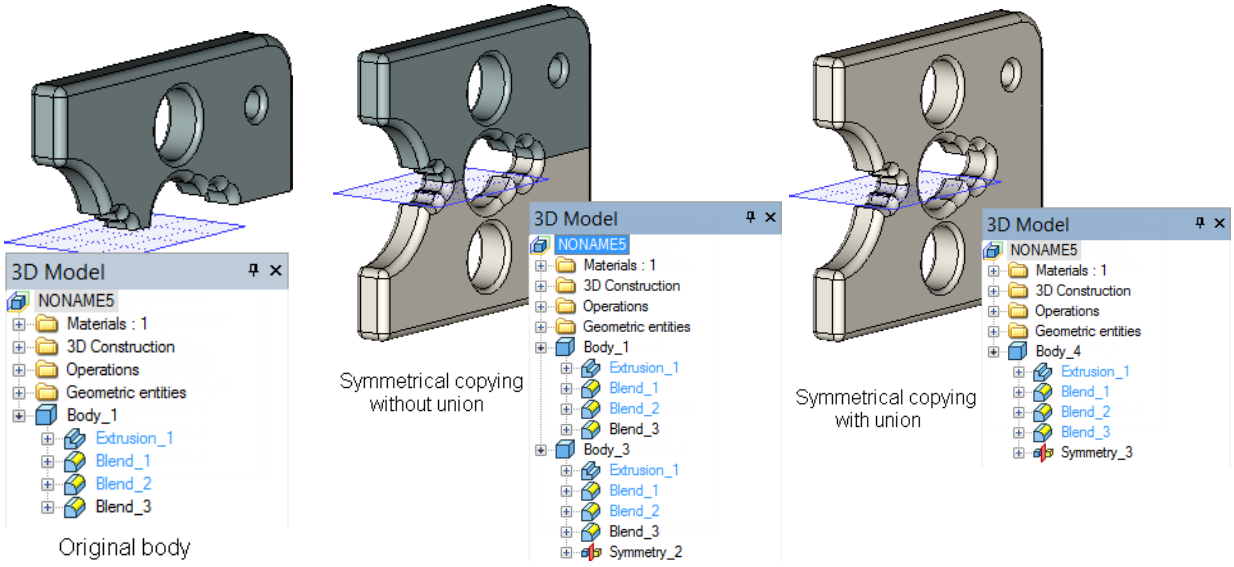
Symmetry of operations allows a user to symmetrically copy the existing operations of the model. As a result, new Bodies appear – symmetrical copies of the initial operations.

In the structure of a 3D model, the result of symmetry of operations is shown in the following way: if for creating the symmetry only one operation was selected, then the last operation of the created Body-copy will be the symmetry operation itself. On the other hand, if the symmetry was created for several operations simultaneously, then the references to the symmetry operation will serve as the basic operations of new Bodies.

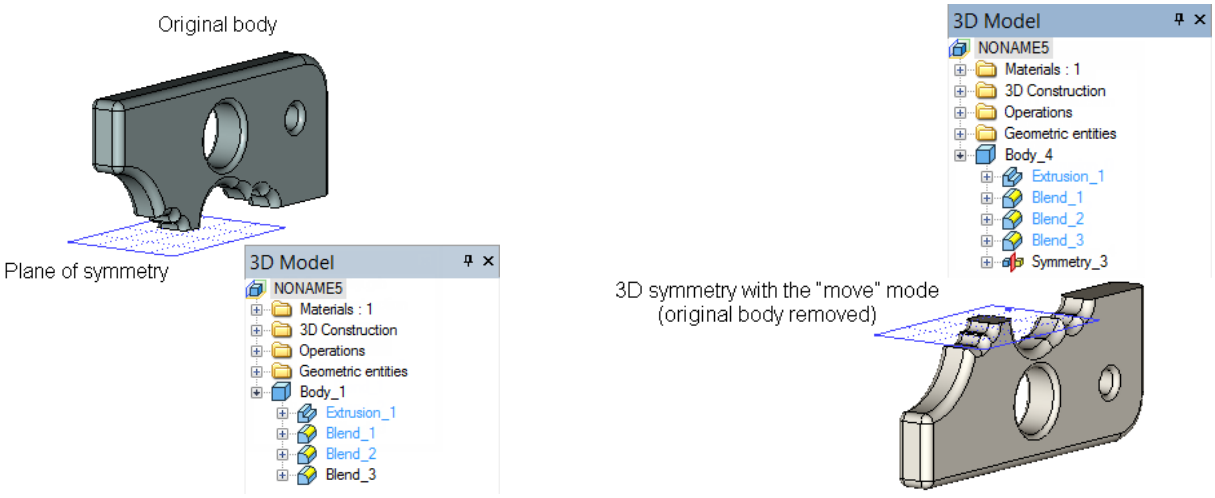




Optional mode of *copying with union* automatically combines the created copy with the body of the source operation. If the collection of the symmetry objects contains more than one operation, then each created symmetric copy is combined with the corresponding to it source operation. This mode can be useful for designing the assemblies with large number of symmetric parts.



3D copy of operations can work in the move mode. In this case, the body of the source operation is removed, and only created copy remains in the 3D scene and in the tree of the 3D model.



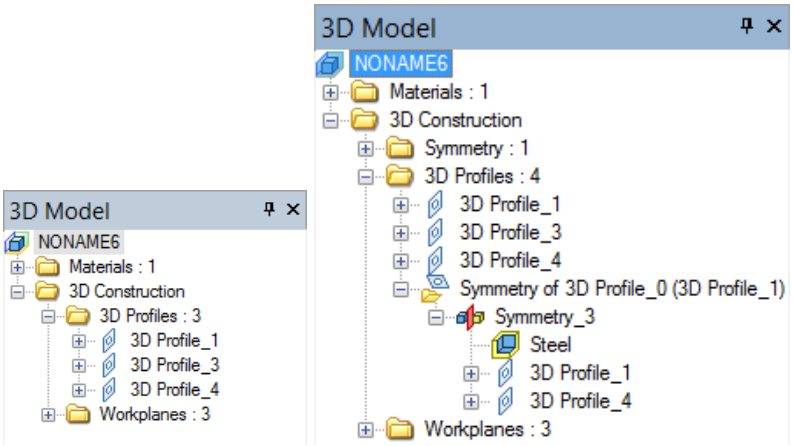
Symmetry of Bodies

Symmetry of Bodes is created in the same way as the symmetry of operations. But while the symmetry of operations copies only the results of specific operations, the symmetry of Bodies copies the whole Body obtained by using the selected operation. Later when this Body is modified, the symmetry will be regenerated taking into consideration new operations entered into the Body history.

Symmetry of 3D Construction

The symmetry operation allows a user to symmetrically copy any 3D construction elements except workplanes and LCS. As in the previous case, in the framework of a single operation, several 3D construction elements can be simultaneously subjected to the symmetry.

In the tree of the 3D model, the created operation is put into a special folder "3D Construction\ Symmetry". The elements-copies, obtained as a result of using this operation, are also put into the corresponding folders of the tree. For example, when creating the symmetry of the 3D profile, the resulting profile-copy will be put into the folder "3D Construction\3D profiles".

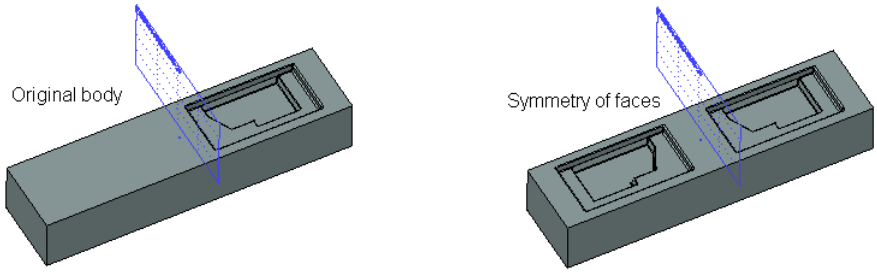


The tree of the 3D model before and after creating the 3D symmetry of construction elements

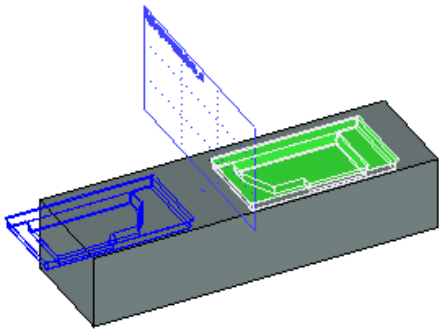
Symmetry of Faces

The symmetry of faces, as well as of the array of faces, and 3D copy of faces, allows a user to add to the bodies present in the 3D model the holes and raised portions which duplicate already existing structural elements.

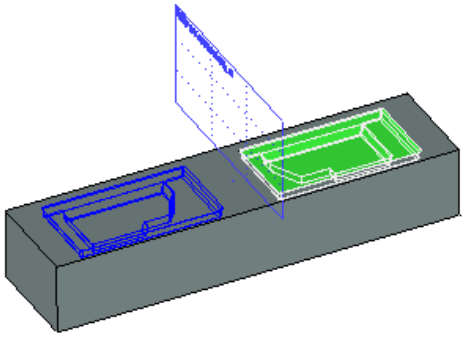
All copied faces of a single symmetry operation have to belong to the same body. The outcome of creating the symmetry operation – collection of faces-copies – is included into the source body as well.



The collection of faces selected for creating the symmetry has to fully describe the geometry of the copied structural element of the part. Faces-copies have to lie within the bounds of the source body and intersect with its other faces in the same way as the faces of the original collection.



Symmetrical faces intersect the body differently compared to the original collection of faces (symmetry of faces will not be created)



Symmetrical faces intersect the body the same way as the original collection of faces (symmetry of faces will be created successfully)

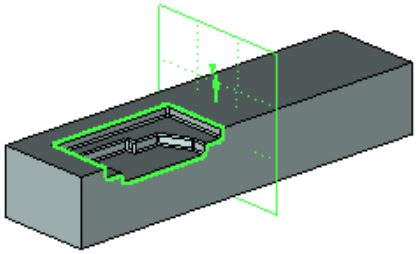
Specifying the initial collection of faces is carried out in the same way as in the arrays of faces by:

Selection of closed sequence of edges bounding the desired collection of faces.

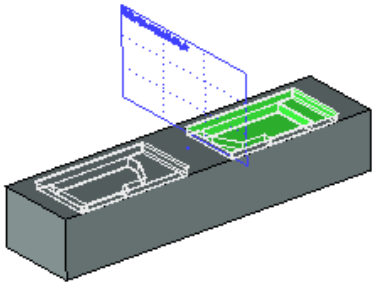
Direct selection of all desired faces;

Selection of the operation that created the desired collection of faces.

In addition, one more additional condition is imposed on the source collection of faces of symmetry: all edges bounding the collection of faces being copied have to belong to one “base” (not included into the list of copied faces) face of the body.



Symmetry of faces will not be created, because edges bounding the collection of faces belong to more than one face used as "base"



Symmetry of faces will be successfully created, because all edges bounding the collection of faces belong to "base" face

The following additional modes can affect the outcome of the face symmetry operation:

Copying to one face. The use of this mode significantly reduces the symmetry regeneration time;

Extend surfaces (is used only along with the mode “Copying to one face”). The system will automatically extend or trim the faces being copied for assuring their proper insertion into the body;

Check intersection with other faces (is used to prevent creation of self-intersecting body).

> More detailed description of these modes can be found in the chapter “Arrays”.

Creation of Symmetry

For creating symmetry operation, the command **3SY: Create 3D Symmetry** is used:

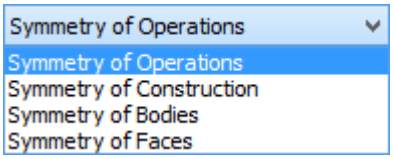
Icon	Ribbon
	3D Model → Create → Symmetry
Keyboard	Textual Menu
<3SY>	Operation > Symmetry

For creating the operation of symmetry, these steps have to be followed:

- ✓ Select the source objects of symmetry;
- ✓ Select the plane of symmetry;
- ✓ Select the parameters of symmetry (for example, the mode of symmetry);
- ✓ Finish creation by pressing in the automenu or in the command's properties window.

Selection of Symmetry Type

The type of the operation being created is selected from the command's properties window with the help of drop-down list: **Symmetry of Operations**, **Symmetry of Construction**, **Symmetry of Bodies**, **Symmetry of Faces**.



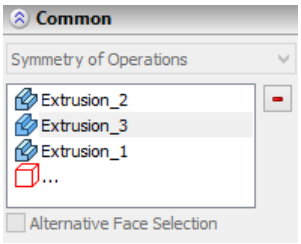
Depending on what type of the symmetry is selected from the list, the set of options in the command's automenu and the set of parameters in the properties window will be changing.

Selection of Source Objects of Symmetry

For selection of the source objects of symmetry, one of the following options will be available (depending on the type of symmetry selected) in the automenu:

	<N>	Select source 3D operation (for symmetry of operations)
	<N>	Select source 3D operation (for symmetry of Bodies)
	<N>	Select Construction
	<N>	Select Faces to Copy or their boundary Edges or Loops

Selection of Source Objects for Creation Symmetry of 3D Operations, Bodies, 3D Construction



The objects can be selected directly in the 3D scene or in the tree of the 3D model. Selected objects are put into the list of the source objects of symmetry in the operation's properties window.

For selection of a 3D Body (also either in the 3D scene, if it is possible, or in the tree of the 3D model) *any* operation taking part in the creation of the given Body can be chosen. The *last* operation of the given Body will be automatically put into the list of the selected objects.

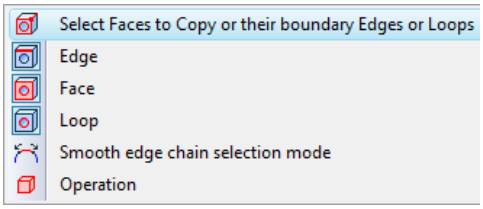
After selection of the first object (operation, construction element), the option will be automatically turned off and the system will move to specifying the symmetry plane. For selecting several objects of symmetry, a user has to independently reactivate the option for selection of the objects of copy in the command's automenu or place the cursor in the row "..." found in the list of the objects of copy (in the command's properties window).

To deselect any of the previously selected objects, indicate it in the list and press the button .

> After selection of the first object of copy, the entry for specifying the operation type will be inaccessible for editing. To change the type of the copy, it is necessary to remove all 3D objects already put into the list of the source objects. Only after that, another operation type can be selected.

Selection of Source Objects for Faces Symmetry

Selection of the source collection for copying faces is done in the same way as selection of the source faces for the array of faces (see the chapter "Arrays").



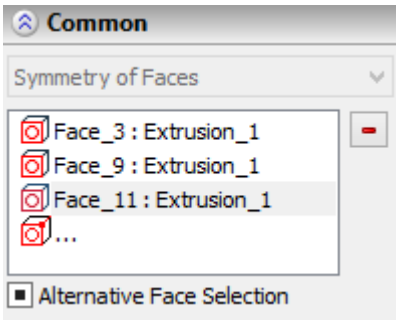
The drop down list of the option for selection of the collection of faces to copy allows a user to specify the selection approach: selection of the faces themselves, selection of the edges bounding the faces, selection of the operation (the same filters are duplicated in the system toolbar).

Selection of operations, bounding edges or the faces themselves is done in the 3D window. Operations can be also selected in the tree of the 3D model.

When specifying the collection of faces with the help of bounding edges, the flag **Alternative Face Selection** shows which of the two collections of faces defined by the specified sequence of edges will be selected as the source object of symmetry. The flag has three states:

Turned on by default ■. The system itself chooses the collection of faces (from the least area condition). If during the process of subsequent manipulations with the 3D model, the balance of the areas changes, the system will automatically change the original collection of faces;

Turned on ☑**/Turned off** ☐. Each of these two states will correspond to a specific collection of faces. The collection of faces corresponding to a specific state of the flag is chosen by the system arbitrarily after specifying the collection of faces. With any further modifications of the model, the collection of faces being copied does not change.



When creating the symmetry of faces, the option remains active until a user independently activates the option for selection of the symmetry plane.

Note! To receive correct model, faces-copies have to lie within the bounds of the source body and intersect with its other faces in the same way as the faces of the original collection.

Selection of Symmetry Plane

Symmetry plane is selected with the help of the following option of the automenu:

	<S>	Select reflection plane

When creating symmetry of 3D operations, Bodies, 3D construction elements, this option is turned on automatically after selection of the first object of symmetry. If at that moment a user turns off this option (for selecting several objects of symmetry), then after selecting all desired objects, a forced activation of this option will be required.

When creating the symmetry of faces, the option always requires independent activation by the user.

The option contains the drop-down list of filters for selecting 3D objects capable of defining the plane. Selection is performed in the 3D window or in the window of the tree of the 3D model.

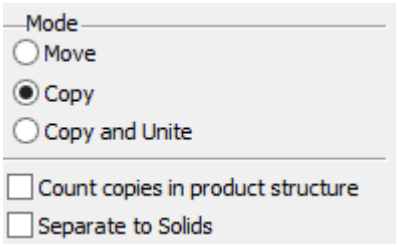
To cancel selection of the symmetry plane, the following option can be used:

	<K>	Cancel selection of reflection plane

Specifying Symmetry Parameters

Specifying Parameters for Symmetry of 3D Operations, Bodies and 3D Construction

The mode of symmetry is selected with the help of a toggle **Mode** in the group **Options** in the properties window of the command: **Move**, **Copy**, **Copy and Unite**.



In some cases, the options "Move" and "Copy and Unite" may not be accessible (see section "Limitations of Symmetry Operations", as well as description of features of various types of symmetry).

Additional flag **Count Copies in product structure** is used only when copying 3D fragments included in the product structure. When checking this flag, the copies of the fragments will be also taken into account in the BOM. When this flag is turned off, only the original fragment is included in the product structure.

Flag **Separate to solids** is used when copying several operations at once. A separate Body-copy is created for each of them.

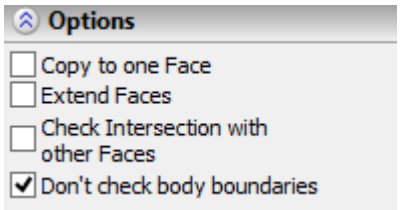
Specifying Parameters for Symmetry of Faces

Optional parameters of symmetry of faces differ from the parameters of symmetry of other types:

Copy on one Face. This flag is set on when the mode "Copying to one face" needs to be turned on.

Extend Faces. This flag turns on the mode of the same name. It is always used along with the previous flag.

Check Intersection with other Faces. This flag allows a user to turn on the mode of checking for intersections of the created copies with already existing faces of the body.



Don't check body boundaries. This flag allows to create symmetry of faces without checking self-intersection and correctness of created geometry. It significantly increases performance. This option should be used only if you are sure in the correctness of the resulting model.

CUT BY SECTION

The cut by section operation allows creating a 3D body by cutting some portions off the original body, or getting two new bodies by cutting the original body into two parts. Operation can be applied either to solid or to sheet bodies, resulting in a body(s) of the same type.

DEFINING CUT BY SECTION OPERATION

The cut by section operation is called by the command **3CU: Cut By Section**:

Icon	Ribbon
	3D Model → Modify → Cut by section
Keyboard	Textual Menu
<3CU>	Operation > Cut by Section

Upon calling the command, the automenu provides both the general system options and the specific ones for creating a cut by section and a separation:

	<B>	Select 3D operation
	<S>	Select section
	<W>	Select intersecting Workplane
	<P>	Select intersecting Profile
	<K>	Cancel selection of cutting surfaces
	<Tab>	Switch view direction
	<M>	Separate into two Solids

Creating a Cut by Section

The cut by section operation definition includes the following steps:

- ✓ **Selection of a body** (the option is activated automatically). Select a body to subject to the cut by section operation in the 3D view window or in the 3D model tree. The selected body will be highlighted.

- ✓ **Selection of the cutting element or set of such elements**.

  You can use the following cutting elements: a 3D section, a worksurface, a workplane, a 3D profile, a face of a body or a sheet body. The body is slashed along the surface of the cutting element.

  You can specify a set of cutting elements. The set can include faces, 3D profiles, sheet bodies (selection of several 3D sections or workplanes is not allowed). In this case, the elements are sewed first into one sheet body, and then the cut by section is performed over the original body by the resulting sewed body. When using a set of cutting elements, the necessary conditions on these elements are: first, it should be possible to sew the elements; second, the resulting sewed body should fully intersect the original body.

  Selection of the cutting elements is done using the automenu options. The 3D section and the workplane are selected using the options and , respectively. The set of cutting elements (faces, 3D profiles, sheet bodies) is defined by the option . A stand-alone 3D profile (face, sheet body) is specified as a set consisting of one element.

  Upon selecting the cutting element, arrows are displayed on its sides pointing towards the portion of the body that will be left in the scene upon performing the cut by section. The direction of the arrows can be flipped by the option <Tab> (the icon).

- ✓ **Definition of the general operation parameters** (*optional)* by the option . The detailed description of the general operation parameters is provided in the chapter "General parameters of 3D elements".

- ✓ **Confirmation for the operation creation** (in the automenu or in the property window).

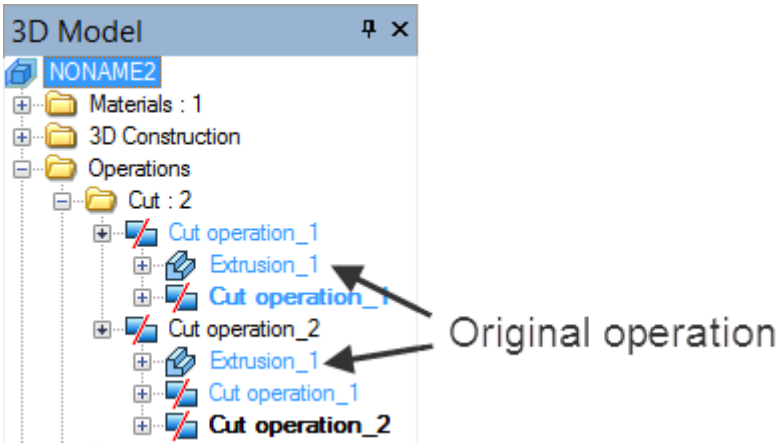
Upon confirming the operation creation, the new 3D body will be displayed in the 3D view window.

Making Body Separation

To divide a body into two parts, additionally set the option:

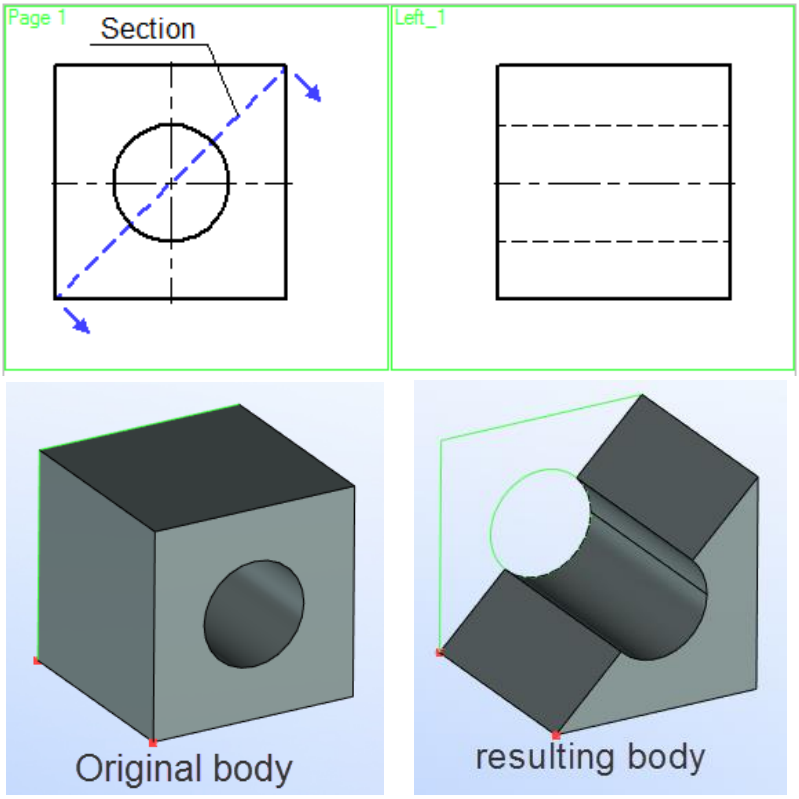
	<M>	Separate into two Solids

In this case, two cut by section operations with identical parameters are created automatically on the original body. Editing of such operations is also done jointly. To define parameters of one of the operations separately, call its parameters dialog box without entering the editing mode (for example, by using the context menu for this operation). The two operations are deleted separately.



Examples of Creating the Operation

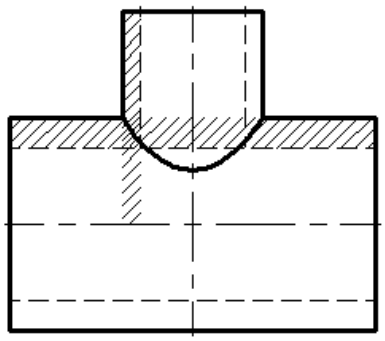
Cutting Body by 3D Section

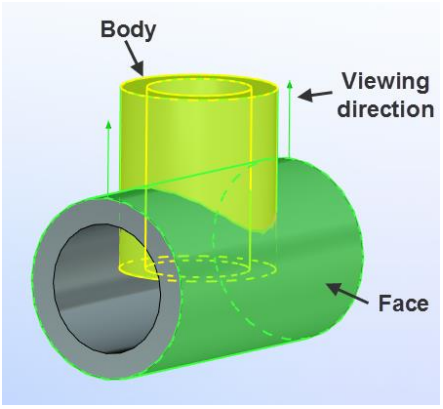


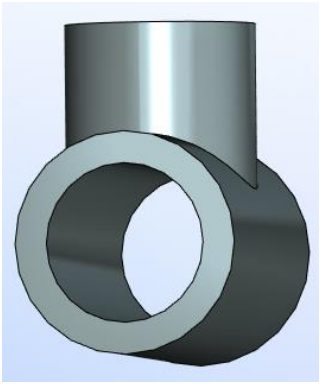
This drawing is provided with the library *"Examples"*, the folder *"\3D Modeling\Cut operation\ Cut by section.GRB"*.

Cutting Body by Face

Defining the body cutting operation by a cutting face provides interesting opportunities. Let's review the following example:





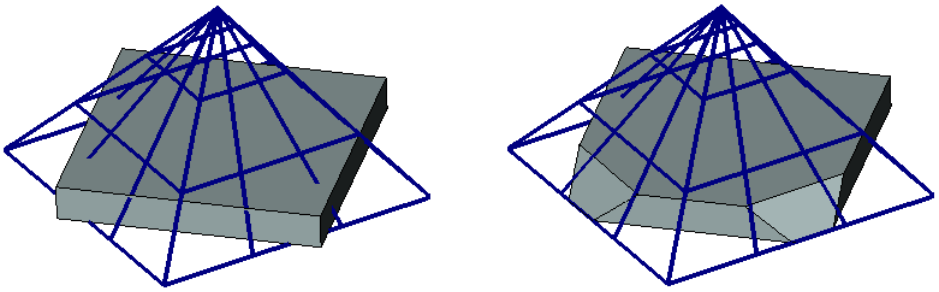


This drawing is provided with the library *"Examples"*, the folder *""\3D Modeling\Cut operation\ Cut by surface.GRB"*.

The drawing shown on the diagram represents an intersection of two pipes. The 3D bodies were created by two Rotation operations. Next, the required cut was made along a face belonging to one of the bodies.

Cutting Body by Set of Cutting Elements

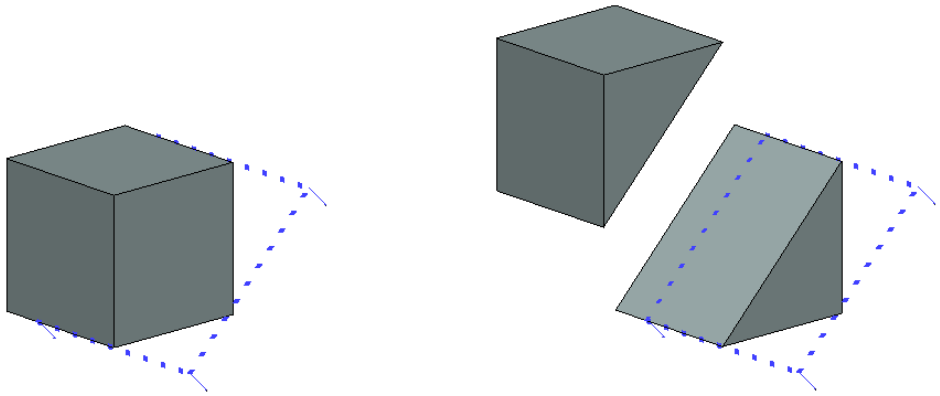
The drawing shows an example of cutting the body of a plate by two front faces of a pyramid.



This drawing is provided with the library *"Examples"*, the folder *""\3D Modeling\Cut operation\ Cut by set of faces.GRB"*.

Separation of Body in Two Parts

The diagram below shows the original body along with two resulting bodies after the Separation operation (for clarity, the resulting bodies were moved apart in the space using the command **Tools** > **Transformation**).



This 3D model is located in the folder *"\3D Modeling\Cut operation\ Cut with separation into two solids.GRB"* **.**

DIVISION INTO BODIES

This command is intended for dividing operations into separate bodies, that unite several bodies (solid and sheet bodies alike). Thus, for example, the result of creating a linear or circular array or a Boolean operation will be divided by this operation into geometrically separate components. The command can also be used when handling a model imported from another system, that consists of several bodies.

DEFINING DIVISION

To divide a body, use the command **3SD: Divide Solid**:

Icon	Ribbon
	3D Model → Modify → Divide
Keyboard	Textual Menu
<3SD>	Operation > Model Composition > Divide

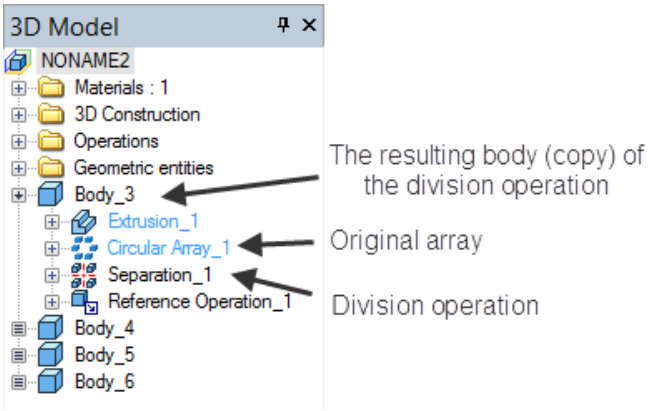
There is yet another way of calling the command: move the cursor to one of the model operations and right click, then in the coming up menu select the item “Create”. The list of available commands will appear, that can be called while that operation is selected. Select the item “Divide Solid”, at this moment the selected body will be highlighted. Selection can be redone.

The operation is created by selecting an object uniting several bodies. The object will be divided into these bodies. You need to confirm the division.

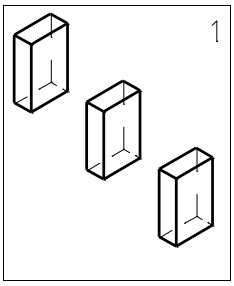
The object selection is done by the option:

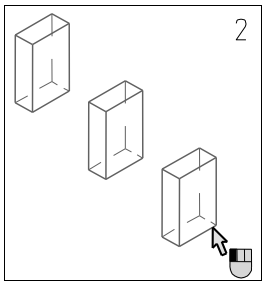
	<F>	Select Solid

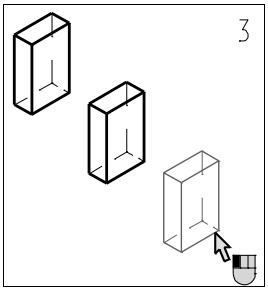
This option is turned on automatically upon calling the command. The examples of an object to be divided are: the result of the array or circular array operation, a Boolean operation, an import operation from other systems. As a result, the selected object will be divided into several separate bodies. The division is done by creating copies of all bodies making up the original operation. The copies are actually displayed in the 3D window. The respective entries in the 3D model tree appear as follows:



The diagram 1 below shows a model made up by a linear array. As we know, all bodies in the array are actually a single body. When pointing to one of the array bodies, the whole array is highlighted, as can be seen on the diagram 2. The diagram 3 shows the result of the division. In this case, after pointing to one of the bodies, only this body is highlighted.



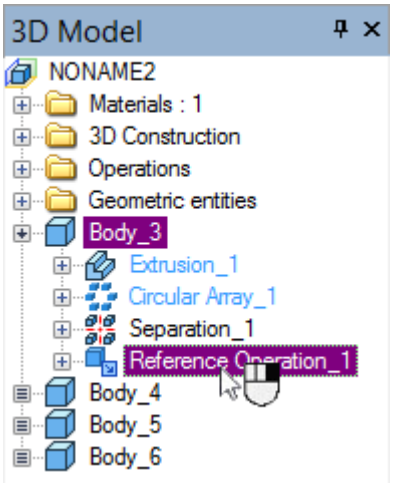




Division Parameters

No parameters are explicitly defined for the operation. However, the resulting bodies (copies) of the operation can be assigned their own parameters by calling the respective dialog box. In this case, the dialog box can be called upon completing the division operation from the context menu by right clicking over the desired body (copy) in the 3D window or in the 3D model tree.

The produced dialog box is typical across various system operations. In it, you can define the general system parameters and specific parameters, as well as the transformation parameters.



Operation Name. The following names are automatically created for the bodies resulting from the "Divide Solid" operation: Reference Operation_1, Reference Operation_2, etc.

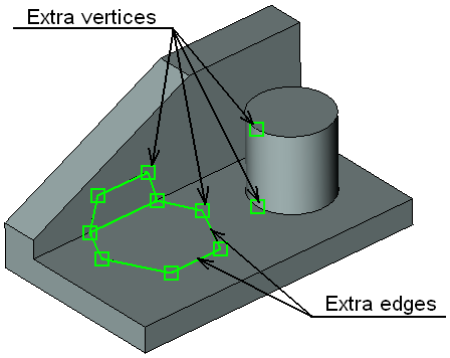
MODEL SIMPLIFICATION

The command «Simplify Model» serves for deleting extra edges or vertices from the specified operation body, as well as for simplifying the geometrical base of faces and edges. The extra entities can occur in the process of modeling, while performing certain operations. Presence of such elements in a model is not an error; however, deleting those can be helpful in certain situations. This operation is specifically intended for tackling problems in complicated cases, when extra edges or vertices obstruct the target design (for example, in blend creation); additionally, this functionality can be used for simplifying the contents of imported models.

GENERAL INFORMATION

Deleting Extra Entities

When model is being simplified, only vertices or edges can be deleted. An extra vertex can usually be excluded from the model when it belongs to a single closed edge (usually this would be an edge of a surface of revolution) or belongs to two adjacent edges lying on one curve. An edge is usually deleted if it is a part of an isolated open chain of edges within one face, or if it is an edge between two adjacent faces lying on one surface.



The operation allows selecting for processing vertices, edges, faces and operations. The admissible entity types are processed as follows:

- a vertex will be deleted if it is identified as extra;
- if an edge is identified as extra, it will be deleted. If vertices of this edge are also identified as extra, then those will be deleted along.
- upon selecting a face, any extra edges on it must be deleted. The extra vertices are then deleted as well.
- if an operation is selected, all extra edges and vertices are subsequently deleted up until no such are left. If the operation's body contains spline faces or curves lying on one surface or one curve, then those are merged to one face or one edge.

If the user selected an entity, which the system cannot exclude it from the model, it will be ignored.

Simplifying Topology

Along with deleting extra topology, certain topological elements can be simplified. If spline edges or faces lie on simple curves or analytical surfaces (plane, cylinder, torus or sphere), then the geometrical base of those elements is substituted by simpler one. This functionality is identical to the option «simplify geometry» that is available in certain operations (for example, «Sweep» or «Loft»).

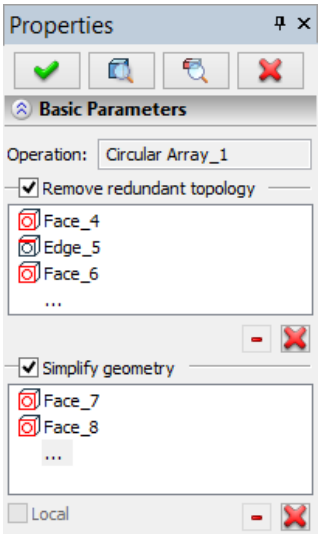
As the input data for such processing, the user can select separate faces or a whole operation (as for processing together).

Working with the Command

The command is called by the following means:

Icon	Ribbon
	3D Model → Modify → Simpify
Keyboard	**Textual Menu**
<3SI>	Operation > Model Composition > Simpify

This operation allows turning on each type of processing separately – either the model simplification or deletion of extra elements – with the help of the appropriate flags in the properties window. A specific list of entities to be processed can be composed for each type of processing. If a processing is disabled then this type of processing is not performed and the entities list is inaccessible.



For processing together, the user can select a whole operation (recommended). To select an operation, you can use the automenu option:

	<O>	Select Operation

The name of the operation appears in the «Operation:» field. The entities list for all types of processing will remain empty in this case. If a list for processing is empty, that means the system will investigate all topological elements of a given operation when performing the processing.

If the list of «Simplify geometry» processing is empty, the option «Separately» is accessible in the properties window. Enabling this option activates a special mode that allows performing subsequent simplification of a set of faces lying on one geometrical surface. Otherwise, simultaneous processing of all faces is performed. For example, consider a set of faces lying on a geometrical spline surface that is close to planar within a certain area. During the separate processing, the system would be able to

simplify the faces that happen to be within the «planar» area. In the simultaneous processing, the system would not be able to simplify any face.

In certain cases, it might be possible for the user to do local changes to the model topology. The list of entities for various processings are composed by the automenu options:

	<R>	Select element to delete redundant
	<G>	Select face to simplify geometry

An entity can also be selected by placing the pointer in the list pane in the properties window. The respective automenu option will turn on automatically.

To cancel all elements selection use command:

	<F>	Cancel selection

The command also provides a useful feature for watching what model entities among the currently selected are extra or simplifyable. The following automenu option is used for enabling such view:

	<V>	Show elements to be removed or simplified

While in this mode, the system will highlight all edges, their cases and faces of the selected operation that can currently be subjected to processing. Deletable and simplifyable entities are highlighted in different colors.

> The described mode is mostly useful for analyzing geometry of an operation as a whole. After analyzing topology of the whole operation, the user would be able to use capabilities of local processing with more confidence, if necessary at all.

FACE TAPER

This operation allows the user to taper one or several selected faces at a specified angle, with automatic correction of the adjacent faces.

Face taper can be used, for example, for designing parts manufactured by casting method under pressure. Addition of tapers to the part enables easy extraction of the part from the mold.

GENERAL CONCEPTS AND OPERATION CAPABILITIES

Taper Direction

The direction, in which the taper (slant) angle of the faces is counted, is determined by a vector. This vector can be defined in two ways.

The first way requires selecting an object, whose geometry will determine the taper direction: a normal to a face, a flat edge, etc.

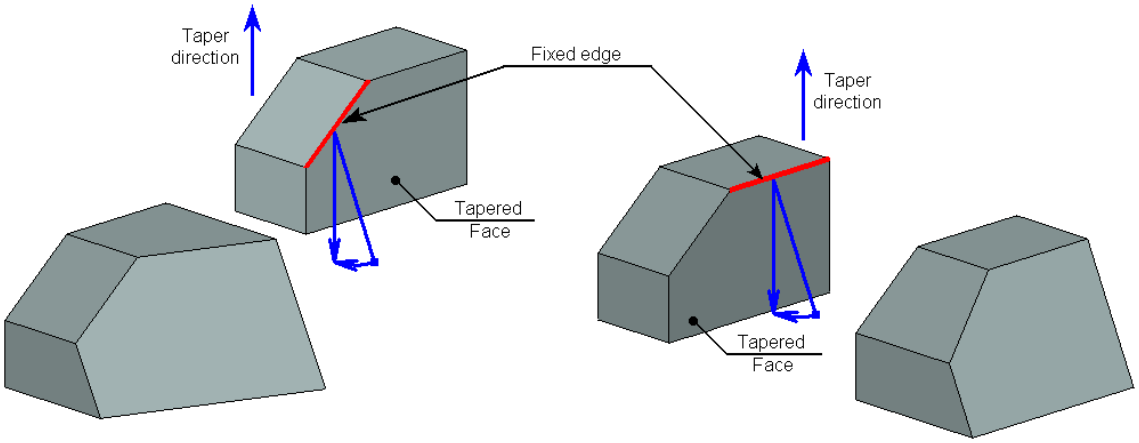
Alternatively, the vector can be defined by two 3D points. The second 3D point defines the vector direction.

When designing parts made by parts made by molding, it is necessary to specify the direction of the taper along the direction of the mold separation.

Fixed Edge/Fixed Taper Face

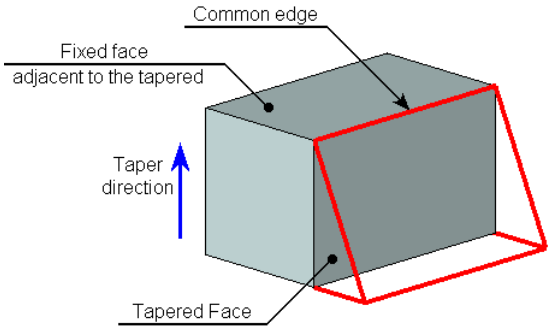
For each tapered face, it is required to specify a fixed edge or a face.

A fixed edge defines a fixed taper line – a line, lying on a tapered face, which does not change its position when creating a taper. Construction of a taper surface takes place with respect to a fixed line. A fixed edge must belong to the same body as a tapered face.

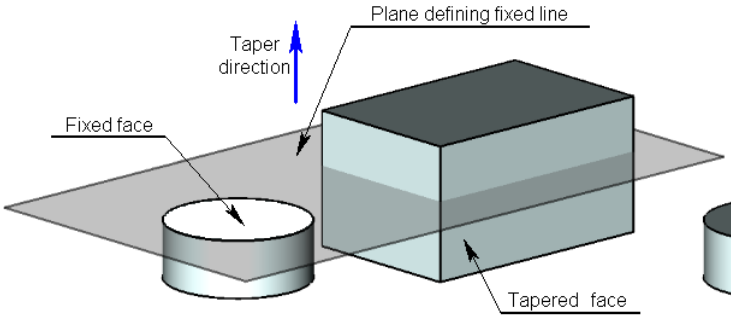


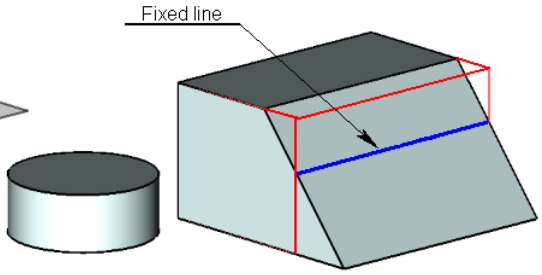
Instead of an edge, it is possible to indicate a fixed face. Intersection of the selected face with the tapered face defines a fixed taper line.

Unlike the fixed edge, the fixed face can be selected on a different body. Once the fixed face is selected, the command creates the plane based on that face, and projects it to the intersection with the face to taper. The intersection line between this line and the face being tapered is the fixed line about which the face tapering will occur.



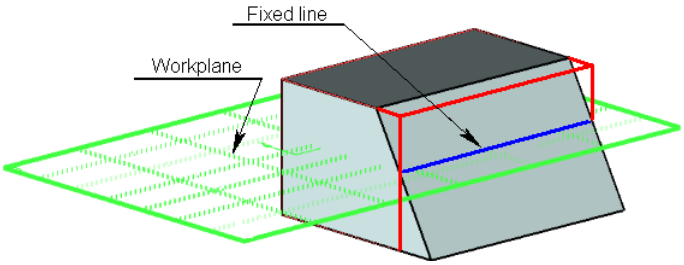
A fixed face can be adjacent to the tapered face. In this case the common edge will be a fixed edge with respect to which the face will be tapered.





Fixed edges and fixed faces are defined separately for each tapered face.

Instead of collection of fixed edges, it is possible to specify one workplane. It will define a fixed line for all tapered faces of the current operation at once. A fixed line of each tapered face will be defined as an intersection of selected workplane with the given face.



Calculating Taper Angle

Taper angle of face can be evaluated according to the following algorithm:

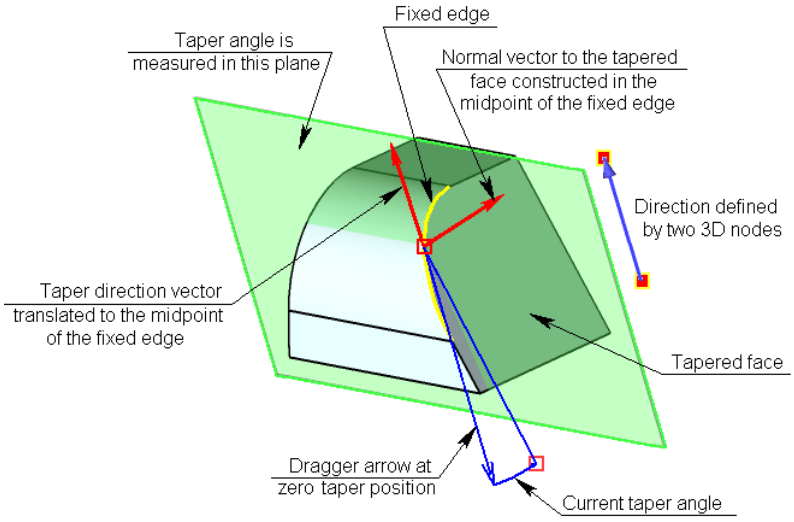
A normal to the face to taper is constructed at each point of the fixed edge.

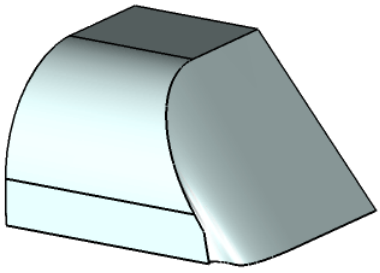
The taper direction vector is translated to that point.

Based on those two vectors, a plane is constructed, in which the taper angle will be counted.

Next, the vector is determined, that sets the zero taper angle.

The taper angle is counted from the formerly described vector in the plane. This angle defines the resulting vector, through which the surface of the tapered face will be passing.



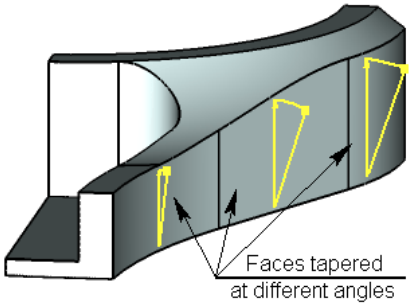


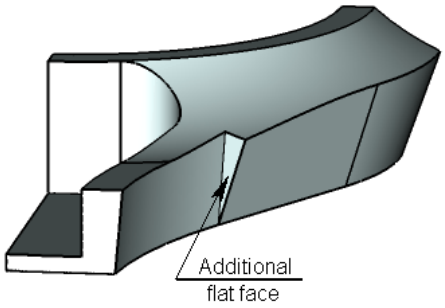
Resulting face taper

Processing of Junctures between Tapered and Adjacent to Them Faces

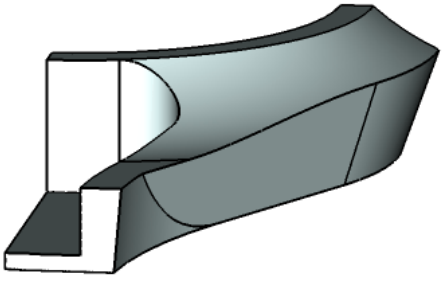
When creating a taper, there is a possibility of creating a gap between the tapered and adjacent to them non-tapered faces. To process these gaps, it is possible to use two methods:

- Creating auxiliary flat faces;
- Extending the surface of the non-tapered face.





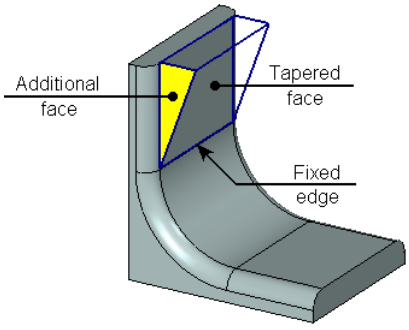
Taper with creation of additional face



Taper with preservation of connection between faces

The first way implies creation of an additional flat face for patching the gap at the connection. The second way allows preserving transition between those faces. In this way, a portion of the tapered face will be trimmed by the surface created by extending the adjacent face.

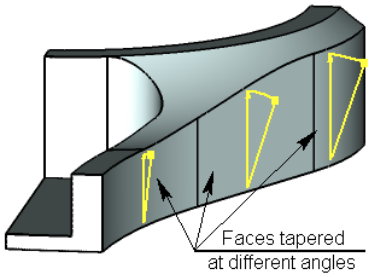
In some cases, creation of the additional face is the only possible way of closing the gap at the faces connection. This exact case is shown on the right-hand diagram.

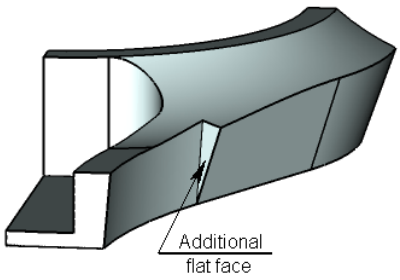


Processing of Junctures between Tapered Faces

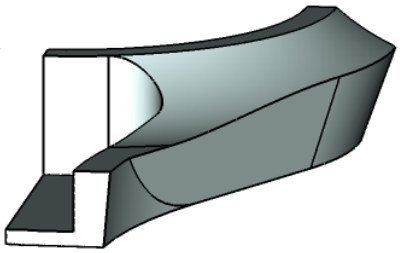
When the adjacent faces are tapered at different angles, a gap can be created between them. For correcting this situation, it is possible to use the following methods:

- Creating additional flat faces;
- Extending the surface of non-tapered face;
- Preserving smooth connection.





Taper with creation of additional face



Taper with preservation
of smooth connection between faces

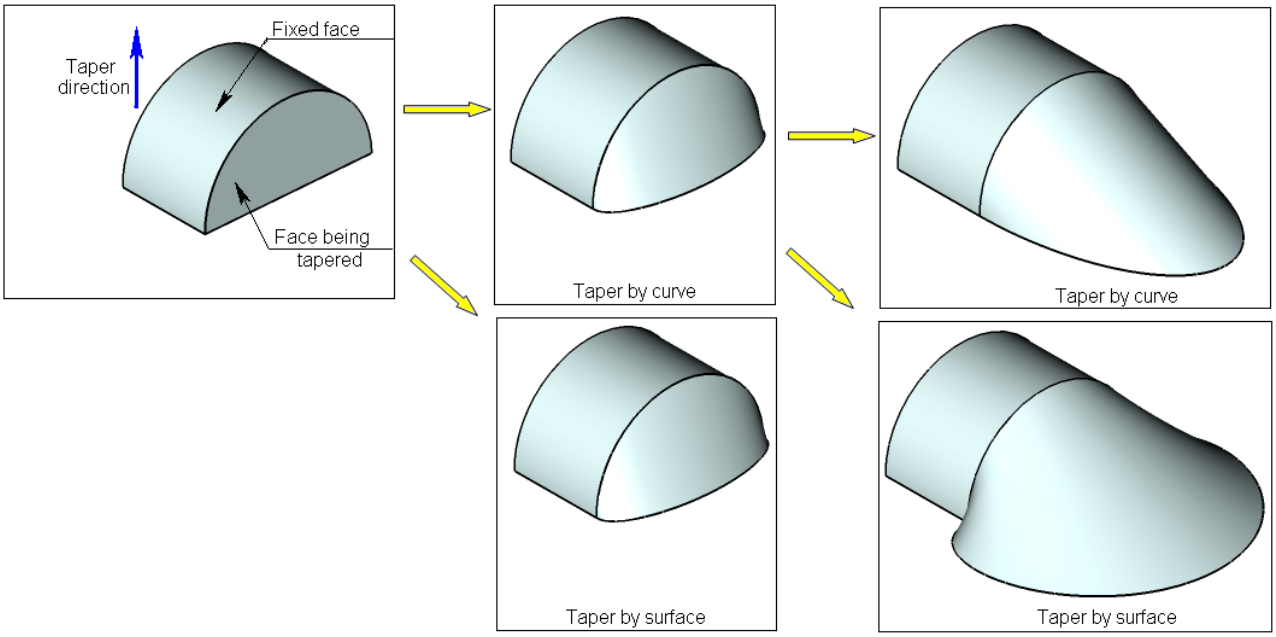
Methods of Creating Taper

The operation provides various methods of shaping up the tapered face:

- ✓ *Standard;*
- ✓ *By offset;*
- ✓ *By curve;*
- ✓ *By surface (see section below "Taper by Offset ").*
- ✓ By default, a standard method for creating tapers is used. It allows you to construct the tapered surface exactly matching the shape of the original surface and a specified taper angle.

In several cases, the standard method does not allow you to create a required taper. In these cases it is possible to use methods "By curve" and "By surface". These types of taper match the shape of the original tapered face in a less stringent manner which permits you to construct the taper in cases when the standard method of taper creation does not work. Methods "By curve" and "By surface" differ by the method of resolving ambiguity upon creating the tapered surface. In the first case, for constructing a taper, a tangent to the curve of the fixed edge is utilized, in the second case – the normal to the tapered surface. The tapers "By curve" and "By surface" are usually used for giving a part more aesthetic shape.

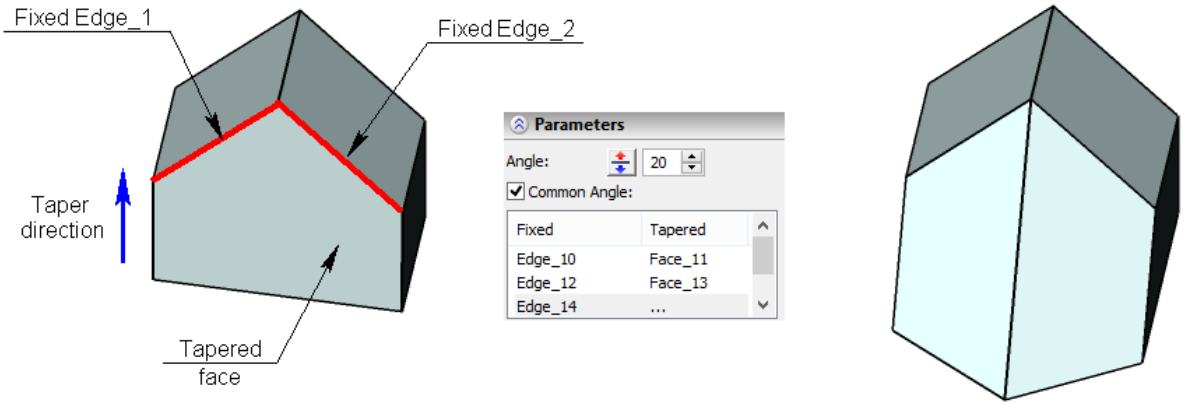
In many cases the methods "By curve" and "By surface" give similar outcome. For example, in the example on the figure below, the difference between these methods is noticeable only upon repeated use of the taper operation.



Additional Capabilities of Taper

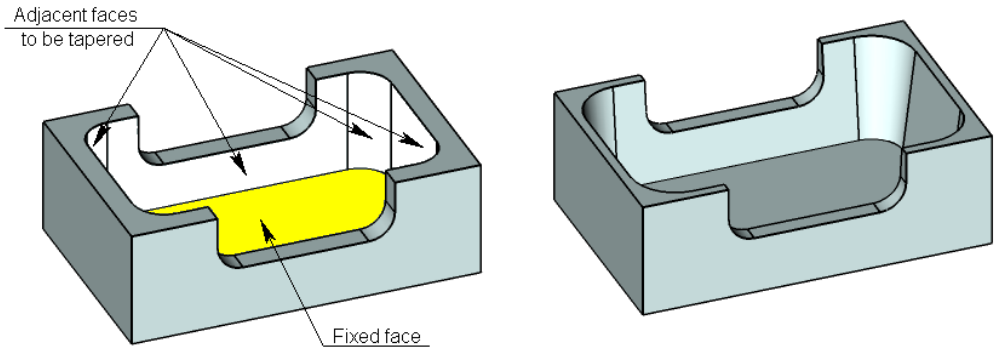
Taper of Face Relative to Multiple Fixed Faces/Edges

The operation allows tapering the selected face relative to multiple fixed faces or edges. As a result of such operation, the tapered face will be split into two parts. The edges of the resulting faces will have the specified taper angle.



Tapering All Adjacent Faces

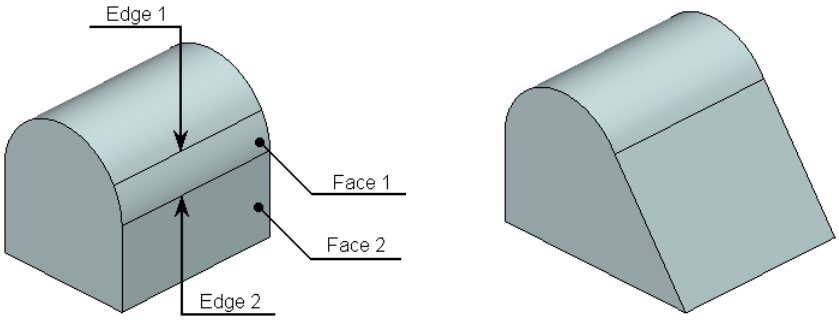
The mode of tapering all faces adjacent to the fixed one is provided for quick taper creation. In this case, all you need is just to select the fixed face. The command will automatically determine all adjacent faces for tapering.



Combined Processing of Adjacent Faces

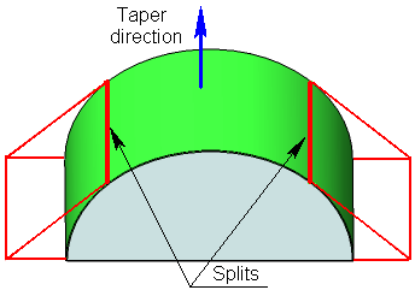
The mode of combined processing of adjacent faces allows the user to create a simultaneous taper for a pair of adjacent faces, when one of the tapered faces serves as a fixed face for another. In a standard mode, an attempt to create such a taper will lead to an error.

On the figure below an example is shown in which it is desired to simultaneously taper two faces: **Face 1** and **Face 2**. For the face **Face 1**, the edge **Edge 1** is selected as a fixed edge. For the **Face 2** a fixed line has to be defined by the face **Face 1** or the edge **Edge 2**. But both **Face 1** and **Edge 2** will be modified upon taper creation. In this situation, the mode of combined processing of faces can help. By turning it on and indicating **Face 1** as a fixed object for the face **Face 2**, the user can obtain a correct taper of two faces at the same time.



Splitting Faces

This parameter is used in cases when it is necessary to create a taper of the curvilinear face. The command splits the face into patches defining the position for which it is possible to create a taper surface. The tapered patches will pass tangentially to the remaining curvilinear part of the face at a specified angle to the taper direction.



COMPLEX FORMS OF FACE TAPER

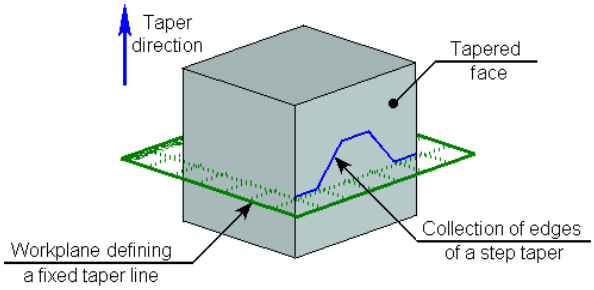
Step tapering

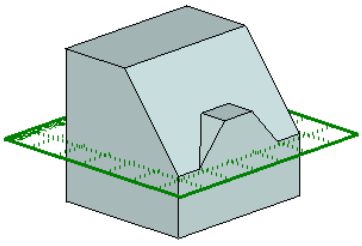
For creating step tapering, the user specifies:

- ✓ Taper direction;
- ✓ Fixed face or workplane;
- ✓ Tapered face;
- ✓ A continuous sequence of edges, lying on the tapered face, whose geometry must be preserved upon taper creation. All edges of the specified sequence must be non parallel to the taper direction.

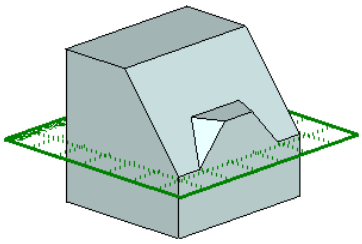
For adding the edge to the tapered face, the command **3ZS: Imprint Elements** can be used.

As a result, the tapered face "turns around" relative to the fixed face/workplane. Along the line generated by a specified sequence of edges, a "step" is formed. Additional faces forming the "step", can be created by two methods: perpendicular to the original face or at a specified taper angle.



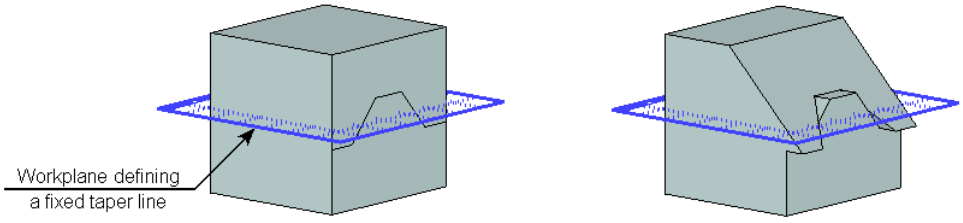


Step taper with a "step" formed perpendicular to original face



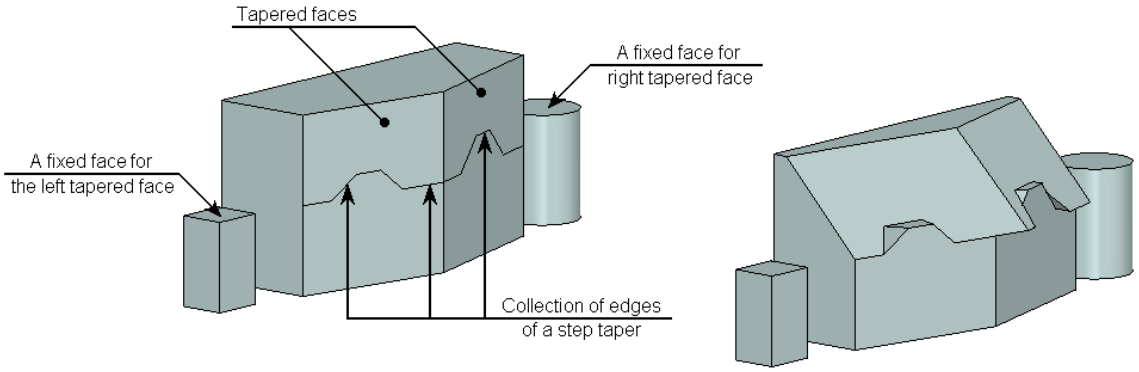
Step taper with a "step" formed according to a specified taper angle

When creating a step tapering, it is possible to create a two-sided "step".



Step taper with two-sided "step"

A step tapering can be created simultaneously on several faces of the body.

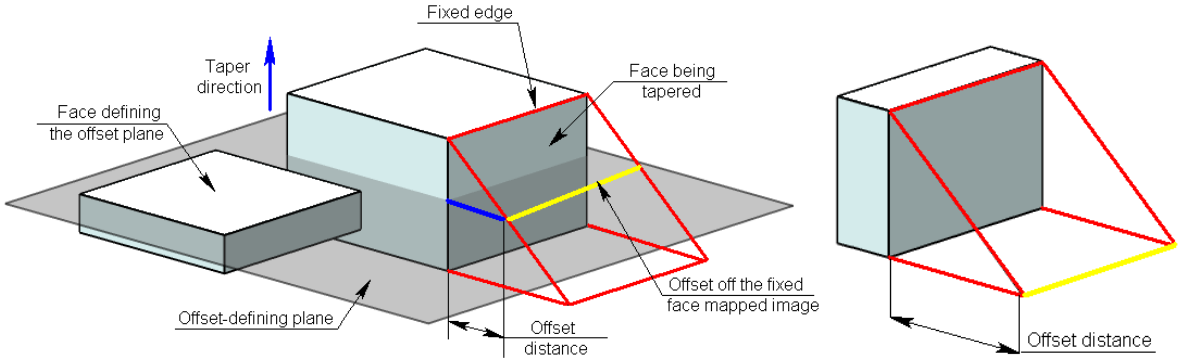


Taper by offset

The method of specifying the taper "by offset" allows the user to determine the magnitude of a taper not by an angle but by an offset of the tapered face with respect to the initial position.

The algorithm of creating the taper "by offset" method is as follows:

- A fixed face or edge is mapped along the taper direction on the plane selected for defining the face offset.
- A specified-distance offset from the mapped face or edge is constructed. In this case, the taper parameter is the offset distance.
- The resulting surface will be passing through the offset off the fixed edge's mapped image.



CREATING FACE TAPER

The command for creating "Face Taper" operation can be called by one of the following means:

Icon	Ribbon
	3D Model → Modify → Face Taper
Keyboard	Textual Menu
<3TA>	Operation > Taper > Face Taper

Creating Face Taper

After invoking the command for creating a taper (except a step taper and a taper by an offset) it is required:

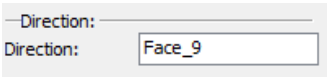
- ✓ Indicate taper direction;
- ✓ Specify a required number of pairs fixed line/tapered face;
- ✓ Specify taper angle (or a set of taper angles when tapering several faces);
- ✓ Select method of tapering (if necessary);
- ✓ If required, specify additional parameters (the way of filling the juncture between the tapered and non-tapered faces; between the faces tapered at different angles);
- ✓ Complete operation creation with the help of .

Specifying taper direction

For specifying a taper direction by one 3D object, the following option is used:

	<D>	Set tapering direction

This option allows selecting objects in the scene, whose geometry is suitable for defining the taper direction vector. Once an object is selected, its name appears in the properties window.



For defining a taper direction by two 3D points, the following options are used:

	<1>	Select first point of tapering direction
	<2>	Select second point of tapering direction

The selected elements are also entered in the properties window. The taper direction is marked by a vector in the 3D scene.

To cancel the direction selection, use the option:

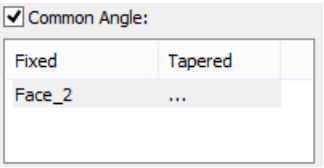
	<X>	Cancel direction selection

Specifying Fixed Object and Tapered Face

After defining a taper direction, the command will prompt you to define a fixed line of the first tapered face. The following automenu options are used for that:

	<T>	Select fixed Face
	<E>	Select fixed Edge
	<W>	Select Workplane

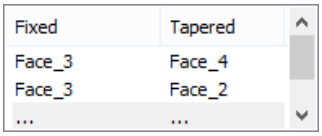
After selecting the respective element, it will be added to the list "Fixed/Tapered" of the command's properties window.



Then the system will prompt you to select a tapered face. The following option will become active in the automenu:

	<F>	Set faces to be tapered

Selected tapered face is also added to the list "Fixed/Tapered" next to the corresponding to it fixed object.



> A fixed line (edge, face, workplane) and a tapered face are specified in pairs. Each tapered face must correspond to a fixed object, and vice versa.

Further action of the command depends on the type of the selected fixed object. If it was a fixed edge, then the system will prompt you to select the next fixed edge. If it was a face or a workplane, the system will wait for specification of one more tapered face. Selected earlier face/plane will be, by default, considered as a fixed object for all subsequently selected tapered faces. To select another fixed object, it is necessary to manually activate the corresponding option (or), select a desired object (face or edge), and then specify a tapered face.

> Note that in the framework of a single operation, it is possible to specify only one workplane – it will become a fixed object for all tapered faces.

For creating the taper of all faces adjacent to a single fixed edge, it is sufficient, after selection of the fixed edge, turn on the following automenu option:

	<A>	Select all adjacent faces for tapering

All faces adjacent to a specified fixed face will be automatically selected as tapered.

To modify already selected fixed objects or tapered faces, it is necessary to select a desired pair in the list **Fixed /Tapered**, turn on the option of selecting a desired object (, or) and select an object again.

It is possible to delete a specified pair fixed object/tapered face by selecting them in a list, pressing the button **[Delete]** or key <Del>.

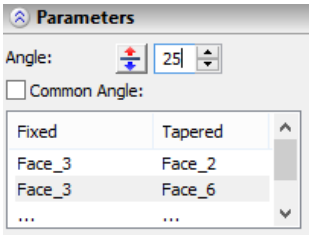
It is also possible to cancel selection of the workplane with the help of the option:

	<X>	Cansel workplane selection

Defining taper angle

The taper angle is defined by a numerical value in the operation parameters. It can be common across the whole set of pairs of fixed objects and tapered faces, or be defined individually for each pair. By default, the common angle is used.

If for a certain pair fixed object/tapered face, it is desired to define a value of the angle different from the common, it is necessary to perform the following steps:



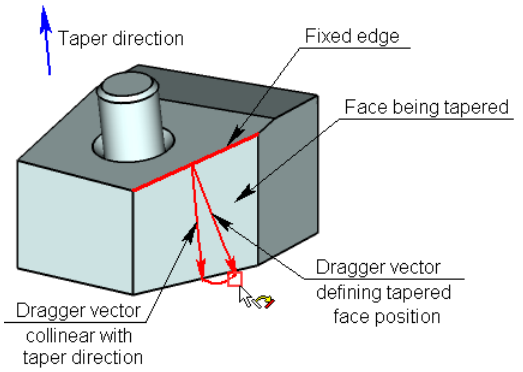
In the properties window, click the desired pair of the fixed and tapered objects.

Clear the check mark on the option **Common angle**.

Specify the angle value for the selected pair.

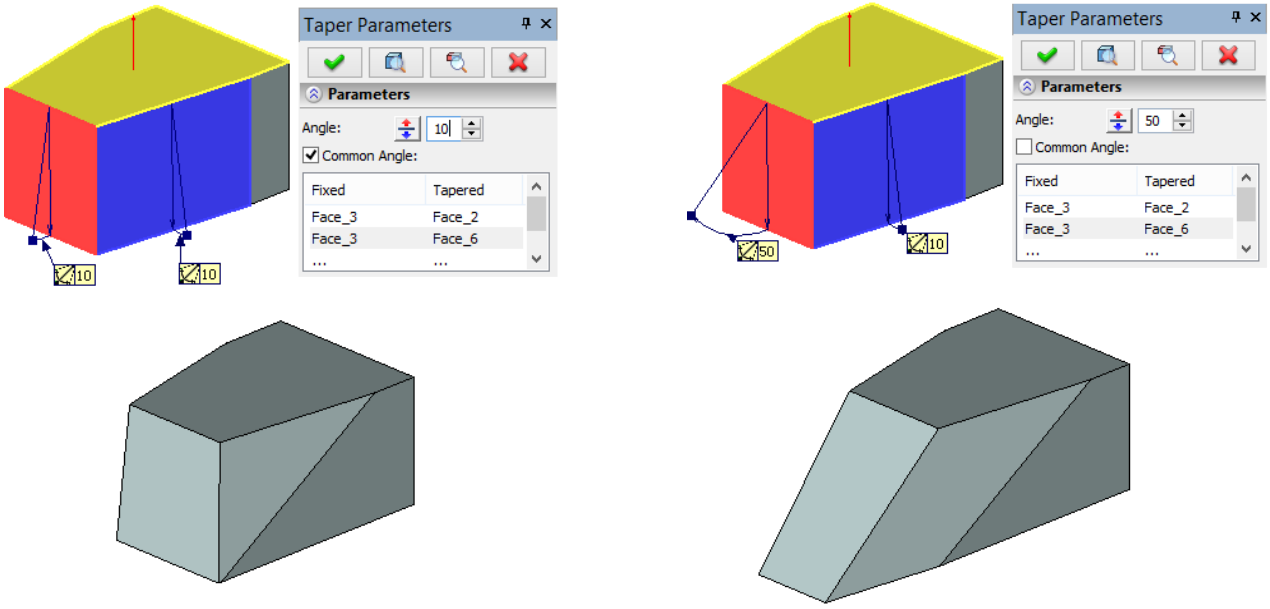
The command provides draggers for easy setting of the face taper angles. By using the draggers, you can dynamically modify the taper angle value.

> In certain cases the draggers may not be displayed.



A dragger for defining face taper angles appears as two vectors connected by an arc. Both vectors originate from the fixed object (a face or an edge). The direction of the first vector coincides with the taper direction. The direction of the second vector defines the resulting position of the tapered face. The taper angle is set by dragging the tip of the second vector.

A separate dragger is provided for each pair fixed object/tapered face in the 3D scene. If a common angle is used for all pairs fixed object/tapered face, then you can modify the taper angle by any of the draggers. If the “Common angle” option is turned off for any of the pairs, then the dragger position will affect the angle of this pair only.



To change the face taper direction, either enter a negative angle value or push the button in the face taper parameters dialog.

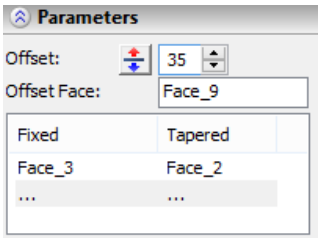
Creating Taper by Offset

For creating *face taper by offset* it is required:

- ✓ Specify a taper direction;
- ✓ Indicate a fixed taper line;
- ✓ Indicate a tapered face or a collection of faces;
- ✓ Select a face in the plane of which the offset will be created;
- ✓ Specify the offset of the tapered face;
- ✓ If necessary, define additional parameters (the method of filling the junctures between the tapered and non-tapered faces, between faces tapered at different angles, etc.);
- ✓ Complete operation creation with the help of .

Taper direction, fixed taper line and tapered face are specified in the same way as for the standard taper. For specifying the face in the plane of which the offset will be created, it is necessary to turn on the following automenu option:

	<O>	Select Face for creating taper by offset



The name of the selected face will be displayed in the command's properties window (the field **Offset face**). The offset is specified as a numeric value of the parameter **Offset**.

When specifying the face for defining the offset, the parameter **Method** is automatically assigned to the value **Taper by offset**.

Creating Step Tapering

For creating *step tapering of faces* it is necessary:

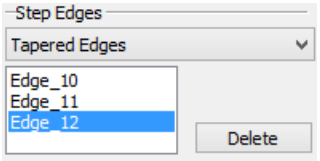
- ✓ Specify taper direction;
- ✓ Specify a fixed taper line;
- ✓ Specify a tapered face or a collection of faces;
- ✓ Specify a sequence of edges on the tapered face;
- ✓ Specify taper angle (or a set of taper angles in case of selecting several tapered faces);
- ✓ Specify the method of creating additional faces (by normal or with a taper);
- ✓ Complete creation of operation with the help of .

Taper direction, fixed taper line and a tapered face are specified in the same way as for the standard taper.

For specifying the edges of a step taper, it is necessary to turn on the option:

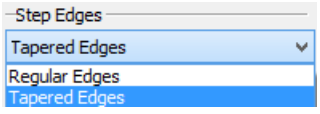
	<G>	Step tapering

When activating this option, the command switches to the mode of selection of edges, with respect to which the step taper will be formed.



Selected edges of the step taper will be added to the corresponding list of the command's properties window. The user can remove selected edge by selecting it in the list and pressing the button **[Delete]**.

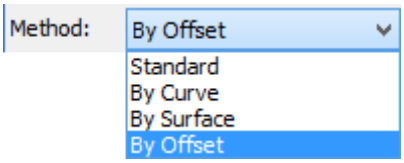
The method of creating additional edges forming the "step" is selected with the help of the list **Edges of step taper**. If the value **Normal** is chosen, the edges will be formed perpendicular to the original face. For the value **Tapered** the faces will be formed at a specified taper angle.



Advanced Operation Parameters

Taper creation method

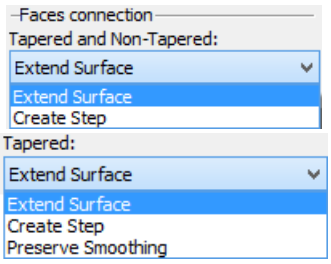
The method of forming the taper is selected from the drop down list of the parameter "Method".



The taper creation method "By offset" can be set automatically when turning on the option of selecting the offset face (see section "Creating taper by offset").

Processing connection between faces

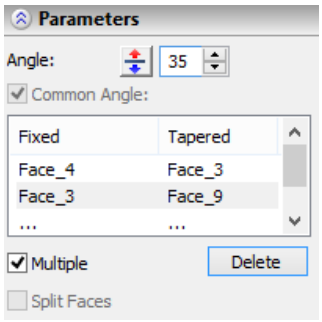
You can select a method for connecting the tapered face and its adjacent faces, using the parameter **Faces Connection/Tapered and Non-tapered** in the properties window.



Similarly, you can select a method for connecting the tapered faces.

Processing faces together

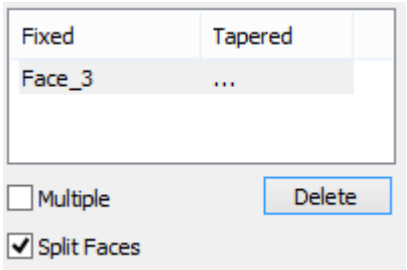
For combined processing of two adjacent faces, it is required to select one of them as a fixed face for another, and turn on the flag **Multiple** in the dialog of the command's properties.



> In the same way, it is possible to create a taper for three and more faces.

Splitting face

For creating the taper with face splitting, the user has to specify a face which is to be split as a fixed face (a tapered face in this case is not specified) and activate the flag **Split Faces**.



> Note that the functionality of splitting faces remains in the command for compatibility with earlier versions of T-FLEX CAD. The currently available **3ZS: Imprint Elements** command provides various ways of dividing a curvilinear face into pieces for further tapering.

BODY TAPER

Body taper operation allows the user to create two-sided tapers of faces of a body relative to a specified taper direction and a parting body. A parting body divides the original body into two parts: "top" and "bottom", which are tapered with respect to each other. A sheet body is usually specified as a parting body, but solid bodies can be used as well.

For "top" and "bottom" tapers, the operation enables specifying different taper angles. If necessary, it is possible to specify only "top" or "bottom" angle, that is, create a one-sided taper of a body.

The use of body taper operation allows the user to significantly simplify the process of designing cast parts and their molds. The parting body corresponds to the surface of separation, and the taper direction - to the direction of the mold separation.

GENERAL CONCEPTS AND OPERATION CAPABILITIES

General Concepts

For creating a body taper, first of all it is necessary to indicate a tapered body itself, a parting body, the sets of "top" and "bottom" fixed edges of the tapered body, taper direction and angle.

Taper Direction

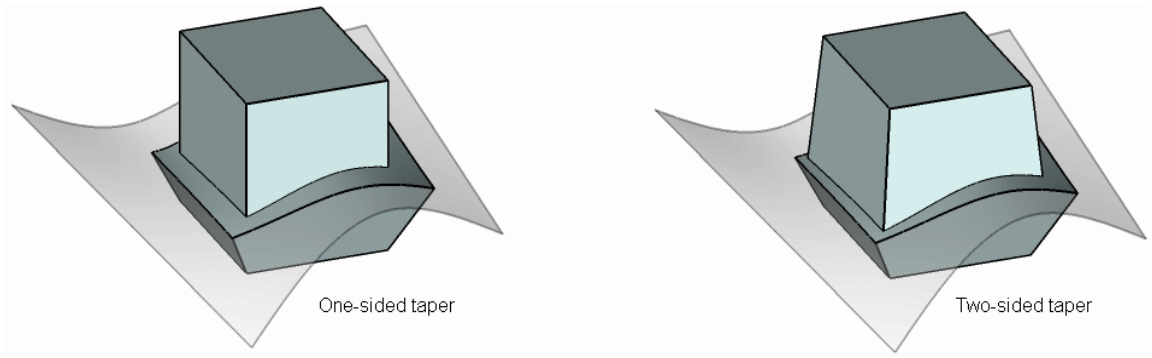
Taper direction is an oriented in space vector, relative to which the taper angles of the faces will be measured.

Taper direction vector can be defined by two methods:

- by one 3D object whose geometry can define a vector;
- by two 3D points defining the start and end points of a vector.

Parting Body

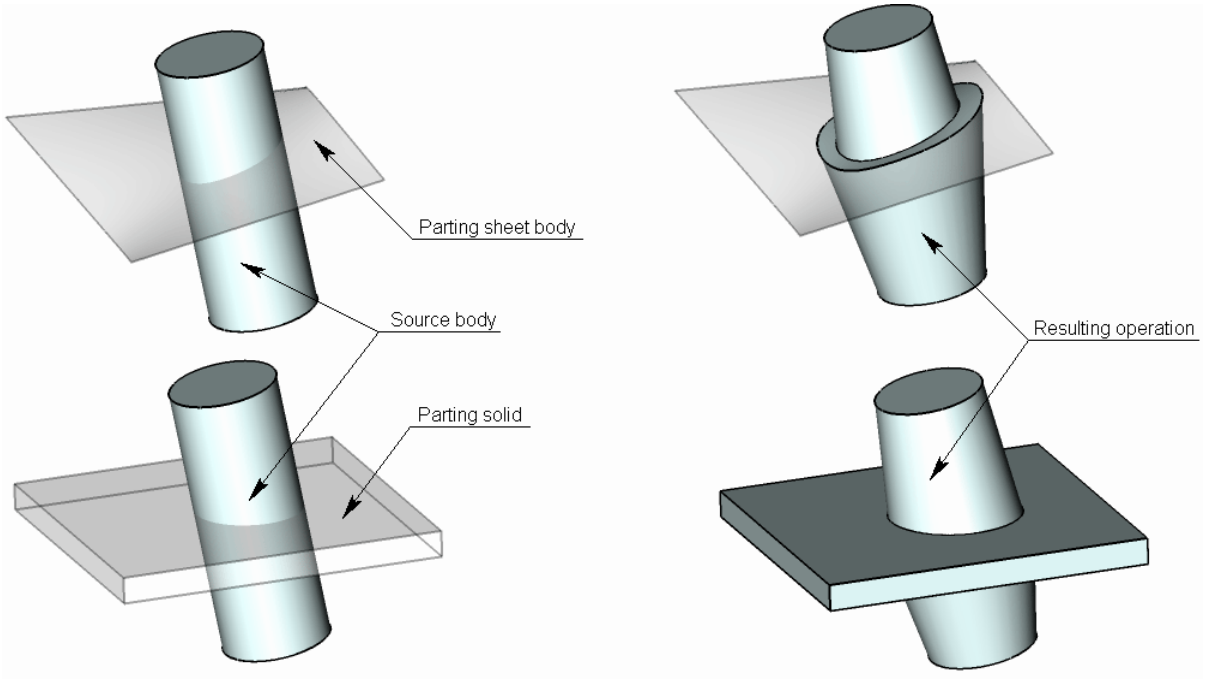
A parting body is used for dividing the original body into two parts, each of which can be tapered individually. When tapering faces at both sides from the parting body, a *two-sided taper* is created. When tapering faces located at one side from the parting body, a *one-sided taper* is created.



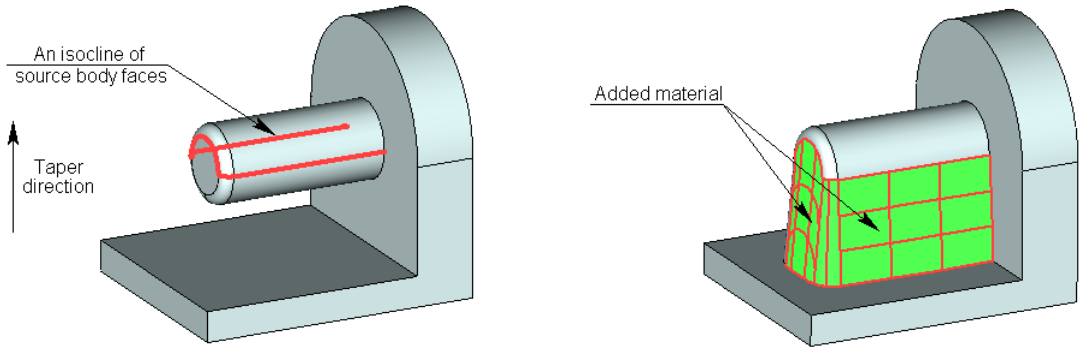
Both sheet and solid bodies can be used as a parting body. In most cases, a sheet body is used as a parting body. After executing the operation, a parting body is remained in the 3D scene.

The parting body may not have through holes. Additionally, the parting body must be large enough to fully intersect the target body in the taper direction.

If a solid was selected as the parting body, then use of the “Body Taper” operation will result in one body obtained by merging the tapered and the parting body.



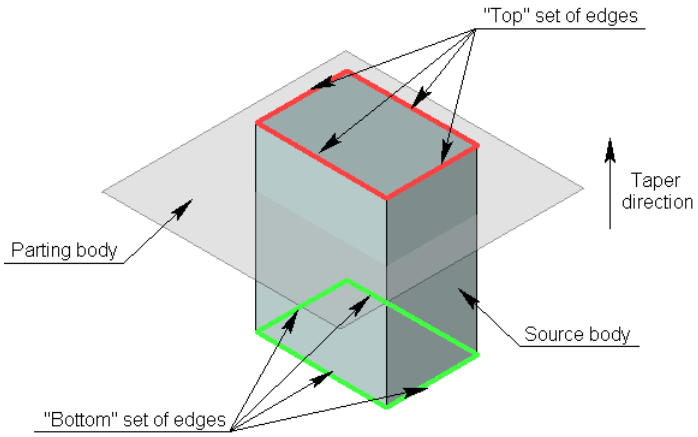
The source body itself can be used as the parting body. On the figure below, an isoclinal line of the faces of the original body was selected as the source edges for creating the taper. The original body itself serves as the parting body.



Fixed Edges/Faces

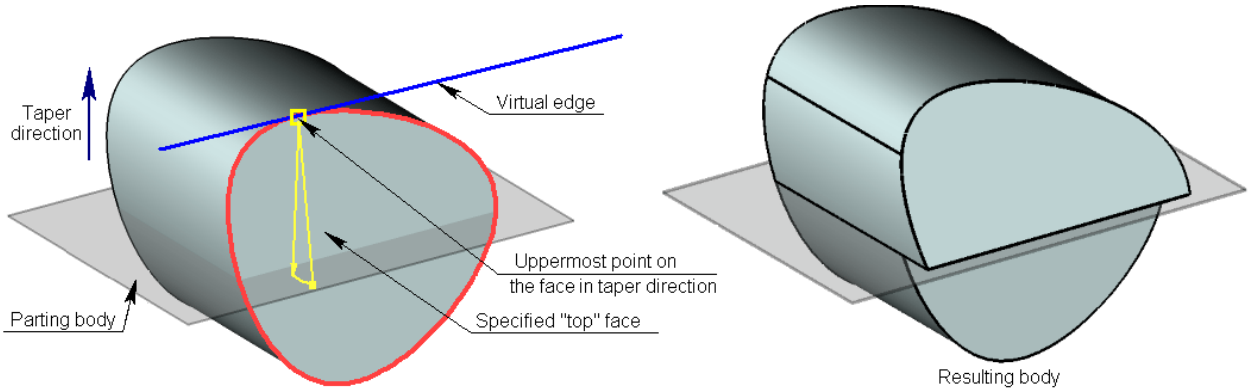
For creating two-sided tapers of a body, it is required to specify two sets of edges: “top” edges and “bottom” edges.

The so-called “top” and “bottom” sets of base edges are respectively those located above and below the "parting body", with respect to the taper direction. The edges define the faces for which a taper needs to be created, and also define the fixed lines of these faces.



It is possible to select only one set of fixed edges (“top” or “bottom”). In this case, a one-sided taper will be created.

Instead of source edges, the user can specify the source faces themselves. For example, when creating a body taper, it might be impossible to select base edges of the “top” and “bottom” sets. Those edges might either not exist, or have the geometry that would not allow creating a body taper. In this case, for creating a taper the user indicates the sets of source faces based on which the virtual edges for constructing the body taper are defined. If, for example, a face was selected as “top”, then the command will automatically find the uppermost point on this face with respect to the taper direction. A virtual edge is defined at this point, passing tangentially to the edge of the selected face and perpendicular to the taper direction vector.



Taper Angle

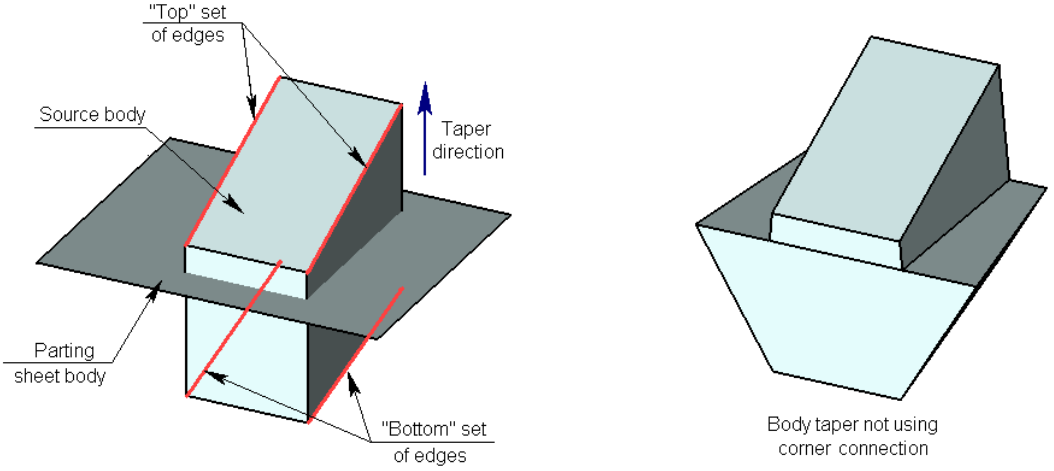
Taper angle is specified separately for the “top” and “bottom” set of edges, i.e. for the “top and “bottom” faces. Within each set, all faces will have identical taper angle.

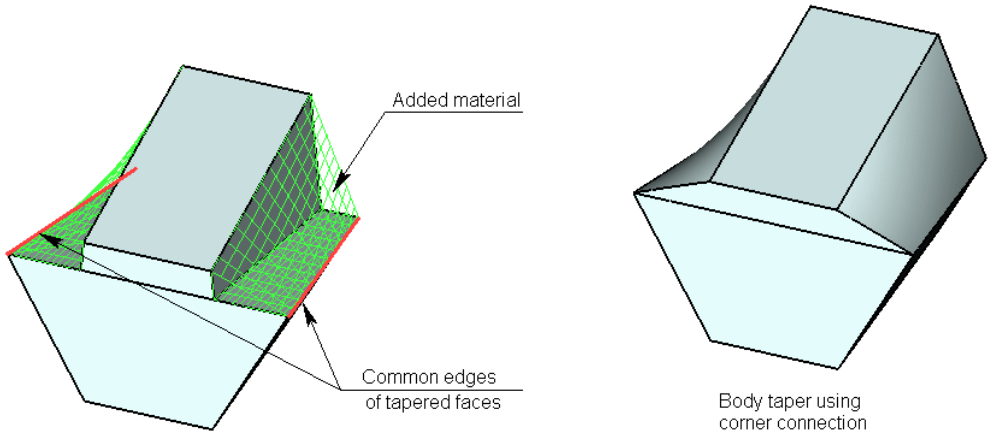
Corner Connection

As a result, of performing a two-sided taper, along the line separating the tapered faces of the body, a gap-step can be created due to non-uniform displacement of the “top” and “bottom” faces. It is possible to avoid creation of this step by using the mode of *corner connection*.

When creating an corner connection, the system automatically adjusts the displacements of the “top” and “bottom” tapered faces in such a way that they have a common edge (on the parting body). Correction of displacements is always carried out by adding material.

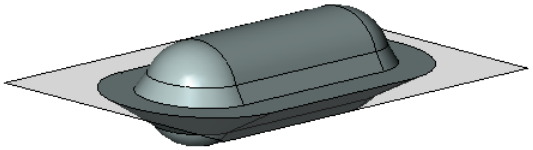
A corner connection can be created only upon constructing a two-sided taper with the sheet parting body.



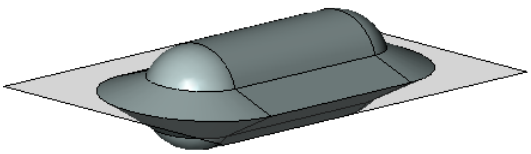


Creating the corner connection can be performed by one of the following methods:

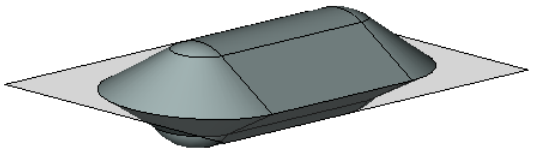
- *"From edge"* – the material is added from the edge determining the given face. The boundaries of the corrected face will include the source edge specified for this face (or a part of the edge);
- *"Tangent to face"* – the material is added in such a way that the corrected surface becomes tangent to the adjacent faces (where it is possible). Simultaneously, the original edges may be deleted and replaced with the new edges created on the basis of the line of intersection of the corrected face and adjacent face;



Taper without corner connection



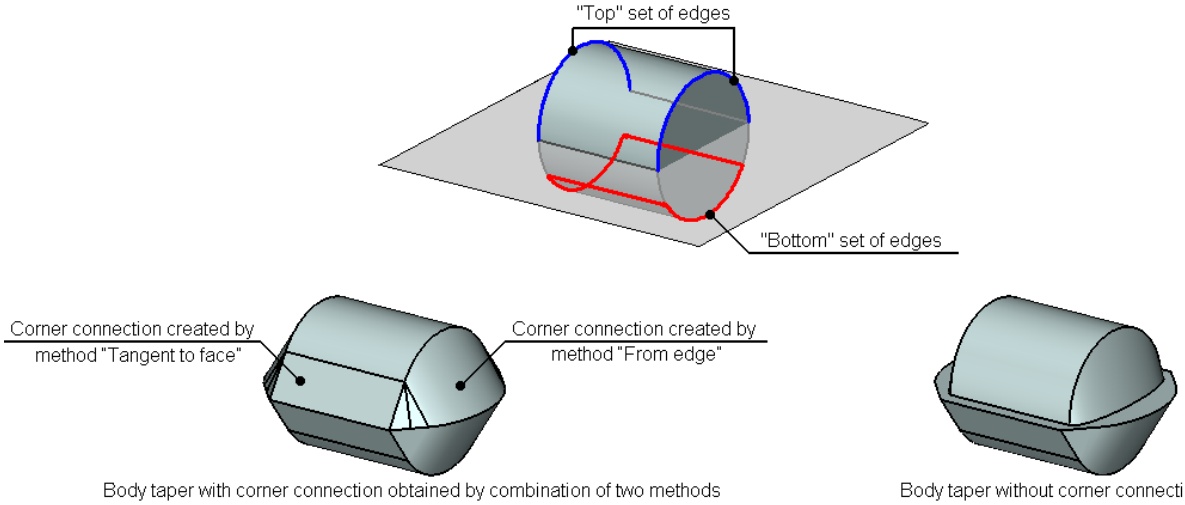
Corner connection created by method "From edge"



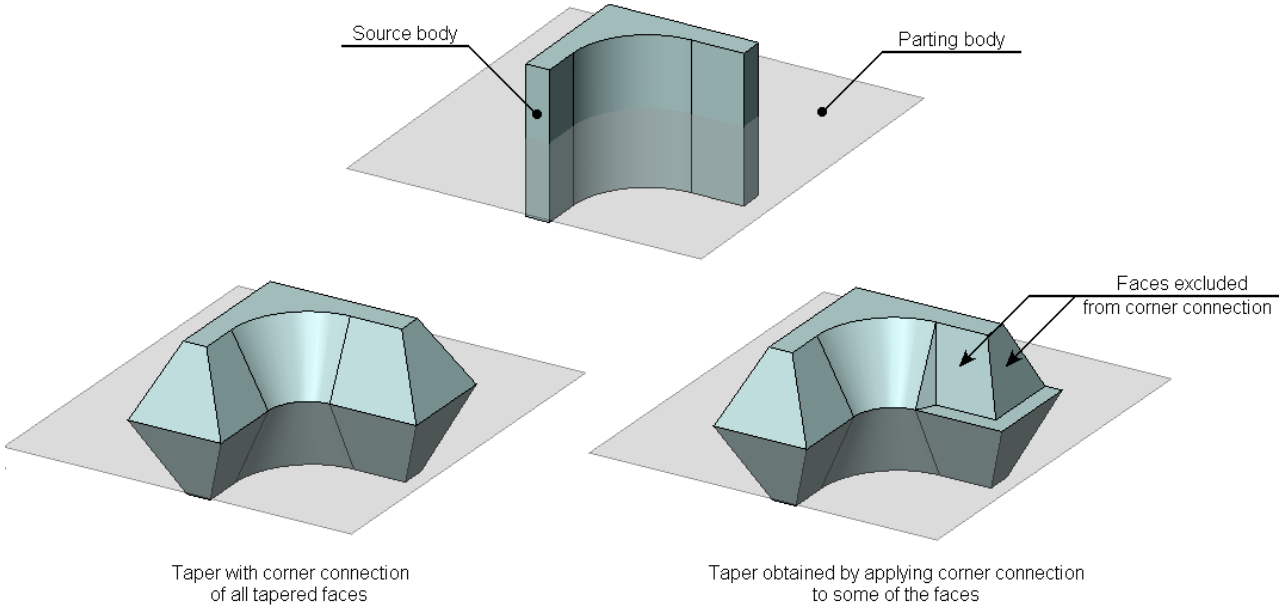
Corner connection created by method "Tangent to face"

- *Combination of first two aforementioned methods* – depending on the geometry of the resulting body, the system itself selects the method of creating the corner connection for each tapered face needing the correction. Priority is given to the method "From edge". If, for some face, this method cannot be used, another method "Tangent to face" is employed.

  On the figure below, an example of creating an corner connection with the use of combination of two methods is shown. The "top" set of taper edges is non-closed, and it is impossible to employ the method "From edge" for the lateral cylindrical face. Thus, for creating a corner connection on the lateral faces of the tapered body, the method "Tangent to face" was used. On the end faces, the correction is carried out by the method "From edge".



By default, in the mode of corner connection, the system adjusts the displacements of all tapered faces. If necessary, it is possible to suppress the displacement correction for any of the tapered faces.



Taper creation method

The following methods of creating the body taper exist in the command:

"Standard" – allows constructing the taper surface exactly matching the form of the source surface and a specified taper angle.

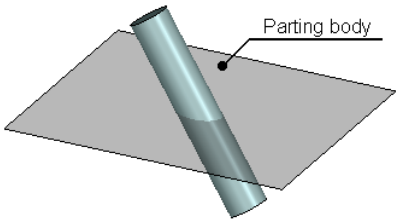
"By curve", "By surface" – matches the form of the source tapered faces in a less stringent manner, which allows constructing the taper in cases when the standard taper construction method does not work.

These algorithms are described in more detail in chapter "Face Taper".

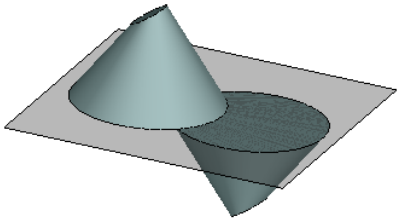
By default, the standard taper creation method is used in the operation. If necessary, it is possible to apply another method of tapering to the entire tapered body as well as to each tapered face individually.

Concave Edge Correction

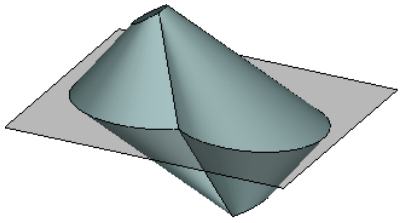
In some cases while creating the taper with the corner connection, concave angles are created on the body, which can interfere with successful creation of taper operation.



Source body



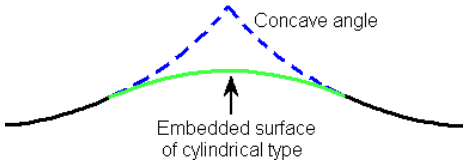
Taper without corner connection



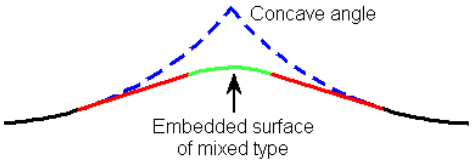
Taper with corner connection
(concave angles were created)

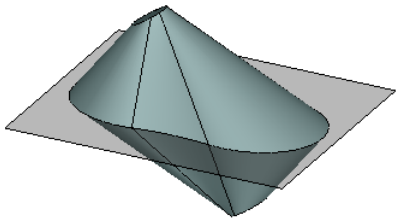
To avoid such a situation various methods of processing the concave angles exist in the operation:

- No correction – created concave angles remain unchanged;
- Cylindrical surface – concave angles are "covered" with a surface of cylindrical type. Simultaneously, in the plane of a parting body the projection of the concave angle is replaced with the circular arc of specified radius.

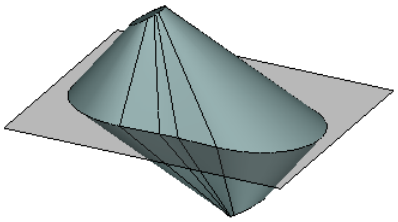


- Mixed surface – the concave angles are "covered" with the surface of mixed type. In the plane of a parting body, the projection of an angle is replaced with a circular arc of a given radius and lines tangent to the surface of the tapered body.





Correction of concave angles
with the help of surface of cylindrical type



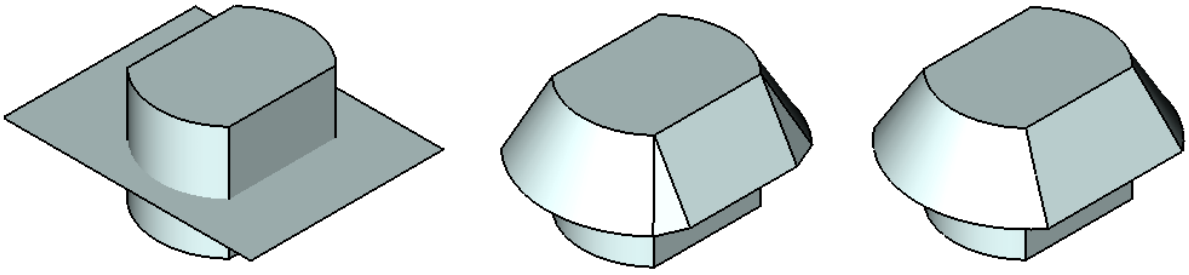
Correction of concave angles
with the help of surface of mixed type

Processing Connections of Tapered Faces

When creating a taper, the command provides options for selecting a method of processing connections between tapered faces. Two processing methods are available:

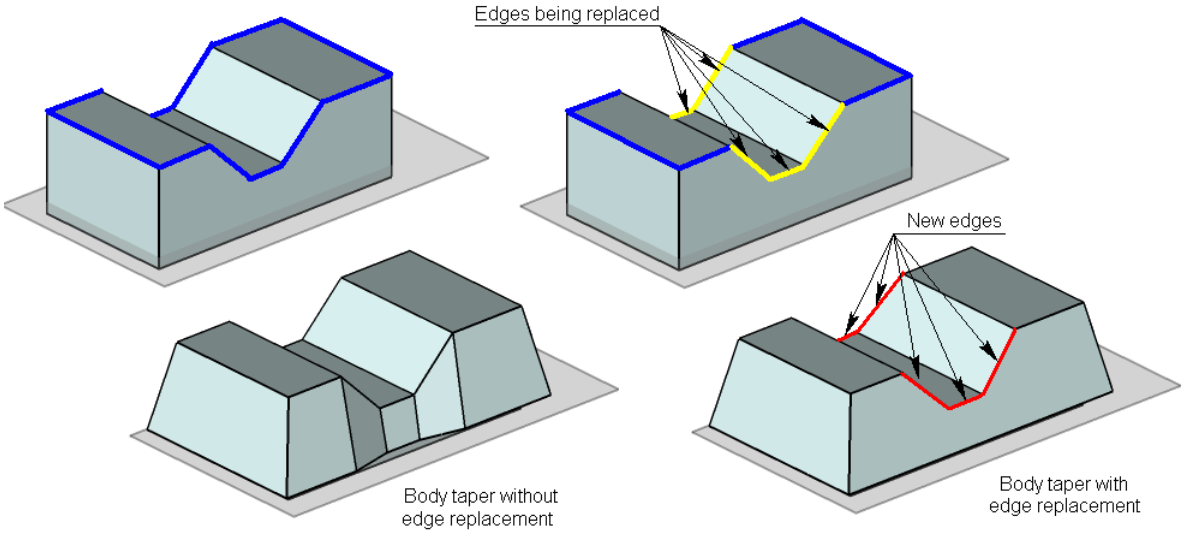
Extend surfaces – in this case, the adjacent tapered faces will be extended to achieve an intersection.

Create plane – in this method, a flat face will be created between two adjacent tapered faces.



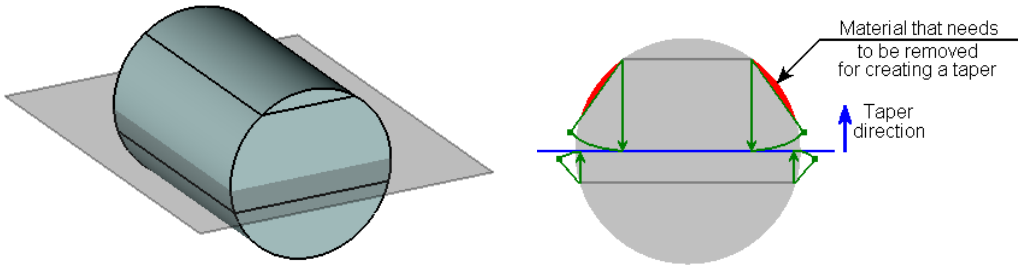
Edge Replacement

In some cases, keeping the shape of edges defined as fixed leads to creation of redundant faces on the tapered body. To improve results of the operation, the user can allow replacement of certain "fixed" edges. Use of edge replacement guards in some cases from creation of extra tapered faces. The replacement edges are created in such away as to make a single slanted face.



Taper with Undercut

In some cases, creation of a taper requires not the addition but the removal of the material of a tapered body. In this situation, it is necessary to use the option of creating an *undercut*. Otherwise, creation of a taper will be impossible.



BODY TAPER CREATION

The “Body Taper” command can be called by one of the following means:

Icon	Ribbon
	3D Model → Modify → Body Taper
Keyboard	Textual Menu
<3TB>	Operation > Taper > Body Taper

To create a body taper, do the following steps:

- ✓ Select the source body.
- ✓ Select the parting body.
- ✓ Define the taper direction.
- ✓ Select the top set of edges.
- ✓ Select the bottom set of edges (required for the two-sided taper).
- ✓ Define the top and bottom taper angles.
- ✓ Define additional parameters of a taper (the method of creating a taper, the method of processing connections between tapered faces, an corner connection, etc.);
- ✓ Complete operation creation with the help of .

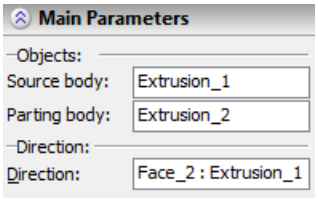
Main Input Data for the Operation

Selecting source body

The default mode of selecting the source body is activated automatically upon entering the “Create Body Taper” operation:

	<S>	Select source body

This source body can be selected either in the scene or in the 3D model tree. After selecting the source body, the properties window will display the respective operation of the source body.

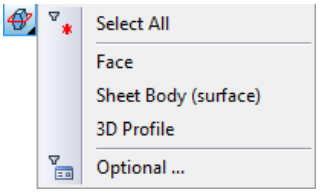


Selecting parting body

Upon selecting the source body, the system proceeds to the mode of selecting the parting body. To enable selection of a solid or sheet as a parting body, the following options are provided in the command automenu:

	<A>	Select parting body
	<H>	Select parting sheet body

When using one of those options for selecting the parting body, the selector is tuned up accordingly. For example, in the sheet body selection mode, the following parting objects can be selected: a face, a 3D profile, an operation-surface. Moreover, filters for choosing the objects can be also customized with the help of the drop-down list of the option .



In the solid body selection mode, only objects with solid geometry can be selected for parting.

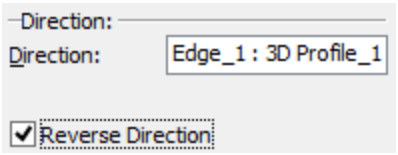
The name of the object used as the parting body appears in the properties window.

Defining taper direction

The next step in the operation creation is selecting an element defining the taper direction. This can be done by the option:

	<D>	Set tapering direction

As common across other operations, the option for selecting the taper direction allows selection of various 3D elements capable of defining a directional vector. The direction-defining object is displayed in the properties window.

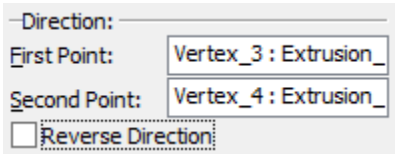


To change the taper direction to the opposite, check the parameter **Reverse Direction**.

Alternatively, the taper direction can be defined by two 3D points. The following options are provided for this purpose:

	<1>	Select first point of tapering direction
	<2>	Select second point of tapering direction

Selected elements are displayed in the "Properties" window.



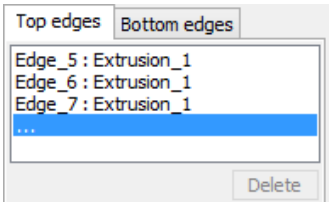
To cancel the direction selection, use the option:

	<K>	Cancel direction selection

Selecting fixed edges

Selection of the set of edges is done directly in the 3D window. To create a one-sided taper, select just the top set of edges. For the two-sided taper, additionally select the bottom set of edges. To do this, use the following options:

	<T>	Select top edges
	<B>	Select bottom edges



Each selected edge will be added to the list of edges in the properties window. Switching between selecting the top and bottom sets of edges can be done either by the automenu options or by clicking the respective tabs in the properties window dialog. To remove an edge from the list, select it in the list, and then press the **[Delete]** button or <Del> key.

Options and can be used in combination with the mode of selecting a sequence of smoothly connected edges, which can be activated by the option:

	<G>	Smooth edge chain selection mode

Let us define a sequence of smoothly connected edges as a set of edges forming closed or non-closed continuous curve which does not have sharp turns (the first derivative is continuous).

When using the given mode, all edges capable of making a smooth sequence with the chosen edge are automatically picked.

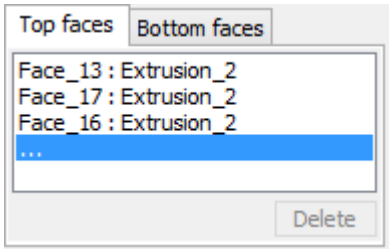
To cancel selection of all edges, use the following option:

	<R>	Cancel selection of all edges

Selecting non-fixed faces

The following options are provided for selecting bottom and top faces:

	<U>	Select top faces
	<O>	Select bottom faces

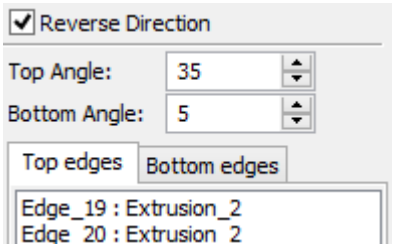


You can activate selection of top or bottom faces by clicking an empty line on the respective tabs of the "Top and Bottom Faces" section in the properties window. Selection of faces, just like edges, is done in the 3D window. The selected faces will be added to the list of faces in the properties window. To remove a face from the list, select it in the list, and then press the **[Delete]** button or <Del> key.

To cancel selection of all faces, use the option:

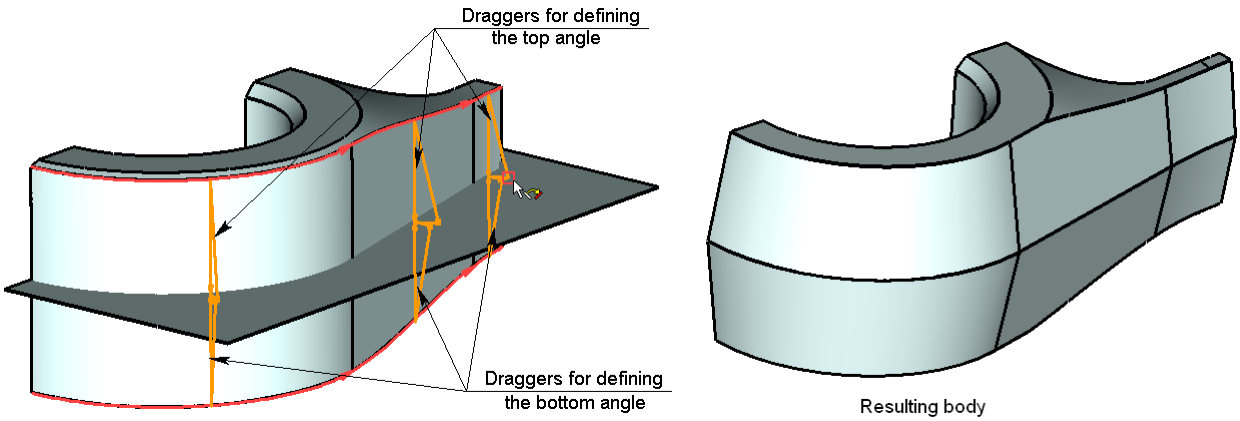
	<Z>	Cancel selection of all faces

Defining taper angles



The command allows defining taper angles for the top and bottom faces of the source body. The angles can be defined by specifying values in the properties window.

To modify taper angle values, you can also use draggers. Once the edges are defined, with respect to which the body taper will be constructed, the draggers appear in the 3D window for modifying the angles. Those draggers work the same way as the draggers defining taper angles in the command **3TA: Create Face Taper**.

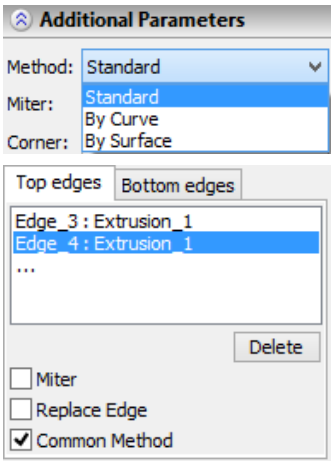


At this stage, the option becomes available in the automenu:

	<Y>	Finish input

Advanced Operation Parameters

Selecting Method of taper creation

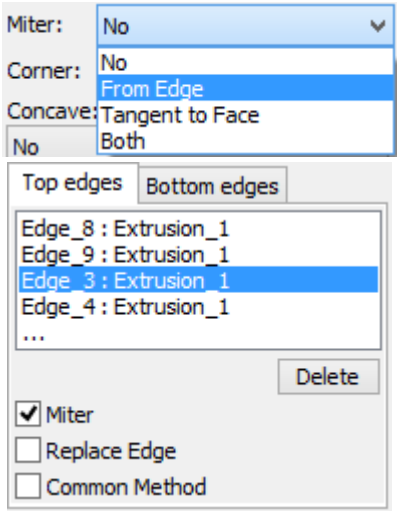


The method of creating a taper is determined by the value of the parameter **Method**. To change the value of this parameter, a drop down list is used. The standard method is used by default.

For all selected fixed taper edges, the flag **Common method** is set by default. This means that the value of the parameter "Method" is unique for all selected fixed taper edges. When changing the value of this parameter, the taper creation method is changed for all tapered faces.

To specify for some tapered face another tapering method (different from the one specified for the rest of the body) it is required: from the list of edges to select the edge corresponding to the given face, turn off the flag "Common method", and with the help of parameter "Method" specify the required taper method.

Creating corner connection



Parameter **Miter** controls creation of corner connection. By default, it is assigned the value **No**, which signifies that the corner connection is not created.

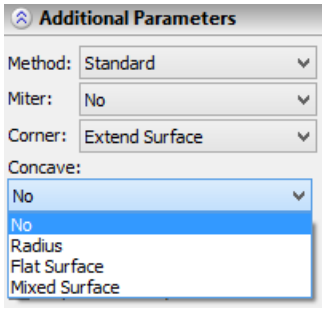
In order to create a corner connection, the user needs to select one of the following values: **From Edge**, **Tangent to Face**, or **Both** from the drop down list. Difference between these three options was described above.

> A corner connection can be created only upon constructing a two-sided taper with the sheet parting body.

By default, the corner connection is created for all tapered faces of the body. To exclude any of the faces from the corner connection, the user needs to select the edge corresponding to the given face from the list, and, for the selected edge, take off the flag **Miter**.

Selecting Method of Correcting Concave Angles

Parameter **Concave** is responsible for correcting concave angles arising upon creation of a taper with a corner connection. By default, this parameter is assigned the value **No**, which means that concave angles remain intact.



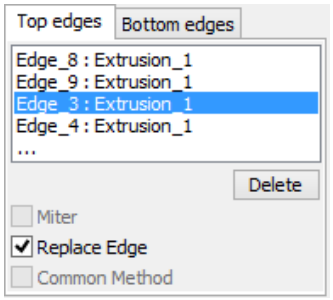
For automatic correction of concave angles it is required to select another value from the drop down list:

- **Radius** – a concave angle is covered with a surface of cylindrical type. In addition, the radius of the surface must be specified in the parameter **Radius**;
- **Mixed Surface** – a concave angle is covered with a surface of mixed type. The radius of cylindrical segment of the surface is specified with the help of the parameter **Radius**.

Controlling Edge Replacement

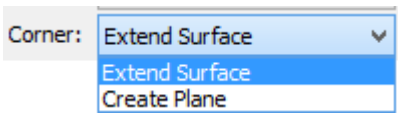
The flag **Replace Edge** controls the capability of replacing fixed edges upon creation of taper. By default, replacement of all selected taper edges is suppressed (flag is turned off for all edges).

To permit replacement of any of the edges, the user should select it in the list of edges and set the flag “Replace Edge”.



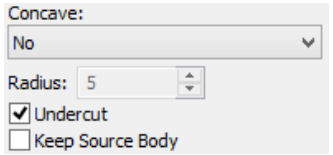
Closing gaps between adjacent faces

The method of processing the connection between tapered faces is selected with the help of parameter **Corner**.



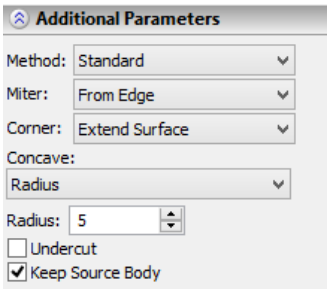
Specifying Undercut

The mode of creating an undercut is activated with the help of the flag **Undercut**. By default, this flag is turned off.



Keep source body

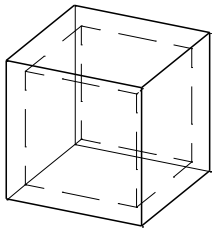
The operation of creating the body taper allows the user to keep the source body in the 3D scene. For that, it is sufficient to set the flag **Keep Source Body**.

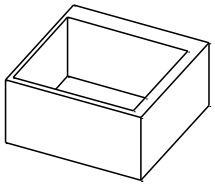


SHELL

MAIN CONCEPTS

The operation "Shell" allows creating offset bodies and shells based on solid bodies. A shell is a hollow, thin-wall body with walls of the specified thickness. When creating a shell, some of the source solid body's faces can be removed.





The operation "Shell" is based on the following principle: an offset surface is created for each face of the source sheet or solid body, at the specified offset distance. Depending on the user preference, the offset can be outwards or inwards of the surface of the processed body (directional or counter-directional to the normal). The offset surfaces can be used as a self-contained result – for creating offset bodies. Alternatively, offsets can be used for making a shell by confining a portion of the inner volume of the source solid body.

SHELL CREATION

The command **3SH: Create Shell** is called by one of the following means:

Icon	Ribbon
	3D Model → Modify → Shell
Keyboard	**Textual Menu**
<3SH>	Operation > Shell

To create a shell, do the following set of steps:

- Select a face to remove or a whole solid body;
- Select a face whose wall thickness is to be defined separately (if desired);
- Define parameters (optional action);
- Finish input.

To create an offset body, do the following set of steps:

- Select the source sheet or solid body;
- Activate the mode of creating offset body;
- Define parameters (optional action);

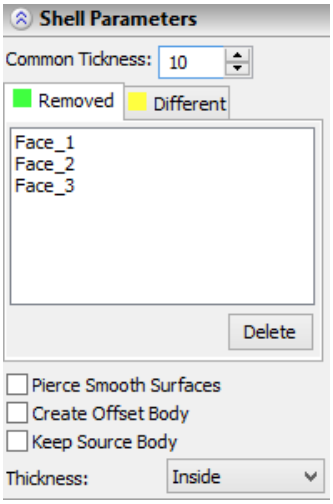
- Finish input.

Selection of a Face or a Body to Remove

To create a solid shell with one or multiple face removal, use the option:

	<R>	Select Face to be removed

This option activates by default upon calling the command. It allows selecting a face or a set of faces to be removed when creating the shell. The parent body of the first selected face to be removed also gets selected. The body itself is not highlighted, while the selected face is highlighted in a specific color.

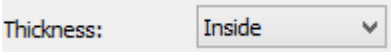


The faces selected for removal are added to the list in the provided pane in the property window on the tab **[Removed]**.

> Placing cursor in the list pane in the property window activates the option for selecting faces to remove.

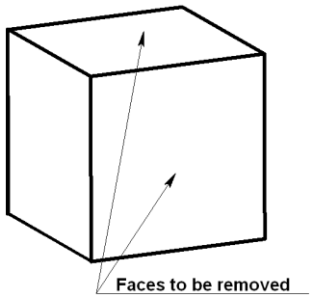
The thickness of other walls of the shell being created is defined in the input box "**Common Thickness**" of the property window. The default value is 1.

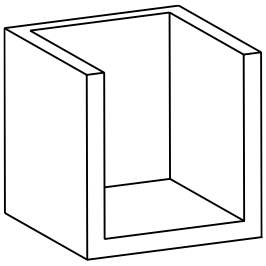
This means the walls of the faces left in the body will be 1 mm thick (the measurement units depend on the document settings).

Thickening the walls of the shell can be done either outwards or inwards of the source body. This is controlled by a special additional parameter in the property window 

.

The wall thickness can be positive or negative. Use of negative values results in shell thickening in the opposite direction.

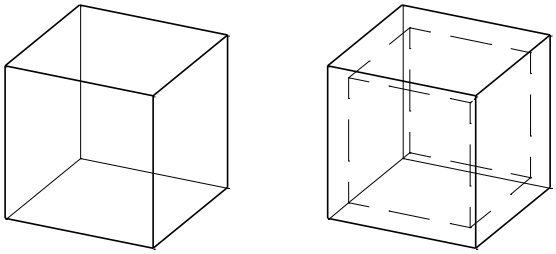




To create a shell without removing any faces, upon entering the command switch to the automenu option:

	<O>	Select Solid

This option allows selecting a solid body to create a hollow body with the specified wall thickness.

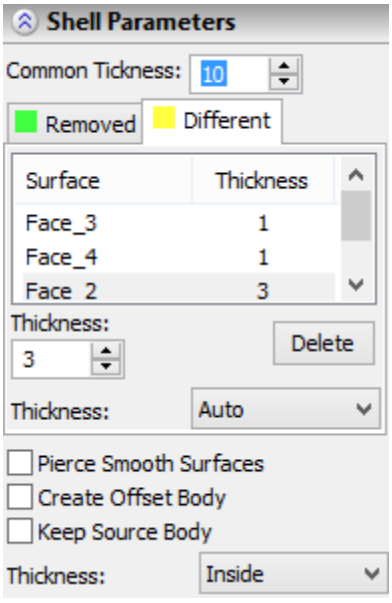


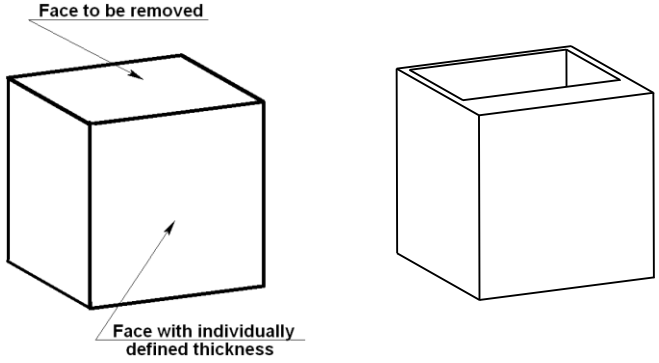
Selection of Faces with Individually Defined Wall Thickness

To define the wall thickness individually for one or several faces, select such faces with the help of the automenu option:

	<T>	Select Face with different wall thickness

This option becomes accessible when the whole body or a face to be removed is already selected. The selected faces are referred to as "Different" and are added to a separate list on the tab **[Different]** in the property window. The thickness of each such face can be defined in the input box "**Thickness**" after selecting one or several faces in the list.





The option:

	<K>	Cancel selection of Faces by pointing to them

is provided for unselecting faces to be removed or faces with individually defined thickness. To cancel a selection, point the mouse at a selected face and click . Faces can also be unselected by manipulating the respective list in the property window. For this purpose, the graphic button **[Delete]** is provided with each list.

All faces can be unselected at once by the option:

	<F>	Cancel selection of all Faces

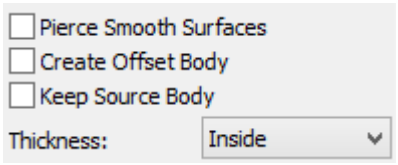
You can confirm the operation creation as soon as the option becomes accessible in the automenu:

	<Y>	Finish input

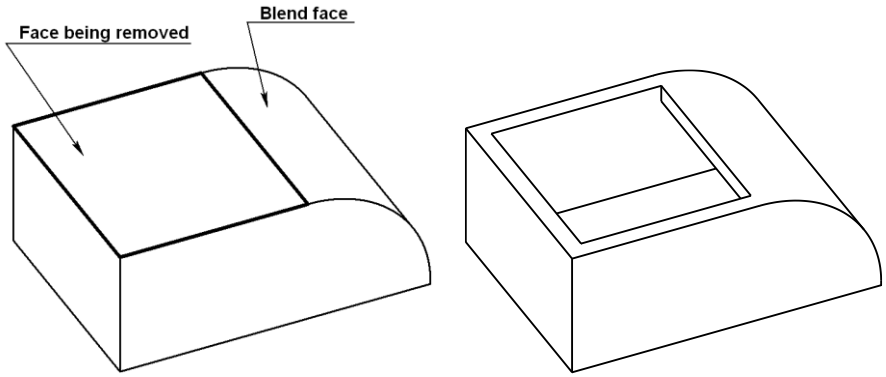
As a result, a shell body will be displayed in the 3D window.

Additional Operation Parameters

Additional operation parameters can be set in the property window, using the following switches:



Pierce smooth surfaces. Setting this parameter allows creating a shell in the cases when the removed face is tangent to adjoining non-flat faces, such as those created by blending (radial fillet).



Create offset body. Setting this flag activates the mode of creating offset (rather than hollow) body. The faces of such body are offset from those of the source body by the specified amount. This mode can be useful for creating workpieces with allowance for machining.

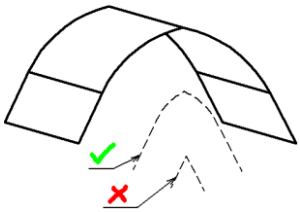
Only sheet bodies can be used for creating offset bodies in the command “Shell”.

Keep source solid. This option allows keeping the source solid body in the scene together with the result. This can be useful when you need to have two models at once, like a workpiece with allowance and an original part.

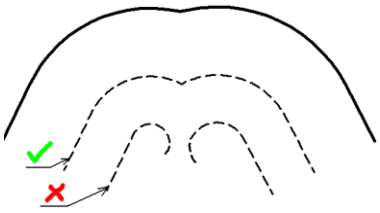
Troubleshooting

When creating “Shell” operation, the system may fail to calculate the result in some cases. A reason for a failure could be particulars of the source body geometry. Let's review most typical situations:

1. **Degenerate faces.** When creating an offset, some faces may disappear. The system may handle some of the cases. However, if there are too many such faces, or several adjacent faces disappear, the system outputs an error.

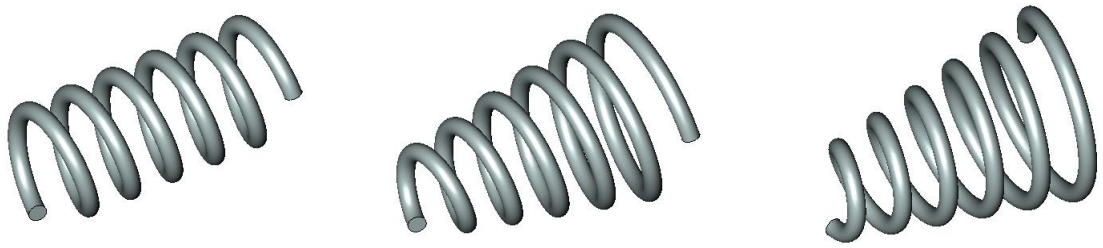


2. **Offset faces do not intersect.** Upon creating offset faces, the system constructs intersection lines between adjacent faces for generating the resulting body. In some cases, two adjacent offset faces may not intersect.



SPRINGS

The operation Spring creates a 3D body as a conic or cylindrical helix-shape spring with round profile. The body is created by sweeping a circular profile along a helical curve defining the spring trajectory.

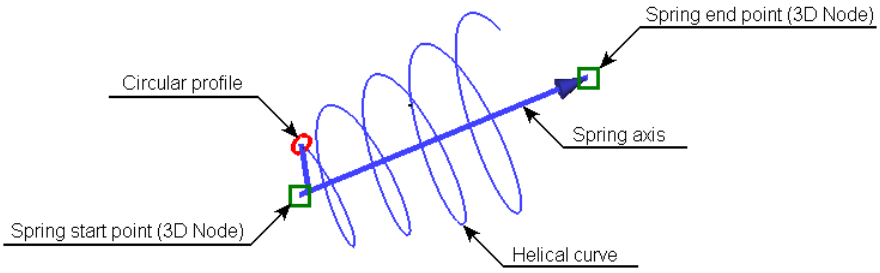


For speeding up modeling, a capability is provided for creating a Boolean operation together with spring creation. (This capability was described in details in the chapter "Extrusion".)

MAIN CONCEPTS AND OPERATION CAPABILITIES

Spring Axis

The helical curve used for sweeping the circle profile is centered about the spring axis. The axis is defined by two 3D points: the start point and the end point. 3D points can be defined by 3D nodes, as well as by any 3D object suitable for defining a point in the space. The vector defining the spring axis will be directed from the first point to the second one (those also defines the start and the end of the spring, respectively). The shape of the curve is defined by the spring parameters. The swept profile will be normal to the original curve.



Spring Main Parameters

The shape of the body resulting from this operation is defined by the following parameters:

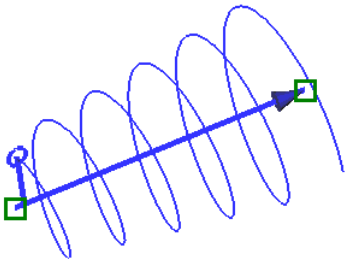
The radius of the spring strip – defines the radius of the circular profile, whose sweeping forms the body;

Mean spring radii at the ends – the radii of the helical curve at the start and end of the spring (mean start and end radii are equal in the case of a cylindrical spring);

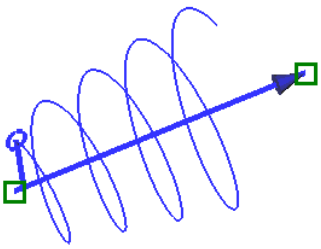
The step (pitch), the number of coils and the free length of the spring;

Spring coiling direction: Clockwise (right) or counterclockwise (left).

The free length of the spring can be defined as the distance between the 3D points defining its axis. In this way, it is sufficient to define just the pitch or just the number of coils – the other parameter will be calculated by the system automatically. In the cases when both the pitch and the number of the spring coils are defined, the spring length will be defined solely by those parameters.



Spring length is defined by the distance between two 3D Nodes defining the axis (additionally, the spring pitch is defined)

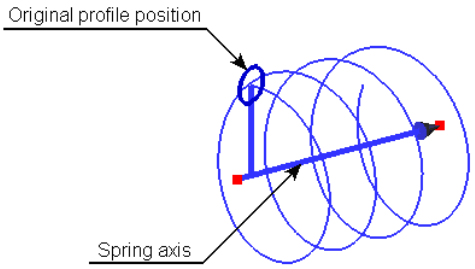


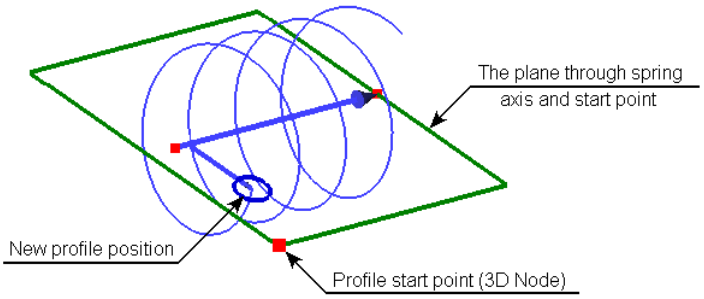
Spring length is defined by the specified values of the pitch and the number of spring coils (n=4)

Initial Position of Spring Profile

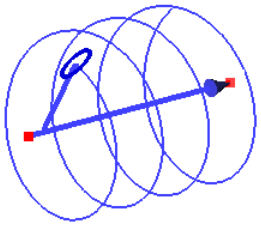
Initial position of the spring profile is set arbitrarily by the system. It can be modified by defining the desired position with the help of an additional 3D point – the start point of the profile – or by specifying the numerical value of the profile rotation angle (with respect to the original position).

Once the start point of the profile is specified, it will snap to the plane passing through the selected point and the spring axis, along the specified start radius vector.

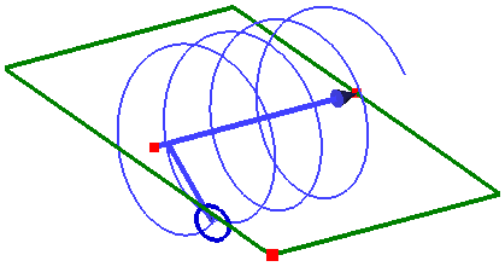




When both the angle of the start point and the numerical value of the rotation angle are specified, the latter is evaluated with respect to the profile position defined by the start point.



Original profile position defined by numerical value of rotation angle (30 degree) without selecting start point



Original profile position defined by start point and numerical value of rotation angle (30 degree)

Compression and Grinding of Spring Ends

If necessary, the end coils of the spring can be compressed (closed) and ground. The diagram below shows a spring with one coil compressed at the start, and one coil compressed and 3/4-circle ground at the end.

Spring Position with Respect to Start and End Points

If the spring length is defined by the pitch and the number of coils, you can force the spring alignment by the start or the end point.

Aligning by start point

Aligning by end point

SPRING OPERATION CREATION

To create the operation Spring, use the command **3SP: Create Spring**:

Icon	Ribbon
	3D Model → Modify → Spring
Keyboard	Textual Menu
<3SP>	Operation > Spring

To create a spring, do the following steps:

- ✓ Define spring axis;
- ✓ Define spring parameters (the radius of the wire strip, the spring main radius, etc.);
- ✓ Select the start point for the spring profile position (optional);
- ✓ Confirm operation creation.

Spring Axis Definition

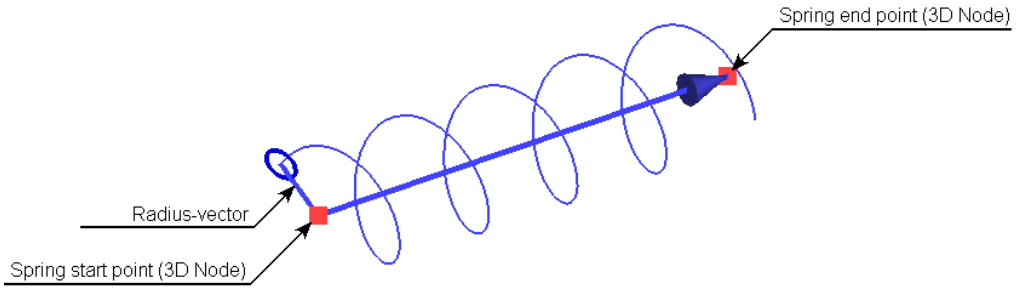
The start point of the spring axis is defined by the following automenu option:

	<F>	Select starting Point of Axis

The end point of the axis is selected option:

	<S>	Select ending Point of Axis

The options activate one after another. Selected 3D objects are highlighted. Once the second point is selected, a vector will be displayed in the 3D scene, defining the axis of the spring, the helical guide and the profile circle. (Initially, those are assigned the default parameter settings.) Additionally, a radius-vector will be displayed, representing the mean radius at the spring start.



At this stage, you can already complete the spring operation by pressing in the property window or in the automenu.

The spring parameters can be defined in the command property window or in the operation parameters dialog box, accessible by the option .

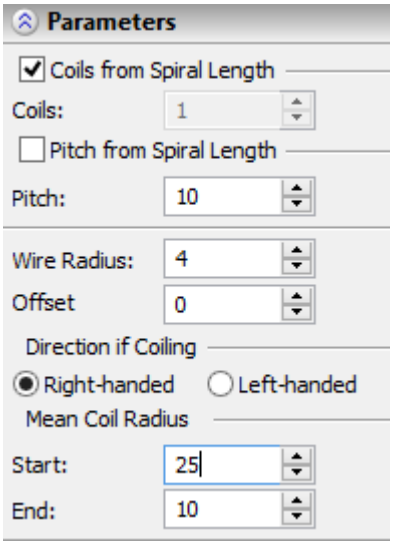
Spring Operation Parameters Definition

You can define main parameters of the spring in the top section of the property window – **(non-optional) Parameters**. Those are wire (strip) radius, the mean radii of the spring and the direction of coiling.

To have the spring length calculated by the distance between the axis-defining 3D points, set one of the flags: **Coils from Spiral Length** or **Pitch From Spiral Length**. In this case, the system will automatically calculate the value of the parameter with the respective flag set (either the number of coils or the pitch). At the same time, you have to specify a numeric value of the other parameter (either **Coils** or **Pitch**). If none of the flags is set, you will have to specify both the number of coils in the spring and the pitch.

The radius of the spring strip (wire) is specified in the box **Wire Radius**. The winding direction of the spring is controlled by the radio group **Direction of Coiling**.

The start and end mean radii of the spring are defined by the parameters **Mean Coil Diameter/Start** and **Mean Coil Diameter/End**. To create a cylindrical spring, specify equal values for both radii.



Selection of the Start Point for Spring Profile Position

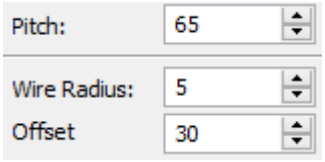
The start point of the spring profile can be specified by the following automenu option (the option activates upon selecting the second axis point):

	<B>	Select 3D Node as profile starting location

To undo a 3D point selection, simply select another one with the same option, or cancel the selection by the option:

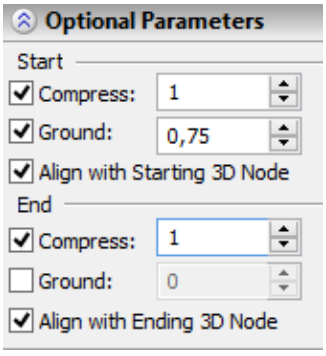
	<K>	Delete starting 3D Node

The profile will then revert to its original position.



You can enter a numerical value of the profile rotation angle with respect to the original position in the operation property window, by the parameter **Offset Angle** (**(non-optional) Parameters** section).

Defining Compression and Grinding of Spring Ends



Parameters for compression and grinding of end coils of the spring are defined in the other section of the property window – in **Optional Parameters**.

To compress coils at the spring start, set the flag **Start/Compress** and enter in the adjacent box the desired number of coils (for example, *"0,75"* or *"1"*). Setting the flag **Start/Ground** allows defining the portion of the first coil to grind. The compression and grinding parameters at the end of the spring are defined in a similar way – by using the flags **End/Compress** and **End/Ground**.

Spring Alignment by Start or End Point

To align the spring by the start or end point, set the following flag in the property window: **Align with Starting 3D Node** or **Align with Ending 3D Node**. In the case of ambiguity (when both flags are set), the system aligns the spring by the start point.

Using the created spring in a Boolean

To create a Boolean operation, do the following:

Turn on the Boolean creation mode by the automenu option:

	<Ctrl+B>	Select original body for Boolean operation

A Boolean is created when the icon is pushed.

Select the type of the operation from the pull-down list under the above option:

	<Ctrl+'+'>	Addition
	<Ctrl+'-'>	Subtraction
	<Ctrl+'*'>	Intersection
	<Ctrl+I>	Smart mode

Operating principles of the Smart mode:

- ✓ If the body being created and the existed selected body have intersection of volumes - the type of Boolean operations: subtraction.
- ✓ If the body being created lies entirely within the selected body - the type of Boolean operations: subtraction.
- ✓ If the body being created touches and selected body touch each other - the type of Boolean operations: addition.
- ✓ If conditions set out above are not met, or a selected body lies inside the created, or error occurred determining the type of penetration, then, Boolean type is not defined - the Boolean operation will not be created.

Select the target body for the Boolean (optional in some cases), using the automenu option:

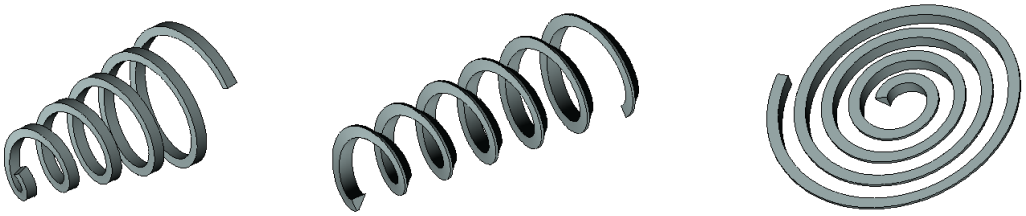
	<Ctrl+T>	Select Target Body for Boolean

If only one body exists in the scene, it is selected automatically. The new body created by the spring operation becomes the tool body of the Boolean.

Upon confirming operation creation, first the body is created by the spring operation, and then the specified type Boolean is performed.

SPIRAL

The operation Spiral creates a 3D body as a conic or cylindrical helix-shape spring with arbitrary profile. This functionality is mostly similar to the Spring operation functionality. Spiral is created by sweeping an arbitrary closed 3D profile along a helical curve. The profile can be oriented perpendicular to the spiral trajectory or perpendicular to the helical curve.

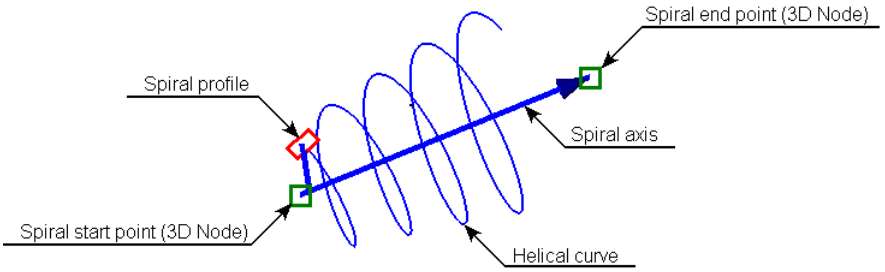


For speeding up modeling, a capability is provided for creating a Boolean operation together with spiral creation (this capability was described in details in the chapter "Extrusion").

MAIN CONCEPTS AND OPERATION CAPABILITIES

Spiral Axis

The helical curve used for sweeping the profile is centered about the spiral axis. The axis is defined by two 3D points: the start point and the end point. 3D points can be defined by 3D nodes, as well as by any 3D objects suitable for defining a point in the space. The vector defining the spiral axis will be directed from the first point to the second one (this also defines the start and the end of the spiral, respectively). The shape of the curve is defined by the spiral parameters. The swept profile will be normal to the original curve.

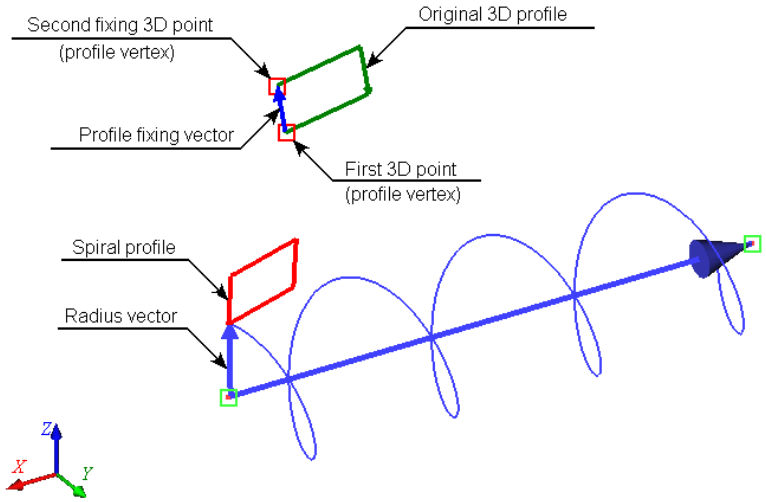


Spiral Profile and its Orientation in Space

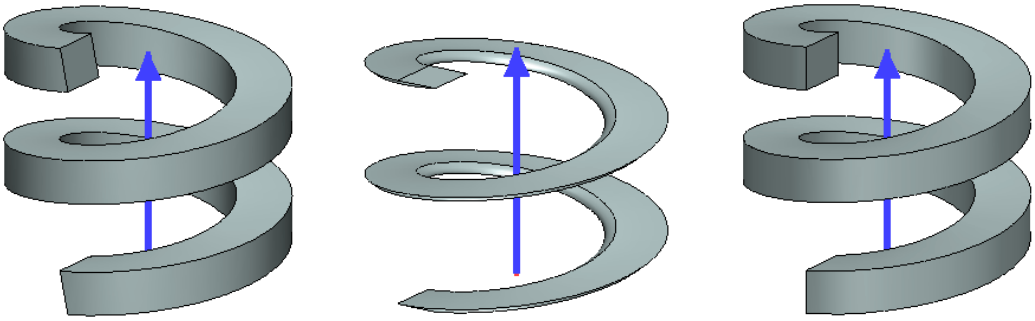
You can select any closed single-contour 3D profile as the spiral profile.

The profile position in the space is defined by a fixing vector. The spiral fixing vector is defined by two 3D points fixing the profile. The profile fixing vector will be directed from the first 3D point to the second one. If the profile is defined by 2D constructions, the profile fixing vector can be defined by two 2D points in the profile plane.

Once the spiral is created, the first point of the profile fixing vector is snapped to the end point of the radius vector at the spiral start. The direction of the vector is collinear with the radius vector.

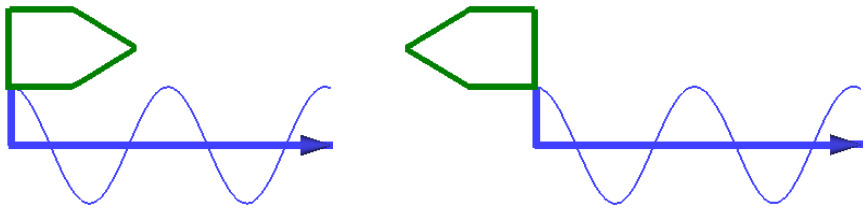


After that, the profile is rotated about the fixing vector to achieve one of the following orientations: perpendicular to the helical curve, perpendicular to the axis or parallel to the axis.



Perpendicular to spiral helix | Perpendicular to spiral axis | Parallel to spiral axis

If necessary, the profile of the spiral can be rotated about the radius vector by *180 degree*.



Spiral Main Parameters

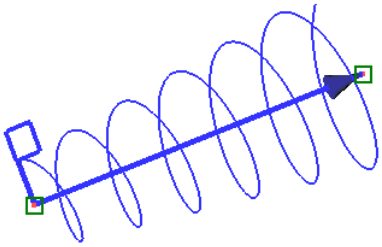
The main spiral parameters are:

Mean spiral radii at the ends – the radii of the helical curve at the start and end of the spiral;

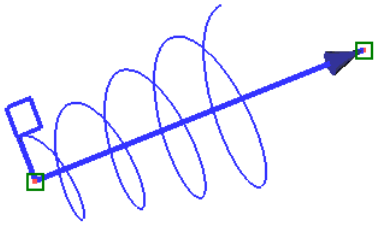
The step (pitch), the number of coils and the free length of the spiral;

Spiral winding direction (hand of coiling): Clockwise (right) or counterclockwise (left).

The free length of the spiral can be defined as the distance between the 3D points defining its axis. In this way, it is sufficient to define just the pitch or just the number of coils, while the other parameter will be calculated by the system automatically. In the cases when both the pitch and the number of the spiral coils are defined, the spiral length will be defined solely by those parameters.



Spiral length is defined by the distance between two 3D nodes defining the axis (additionally, the pitch is defined)



Spiral length is defined by the specified values of the pitch and the number of spiral coils (n=4)

The radius of the spiral can also be calculated automatically by the system. It will be equal to the distance from the spiral axis to the first profile attachment point.

Initial Position of Spiral Profile

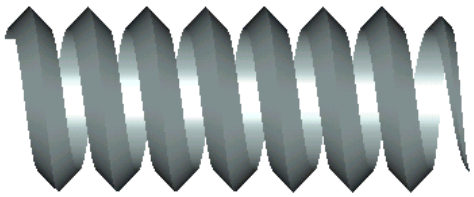
The initial position of the spiral profile is defined by the system as follows: The spiral start radius vector lies in the plane passing through the spiral axis and the closing point of the original 3D profile. If necessary, a different starting position can be specified. The desired starting position can be defined with the help of an additional 3D point – the start point of the profile – or by specifying the numerical value of the profile rotation angle (with respect to the original position).

Once the start point of the profile is specified, it will snap to the plane passing through the selected point and the spiral axis, along the specified start radius vector. This capability was described in the chapter "Spring".

When both the angle of the start point and the numerical value of the rotation angle are specified, the latter is evaluated with respect to the profile position defined by the start point.

Tapering

If necessary, the ends of the spiral can be tapered at a specified rate. This capability is provided for defining thread tapering when using Spiral operation for creating thread bodies.



When tapering, the spiral profile reduction is done against the direction of the profile fixing vector.

STEPS OF SPIRAL OPERATION CREATION

To create a Spiral operation, use the command **3SR: Create Spiral**:

Icon	Ribbon
	3D Model → Modify → Spiral
Keyboard	Textual Menu
<3SR>	Operation > Spiral

To create a spiral, do the following steps:

- ✓ Define the spiral axis by two 3D nodes.
- ✓ Select a 3D profile.
- ✓ Define the profile orientation in the space.
- ✓ Define the starting position of the profile (select the start point for the profile position – *optional*).
- ✓ Define spiral parameters (the spiral radius, etc.).
- ✓ Confirm the operation creation (in the property window or in the automenu).

The spiral parameters are defined in the command property window or in the operation parameters dialog box, accessible by the option .

Spiral Axis Definition

The start 3D point of the spiral axis is defined by the following automenu option:

	<F>	Select starting Point of Axis

The end point of the axis is selected option:

	<S>	Select ending Point of Axis

The options activate one after another. Selected 3D objects are highlighted. Once the second point is selected, a vector will be displayed in the 3D scene, defining the axis of the spiral, the helical curve and the radius vector indicating the radius at the start of the spiral.

Spiral Profile Selection

Upon defining the spiral axis, the following options become available in the automenu:

	<C>	Select 3D Profile

To select the spiral profile, in the 3D view window move the pointer to a 3D profile and click . To select a 3D profile based on a 2D hatch, you can pick the original 2D hatch in the 2D view window (or on the activated workplane).

Defining 3D Profile Orientation in Space

The 3D points of the profile fixing vector are defined by the following automenu options:

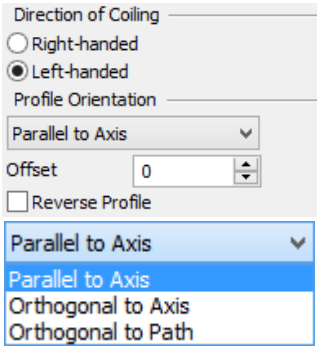
	<R>	Select First Attachment 2D Node
	<T>	Select Second Attachment 2D Node

To define the fixing vector by 2D nodes (in the case of the 3D profile based on 2D elements), use another pair of options in the command automenu:

	<N>	Select first attachment 2D Node
	<A>	Select second attachment 2D Node

Selected 3D points (or 2D nodes) are highlighted; however, the profile fixing vector is not displayed in the scene.

The way of orienting the profile is defined in the operation parameters, in the group **"Profile Orientation"**. The combo box contains the following options:



Parallel to Axis – the spiral profile plane stays parallel to the spiral axis as it is swept along the helical trajectory;

Orthogonal to Axis – the spiral profile plane stays perpendicular to the spiral axis;

Orthogonal to Path – the spiral profile plane stays perpendicular to the helical trajectory.

To rotate the spiral profile about the radius vector by *180 degree, set the flag* **Reverse Profile**.

At this stage, you can already complete the spiral creation by pressing in the property window or in the automenu.

Spiral Parameters Definition

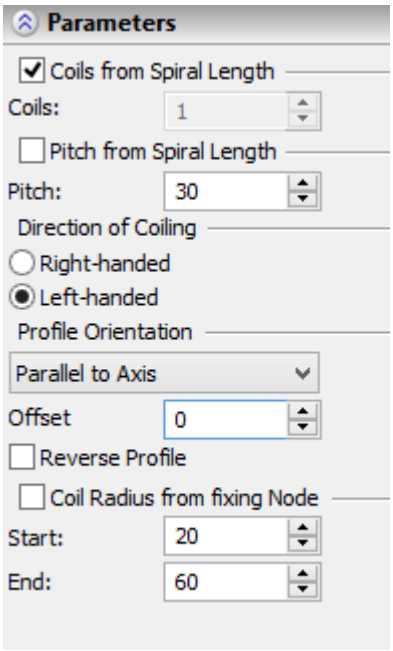
The main spiral parameters are defined in the first section of the property window – **Parameters**. Those include the number of coils and the pitch, the start and end mean spiral radii, the spiral direction, etc.

To have the spiral length calculated by the distance between the axis-defining 3D points, set one of the flags: **Coils from Spiral Length** or **Pitch from Spiral Length**. In this case, the system will automatically calculate the value of the parameter with the respective flag set (either the number of coils or the pitch). At the same time, you will have to specify a numeric value for the other parameter (either of **Coils** or **Pitch**). If none of the flags is set, you will have to specify both the number of coils in the spiral and the pitch.

The winding direction of the spiral is controlled by the radio group **Direction of Coiling**.

The spiral radius will be calculated automatically (by the distance between the axis and the first attachment point of the spiral profile), if the flag is set, **Coil Radius from fixing Node**. To define an

arbitrary radius at the spiral start and end, cleared the flag and enter the desired values in the boxes **Start** and **End**.



Selection of the Start Point for Spiral Profile Position

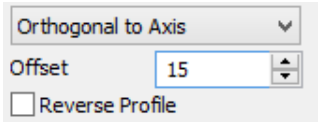
The start 3D point of the spiral profile is defined by the following automenu option:

	<B>	Select 3D Node as profile starting location

To undo a 3D point selection, simply select another one with the same option, or cancel the selection by the option:

	<K>	Delete starting 3D Node

The profile will then revert to its original position.



You can enter a numerical value of the profile rotation angle with respect to the original position in the operation property window, by the parameter **Offset Angle** (**Parameters** section).

Using the created spiral in a Boolean

To create a Boolean operation, do the following:

Turn on the Boolean creation mode by the automenu option:

	<Ctrl+B>	Select original body for Boolean operation

A Boolean is created when the icon is pushed.

Select the type of the operation from the pull-down list under the above option:

	<Ctrl+'+'>	Addition
	<Ctrl+'-'>	Subtraction
	<Ctrl+'*'>	Intersection
	<Ctrl+I>	Smart mode

Operating principles of the Smart mode:

- ✓ If the body being created and the existed selected body have intersection of volumes - the type of Boolean operations: subtraction.
- ✓ If the body being created lies entirely within the selected body - the type of Boolean operations: subtraction.
- ✓ If the body being created touches and selected body touch each other - the type of Boolean operations: addition.
- ✓ If conditions set out above are not met, or a selected body lies inside the created, or error occurred determining the type of penetration, then, Boolean type is not defined - the Boolean operation will not be created.

Select the target body for the Boolean (optional in some cases), using the automenu option:

	<Ctrl+T>	Select Target Body for Boolean

If only one body exists in the scene, it is selected automatically. The new body created by the spiral operation becomes the tool body of the Boolean.

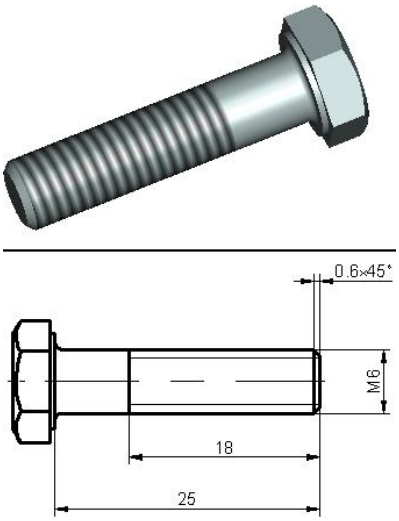
Upon confirming operation creation, first the body is created by the spiral operation, and then the specified type Boolean is performed.

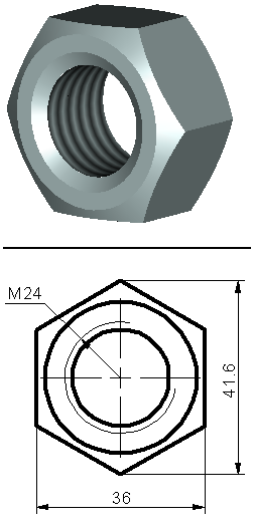
THREAD

T-FLEX CAD supports creation of cosmetic threads on cylindrical and conic faces of a three-dimensional model. A cosmetics thread is not the exact geometrical representation of an actual thread that can be created in the command **3SR: Create Spiral**. Nevertheless, it is quite suitable for most of the tasks arising in thread modeling, while consuming much less computational resources.

A cosmetic thread is a texture applied to the selected face. The simulated image reflects the pitch, direction and boundaries of the thread.

When creating drawings by projecting cosmetic threads, the standard thread notation is automatically generated on the projections.





MAIN CONCEPTS. OPERATION CAPABILITIES

Cosmetic threads are created based on a special database *"Threads.mdb"*, located in the folder ...*Program*. The database contains geometrical parameters of standardized threads according to the engineering standards. If necessary, the database can be appended with custom threads. You can also create nonstandard threads directly in T-FLEX CAD, without using the database.

When creating a cosmetic thread, you need to select the base face (cylindrical or conic), define the thread type (standard) and side (outer or inner) and the main parameters. To define the thread parameters, use either the property window or the standard parameters dialog box. All of the thread parameter values can be defined using variables. Additionally, you can specify the thread offsets from the ends of the base face or from an arbitrary 3D object. The position of the thread end can also be defined relative to its start, by actually specifying the thread length.

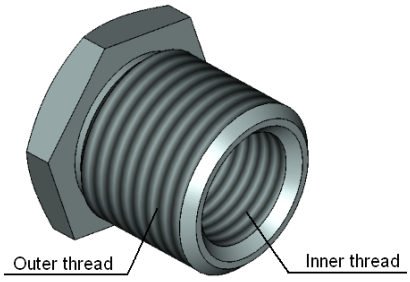
The parameters of the standard thread – the pitch, the working profile height and the class – can either be set by the system automatically or manually defined by the user.

In the Autoadjust mode, the system sets the thread parameters from the database, based on the diameter of the base face. When the base diameter is modified, the thread parameters are adjusted automatically.

In the manual mode, the user defines the thread parameters by numerical values or by using variables. The entered values are checked against those in the thread database. If the input diameter or pitch is not defined for the respective thread type, the value will be taken from the database nearest to the specified one. In this case, the thread parameters will not adjust to the base face diameter modifications. When creating a nonstandard thread (not existing in the thread database), all its parameters are defined manually.

The combination of parameters and the thread behavior depends on its property – being an outer or an inner thread. For an outer thread, the base face diameter is considered the outer diameter, while for an inner thread it is the inner diameter. For the same base face diameter, the system-defined parameters will be different for outer and inner threads. The thread side (outer or inner) is a characteristic constant and can be defined in two ways.

In the first way, the thread side is defined automatically at creation. Inner threads are created on concave faces, while outer ones – on the convex faces. If the face later changes from convex to concave or vice versa due to Boolean operations or other body modifications, the thread will disappear. The automatically defined thread side is suitable for most of threads. In the second way, the thread side is explicitly defined by the user at creation, regardless of the current geometry of the base face. This may be necessary, for example, when creating fragments of holes.



In the fragments, the thread is applied to the convex face, yet is marked as inner. Upon assembling the fragment, performing a Boolean subtraction makes the base face of the thread concave. Thus, after the Boolean operation the thread is carried over to the resulting inner cylindrical face of the hole.

When creating a 2D projection, the necessary graphic lines are created automatically to mark the thread. The parameters of such graphic lines are defined separate from the rest.

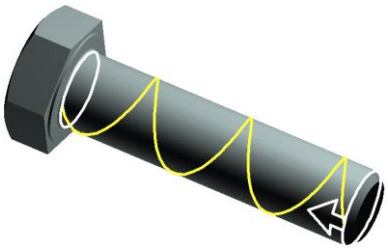
CREATING THREAD

To create a thread, use the command **3AT: Create Thread**:

Icon	Ribbon
	3D Model → Modify → Thread
Keyboard	Textual Menu
<3AT>	Operation > Thread

First, you need to select a cylindrical or conic face in the 3D window to which the thread will be applied. Upon entering the command, the following automenu option is active:

	<F>	Select cylindrical face



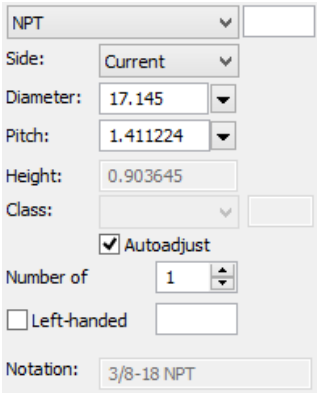
Upon the selection, a schematic image of the thread spiral is displayed on the face, along with the marks for the thread start and end. By default, the thread is applied to the full extent of the selected face. The arrow on the thread image shows the direction of cutting the thread. The cutting direction is defined by the face selection: the thread start is set at the face end nearest to the pointer at the time of clicking for face selection. It is necessary to know the cutting direction in order to correctly define the boundaries of the thread.

Upon selecting the first face, the option stays active, allowing to select another face of the same type and diameter for extending the thread. This allows creating a thread spanning several faces.

Defining Main Thread Parameters

Among thread parameters (in the property window or the parameters dialog box of the option), select the thread standard: one of the standards provided in the database, or non-standard. An input box next to the combo box of the thread the standards allows defining the thread standard, using a numerical value or a variable (this number corresponds to the record number of the thread in the database).

The parameter **Side** defines the thread property being outer or inner. One of the three choices can be selected from the combo box:



Current. The side of the thread is determined by the type of the base face at the time of the thread creation. If the type of the base face changes to the opposite later in modifications (for example, as a result of a Boolean operation), the thread will be deleted as inappropriate for the face.

Outer. The thread is considered outer (regardless of the type of the base face). When using automatic setting of parameters, the face diameter is considered the outer diameter of the thread.

Inner. The thread is considered inner (regardless of the type of the base face). When using automatic setting of parameters, the face diameter is considered the inner diameter of the thread.

The preferred setting for most threads is “Current”. The choices “Inner” and “Outer” should be used on faces, whose type is expected to flip later. For example, when creating fragments of holes

that are subtracted from the base body when assembled, the "Inner" thread is created, even though it is the outer face in the fragment.

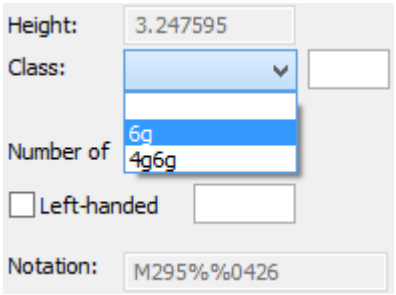
When selecting the nonstandard thread type, all its parameters, including the notation on the drawing, are always defined by the user manually by numerical values or by variables.

For standard threads, Autoadjust mode is turned on by default (the flag **Autoadjust**). In this case, the thread parameters are set according to the diameter of the base face. The nearest to the actual diameter value is picked from the set of diameters of the specified thread. Should two satisfactory values be found, the one is picked with the higher priority according to the standard guidelines. Then, the most suitable pitch is picked among the ones corresponding to the chosen diameter (for a metric thread, this will be the large pitch, for other threads – the one recommended by the respective standard).

The Autoadjust mode is not limited to the thread creation. When modifying the diameter of the base face with the Autoadjust mode turned on, the thread parameters will be adjusting automatically as well.

The selected diameter and pitch are displayed in the respective boxes of the parameters dialog (**Diameter**, **Pitch**). The box **Height** reflects the height of the thread working profile for the given pitch. If necessary, you can select another pitch from the system-offered list. In this case, when performing automatic adjustment of the face diameter modifications, the nearest to the manually defined pitch will be set, rather than that recommended by the standard.

When the Autoadjust mode turned off, you can specify arbitrary parameter values or variable names for the standard thread parameters in the parameters dialog box. Nevertheless, even in this case, the thread parameters are taken from the database. The values are selected nearest to those input by the user. Such parameters do not change with modifications of the base face diameter.



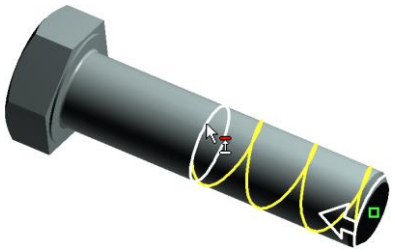
The additional "Class" field allows defining a tolerance or a precision class for the thread (in some standards). The value of this parameter can be selected from the pull down list, if the respective tolerance or class of precision values were defined in the thread database. The contents of such list are based on the values entered in the database, respective to the thread type and standard.

The field **Number of threads** defines the number of thread sets.

The threads are created right-handed by default. If you need to create a left-handed thread, turn on the flag **Left-handed**. The input box on the right of the flag allows controlling its state by using a numerical variable (the value "1" means the flag is on, "0" – off).

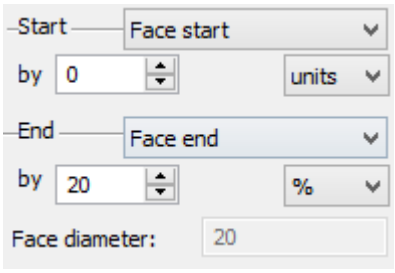
The **Notation** field displays the thread notation that will be used on a projection.

Defining Offsets



By default, the thread is applied to the whole extent of the selected face. That means, zero offsets from the face ends are used. If necessary, you can specify a different offset value or modify the way of defining the offsets – with respect to the face ends, relative to an arbitrary 3D object, etc.

The offset values can be defined either by a numerical value in the property window or by hand in the 3D view window. In the latter case,



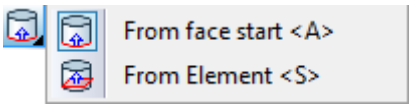
move the pointer to the dragger defining the position of one of the thread ends. The pointer will change to (at the start of the thread) or (at the end of the thread). After that, you can define the new position of the respective thread end by dragging the mouse with the depressed . The property window will be reflecting the current offset value. If necessary, it can be adjusted by entering a new value.

The way of defining the thread offsets can be modified either in the parameters dialog box or by using the automenu options:

	<A>	Select thread start type
	<D>	Select thread end type

Each of the options contains an enclosed list of choices for defining the thread start and end, respectively.

To define the start boundary of the thread, you can select:

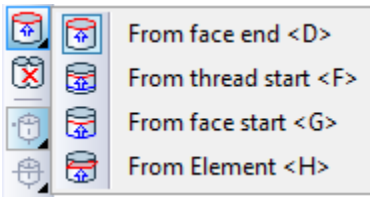


From face start. This is the default setting. The offset for the thread start is counted from the start boundary of the base face. The start is considered the end of the face nearest to the pointer position of the

time of selecting the face.

From object. The thread start offset will be counted from the additionally selected 3D object. Upon selecting this option, you need to specify the desired object.

Define the end boundary of the thread as:



From face end. This is the default setting. The offset for the thread start is counted from the end boundary of the base face.

From thread start. The offset is counted from the start of the thread, thus defining the actual thread length.

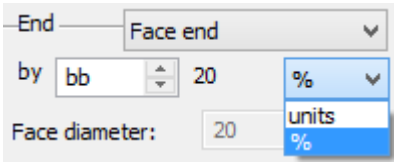
From face start. The offset for the thread end is counted from the start boundary of the base face.

From object. The offset for the thread end is counted from an additionally selected 3D object. Upon selecting this option, you need to specify the 3D object.

The mode of defining **From object** can only be set by the automenu options.

In the bolt example, one of the thread parameters is its length, counted from the end of the bolt stem. Often, the thread is applied to a face already trimmed by a chamfer. To account for the chamfer when defining the thread length, use the option "From object" for the start boundary and select the end face of the bolt stem or an element belonging to this face (such as a 3D node) as the object. The end boundary of the thread can be defined in this case as "From thread start", entering the value of the exact thread length as the offset.

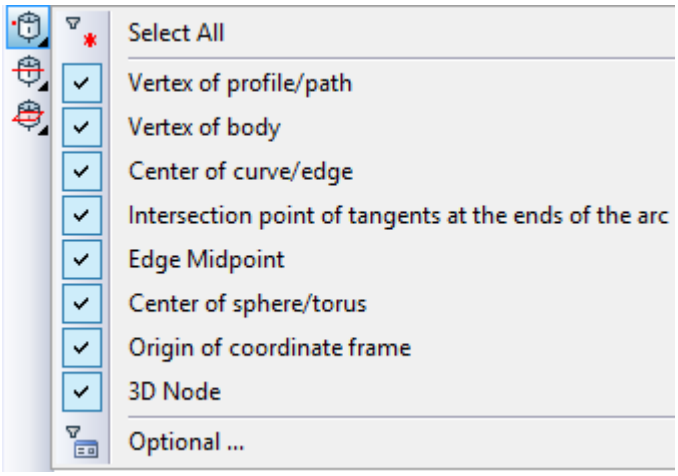
The amount of default offsets is defined among the model units (the command **ST: Set Document Parameters**, the tab "3D"). It is possible to define the offsets in percent of the face length or of the distance between the start of the thread and the end of the face (if using the option "From thread start" for the end boundary of the thread).



When selecting any of the thread boundaries in the option "From object", the options for selecting 3D objects become accessible in the automenu:

	<V>	Select points
	<L>	Select lines
	<U>	Select planes

You can select any 3D elements as the object, that define a point, line or plane. Any of those options contains a list of filters for selecting 3D elements, that tune the selector for picking the desired geometry type. Use of those filters is described in details in the chapter "Basic 3D Terms and Concepts of Modeling with T-FLEX CAD".



To select a different object for defining the position of the thread boundary, simply repeat the selection. You can cancel relations between thread boundaries and selected objects by modifying the thread offset option in the automenu or in the property window.

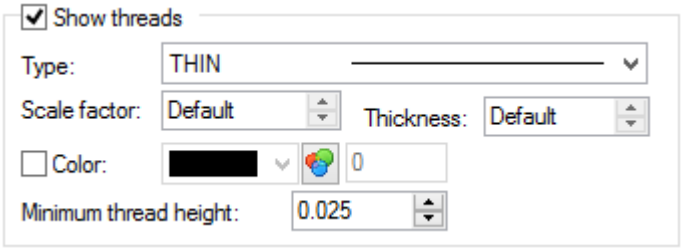
To cancel the selection of the base face and the reference objects for the thread boundaries in one move, use the automenu option:

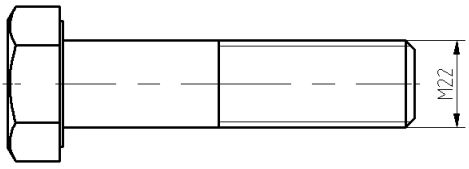
	<R>	Cancel selection of all Faces

Complete the thread creation by pressing the option . The thread will then be schematically displayed on the selected face.

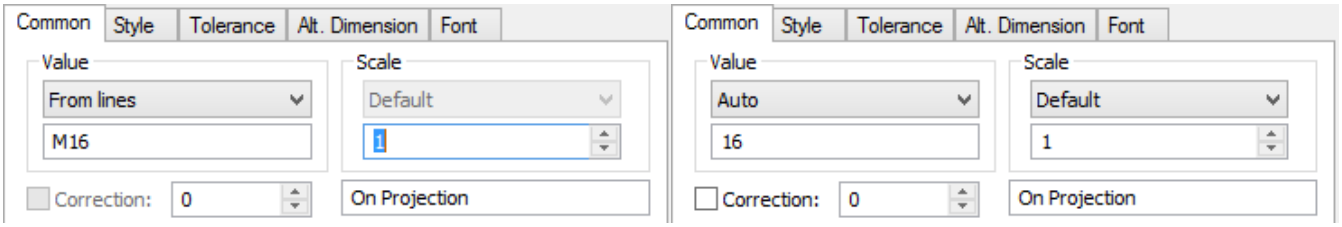
Thread Display on 2D Projections

By default, the thread-marking lines are automatically created on 2D projections of bodies with threads. The line display is controlled via the projection properties, the tab "Lines".

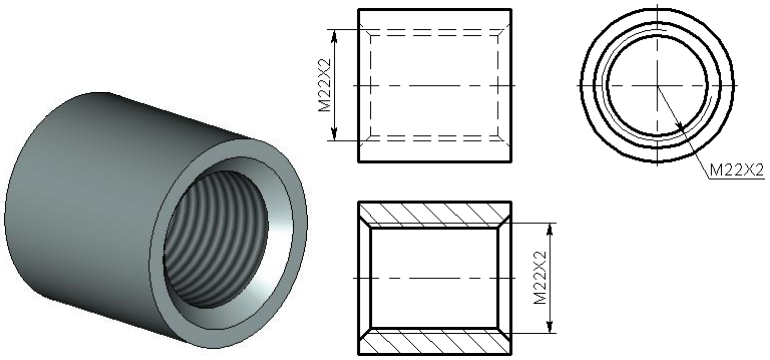




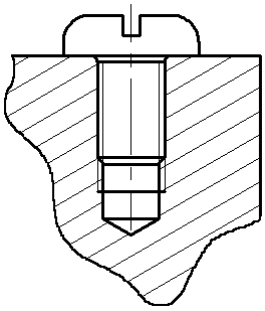
When dimensioning the thread lines, the thread notations are applied automatically. To cancel this mode (of creating a normal dimension), set parameter "Value" to "Auto". By default the value is set to "From lines" for dimensioning the thread lines.



Hidden threads can be displayed on 2D projections by dashed lines, when the “show hidden lines” mode is turned on. When creating a section or a local view, the thread is displayed in the usual way. In those cases, thread notations can be applied automatically when dimensioning thread lines.



Sections of the threaded connections require setting of the “Fill Element Outline” flag in 2D Projection parameters (see more details in “2D Projections. Creating 2D Drawings of the 3D Models” chapter). When flag is set, lines of the section view will be constructed correctly. Otherwise projection lines of bolts and holes will overlap each other due to interpenetration.



HOLES

The operation "Hole" helps creating standard holes of various types. The operation capabilities span from creating isolated holes in a single body to cutting holes or their arrays through multiple separate bodies. When threaded holes are created, cosmetic threads are applied automatically.

MAIN CONCEPTS AND OPERATION CAPABILITIES

The operation "Hole" creates a set of holes. All holes in a set will be of the same type, size and depth. A set may consist of one or more holes.

For each hole in a set, you specify an *attachment point*, in the body face where creating the hole. The face hosting the attachment point is called *base* face of the hole.

Base faces of the holes in one set (that is, holes belonging to the same operation instance) can be same or different, or even belong to different bodies.

> A set of holes on several bodies yields a single multi-body operation. To use each body separately in the future, apply Divide operation after creating the holes.

Holes in one set can have different orientation. Hole orientation means orientation of the hole axis in the space and the hole rotation angle about its axis.

The fixing point and orientation of any element in a set can be modified at the operation creation or editing. Any hole can be deleted from a set.

Hole Templates

Two mechanisms are used for holes creation: primitives or fragments (templates).

Primitives – a mechanism that creates a special operation to define the shape of the hole. No external objects are used. This mechanism is used to create holes of the following types: *square*, *for fasteners* and *oblong*.

When using a template mechanism, hole creation is realized as an automated process of inserting a special 3D fragment – a hole template.

A template has a set of parameters that define the type and dimensions of a hole created with its help.

Hole templates are stored in the parametric library *"Hole Features"*, among "Utilities". If necessary, the user can add custom elements to this library.

> If inch units are set in the command **ST: Set Document Parameters**, on the tab "3D", then the library *"Hole Features Inch"* is used for creating holes.

The standard library includes hole templates of the following types:

Holes for check screws (Type 1, 2, 3);

Threaded;

Center hole (shape A, B, C, E, F, H, R, T).

To add custom holes to the library, create a template for the new hole type in a separate document, following certain rules, and add it to the respective library ("*Hole Features*" for those using metric units, "*Hole Features Inch*" - inches).

Hole Attachment Point

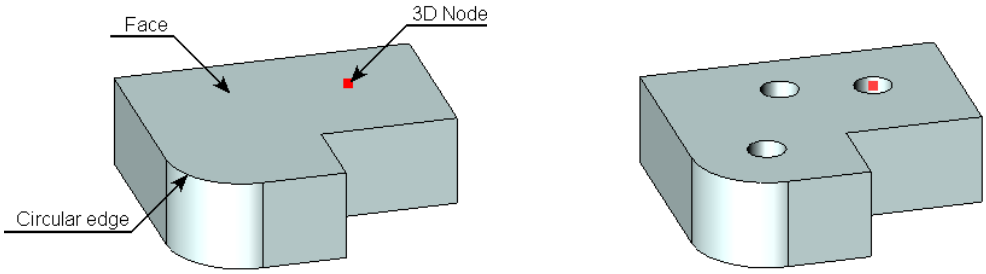
To specify position of holes being created, specify an attachment point for each hole, one at a time. One hole is associated with one point. The base face of each hole is defined by the system automatically. By default, the axes of the holes are normal to the respective base faces.

Attachment points of the holes can be defined by one of the following 3D elements:

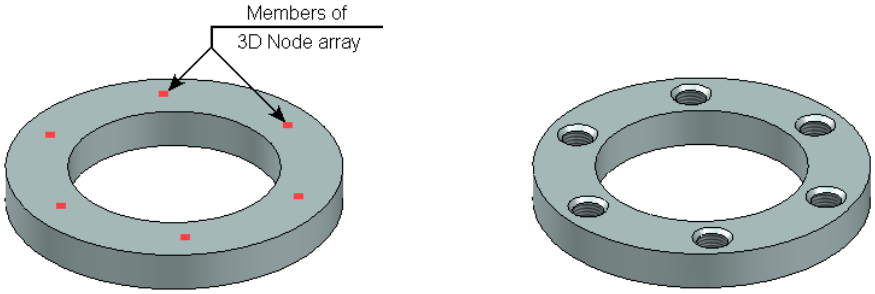
Face of a body. The specified face becomes the base face of the hole; the hole center will be positioned in the location of the mouse click that selected the face;

3D node belonging to one of the body's faces. The face of the node will be set as the base face; the hole center will be at the node;

Circular edge belonging to one of the body's faces. The face of the circular edge's plane becomes the base; the hole center snaps to the center of the edge's underlying circle.

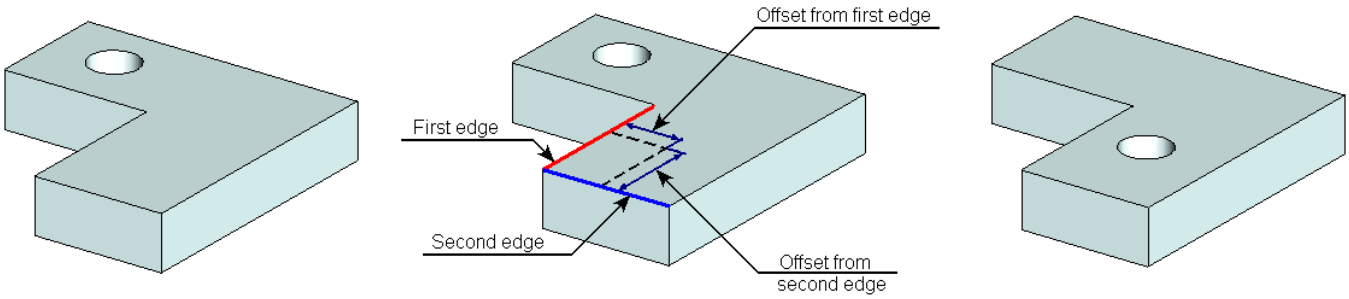


To quickly define a set of holes, you can select an array of 3D nodes for the attachment points. Holes will be automatically created at each 3D node of the array.



The attachment point of any hole in a set can be modified by one of the following means:

1. Define a totally new attachment point (new base face or new attachment 3D element for the hole center);
2. Modify the hole position on the current face (maintaining same base face) – by defining *offsets* of the hole center from the two side edges of the base face. The hole center will be attached to the point on the base face, defined by the specified offsets.



Original hole position | Offset from side edges | New hole position

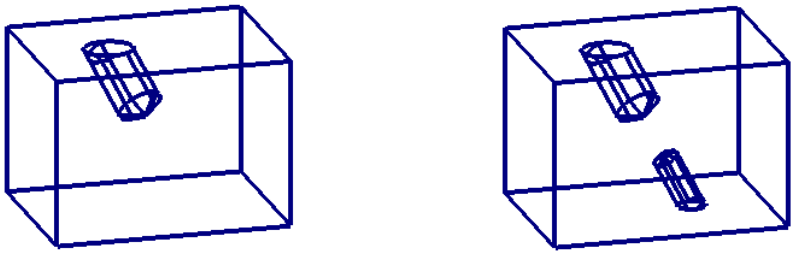
3. Create coaxial hole with automatic adjustment of the hole position on the base face (see the topic "Coaxial holes").

Coaxial Holes

Any hole can be created as coaxial (using the same axis) with an arbitrary cylindrical or conic face. A hole can also be created coaxial with an existing round hole.

The location of the base face should permit creating a coaxial hole. For example, when creating a hole coaxial to another hole, the two holes may not be placed on the same face.

The axis directions of the new hole will coincide with that of the source face.



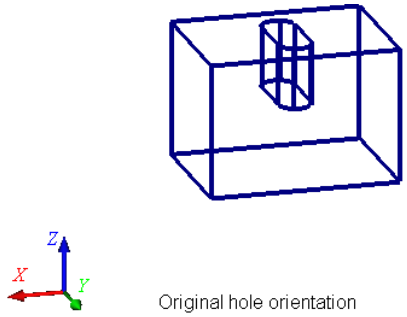
Source hole | Coaxial hole

Hole Orientation

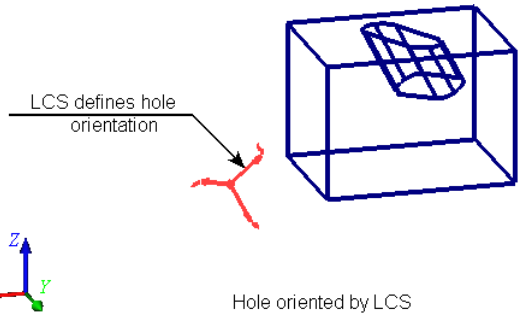
By default, all new created holes are placed by the system in such a way, that the axis of each hole (X-axis of the hole template coordinate system) is directed along the normal to the base face at the attachment point. The orientation of the hole cross section is set by the system arbitrarily. (This is the angle of the hole cross section rotation about the Y-axis of the hole coordinates.)

If necessary, you can modify orientation of each hole in a set separately. A hole's orientation can be modified in one of the two ways:

By auxiliary LCS – a specified LCS is used for defining the direction of the axes of the coordinate system of the hole (existing in its template). The hole is rotated in such a way that its coordinate system axes all get aligned with the direction of the respective axes of the selected LCS;



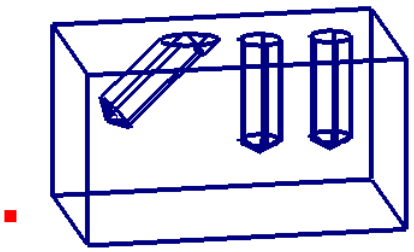
Original hole orientation



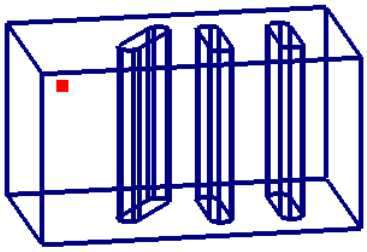
Hole oriented by LCS

By 3D nodes. In this case, the hole's orientation and the angle of rotation about its axis are defined separately. You can modify just one of these parameters, for example, only rotation of the hole about the axis, in order to turn a non-round hole about its axis. In this case, the axis itself can be left unchanged (in the system default position).

When using a 3D node for orienting the hole axis, the axis will be directed from the hole attachment point to the 3D node. When specifying rotation of the hole by a 3D node, the Y-axis of the hole coordinate system will be turned so as to point at the selected node.

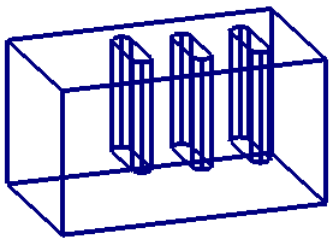


The axes of one of the holes is oriented by a 3D Node

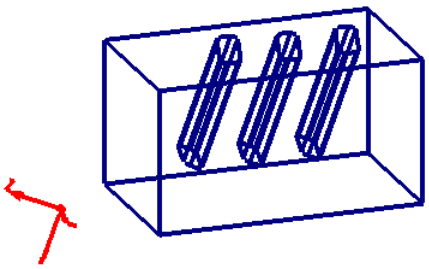


The rotation angle above the axis of one of the holes is defined by a 3D Node

When orienting by an LCS, you can do this to a whole set of holes simultaneously.



Original position of a set of holes



All holes in a set are reoriented by an LCS

Hole Depth

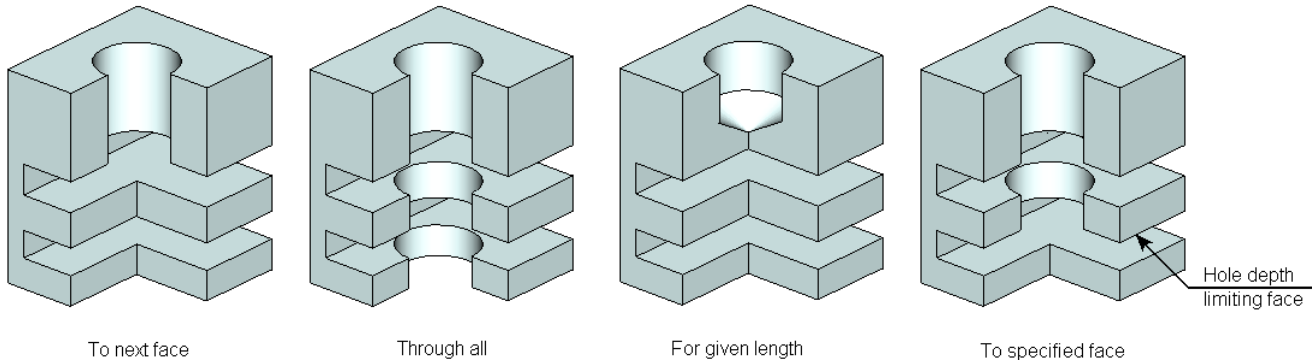
Depth of a set of holes can be specified by a numerical value or calculated automatically, depending on the current mode:

To next face. Holes are cut up to the intersection with the next face.

Through all. Holes are cut throughout the selected body (a set of bodies).

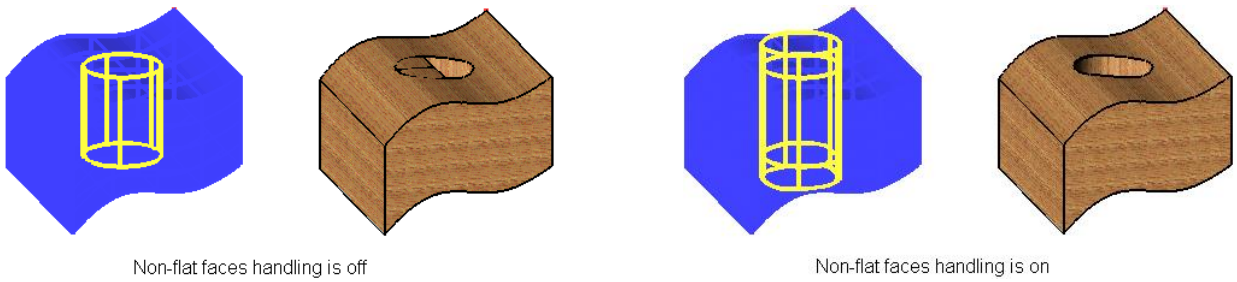
For given length (depth). The depth of the holes is defined by a numerical value.

To specified face. Holes are cut up to the specified face. (The selected face must intersect with the holes being created.)



The holes for check screws and center holes of the shape A, B, C, E, R, T permit the option "For given length" only.

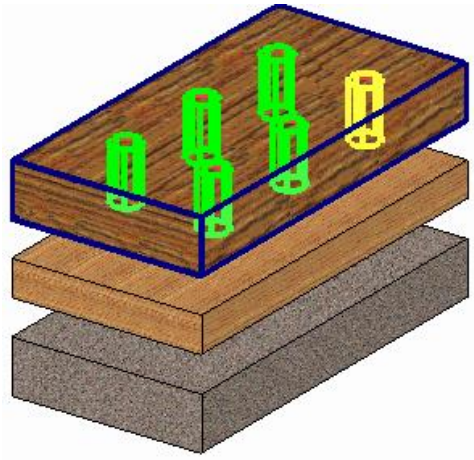
If a hole axis is not perpendicular to the intersected faces of the body, or if faces intersected by a hole are not flat, additional manipulations may be required to get a correct hole entry. In such a case, we recommend using the mode of *considering (handling) non-planar faces*. In this mode, holes are extended by an allowance amount at either side, and then trimmed by body faces.



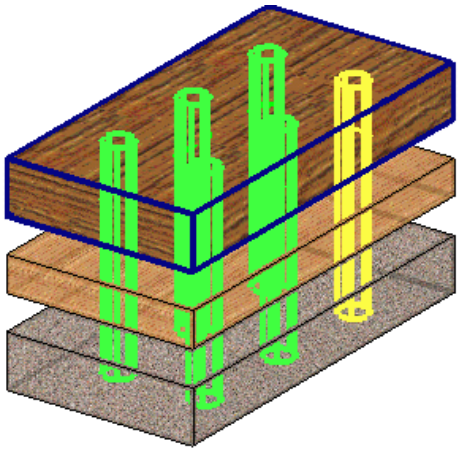
Holes through Multiple Bodies

Holes can be cut through several solid bodies at once. Such sets of holes create multi-body operations.

To use each model body separately in the future, use Divide operation after creating the holes.



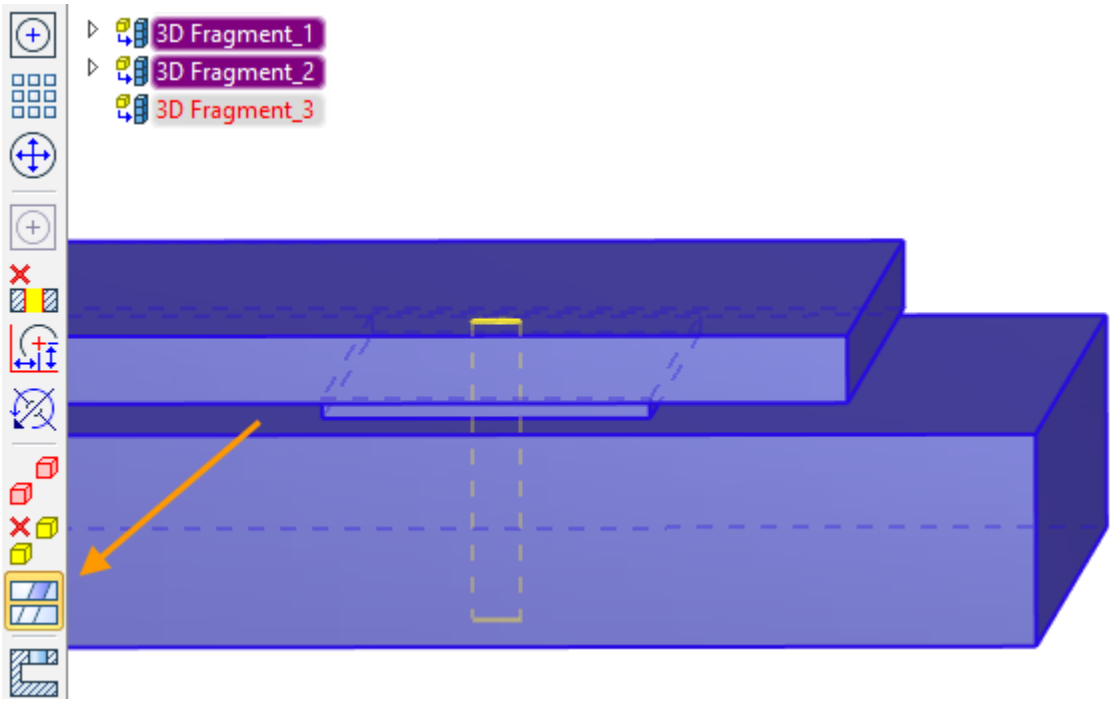
A set of holes created through one body



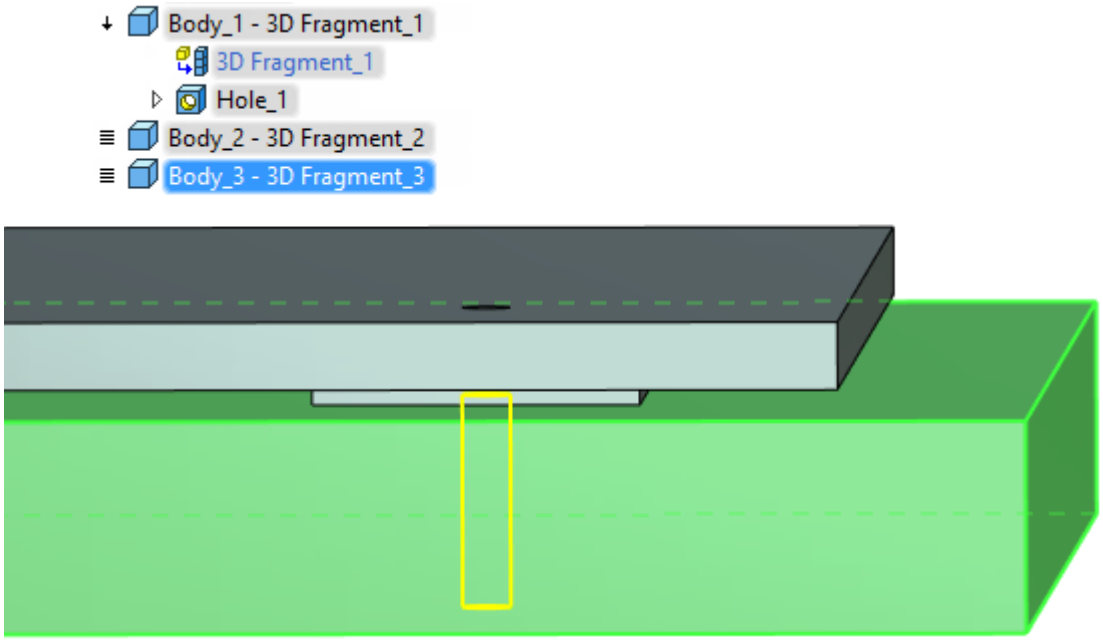
A set of holes through multiple bodies

You can create coaxial holes in several bodies or fragments and such holes will not be united in one resulting body. Use the **Create coaxial holes** option for this purpose.

	<H>	Create coaxial holes



A separate hole will be created for each of the bodies.



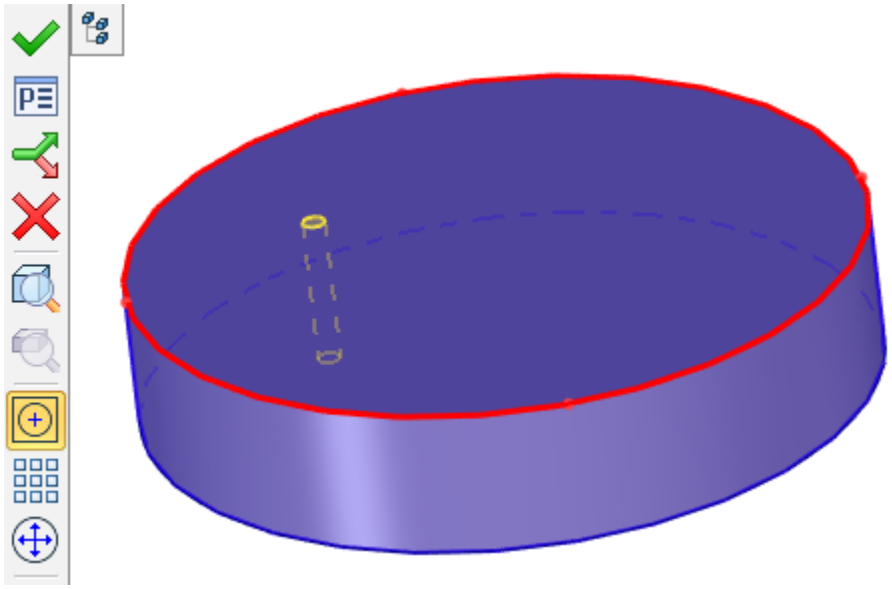
Parameterization is via the common variables. Thus if you specify hole parameters using variables, they will be retained for each of the holes.

Create Holes With Specifies Direction

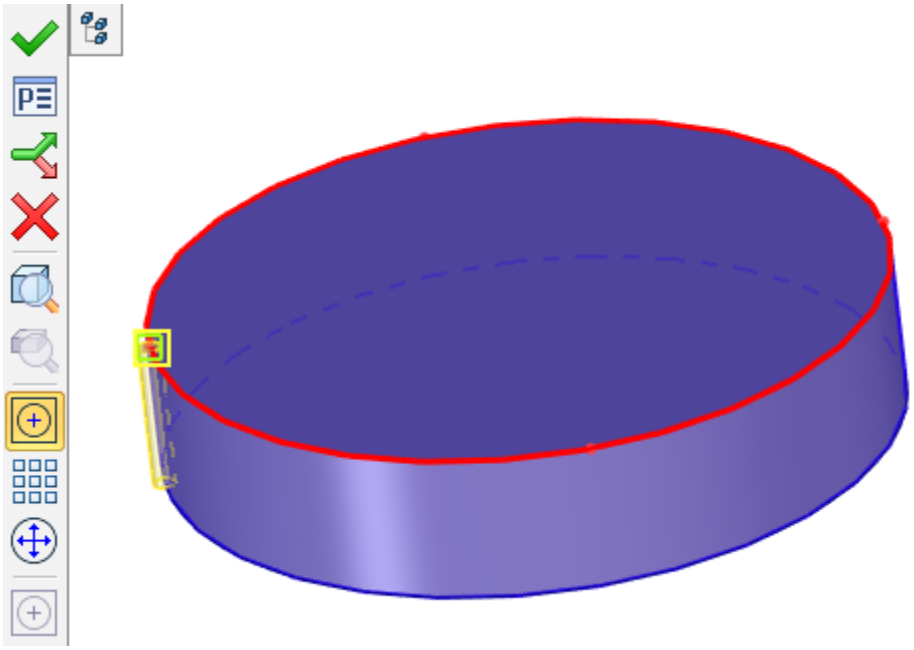
Nodes for the holes are located on the boundary of two or more faces. It is required to create holes on the nodes according to certain direction. Since the system cannot automatically determine the direction, it selects the first appropriate solution. However, the user can independently set the direction by performing the following sequence of actions.

In this example, we will build an array of holes using the nodes created on the edge of the body. The nodes are already created.

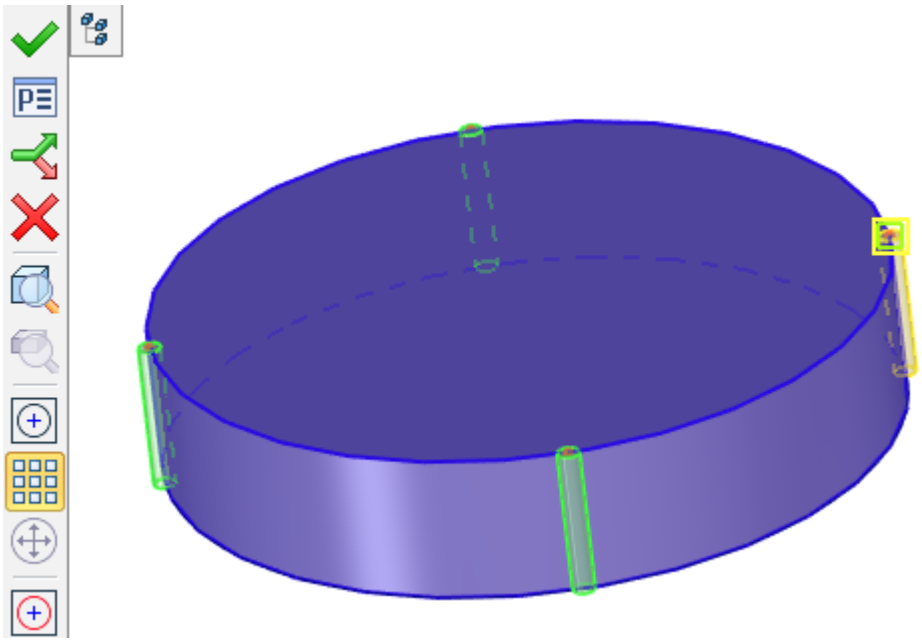
Step 1. Select the face for the hole. The face will define the direction for the hole.



Step 2 – Select **A: Enter hole center** command and select one of the nodes for which holes should be created. The system will construct holes according to the selected face.



Step 3 - Select the rest of the array.



HOLE CREATION

To create standard holes, use the command **3H: Create Hole**:

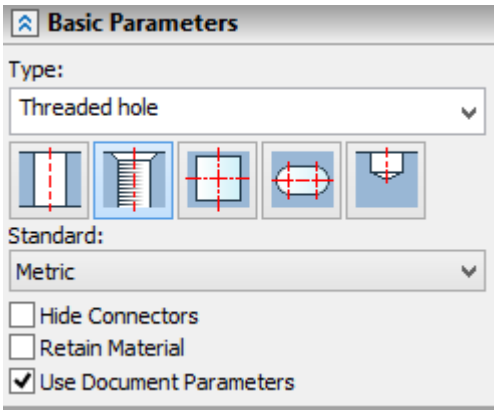
Icon	Ribbon
	3D Model → Modify → Hole
Keyboard	Textual Menu
<3H>	Operation >Hole

To create a hole, do the following steps (the order of steps may vary):

- ✓ Select the type and main geometrical parameters of the set of holes to be created;
- ✓ Define attachment points for the holes;
- ✓ Specify the desired hole depth;
- ✓ Modify default axis orientation and rotation angles of the holes (optional);
- ✓ Finish hole creation (use in the automenu or in the property window).

Selecting Type and Main Geometrical Parameters of Holes

Select the type of the holes being created in the operation property window. The list of holes contains all elements in the hole utility library (*"Hole Features"* or *"Hole Features Inch"*).



Type of a hole can be selected either from the quick access drop-down list or by clicking the corresponding icon. If you select type from the list and it is not presented among icons below, the new icon will replace the last right icon.

Item "Type" in the drop-down list allows you to select element from custom libraries added to selection list.

The content of the selection list is linked with the value of the **Standard** field and will update accordingly.

You can select holes library using option:

	<B>	Library configuration

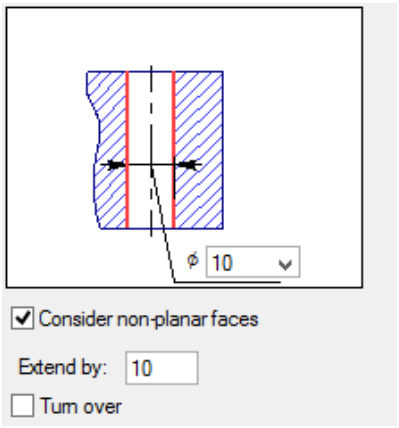
More information about configurations can be found in section "Library configuration".

A schematic chart of a hole of the selected type is provided in the lower portion of the property window, with input boxes for all its geometrical parameters. To specify any dimension, just placed the cursor in the respective input box in the chart and type in the desired value (or select one from the list). Variables may be used as values. In this case the value of parameter will be assigned to the nearest value from the appropriate list.

Hide Connectors. Flag allows hiding connectors of created holes.

Retain material. When active, body's basic material will be applied to the created hole faces. Otherwise, the coating material will be used.

Use Document Parameters. The option is used for holes created using the fragments mechanism. If this option is enabled, the hole uses document parameters, for example, the units. If this option is disabled, the hole will use parameters from the fragment.



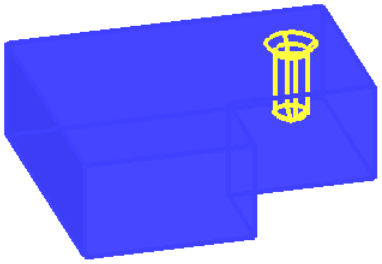
Reach-through asymmetric hole (e.g. with one side chamfer) may be turned upside-down by its axis direction with option "Turn over". Often this flag is used for mold-base design. Note that this flag will affect all holes from the set being created.

The initially defined type and dimensions of holes can be modified at any time while creating or editing holes (before the confirmation by pressing).

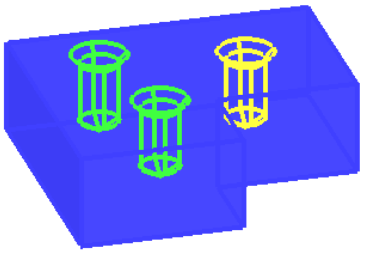
Defining Attachment Points of Holes

Hole attachment points can be defined upon activating the automenu option (the option is active by default upon entering the command):

	<A>	Enter hole center



With the option active, simply select the desired face, 3D node or a circular edge on a face in the 3D window. The parent body of the selected element is highlighted, and a wireframe preview of the hole being created is displayed.



After defining the attachment point for the first hole, the option stays active. This helps creating a set of holes without additional actions. Just keep picking attachment points after selecting the first one.

As a result of defining a number of attachment points, a preview of a set of holes is displayed in the 3D scene. The type and geometrical parameters of all elements are the same.

Creation of a set of holes from an array of 3D nodes is done with the option:

	<M>	Add Hole Array

Upon activating the option, specify the source array of nodes. You can select an array either in the 3D model tree or directly in the 3D scene by clicking any array element with .

Notice! When you create a holes array, you need to set maximum number of the array elements. If this rule is not met, offset or disappearance of new holes in array can occur after the holes number increase.

Defining Hole Depth

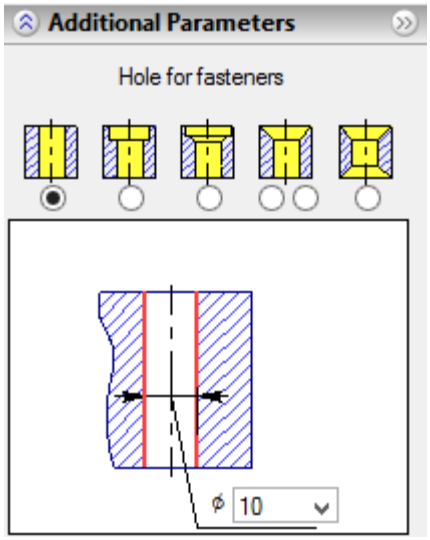
The desired mode of defining the hole depth is controlled by the following automenu options of the command:

	<E>	To next face
	<F>	Through all
	<J>	For given length
	<S>	To specified face

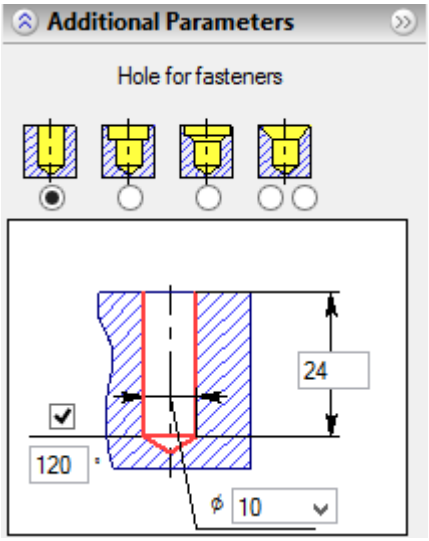
For some types of holes, certain options may not be available.

Selecting the first two modes does not require any additional actions. The depth of the holes will be determined by the system automatically.

When in the mode , the hole chart presents additional items for defining the hole depth.



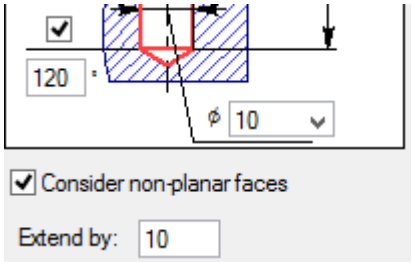
Appearance of property window in the case of "Hole for fasteners" hole type when defining depth "Through all", "To specified face", "To next face"



Appearance of property window in the case of "Hole for fasteners" hole type when defining depth "For given length"

In the last mode (the option), you need to additionally select a face in the 3D scene, up to which the holes will be cut. The selected face will be highlighted (by default - *red*).

The additional mode of handling non-flat faces is turned on by the flag **Consider non-planar faces** in the lower portion of the operation property window. Setting the flag brings up an additional item in the property window, **Extend by**, where the amount of allowance for the hole extension is to be input.



Creating Holes through Multiple Bodies Simultaneously

Additional bodies can be selected by the option:

	<K>	Select additional bodies

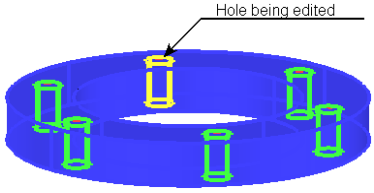
The bodies can be selected either in the model tree or directly in the 3D scene.

Selection of bodies can be canceled by the option:

	<L>	Cancel additional bodies selection

Hole Selection for Modifying Hole Position and Orientation

Attachment points and orientation can be modified separately for each hole in a set being created. The hole currently available for editing is highlighted in the 3D scene. The default highlighting color is *yellow*. Other holes in the set are highlighted *green*.



Normally, the last created hole is instantly made available for editing. A different hole in the set can be selected by the option:

	<D>	Select Hole for edit

Upon activating this option, simply point the mouse at one of the created holes and click . The selected hole will be highlighted.

The selected hole can be deleted by the option:

	<X>	Delete selected Hole

Modifying Hole Attachment Point

To define a new attachment point from scratch, use the option:

	<R>	Change Hole position

Upon calling this option, select a new attachment point for the hole being edited.

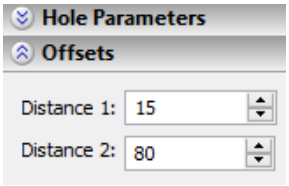
A hole position can be modified within the current base face by the option:

	<C>	Set offsets for Hole

Upon activating this option, more options will appear in the automenu for selecting edges to offset from and for defining the offset itself:

	<D>	Select Edge that sets the first offset
	<F>	Select Edge that sets the second offset
	<G>	Set Hole offset

To define offsets, subsequently select two side edges of the base face in the 3D scene by the options and . The offset values can be defined by simply clicking the mouse at the desired position of the hole center on the face (with the option active) or by entering exact numerical offset values in the operation property window.



To quit the mode of defining offsets, use the option:

	<R>	Cancel offset definition for Hole

Modifying Hole Orientation. Creating Coaxial Hole

To reorient a hole, use the following option in the command automenu:

	<T>	Set Hole direction

Upon activating this option, additional options appear in the automenu for defining new orientation of the hole being edited.

A hole can be oriented by an LCS by the option:

	<L>	Select LCS to define Hole direction

Upon calling the option, specify the desired LCS. The hole being edited will be oriented by the axes of the selected coordinate system.

A hole axis can be oriented by a 3D node with the help of the option:

	<F>	Select 3D Node for Hole direction

Rotation of a hole about the axis in its current position is done by the option:

	<G>	Select 3D Node for Hole turn angle

To quickly reorient all holes in a set, use the option:

	<A>	Select LCS to define directions of all Holes

A hole coaxial with any cylindrical/conic face is created by the option:

	<O>	Select Hole to create coaxial Hole

Upon activating this option, select a face in the 3D scene, whose axis is to be used for creating the coaxial hole.

To undo a defined orientation, use the following options:

	<D>	Cancel Hole turning
	<Q>	Cancel turning for all Holes

To quit the mode of defining offsets, use the option:

	<R>	Return from setting Hole direction

Library configuration

Library Configuration option displays structure of libraries that are used in the current operation. The option allows to select fragments that will be added to the selection drop-down list. Also it allows to add custom libraries that contain additional fragments.

Configurations window appears when you call the option with the tree that contain the following configurations folders:

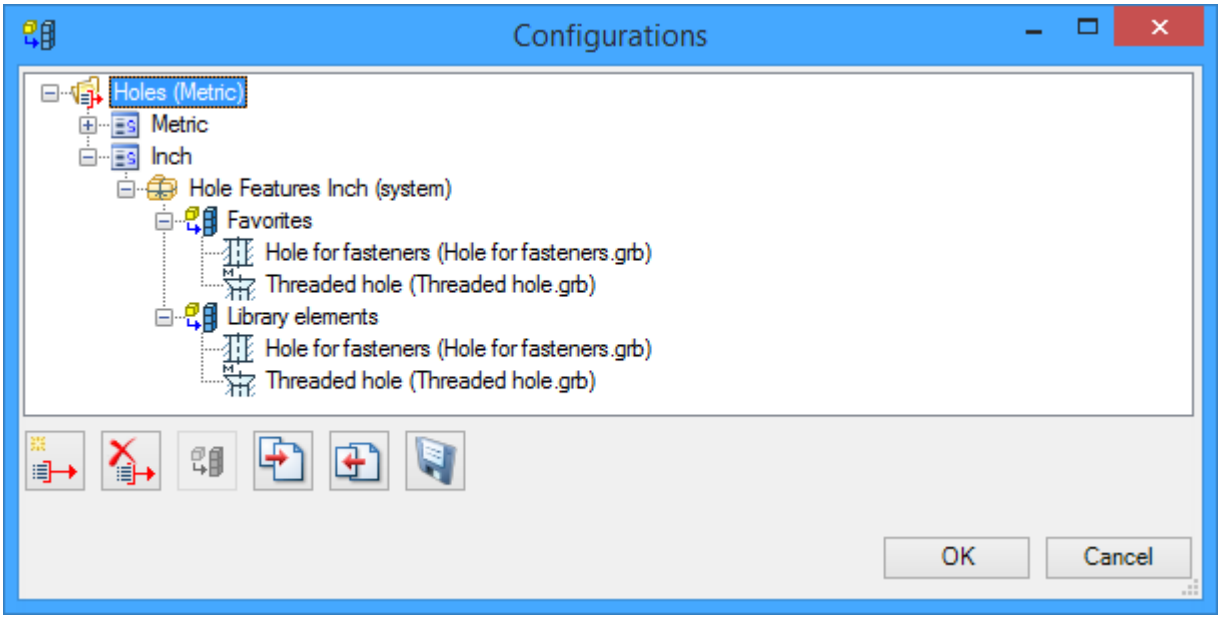
Configuration name. Corresponds to the current operation that we configure.

Standard. Defines subset of libraries that will be linked to this standard.

Library. Contains files of fragments that will be added to the selection list.

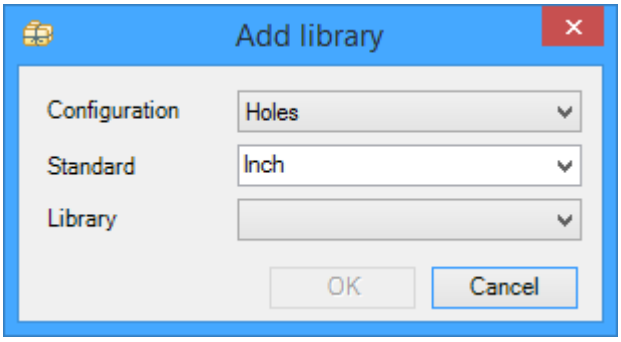
There are “Favorites” and “Library elements” groups inside each library.

All fragment files that exist in the selected library are displayed in its “Library elements” group. The fragments that are included in the quick access drop-down list are displayed in the “Favorites” group.



New library. **Add Library** window appears when you call the option.

You need to select configuration, standard and the library that will be added. There are two standards by default: metric and inch. To add a new standard you need to enter its name in the **Standard** field. Drop-down list in the **Library** field includes all libraries, that are displayed in the **Documents Menu** window.



Delete element. Deletes selected configuration, standard or library.

Add/delete fragment as an element by default. Allows to select fragments that will be added to the quick access drop-down list and in the “Favorites” group.

Export library configuration. Allows to export configurations in external *.xml file.

Import library configuration. Allows to import configurations from external *.xml file.

Save changes. Allows to save changes in the configurations.

DEFORMATION

The deformation operations allow a user to perform modification of solid and sheet bodies by various means: skewing, bending, scaling/twisting etc.

When creating deformation operations, a function producing volume deformation of the given body is formed in the model on the basis of parameters specified by the user. This function can be defined over the entire volume of the given body or only its part. Applying this function in a continuous manner deforms the entire volume of this body (or its part). Topology of the deformed part of the body does not change. The number of faces, edges, vertices, etc. remains the same. If necessary, faces and edges of special type (planes, segments, cylinders, arcs of circles etc.) are automatically replaced with spline surfaces and curves.

Commands of deformation can be applied to both solid and sheet bodies (surfaces).

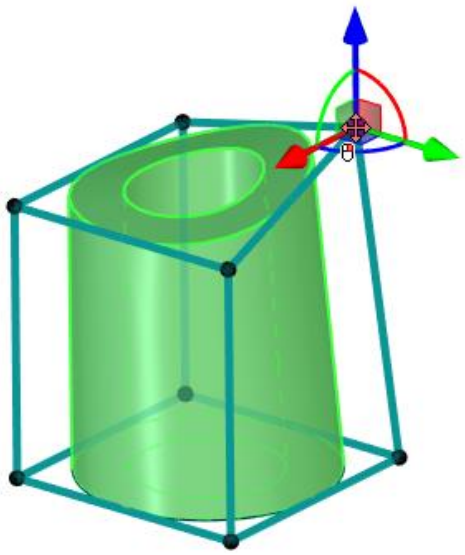
In the process of specifying all parameters of deformation operations, the future result of applying the current operation is dynamically displayed in the 3D window. To speed up the work, this dynamic view can be turned off.

SKEW

General Information

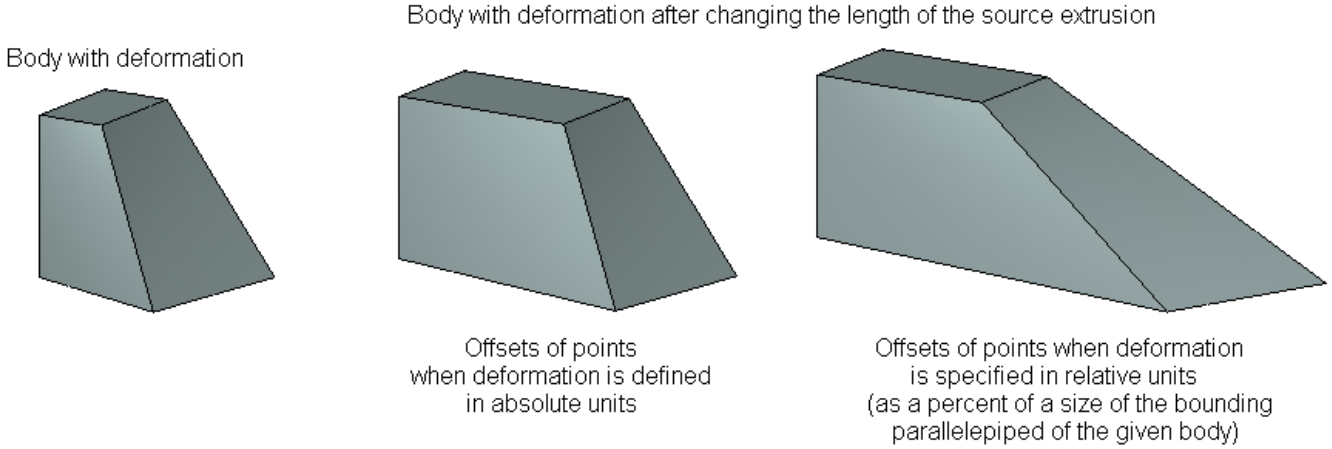
In the skew deformation, the deformation function is specified by displacing the vertices of the bounding parallelepiped of the given body. The bounding parallelepiped is calculated with respect to the axes of the global coordinate system or additionally-specified LCS.

The offset of vertices of the bounding parallelepiped can be realized along any axis of the coordinate system (the axis with respect to which the bounding parallelepiped is constructed), along the edges of the bounding parallelepiped, along the face diagonals of the bounding parallelepiped. For specifying the offsets in the command, the draggers as well as the dialog of the command's properties can be used.

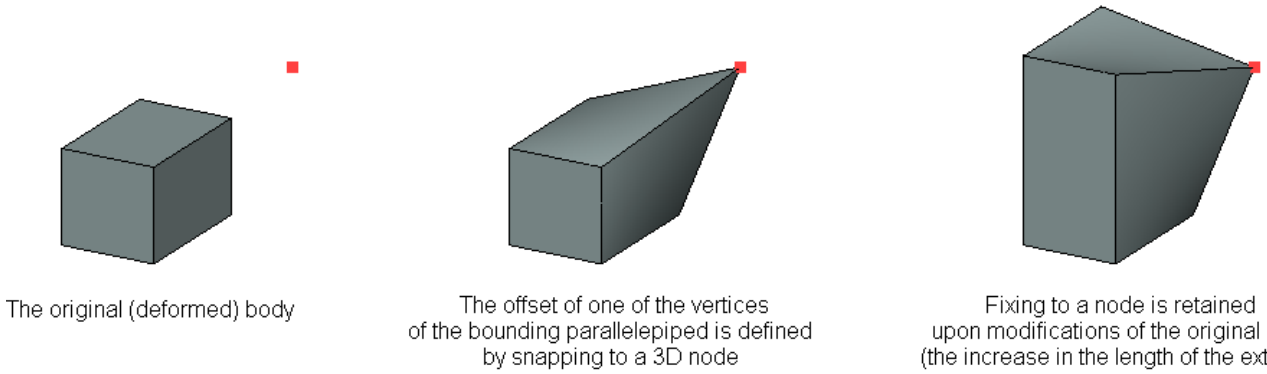


The magnitude of the vertex offset for a bounding parallelepiped can be specified both in absolute units – units of model and in relative units – in fractions of length (in percents) of the bounding parallelepiped along the respective axes. The relative coordinates can be used in cases when it is necessary to preserve proportions of the deformed model no matter what changes in the source operations are made. For example, the picture below shows how the deformed model will be modified when changing the length

of the source extrusion depending on in which units the offsets of the vertices of the bounding parallelepiped have been specified.



The vertices of the bounding parallelepiped can be snapped to the 3D points. In this case the offset does not depend on modification of the deformed body or on what units of length have been chosen for specifying the offsets.



When attaching the vertex to a 3D point, the offsets of the vertex with respect to the selected 3D point can be specified additionally.

Creation of Skew Deformation

Skew deformation is performed with the help of the command **3DRS: Skew Deformation**:

Icon	Ribbon
	3D Model → Special → Deformation → Skew Deformation
Keyboard	Textual Menu
<3DRS>	Operation > Deformation > Skew Deformation

For specifying this deformation it is necessary:

- ✓ Select a body being deformed;
- ✓ Define LCS with respect to which the bounding parallelepiped will be formed *(optional step)*;

- ✓ Specify the required offsets of the vertices of the bounding parallelepiped;
- ✓ Complete the operation creation by pressing .

Selection of Deformed Body and LCS of Deformation

For selecting the deformed body, the following option of the automenu is used:

	<O>	Select Body to Deform

This option is turned on by default upon the entry into the command. The deformed body is selected with the help of in the 3D window or in the tree of the 3D model. The selected body is highlighted, and the contour of the bounding parallelepiped appears.

By default the bounding parallelepiped is calculated with respect to the axes of the global coordinate system. It is possible to select another coordinate system for calculation of the bounding parallelepiped with the help of the option:

	<L>	Select source LSC

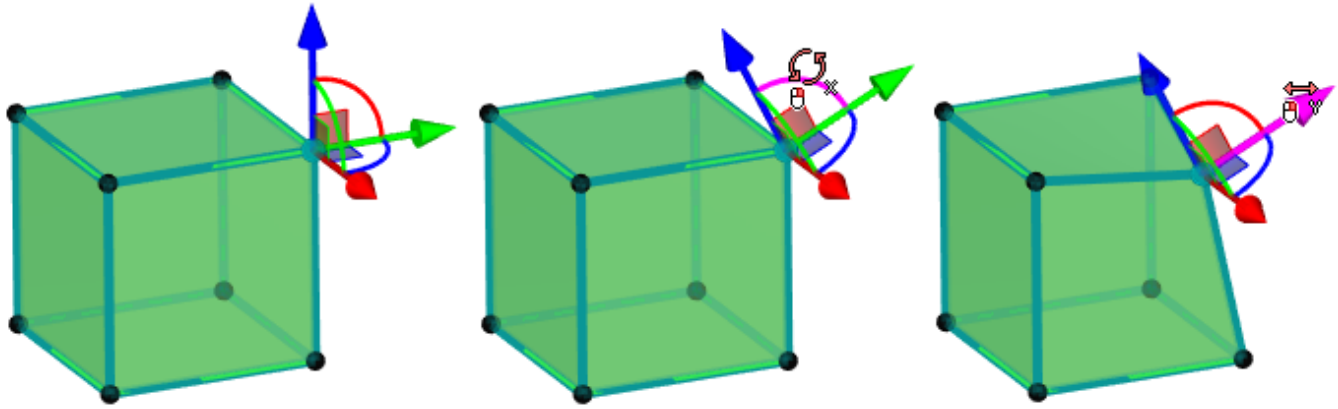
After choosing the LCS, the bounding parallelepiped will be automatically reconstructed in accordance with the axes of the LCS.

Cancellation of the selected LCS can be done with the help of the option:

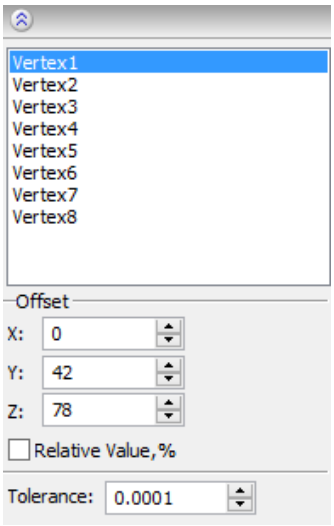
	<C>	Cancel LSC selection

Specifying Offsets for Nodes of Bounding Parallelepiped of Deformation

After constructing the bounding parallelepiped, in the 3D window a user can select the nodes of the parallelepiped in any order and specify their offsets (with respect to initial position). To specify the nodal offsets, the 3D draggers are used.



The offsets of the vertices of the bounding parallelepiped (with respect to their initial position) can be also specified in the dialog of the command's properties. After selecting the deformed body, the list of all vertices of the bounding parallelepiped will appear in the dialog. Under the list, there will be entry fields for input of the offset values (the parameters group **Offset**). To specify the offsets, it is sufficient to select the desired node-vertex from the list (it will be highlighted in the list and in the 3D window), and specify the offset values along the axes X, Y, Z in the entry fields of the group **Offset**.



Units in which the offsets of vertices are specified are controlled by the status of the flag **Relative Value, %**. By default this flag is turned off, and the specified offsets are absolute (i.e., are measured in units of model). If the flag is enabled, the specified offsets will be recalculated in relative units.

Additional parameter **Tolerance** specifies the value of the precision used in evaluations of the deformation function.

For fixing a vertex of the bounding parallelepiped to a 3D point, select the desired vertex (in the 3D window or in the list of vertices in the dialog of the command) and turn on the following option of the automenu:

	<Z>	Set Fixing to Point

After activating this option, 3D fixing point has to be selected. The drop down list of this option contains filters for selecting various 3D objects capable of defining a 3D point. The selected 3D object is specified in the list of vertices next to the name of that vertex of the bounding parallelepiped whose location is defined by the object.

For a vertex of the bounding parallelepiped fixed to a 3D point, the offsets entry fields in the properties window will remain accessible. The offsets specified in these entry fields will determine the vertex offset with respect to the 3D fixing point. The offsets with respect to a fixing point can be also specified in the 3D window with the help of the draggers.

To cancel a fixing to a 3D point, select the desired vertex (in the 3D window or in the vertices list in the dialog of the command) and invoke the following option of the automenu

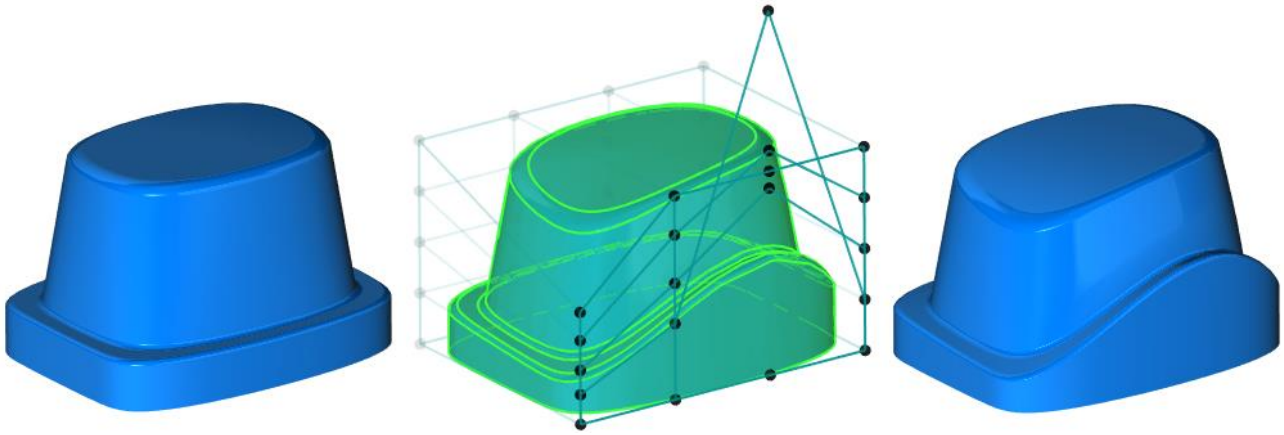
	<D>	Cancel Fixing to Point

After invoking this option, the vertex is untied from the 3D point. Note that the offset of the given vertex does not change at that. The vertex will remain in the position specified by the 3D point. However, later, when modifying the original model or location of the 3D point, the offsets of the given vertex of the bounding parallelepiped will be recalculated as if they were specified by numeric values from the beginning.

SCULPT DEFORMATION

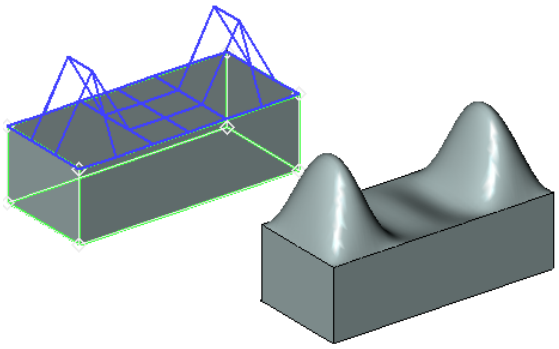
General Information

In the sculpt deformation, the deformation function is defined by displacing the nodes of the regular mesh on one of the faces of the bounding parallelepiped of the deformation area.

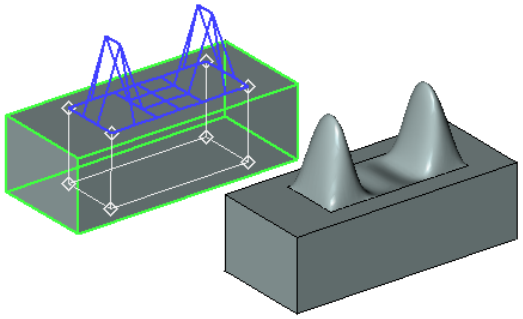


The bounding parallelepiped, as in the previously considered deformation type, can be built with respect to the axes of the global coordinate system or additionally-specified LCS.

By default, the parallelepiped of the deformation area spans the entire body. If necessary, its dimensions can be modified, thus altering the deformation area. The offset of the faces of the parallelepiped relative to their initial position can be defined by specifying both an absolute and relative value and also by snapping to a 3D point.

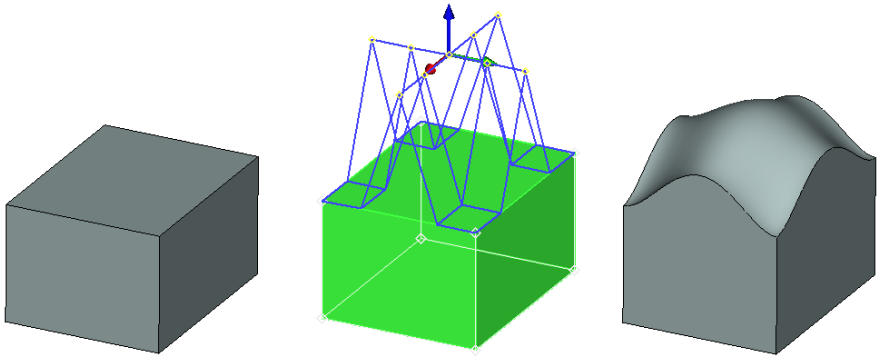


Deformation area spans the entire body

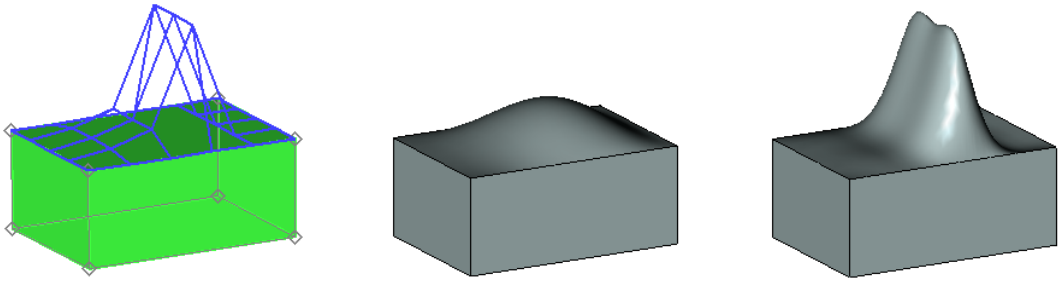


Deformation area spans only part of the body

The regular mesh is automatically created on a face of the bounding parallelepiped which is selected for creating deformation. Dimensionality of the mesh is specified by the user. Any node of the mesh can be displaced from its initial position by the user-specified value. Several nodes can be selected and moved simultaneously.



As a result of displacing the nodes of the mesh, the flat face of the bounding parallelepiped is transformed into a space spline surface which forms the required transformation law of the body. The spline surface being formed can be of two types: control polyhedron type (the nodes of the mesh are used for constructing the control polyhedron of the surface) and interpolation type (going directly through the nodes of the mesh).

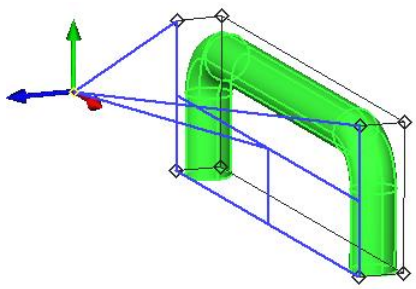


Deformation by means of creating the spline surface by control polyhedron

Deformation by means of creating the spline surface by interpolation over points

The sculpt deformation has three modes:

- **One side** – only the nodes located on the selected face of the bounding parallelepiped are displaced;
- **Both sides** – the nodes located on the selected and the opposite faces of the bounding parallelepiped are displaced. The nodes on the face opposite to the selected one are displaced in the same direction by the same distance;
- **Symmetric** – the nodes located on the selected and the opposite faces of the bounding parallelepiped are displaced. The nodes on the opposite face are displaced symmetrically with respect to the symmetry plane of the parallelepiped.

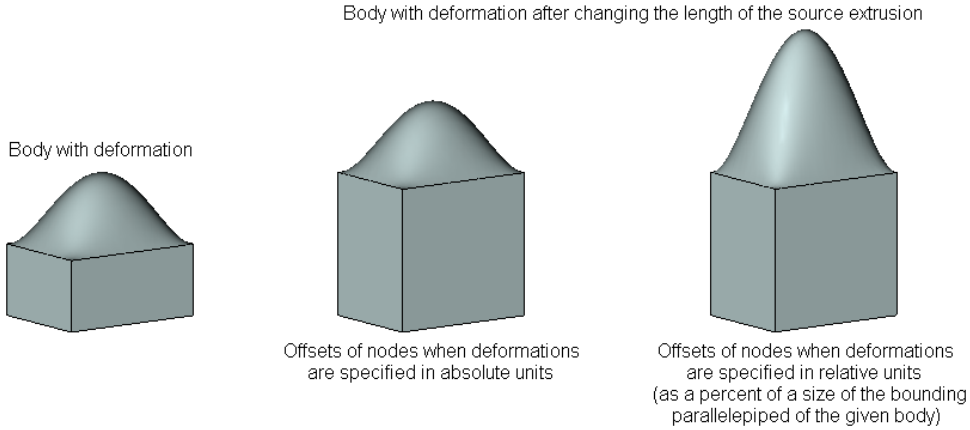


The mode of one-sided deformation

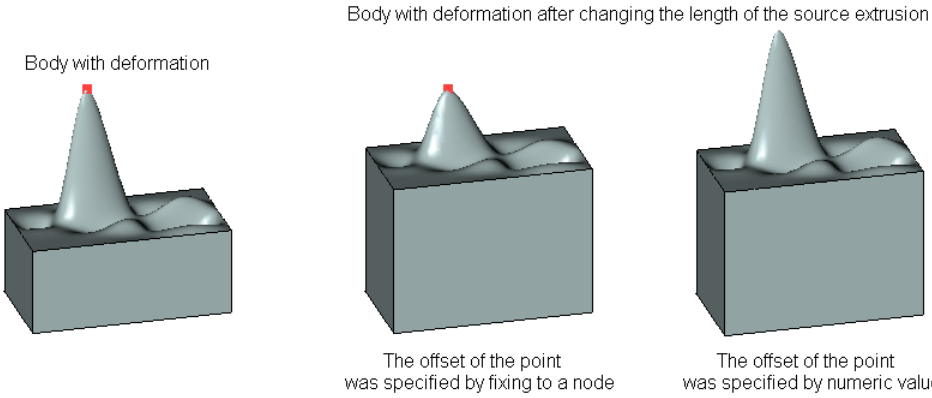
The mode of two-sided deformation

The mode of symmetric deformation

The value of the mesh points offsets from their initial position can be specified both in absolute units – units of model and in relative units – in fraction of length (in percents) of the bounding parallelepiped along the respective axes. The relative coordinates can be used in cases when it is necessary to preserve proportions of the deformed model no matter what changes in the source operations are made. The picture below shows how the deformed model will be modified when changing the length of the source extrusion depending on in which units the offsets of the mesh points have been specified.



A snap to 3D points can be also specified for the nodes of the mesh. In this case the offset is determined by the location of the 3D snap point and does not depend on modification of the deformed body or on units in which the offsets are specified.



When fixing the node of the mesh to a 3D point, the node's offsets with respect to the selected 3D point can be specified additionally.

Creation of Sculpt Deformation

For creating this type of deformation, the command **3DRC: Sculpt Deformation** is used:

Icon	Ribbon
	3D Model → Special → Deformation → Sculpt Deformation
Keyboard	Textual Menu
<3DRC>	Operation > Deformation > Sculpt Deformation

After invoking this command, a user has to:

- ✓ Select the deformed body;
- ✓ Define LCS with respect to which the bounding parallelepiped will be formed *(optional step)*;
- ✓ Define the deformed face of the bounding parallelepiped;
- ✓ Specify the boundaries of the deformation area (of the bounding parallelepiped of the deformation area) *(optional step)*;
- ✓ Specify mesh parameters on the selected face *(optional step)*;
- ✓ Specify the mode of deformation: *one-sided*, *symmetric* or *two-sided* deformation *(optional step)*;
- ✓ Define the desired displacements for the nodes of the mesh;
- ✓ Complete the operation creation by pressing .

Selection of Deformed Body and LCS for Deformation Area Construction

For selecting the deformed body, the following option of the automenu is used:

	<O>	Select Body to Deform

This option is turned on by default upon the entry into the command. The deformed body is selected with the help of in the 3D window or in the tree of the 3D model. The selected body is highlighted, and the contour of the bounding parallelepiped appears on the screen.

By default the bounding parallelepiped is calculated with respect to the axes of the global coordinate system. It is possible to select another coordinate system for calculation of the bounding parallelepiped with the help of the option:

	<L>	Select Source LSC

After choosing the LCS, the bounding parallelepiped will be automatically reconstructed in accordance with the axes of the LCS.

Cancellation of the selected LCS can be done with the help of the option:

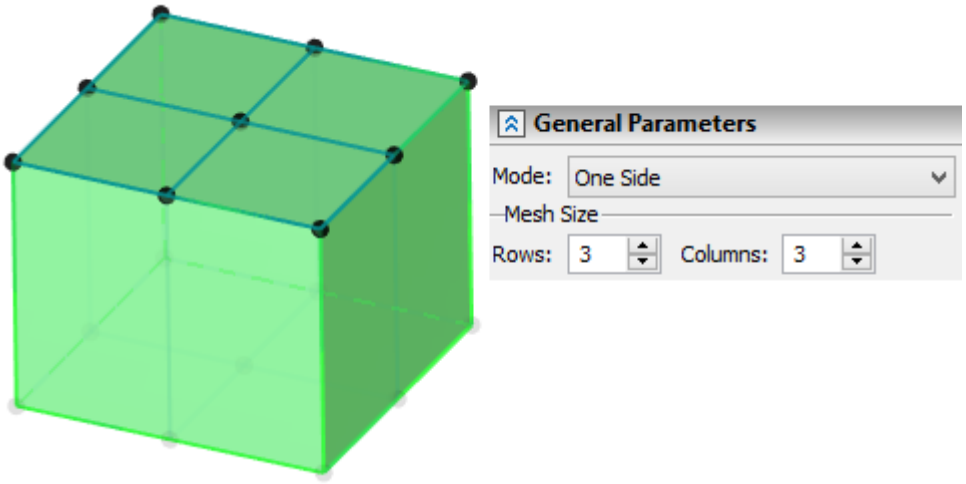
	<C>	Cancel LSC selection

Selection of Deformed Face and Deformation Mode

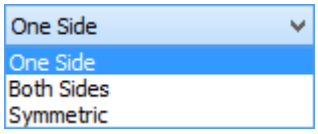
After specifying the bounding parallelepiped, the user has to indicate the face which will be subject to deformation. To do it, it is sufficient to bring the cursor to the desired face (it will be highlighted) and press . The option of face selection will become active at this moment in the automenu:

	<T>	Select Face

The mesh will appear on the selected face (by default it is a 3x3 mesh). The mesh parameters can be changed in the command's properties window (parameters group **Mesh Size**). The maximum allowable size of the mesh – 20x20.



By default, the deformation is created in the one-sided mode, i.e. only the nodes of the selected face of the bounding parallelepiped are displaced. The mode can be also changed in the command's properties window with the help of the parameter **Mode**.



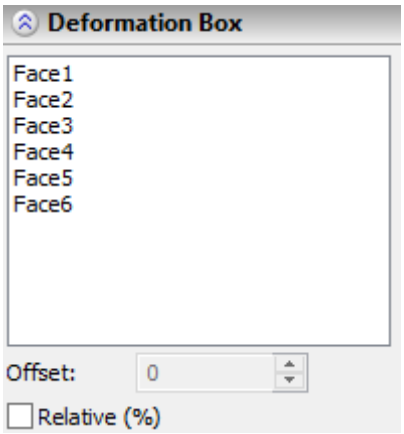
Modifying Size of Deformation Area

The bounding parallelepiped of the deformation area initially spans the entire body. To modify the size of the deformation area, the following option of the automenu is used:

	<D>	Change Deformation Box

After invoking this option, the command will turn to the mode of modifying the boundaries of the deformation area. Highlighting of the deformed face and the mesh temporarily disappear from the

screen. The list of the faces of the deformation area bounding parallelepiped appears in the command's properties window.



The offsets for the faces of the bounding parallelepiped can be specified directly in the 3D window or in the command's properties window (with the help of the list of the faces).

For moving the face in the 3D window, it is sufficient to point at it with the help of (the selected face will be highlighted) and, without releasing the pressed mouse button, shift the face to the desired position. When selecting the face in the 3D window, it will be automatically highlighted in the list of faces in the properties window. The offset specified for the face will be shown in the entry field of the parameter **Offset**.

The position of the face of the bounding parallelepiped can be also modified in the properties window just by selecting the face from the list (in the 3D window the selected face will be highlighted in color) and by entering the desired offset into the entry field of the parameter **Offset**.

To specify the offsets of the faces in relative units (relative to the initial size of the deformation area bounding parallelepiped, that is the size of the bounding parallelepiped of the deformed body), set on the flag **Relative Value, %**.

To fix a face to a 3D node, use the option:

	<Z>	Set Fixing to Point

After invoking this option, define a 3D point to which the selected face of the deformation area bounding parallelepiped should be fixed. Any 3D object capable of defining a point (3D node, the vertex of the body, etc.) can be used for specifying this point. The selected 3D object is specified in the list of faces next to the name of that face of the bounding parallelepiped whose location is defined by the object.

A user can also specify the offsets for the face fixed to a 3D point – in this case these offsets will be relative to the 3D point to which the face is fixed.

To cancel the snap of the face to a 3D point, use the option:

	<H>	Cancel Fixing to Point

After canceling the fixing, the face location does not change. However, the offsets of the face will be subsequently recalculated with respect to the body's overall size and will not depend on the location of the former fixing point for the given face.

For quick cancellation of all modifications of the deformation area, the following option can be used:

	<W>	Set Deformation Box by Body Size

This option cancels all specified offsets for the faces of the deformation area bounding parallelepiped.

To exit the mode of modifying the size of the deformation area, in the automenu turn off the option .

Specifying Offsets of Deformation Mesh Points

Offsets of the mesh points can be specified both in the dialog of the properties window and directly in the 3D window.

For specifying offsets in the 3D window, select successively the nodes (or group of nodes) and with the help of the dragger in the 3D window move them to the desired location.

For selecting one mesh point, it is sufficient to bring the cursor of the mouse to it and press . The selected node will be highlighted, and the dragger in the shape of the coordinate system will appear at the node. The offset of the node is specified by moving the dragger along one or two axes at once (the work with such dragger is described in detail in chapter "3D Nodes").

There are several ways of selecting simultaneously several mesh points. For example, it is possible to successively select the desired nodes while keeping the key <Ctrl> pressed. In this case, the dragger will be drawn at the last selected node of the group. When moving the dragger, all points of the group will be shifted in the same direction by the same distance.

All nodes lying on the same mesh line can be selected quickly just by pointing at this line with the help of . While doing that, all nodes lying on the given mesh line will be selected, but the dragger will not appear. To activate the dragger, point with the at one of the selected nodes. If while pressing the key <Ctrl>, a user successively defines two mesh lines, all nodes on the defined mesh lines will be selected, etc.

When editing, you can quickly select a rectangular group of points with <Shift>.

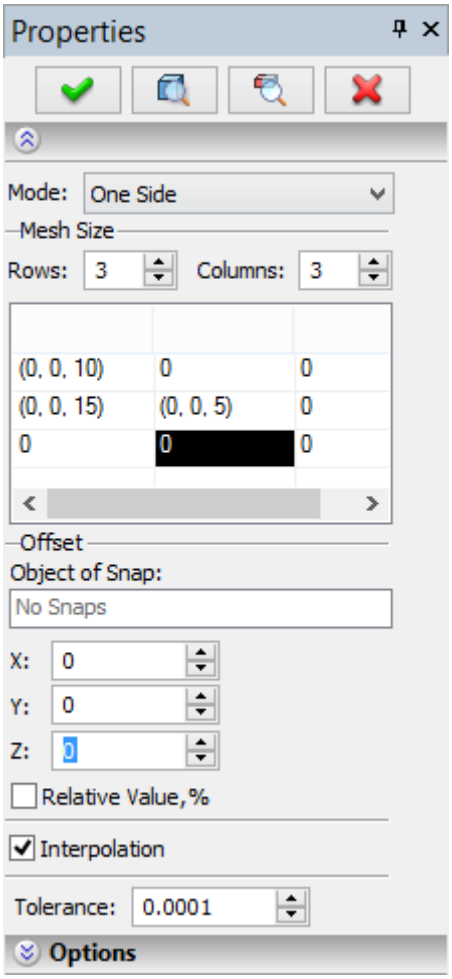
In the **Interpolation** mode, you can add new grid line: hold <Alt> and select one of the existing horizontal or vertical lines in the scene. The deforming principle will not be changed.

For specifying the offsets of the mesh points in the properties window, a table in the parameters group **Mesh Size** is used. Each cell of this table corresponds to one mesh point. The symbol "0" is displayed in the cell if the offset is not specified for the respective node. For the displaced nodes, the specified offsets are shown in the format (X,Y,Z), where X – the node's offset along the X-axis, Y – the offset along the Y-axis, Z – offset along the Z-axis.

To specify offset of any mesh point, in the table select the cell corresponding to the desired node. While doing that, the parameters of the group **Offset** will become accessible for editing below the table. They will define the offsets along the **X**-axis, the **Y**-axis and the **Z**-axis. When modifying the offset values, the node will be moving in the 3D window.

As with the work in the 3D window, it is possible to specify offsets at once for a group of nodes. For selecting several nodes (that is, several cells in the table of nodes), use +<Ctrl>.

The units in which the offsets of the mesh points are specified are controlled by the status of the flag **Relative Value, %**. By default it is turned off, and the specified values are absolute (i.e., are measured in units of model). If the flag is enabled, the specified offsets will be recalculated in relative units.



To fixing a mesh point to a 3D point, select the desired node (in the 3D window or in the nodes table in the dialog of the command) and turn on the following option of the automenu:

	<A>	Set Fixing to Point

After activating this option, select a 3D point to which the current node of the mesh should be fixed. The drop down list of this option contains the filters for selecting various 3D objects capable of defining a 3D point. The selected 3D object is inserted into the **Fix to an object:** field of the parameters group **Offset**.

For the mesh point fixed to a 3D point, it is possible to additionally specify the values of the node's offsets relative to the selected 3D point (with the help of the dragger in the 3D window or in the properties window).

To cancel a fixing of the mesh point to a 3D point, select the desired mesh point (in the 3D window or in the nodes list in the dialog of the command) and invoke the following option of the automenu:

	<B>	Cancel Fixing to Point

After invoking this option, the current mesh point is untied from a 3D point. Note that the offset of the given node does not change at that. The node will remain in the position specified by the 3D point. However, later, when modifying the original model or location of the 3D point, the offsets of the given node of the deformation mesh will be recalculated as if they were specified by numeric values from the beginning.

The type of the formed spline surface (by control polyhedron or by mesh points) is determined by the status of the flag **Interpolation**. When this flag is disabled, the spline surface is created by the control polyhedron, and when the flag is enabled – the created spline surface will go through the nodes of the mesh.

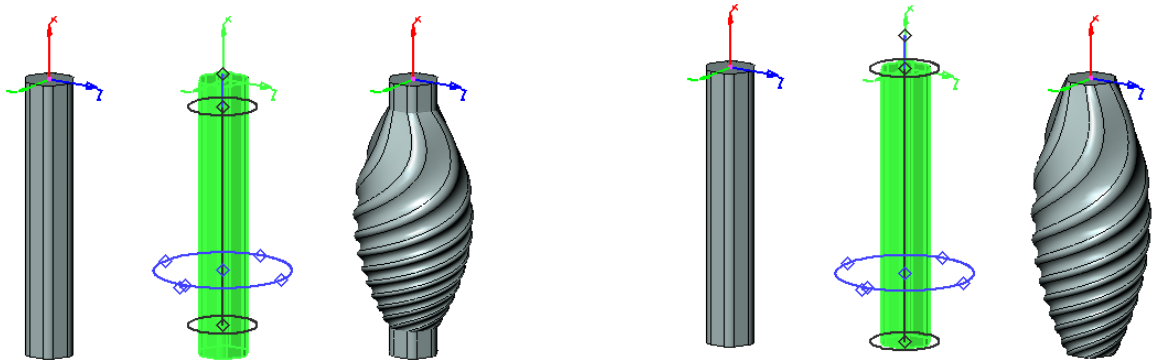
Additional parameter **Tolerance** specifies the value of the precision used in evaluations of the deformation function.

SCALING/TWISTING

General Information

Scale/Twist Deformation is always specified with respect to the coordinate system of deformation (defined with the help of LCS). Twisting is performed only around the X-axis of the coordinate system of deformation, scaling is possible along all axes.

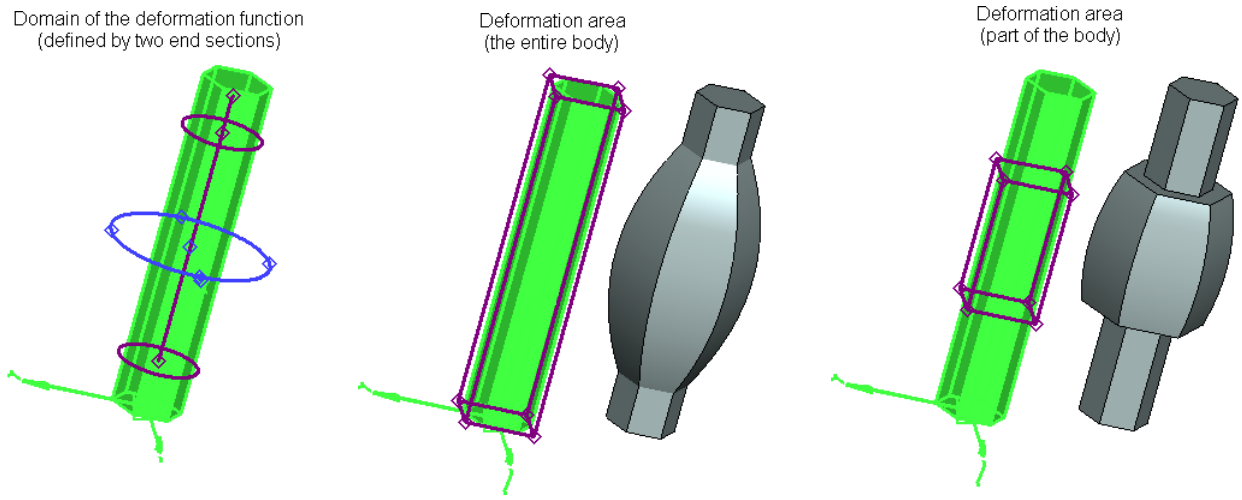
The deformation function is determined on the basis of the parameters (angle of twist and deformation scales) specified for several sections of the deformed body located along the X-axis of the coordinate system of deformation. For different sections a user can specify different scales and angles of twist along the axes of the coordinate system of deformation. For each section the scales along different axes can be also different. The value of any parameter (except section position) can be assigned to a variable.



The deformation function is calculated over the specified *domain of the deformation function* along the X-axis. The domain of the deformation function is defined by the start and end boundaries along the X-axis. The domain of the deformation function can include into itself the entire deformed body, only its part or extend beyond the boundaries of the body.

The computed function is applied to the part of the deformed body defined by the *deformation area*. By default, the deformation area is constructed as the bounding parallelepiped of the deformed body (in the axes of the selected coordinate system of deformation). If necessary, the size of the deformation area

can be modified by moving the faces of the deformation area parallelepiped. It is possible to specify location of the faces by snapping to a 3D point.



Creation of Scale/Twist Deformation

To create the scale/twist deformation, use the command **3DRT: Scale/Twist Deformation**:

Icon	Ribbon
	3D Model → Special → Deformation → Scale/Twist Deformation
Keyboard	Textual Menu
<3DRT>	Operation > Deformation > Scale/Twist Deformation

After invoking this command, it is necessary:

- ✓ Select the deformed body;
- ✓ Define the LCS with respect to which the parameters of deformation will be specified (optionally);
- ✓ Select the domain of the deformation function;
- ✓ Select the deformation area (if necessary);
- ✓ Define the location of the sections of deformation;
- ✓ Specify parameters for the sections of deformation;
- ✓ Complete the operation creation by pressing .

Selection of Deformed Body and LCS of Deformation

For selecting the deformed body, the following option of the automenu is used:

	<O>	Select Body to Deform

This option is turned on by default upon the entry into the command. The deformed body is selected with the help of in the 3D window or in the tree of the 3D model. The selected body is highlighted.

When you select body for the operation, the system is creating LCS for the deformations automatically.

Selection of the coordinate system of deformation is done with the help of the option:

	<L>	Select Source LCS

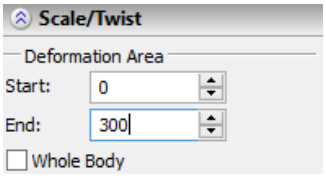
It is recommended to create the LCS of deformation in such a way that its X-axis is directed along the axis of the body since the twist of the body will be performed exactly around the X-axis of the coordinate system.

Cancellation of the selected LCS can be done with the help of the option:

	<C>	Cancel LCS selection

Specifying Domain of Deformation Function and Deformation Area

The domain of the deformation function is specified in the command's properties window (the group of parameters **Deformation Area**).



Parameters **Start** and **End** define the distance from the origin of the LCS of deformation to the start and end boundaries of the domain of the deformation function. The distances are measured along the X-axis of the LCS in units of modal. The system puts the default values into these entry fields so that the domain of the deformation function spans the entire deformed body. A user can narrow or expand the deformation area by specifying other values in these entry fields.

When the flag **Whole body** is enabled, the domain of the deformation function is computed automatically in accordance with the current deformation area.

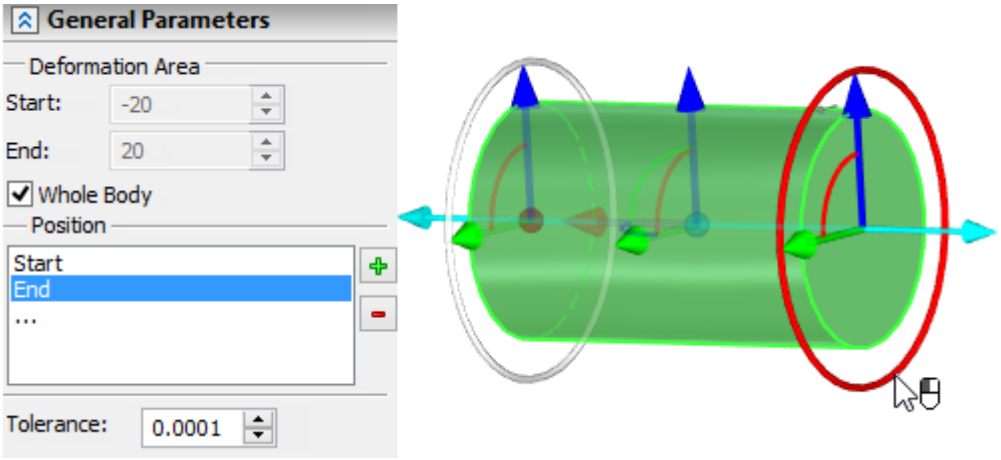
The bounding parallelepiped of the deformation area initially spans the entire body. For modifying the size of the deformation area, as in the sculpt deformation, the following option of the automenu is used:

	<D>	Change Deformation Box

Modifying the deformation area is performed similarly to the sculpt deformation (see above). To exit the mode of modifying the size of the deformation area, in the automenu turn off the option .

Creating Sections of Deformation

After selecting the LCS, the 3D draggers to control location and parameters of the sections of deformation will appear on the screen. By default, only two sections are created: at the start and at the end of the deformation area. These sections cannot removed, their location cannot be modified.



In the dialog of the command's properties in the field **Position** the list of all sections specified for the given deformation is created. The work of the properties dialog is synchronized with the sections' draggers in the 3D window. When selecting the dragger of some section in the 3D window, the same section is automatically selected in the list. And vice versa: the section selected in the field **Position**, will automatically become active in the 3D window. Selected for editing (active) section is marked in the 3D window with color.

To add a new position click the left mouse button on the line that connects centers of two existing positions.

The buttons and in the dialog of the command's parameters allow a user to create new sections of deformation and delete existing ones. The new section by default is placed *after* the section which is active at the moment of pressing the button . The exception is the case when at the moment of pressing the button , the active section is the end section – in this case the new section will be placed *before* the end one.

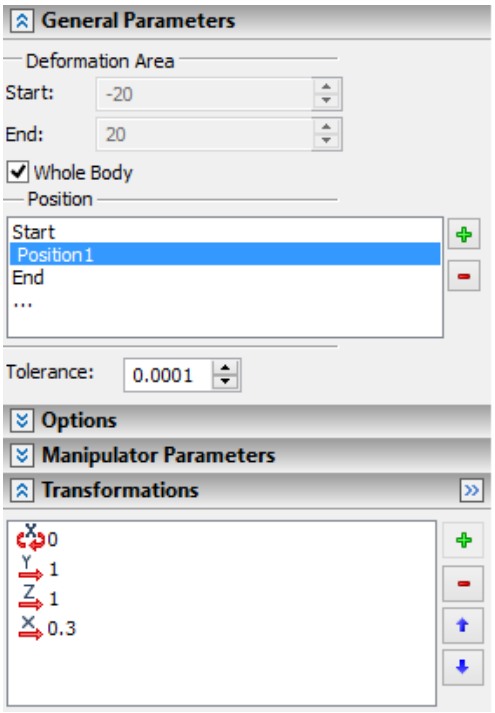
Moving/Scaling along the X-axis is now performed with respect to the origin of the applied coordinate system. Move/scale manipulator is available for both primary ellipses.

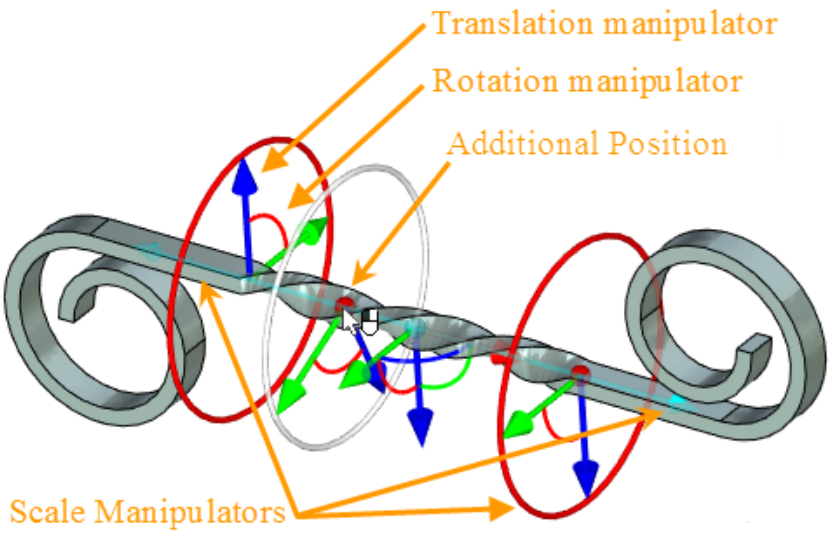
Local coordinate system manipulator can be hidden using the appropriate flag in the **Options** tab.

Specifying Parameters of Sections

For specifying parameters of some section, first it is necessary to make it active by selecting it from the sections list in the properties window or directly in the 3D window. After that, the section parameters can be specified in the **Transformations** section.

To specify the section parameters in the 3D window, the special draggers of the active section are used. Upon bringing the cursor, the dragger is highlighted. For modifying any parameter of the section connected with a certain dragger, bring the cursor to the desired manipulator, press and without releasing the mouse button, move the cursor to the desired location of the dragger.





The dragger of the scale also allows a user to simultaneously change the scales along the two axes (Y and Z). To do it, bring the cursor to an arbitrary point of the active section, press and without releasing the pressed mouse button move the cursor to the desired location.

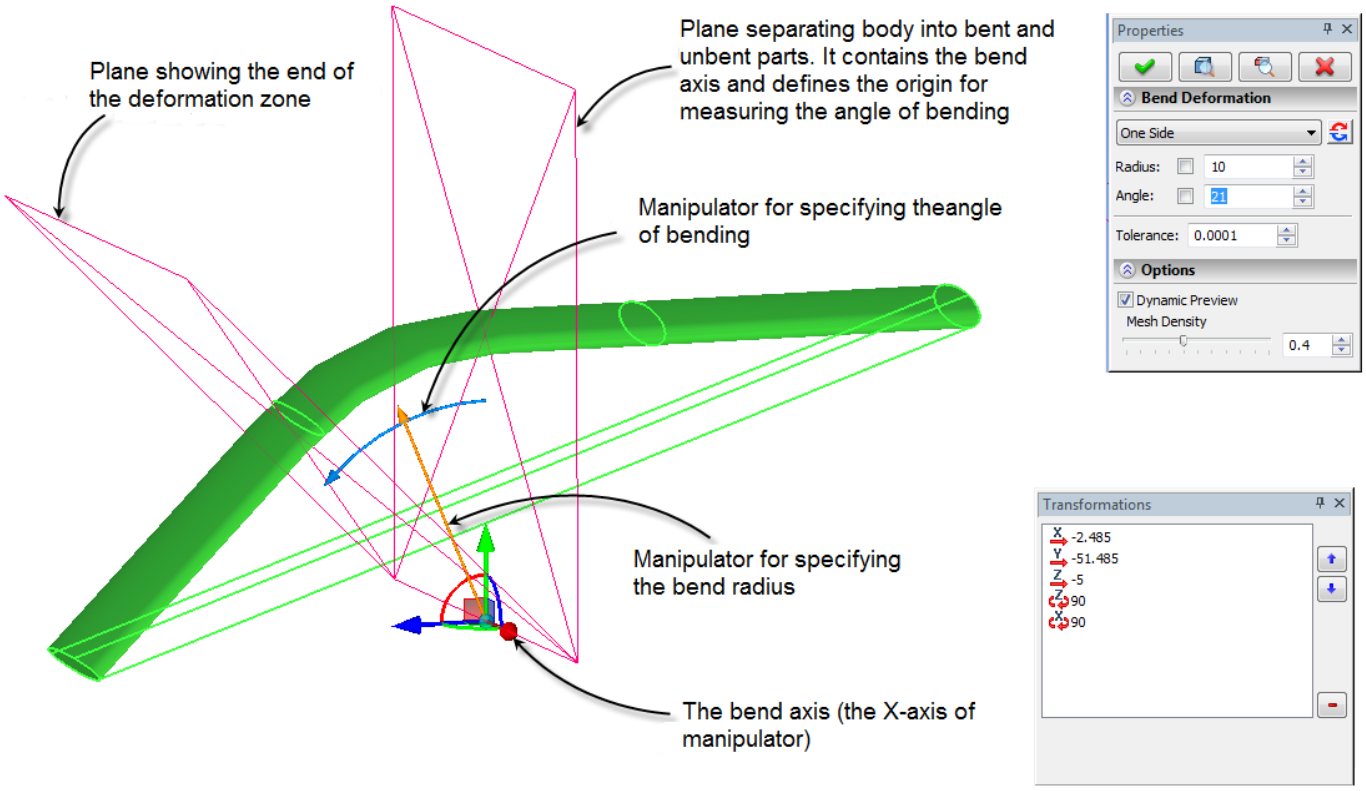
Additional parameter **Tolerance** specifies the value of the precision used in evaluations of the deformation function.

BENDING

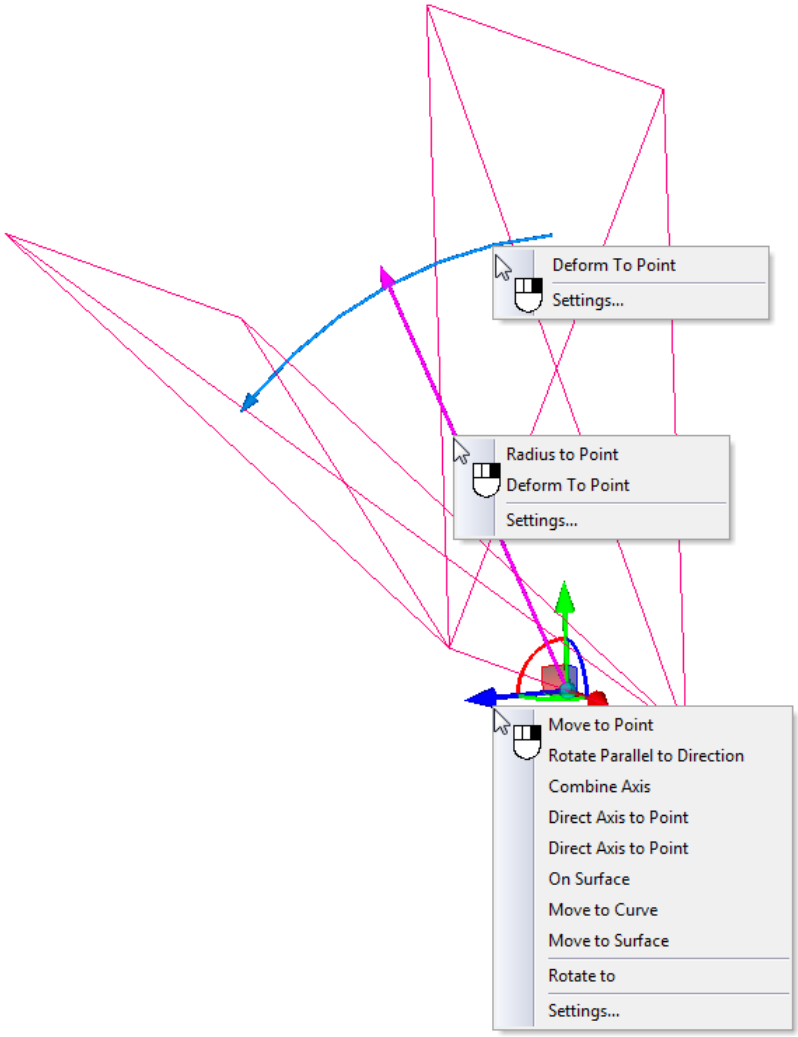
General Information

Bend deformation allows us to bend a selected body with respect to the selected axis by a specified angle. To create this deformation, a user has to specify the deformed body, the origin and the direction of the bend axis, the angle and the bend radius.

To specify the bend axis, the angle and the bend radius, it is possible to use the special multi-element manipulator in the form of LCS with additional elements for specifying the angle and the bend radius.



For all elements of the manipulator of the operation, are available the context menus with the set of additional options that simplify positioning of the manipulator and specification of the parameters of bending.

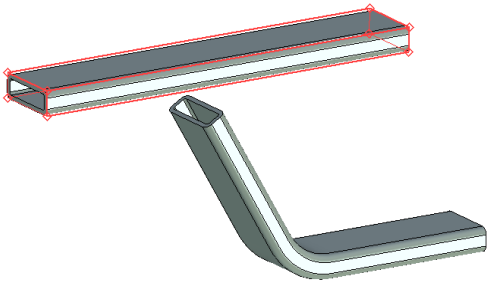


The angle and radius of bending can be specified both with the help of the manipulator and as the numeric values in the dialog of the command's properties window.

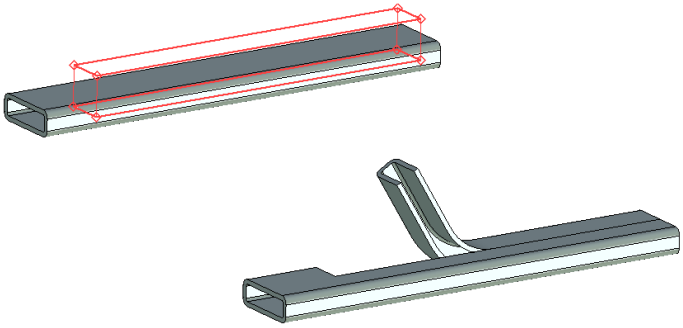
> The bend radius defines the position of the “neutral” surface. For the source body, this is the plane that turns into the cylinder on the bent body. The points lying on this surface do not undergo the tension or compression deformation.

In this operation, it is possible to quickly change the part of the body being deformed, and also, create two-sided bending (with respect to the dividing plane).

As in the sculpt deformation, in the bend deformation it is also possible to modify the size of the deformation area (the size of the deformation area bounding parallelepiped). By default, it spans the entire body.



The deformation area spans the entire body



The deformation area spans only a part of the body

Creating Bend Deformation

For creating the bend deformation, use the command **3DRB: Bend Deformation**:

Icon	Ribbon
	3D Model → Special → Deformation → Bend Deformation
Keyboard	**Textual Menu**
<3DRB>	Operation > Deformation > Bend Deformation

After invoking this command, follow these steps:

- ✓ Select deformed body;
- ✓ With the help of the manipulator, specify the location of the zone of bending and of the bend axis;
- ✓ Select deformation area (if necessary);
- ✓ Specify the bend radius and the bend angle (with the help of the manipulator or in the command's properties window);
- ✓ Select the mode of creating deformation (deformation of the selected part of the body, deformation of the opposite part or two-sided deformation);
- ✓ Complete the operation creation by pressing .

Selection of Deformed Body

To select the deformed body, the following option of the automenu is used:

	<O>	Select Body to Deform

This option is turned on by default upon the entry into the command. The deformed body is selected with the help of in the 3D window or in the tree of the 3D model. The selected body is highlighted.

Selection of Bend Axis and Body-Dividing Plane

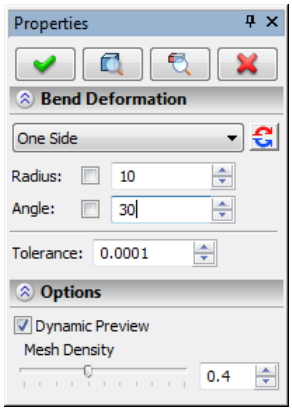
After selection of the deformed body, the manipulator for specifying parameters of bending appears in 3D scene. By default, the axes of the manipulator are oriented along the sides of the bounding

parallelepiped of the bent body. The axis of bending coincides with the X-axis of the manipulator. The plane that separates the bent and unbent parts of the body is defined as the X-Y-plane of the manipulator.

The work with the manipulator of the bending deformation is similar to that of analogous manipulators in other commands of T-FLEX CAD.

Specifying Bend Angle and Radius

A user can specify the numeric parameters of bending deformation – bend angle and bend radius – in different ways. The exact values of these parameters can be specified in the dialog of the command's parameters (parameters **Radius**, **Angle**). Or they can be defined by using the draggers in the 3D window.

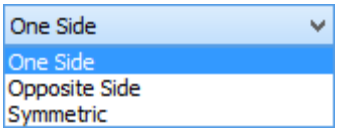


To specify the values of the angle and radius with the help of the draggers, bring the cursor to the corresponding dragger, press and without releasing the pressed mouse button, move the dragger to the desired location. It is also possible to press after selection of the corresponding manipulator. In this case, the manipulator will start follow dynamically the cursor. To fix the desired position of the manipulator, it is required to press again.

In the context menu of the manipulators for specifying the angle and the radius, are also accessible the commands for specifying the values of the radius and the angle with the help of 3D points.

With the help of the button in the dialog of the command's properties, a user can quickly change the direction in which the bend angle is measured.

Selection of Bending Mode. Modifying Size of Bending Area



The mode of creating deformation is specified in the command's properties window with the help of the drop down list **Mode**. By default, the value **One Side** is set, which means that the deformation will be applied only to one part of the initial body (the part which is chosen by the system by default). If a user wants to apply the deformation to another part of the body, he should turn on the option **Opposite Side**.

For creating two-sided deformation, that is bending of both parts of the body, the option **Symmetric** should be used.

The bounding parallelepiped of the deformation area initially spans the entire body. For modifying the size of the deformation area, as in the sculpt deformation, the following option of the automenu is used:

	<D>	Change Deformation Box

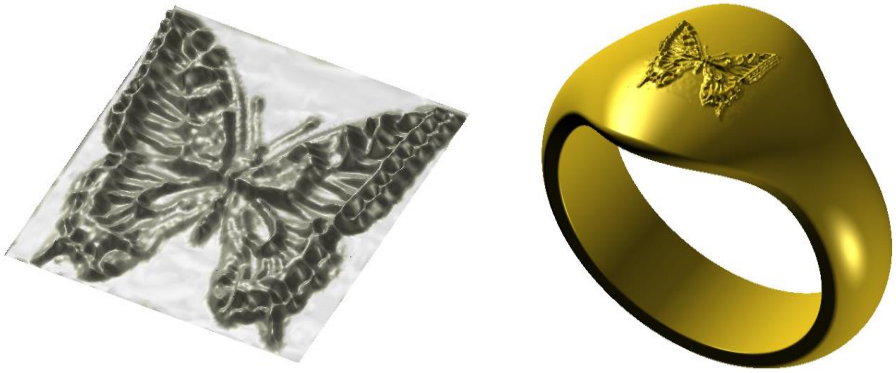
Modifying the deformation area is carried out similarly to the sculpt deformation (see above). To exit the mode of modifying the size of the deformation area, in the automenu turn off the option .

DEFORMATION BY SURFACE

General Information

In the deformation by surface the deformation law is defined by two surfaces: the source and the target one. The deformation function is constructed in such a way that the source surface is reflected onto the target one.

In general case, the source and the target surfaces may not belong to the deformed body. In practice, however, it is often convenient to choose one of the faces of the deformed body to be the source surface. In this case, the deformation function is constructed in such a way as to provide the «bend» of the deformed body onto the target surface.



A solid or a sheet body can be selected as the deformed body.

The source and the target surfaces can be specified by face, 3D profile or a sheet body. The sheet body, specified as the source body, must consists of one face. For defining the target surface, the polyhedral sheet body can be used, however, its faces must have a continuous joint on the tangent plane.

> For a source surface, it is not recommended to select surfaces with a complicated geometry, for example, faces with a picture “pressed out” on them. When transforming such face to the target one, the loss of the face geometry may occur. In such cases, as a source surface, it is better to define, for example, the 3D profile drawn in the plane containing the boundaries of the complicated face.

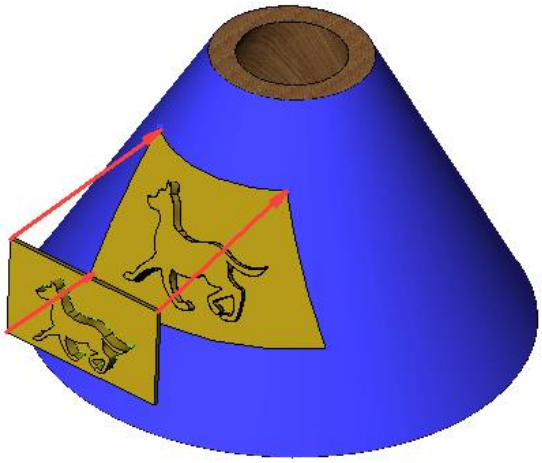
For defining the deformation function more precisely, base points are specified on the source and the target surfaces. For each surface, three points lying on the surface are specified. The points can be specified with the help of 3D nodes, vertices or other 3D objects capable of defining a 3D point. The specified points are not supposed to lie on the same line. The order in which the points on the source and the target surfaces are specified has to coincide.

Two transformation methods are used when performing the deformation by surface:

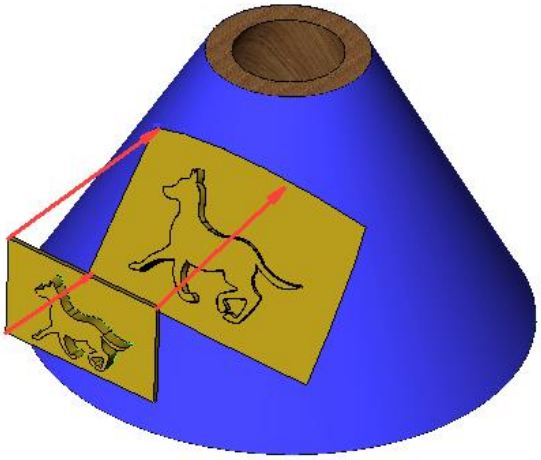
- **By parameters** – transformation with superposition of the base points of the surfaces. With that deformation method, the points on the source surface exactly go into the points on the target surface. When using this method, the target surface cannot have more than one face;

- **With minimal distortions** – transformation with preserved geometric distances between points of the source surface. This method gives minimal distortions, if the distances between the first two points on the source and the resulting surfaces coincide. If the distances do not coincide, the distances are recalculated with the corresponding similarity factor. The distances are measured as the lengths of the normal sections.

  The method "with minimal distortions" can be applied in cases when the target surface consists of a collection of faces (sheet body). It provides the superposition without irregularities which are common for the interior surface parametrization.

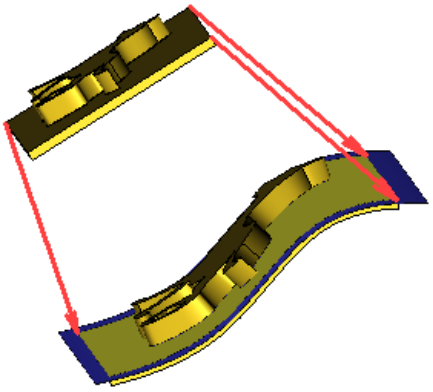


Method "By parameters"

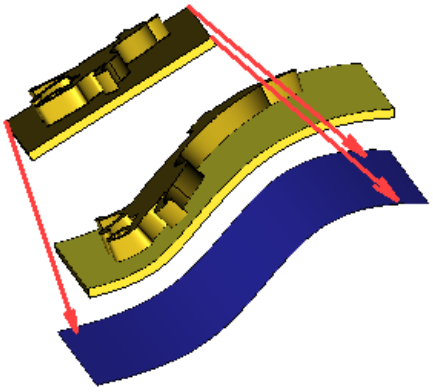


Method "With minimal distortions"

When creating the deformation by surface, the offset from the target surface can be specified. In this case, upon forming the deformation function, the source surface will be transformed not to the target surface, but to the equidistant to it surface with the specified offset.

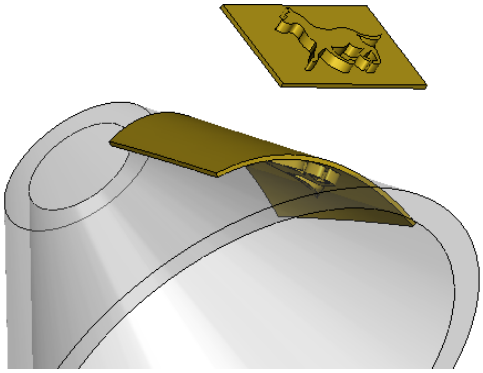


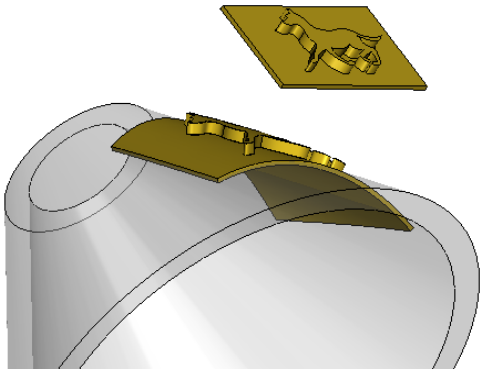
Deformation without offset



Deformation with offset

As a result of the deformation, the resulting body may turn out to lie on either side of the target (or equidistant to it) surface. The placement of the body can be quickly changed with the help of the button in the dialog of the command's properties.





Creating Deformation by Surface

For creating deformation by surface, the command **3DRF: Deformation by Surface** is used:

Icon	Ribbon
	3D Model → Special → Deformation → Deformation by Surface
Keyboard	Textual Menu
<3DRF>	Operation > Deformation > Deformation by Surface

To create deformation by surface, it is necessary:

- ✓ Select deformed body;
- ✓ Select the source surface of deformation;
- ✓ Define the base points on the source surface;
- ✓ Select the target surface of deformation;
- ✓ Define the base points on the target surface;
- ✓ Specify deformation method (“By parameters” or “With minimal distortions”);
- ✓ Specify offset (if necessary);
- ✓ Complete operation creation by pressing .

Selection of Deformed Body and Source Surface

After invoking the command, in the automenu the options for specifying necessary initial data will appear. For selection of the deformed body, the following option is used:

	<O>	Select Body to Deform

This option is turned on by default upon the entry into the command. The deformed body is selected with the help of in the 3D window or in the tree of the 3D model. The selected body is highlighted.

After that, the system will prompt the user to select the source surface of deformation:

	<T>	Select Source Surface

The drop down list of this option contains the list of 3D elements capable of defining the source surface of deformation: face, sheet body, 3D profile. The selected surface will be marked in the 3D scene.

Selection of Correspondence Points on Deformed Body and Source Surface

After selection of the source surface, the options for selecting three base points of the source surface will be successively activated in the automenu:

	<L>	Select First Point on Source Surface
	<K>	Select Second Point on Source Surface
	<G>	Select Third Point on Source Surface

The drop down lists of these options contain filters for selection of the 3D elements capable of defining a 3D point.

For specifying the target surface, the following option of the automenu is used (it becomes active automatically after specifying the base points of the source surface):

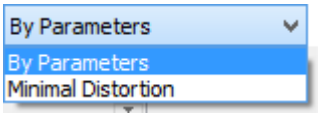
	<M>	Select Target Surface

This option also has a drop down list with the filters for selection of the different types of the 3D elements capable of defining a surface.

After selection of the target surface, its base points need to be specified successively:

	<R>	Select First Point on Target Surface
	<F>	Select Second Point on Target Surface
	<B>	Select Third Point on Target Surface

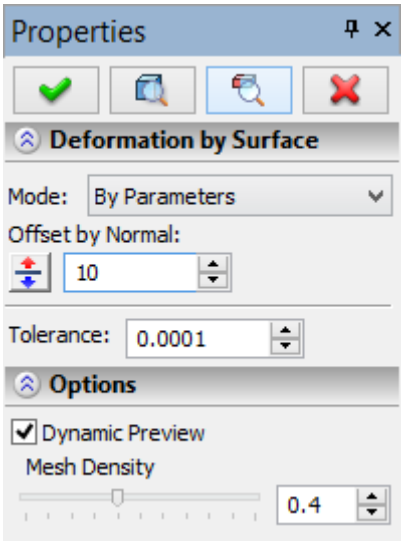
Specifying Deformation Parameters



The deformation mode is selected with the help of the parameter **Mode** in the dialog of the command's properties.

The offset of the source surface with respect to the target one is specified with the help of the parameter **Offset**. The placement of the deformed body with respect to the target surface can be changed with the help of the button found in the dialog of the command's properties (to the left of the parameter **Offset**).

Additional parameter **"Tolerance"** defines the value of the precision used in evaluations of the deformation function.



DEFORMATION BY CURVE

General Information

The law of the deformation by curve is determined by two space curves (source and target). The source curve is identified with the deformed body in its initial state; the target curve – in the resulting state. The space transformation law is constructed in such a way as to provide the transformation of the source curve to the target one.

In general case, the source and the target curves can have arbitrary shapes. Most often, a line (for example, one of the edges of the deformed body or a straight 3D path) is used as the source 3D curve.

When constructing the deformation law, in each point of the two curves the auxiliary $LCS_1(s)$ and $LCS_2(s)$ (s – the curve parameter) are calculated whose pairwise superposition allows a user to define more precisely the space orientation of the resulting body. The auxiliary LCS are computed by the system in the following way: the tangent to the source/target curve at the current point is taken as the X-axis, the normal to the auxiliary surface at the given point of the same curve is taken as the Y-axis, the Z-axis is the vector product of the X and Y axes.

The auxiliary surfaces for the source and target curves can be constructed differently depending on the selected algorithm of the deformation by curve:

- **3D Curve – 3D Curve;**
- **Curve – Spiral;**
- **Pair of Curves – Pair of Curves.**

3D Curve – 3D Curve

When using this algorithm, a user specifies an auxiliary direction vector for each of the curves (source and target). For this purpose, a user defines the LCS one of the axes of which will define the direction vector.

The auxiliary surface for both curves (source and target) is formed by moving the direction vectors along the corresponding curves by using the minimum twist method.

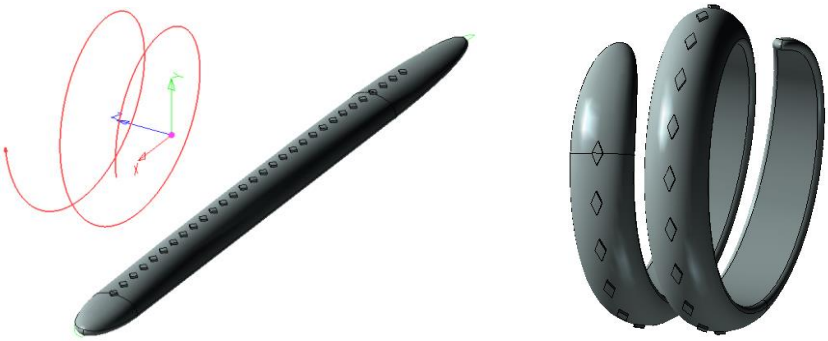
As an example of using this algorithm, consider one of the typical problems solved by this algorithm – wrapping the source body into a ring. The resulting body can be closed.

3D Curve – Spiral

This algorithm is based on the assumption that the target curve resembles a "spiral", that is, there is a space axis around which this curve is "wound". The use of this algorithm of the deformation by curve allows a user to "wind" bodies around the axisymmetric curves in the most natural way.

In this algorithm, a user defines an additional direction vector for the source curve (by selecting LCS and one of its axes). The auxiliary surface for the source curve is formed in the same way as in the previous case by moving the given direction vector along the curve by using the minimum twist method.

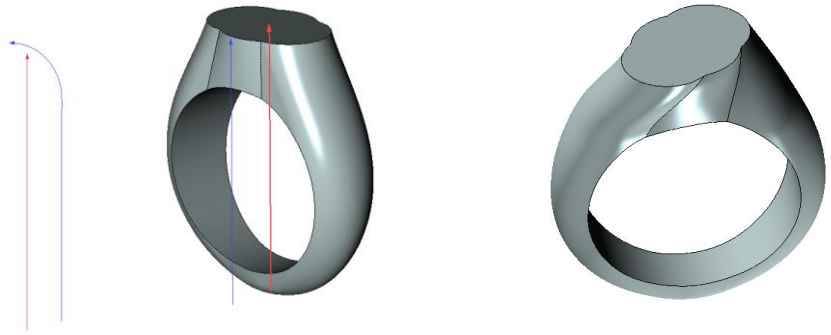
A user also defines the axis of the target curve by selecting the LCS and the desired axis. For the target curve the auxiliary surface is formed as a set of vectors directed from each point of the curve at the axis along the shortest distance.



Pair of 3D Curves – Pair of 3D Curves

In this algorithm, for each of the main curves (source and target) an additional 3D twist curve is specified.

When calculating the transformation law, the auxiliary surfaces for the source and target curves are formed as the set of vectors directed from each point of the source/target curve at the corresponding to it point of the twist curve.



Creating Deformation by Curve

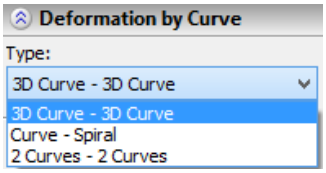
For creating deformation by curve, the following command is used **3DRV: Deformation by Curve**:

Icon	Ribbon
	3D Model → Special → Deformation → Deformation by Curve
Keyboard	Textual Menu
<3DRV>	Operation > Deformation > Deformation by Curve

The first step after invoking this command is the selection of the deformed body:

	<O>	Select Body to Deform

This option is turned on by default upon the entry into the command. The deformed body is selected with the help of in the 3D window or in the tree of the 3D model. The selected body is highlighted.



After selecting the deformed body, in the command's properties window the desired deformation algorithm has to be selected. To do it, use the drop down list of the parameter **Type**.

Depending on the deformation algorithm selected, the following steps sequence will vary.

Creating Deformation of Type "3D Curve – 3D Curve"

If in the list of the parameter "Type" a user selects the value "3D Curve – 3D Curve", then in the automenu the following option becomes active:

	<O>	Select Body to Deform

This option is turned on by default upon the entry into the command. The deformed body is selected with the help of in the 3D window or in the tree of the 3D model. The selected body is highlighted.

After that, the system will prompt a user to select the source curve:

	<T>	Select Source Curve

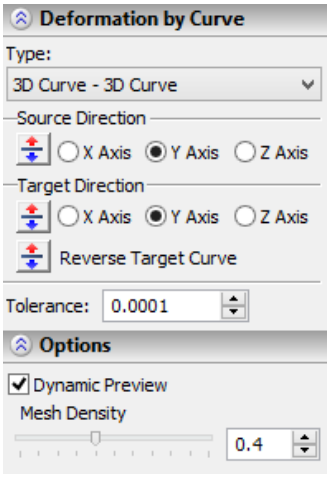
The drop down list of this option contains the list of the 3D elements capable of defining a 3D curve. The selected curve is marked in the 3D scene.

Cancellation of the source curve selection can be done with the help of the option:

	<H>	Cancel Source Curve Selection

After selecting the source curve, a user needs to specify the LCS one of the axes of which will serve as the direction vector for the source curve:

	<L>	Select Source LSC



In the command's properties window, specify the axis of the LCS which will serve as the direction vector for the source curve (parameter **Source direction**). In the dialog one or two axes of the LCS can be unavailable for the selection which means that the selection of the corresponding direction vector will not be possible for the selected source and target curves.

On the basis of the defined direction vector the system automatically computes the auxiliary surface for the source curve and the auxiliary LCS at each point of the curve. For clearness, in the 3D window the auxiliary LCS will be shown for the start point of the source curve.

The button allows a user to change the direction of the obtained direction vector of the source curve, and, therefore, the orientation of the auxiliary LCS.

To cancel the selection of the LCS, which specifies the direction vector of the source curve, the following option is used:

	<C>	Cancel LSC selection

The target curve and its direction vector are defined similarly with the help of the following options:

	<B>	Select Target Curve
	<D>	Cancel Target Curve Selection
	<M>	Select Target LSC
	<N>	Cancel LSC selection

By changing axes which specify the direction vectors for both curves as well as the directions of these axes, it is possible to find the desired choice of the transformation. For obtaining the desired result, it is also possible to use the button [**Reverse Target Curve]**, which allows a user to change the direction of the selected target curve.

Creating Deformation of Type "Curve – Spiral"

If in the list of the parameter **Type** a user selects the value "Curve – Spiral", then in the automenu the following option becomes active:

	<O>	Select Body to Deform

The deformed body is selected with the help of in the 3D window or in the tree of the 3D model.

After that, the system will prompt the user to select the source curve:

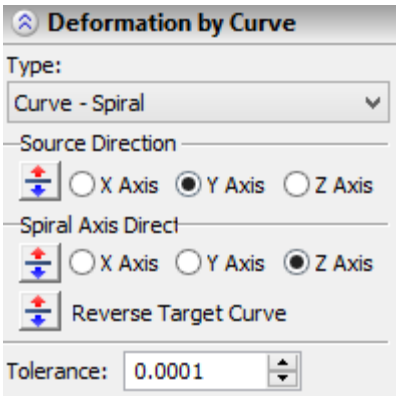
	<T>	Select Source Curve

Cancellation of the source curve selection can be done with the help of the option:

	<H>	Cancel Source Curve Selection

After selecting the source curve, a user needs to specify the LCS the axis of which will serve as the direction vector for the source curve:

	<L>	Select Source LSC



In the command's properties window, specify the axis of the selected LCS which will serve as the direction vector for the source curve (parameter **Source Direction**). In the 3D window the auxiliary LCS will be shown for the start point of the source curve.

The button allows a user to change the direction of the obtained direction vector of the source curve.

To cancel the selection of the LCS, which specifies the direction vector of the source curve, the following option is used:

	<C>	Cancel LSC selection

The target curve and its axis are selected similarly with the help of the following options:

	<B>	Select Target Curve

	<D>	Cancel Target Curve Selection
	<M>	Select Target LSC
	<N>	Cancel LSC selection

As in the previous case, for obtaining the desired result, it is also possible to use the button [**Reverse Target Curve**], which allows a user to change the direction of the selected target curve.

Creating Deformation of Type "Pair of Curves – Pair of Curves"

When choosing the option "Pair of Curves – Pair of Curves", the following option becomes active in the automenu:

	<O>	Select Body to Deform

The deformed body is selected with the help of in the 3D window or in the tree of the 3D model.

After that, the system will prompt the user to select the source curve:

	<T>	Select Source Curve

Cancellation of the source curve selection can be done with the help of the option:

	<H>	Cancel Source Curve Selection

After that, it is necessary to define the twist curve for the source curve of deformation:

	<K>	Select Source Twist Curve

This curve is selected in the 3D window or in the tree of the 3D model.

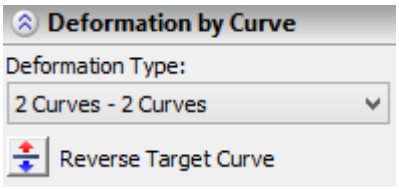
To cancel selection of the source twist curve, use the option:

	<L>	Cancel Source Twist Curve Selection

The target curve and its twist curve are selected similarly with the help of the following options:

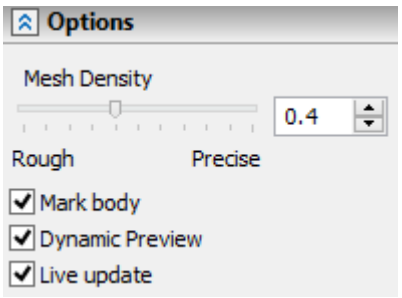
	<B>	Select Target Curve
	<D>	Cancel Target Curve Selection
	<M>	Select Target Twist Curve
	<N>	Cancel Target Twist Curve Selection

In addition, in the dialog of the command's properties the button [**Reverse Target Curve]** is available, which allows a user to change the direction of the selected target curve.



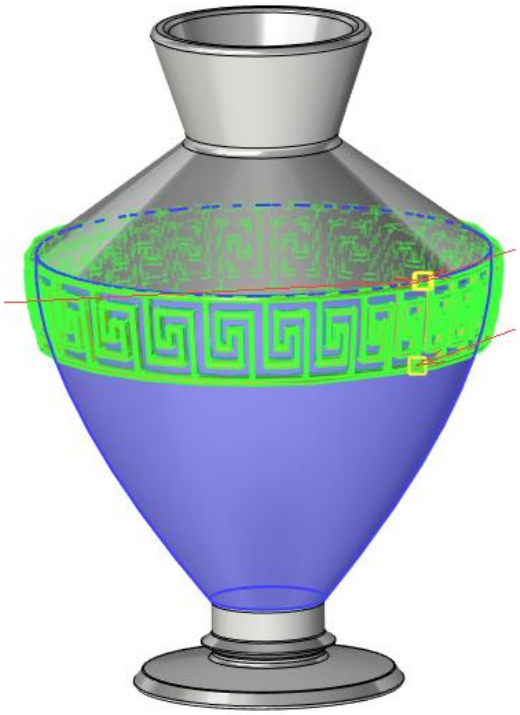
Options in Deformation Operations

In the process of specifying/editing parameters of deformation, the future result of the current operation is dynamically shown in the 3D window. For speeding up the work, the dynamic preview can be turned off by taking off the flag **Dynamic Preview** in the command's properties window.

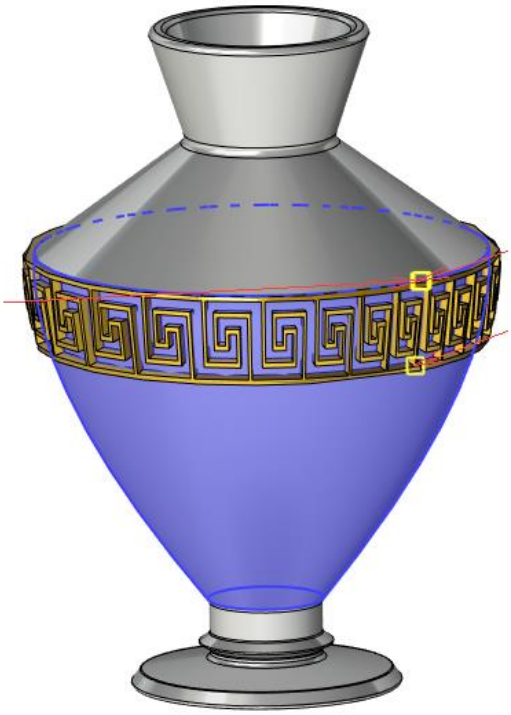


An additional parameter **Mesh Density** controls accuracy of the mesh used for visualization of the result of the deformation. The increase in the mesh density leads to slower regeneration of the bodies in the scene but increases accuracy of the drawing.

If you disable option **Mark body**, the body is no longer highlighted in the scene, which simplifies visual perception of the deformed object.

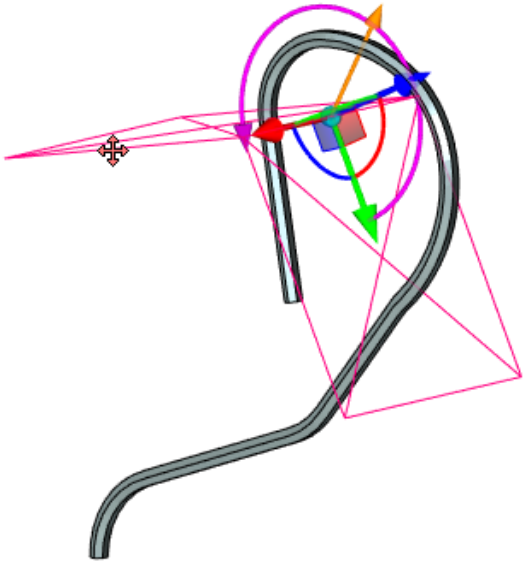


On

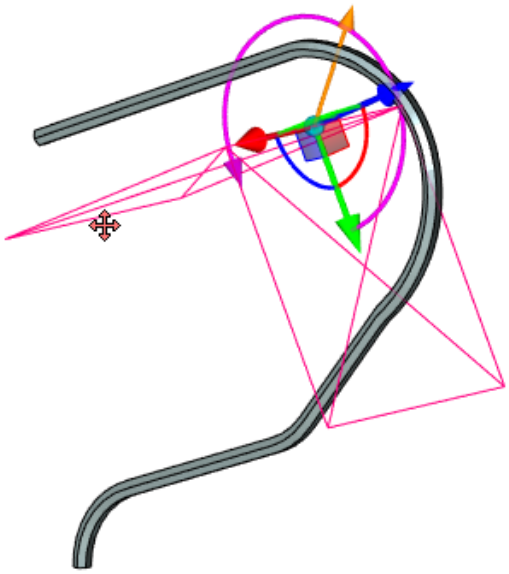


Off

The **Live update** option allows to watch deformations of the body in real time. Deactivating the option accelerates recalculation because solid body updating is not required.



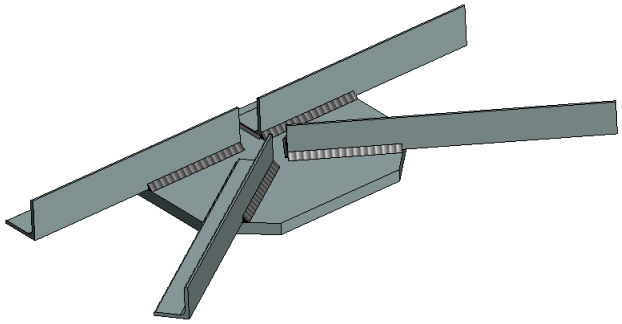
On (the result is displayed)



Off (the result is not displayed, only decorations update)

WELDS IN 3D MODEL

The weld modeling commands of T-FLEX CAD allow creating welds not only in a 2D drawing, but also within a 3D model. Welds appear in the 3D scene as special "cosmetic" bodies with a specific texture.



You can make an annotation for a 3D weld just like you do for a weld in a 2D drawing The annotation can be created directly in the 3D model (it will be a 3D leader note), or on a 2D projection of such model.

When you create the table of welds on one of 2D pages of a given document, the 3D welds are entered in it along with the 2D welds.

BASIC INFORMATION ABOUT 3D WELDS

A 3D weld can be created by various means. All methods of creating 3D welds can be classified in groups:

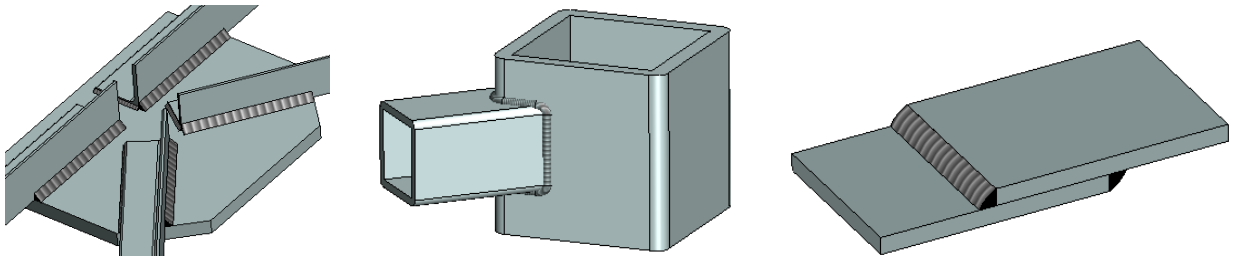
- ✓ 3D welds created on faces of the 3D model;
- ✓ 3D welds created by guides: "by one guide" and "by three guides";
- ✓ 3D welds created on edges, 3D paths, 3D profiles, and operations;
- ✓ 3D welds created by already existing 3D welds.

For 3D welds created by edges and by guides a new "cosmetic" body is created in the 3D scene. These welds do not influence the remaining part of the 3D model and are not entered in the 3D model history. In the 3D model tree, a 3D weld body is put into a separate "Welds" folder.

> The "WeldMaterial" material from the standard T-FLEX CAD materials library is used for the weld body.

If a 3D weld is created by other methods, the new body will not be created. The weld will be represented in the 3D scene by the model elements on which it is based.

By using various methods of creating 3D welds in T-FLEX CAD, you can simulate various types of welds in the 3D model.



For any 3D weld it is possible to specify to which *weld type* it belongs. The type determines all basic characteristics of the weld (type of weld joint, structural elements, annotation). The type of the weld does not depend on the way the weld was created.

> The weld type is created as a separate model element. The weld types defined in the current document are common across both the 2D and 3D welds. A detailed description of weld types is available in the "Weld" chapter of the "Fundamentals. Two-Dimensional Design" manual.

You do not have to specify the type of a 3D weld. Such a 3D weld would be called *nonstandard*. A nonstandard 3D weld is not entered in the welds table. Besides that, nonstandard 3D welds are not displayed in the welds tree of the "Welds" window.

The nonstandard 3D welds can be used to decorate the 3D model. For example, suppose, the user wants to draw a weld on the model simply to make it look more realistic, without an intention to create weld documentation in the future. In such a case, it may not be justifiable to spend time on creating a weld type. You can simply define a nonstandard 3D weld in the 3D model.

> Please distinguish between the "nonstandard" 3D weld and a 3D weld created by the nonstandard weld type, whose parameters were custom-defined by the user. The 3D weld of a nonstandard type is entered in the welds table.

Methods of Creating 3D Welds

Creating 3D Welds on Faces

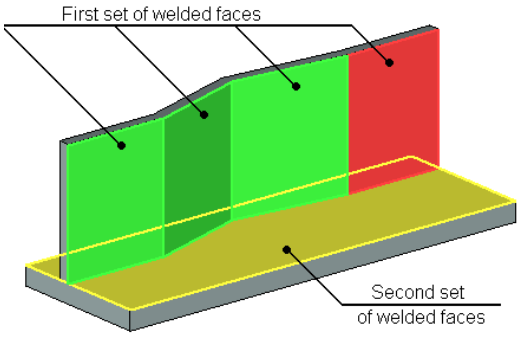
There are three methods to create a 3D weld on faces of the 3D model:

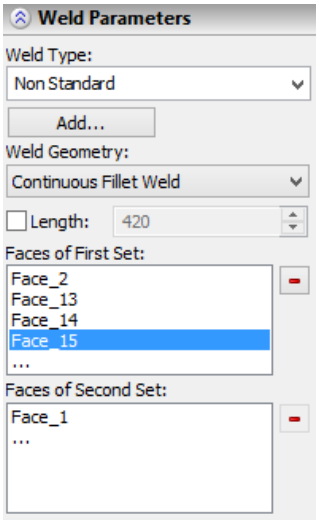
- ✓ Creating a continuous fillet weld;
- ✓ Creating an intermittent fillet weld;
- ✓ Creating a butt weld.

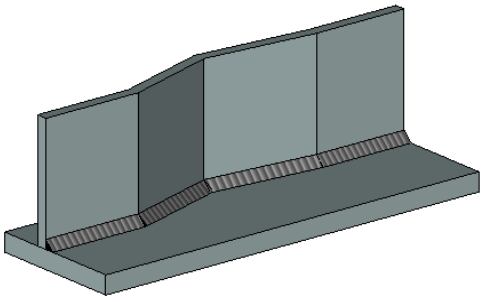
In all such cases, the weld is created based on two sets of faces being welded. The sets define the faces that must be "connected" by the weld. The number of faces in the sets doesn't have to match. The selected faces must belong to one or two Bodies.

Continuous Fillet Weld

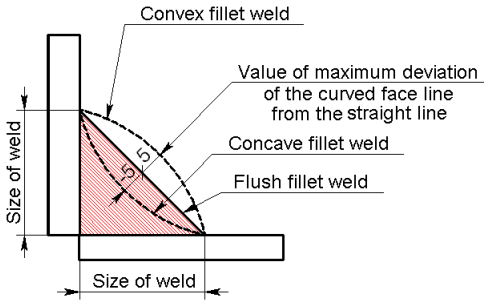
When creating a continuous fillet weld, you need to select two sets of faces being welded, the size of weld and the contour type (flush, convex, concave). The result will be a weld between the surfaces of the specified sets of faces, whose size will correspond to the defined size of weld.

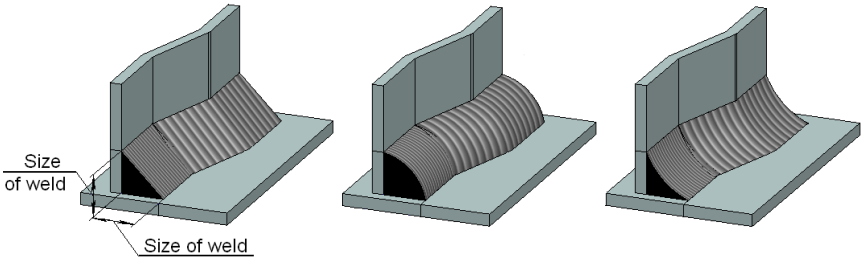






The size of weld is counted from the line of faces' intersection (see the figures). The weld contour being created is specified as the maximum deviation of the curved face line of the weld being created from the straight line. A positive deviation value means convexity of the weld, whereas negative – the concavity. Zero value corresponds to the "Flush" contour.

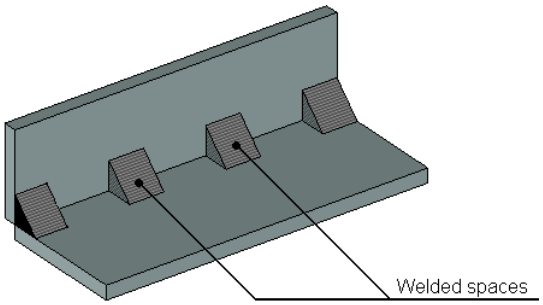


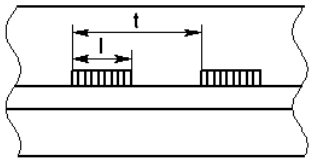


Flush fillet weld | Convex fillet weld | Concave fillet weld

Intermittent Fillet Weld

An intermittent fillet weld is a kind of a fillet weld in which the continuity is broken by recurring unwelded spaces. Therefore, it is generally created in the same way as a continuous fillet weld. Additionally, you need to specify *the length of weld* and *pitch*. Please note that for some types of welds, these parameters may be predefined by the type itself (if so required by the applicable standards, such as GOST). In this case, the values defined for the weld, override the values prescribed by this weld type.



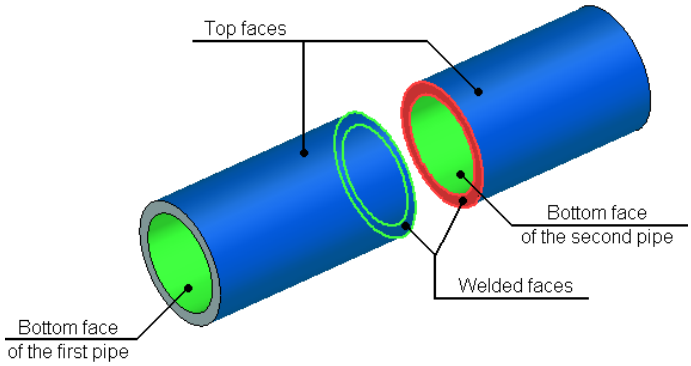


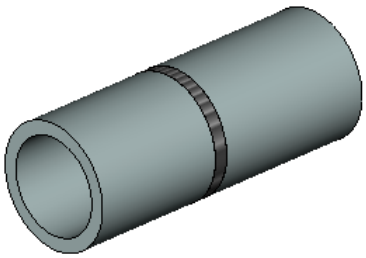
Parameters of an intermittent fillet weld:
l - length of weld, t - pitch

Butt Weld

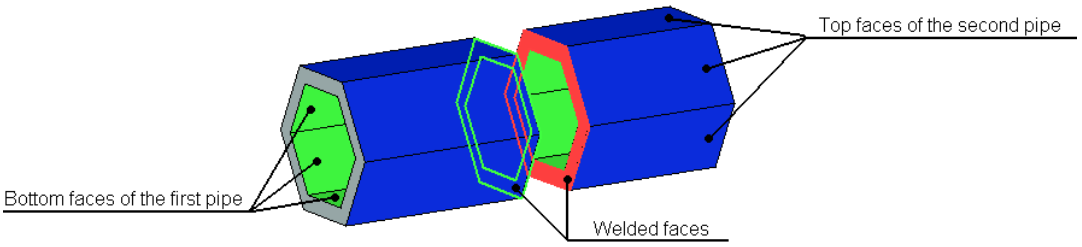
A butt weld connects the entire surface of the two sets of faces being welded. To ensure the correct orientation of the butt weld, you need to additionally specify the *top* and *bottom* faces for each set of faces being welded. Those define the required correspondence to construct the weld body. Specify the top and bottom faces by picking ones among those having common edges with the faces being welded.

For example, when creating a butt weld between two pipes of the round section shape, you select the outer cylindrical faces of both bodies as the top faces. The bottom faces in this case will be the inner cylindrical faces. Or, you can do the opposite, as far as the outer face of one body matches with the outer face of the other body, and so do the inner faces.

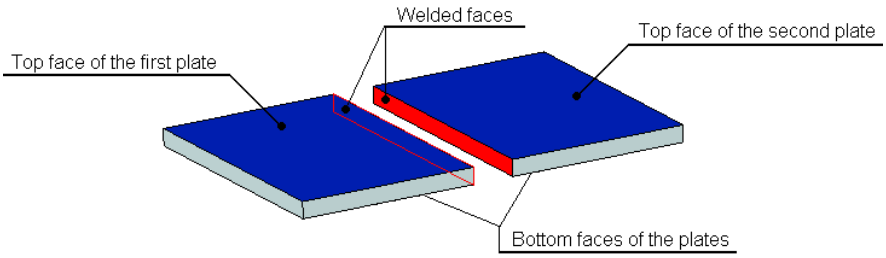




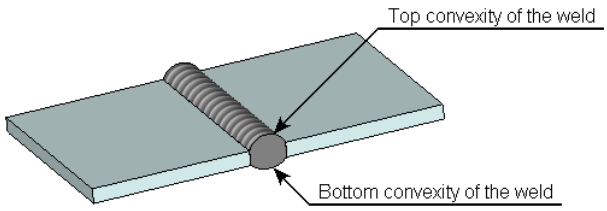
When creating a butt weld between two pipes with a noncircular section shape, select all outer faces of both pipes as the top faces, and all inner faces - as the bottom faces. At the same time, if several top faces are specified on each of the bodies, then the order of selecting faces on one body must match the order of selecting faces on the other body. The same is true in the case of selecting several bottom faces on each of the bodies.



When creating a butt weld between two plates, you select the top and bottom faces as the ones, along which the weld will go. Please note that the top face on one plate must be on the same side as the top face on the other plate.



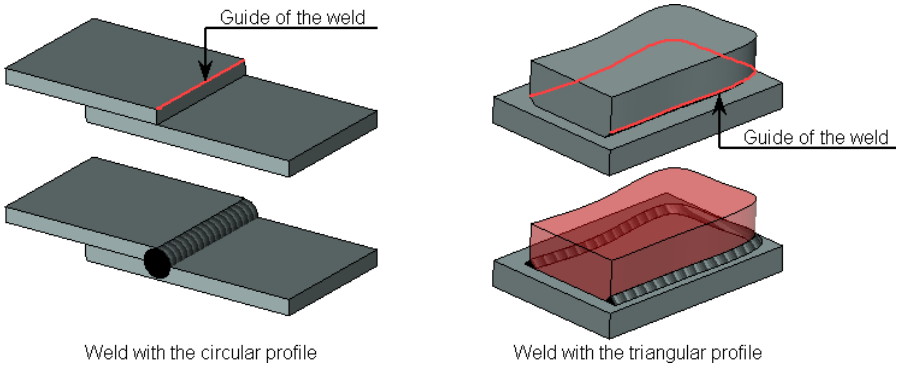
In the case of the butt weld, you can specify the contour type (flush, convex, concave) on either side of the parts being welded (that is, on the sides of the top and bottom faces). The weld contour from each sides of weld is specified as the maximum deviation of the curved face line of the weld from the straight line.



Creating 3D Weld by Guides

Weld Created by One Guide

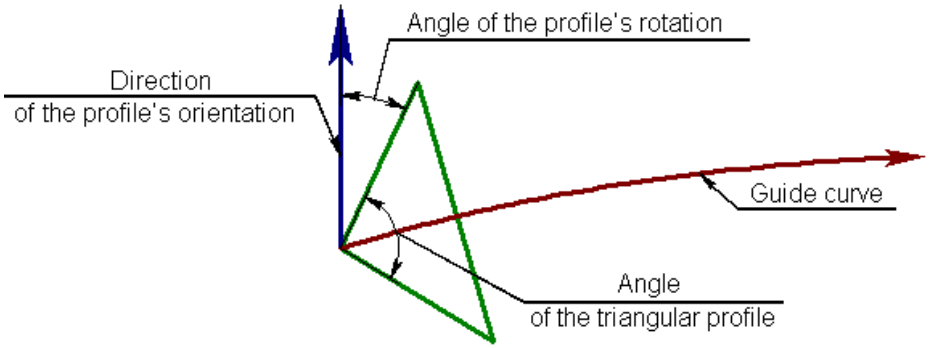
When creating the weld by one guide, the body of the weld is created by translation of the weld's profile along the arbitrary guide. The guide of the weld can be closed.



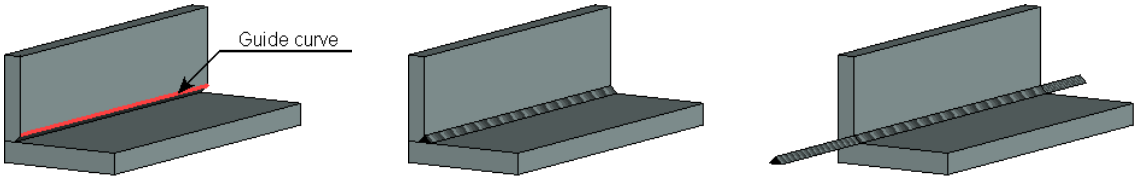
As a guide, the user can select edges, 3D paths, 3D profiles, etc. It is also possible to define the guide by a group of 3D elements (edges, paths, etc.) on condition that all selected elements form a continuous sequence.

The shape of the profile is selected from two options offered by the system: circle or isosceles triangle. For the circular profile, the user defines the radius, for the triangular profile – the size of the leg and the size of the angle between the legs.

For specifying precise orientation of the triangular profile, the user has to define direction of the profile's orientation and the angle of the profile's rotation with respect to this direction. When creating the weld, the vertex of the triangle is superposed with a point on the guide, and one of the sides of the triangle is superposed with the given direction vector, and after that the profile is rotated through a required angle.



The length of the created weld is equal to the length of the guide. In cases where it is desirable to create the weld longer than the guide, the user can specify the magnitude of extensions at the ends of the guide. The body of the weld will be elongated along the tangent to the guide at its end points by a specified extension length.

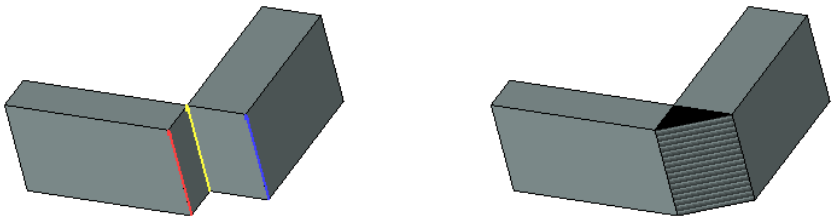


Weld without extensions

Weld with extensions on both sides

Weld Created by Three Guides

For creating the weld of this kind, three guides must be specified. The body of the weld, triangular in cross-section, is formed in such a way that its lateral edges coincide with the guides.

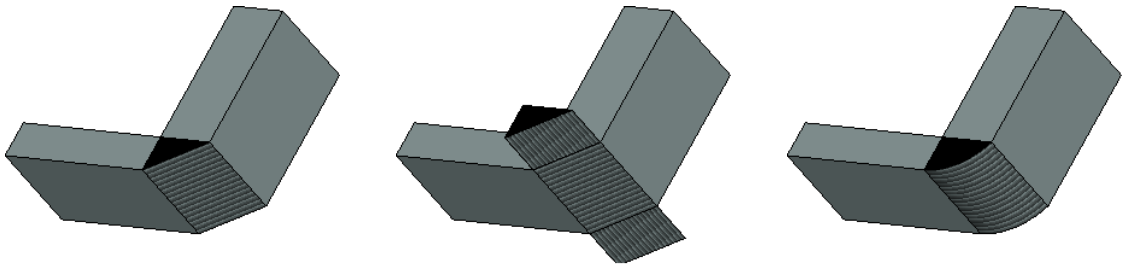


Guide curves are define by edges

For the guide, the user can select edges, 3D paths, 3D profiles, etc. It is also possible to define the guides by a group of 3D elements (edges, paths, etc.) on condition that all selected elements of one guide form a continuous sequence.

The length of the created weld is equal to the length of the first guide. In cases where it is desirable to create the weld longer than the guide, the user can specify the magnitude of extensions at the ends of the guide. The body of the weld will be elongated (by a parallel shift of the end sections) by a specified extension length.

For the weld by three guides it is possible to specify the contour type (flush, convex, concave) for the side lying between the first and the second guides.



Creating 3D Welds on Edges, 3D Paths, 3D Profiles, and Operations

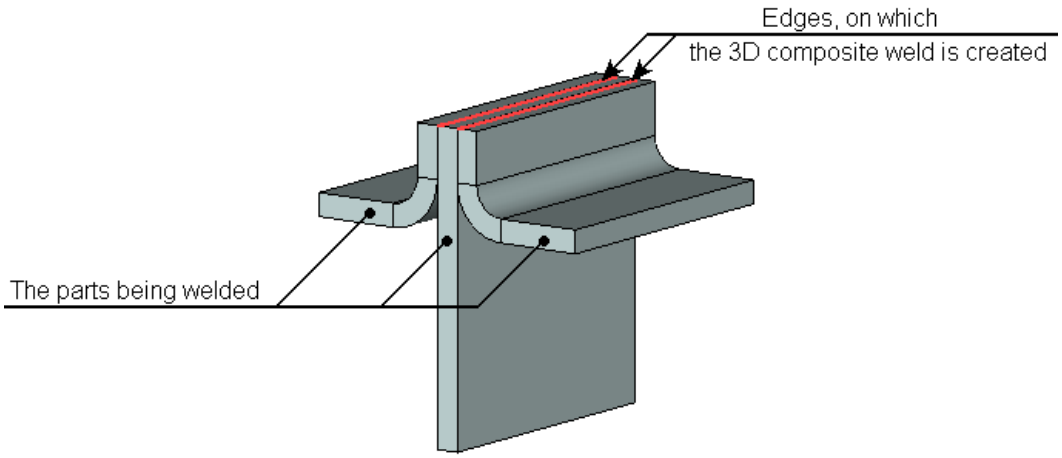
A 3D weld created on edges, 3D paths, 3D profiles, and operations, can be used in the cases when the required appearance of the weld cannot be achieved by building it on the model faces. It serves to indicate that some existing 3D model elements represent welds. The supply more information with the created composite weld, you can specify, what parts it connects. Please note that such a 3D weld cannot be nonstandard.

When the 3D weld is created, no additional "cosmetic" weld body is created in the 3D scene. You can still "see" such a weld – as you select it in the welds window, the respective 3D objects will be highlighted in the 3D scene.

> If necessary, you can manually assign the "WeldMaterial" material to the body representing a 3D weld.

A 3D weld created on edges, 3D paths, 3D profiles, and operations, is entered into the table of welds along with other types of welds. The length of a composite weld is determined in the automatic calculation mode as the sum of the perimeters or lengths of all 3D model elements that are involved in it, except bodies (operations). The bodies that are involved in a composite weld, are not accounted for when automatically calculating its length.

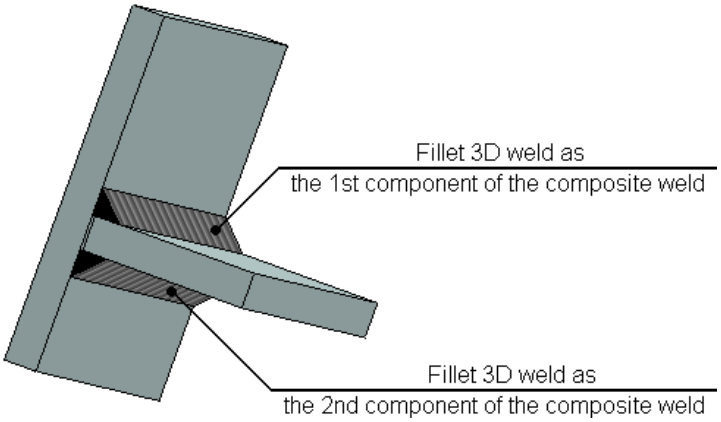
As an example, consider creating a flange-face weld. Suppose, we need to create a weld that connects three parts (see the figure below). Neither the fillet nor the butt 3D weld is suitable in this case. We will therefore create a 3D weld on two edges, and will additionally specify what bodies must be connected by this weld.



Creating 3D Welds by Existing 3D Welds

A 3D weld can be created based on a set of already existing 3D welds. Such a weld cannot be nonstandard. Just like in the previous case, no additional body is created in the 3D scene when such a 3D weld is created.

As an example of creating such a weld consider creating in T-FLEX CAD a two-sided Tee-weld. A Tee-weld can be imitated by a pair of fillet welds. To make this pair of welds work as a single weld in the future, you need to additionally join them into one weld.



To add more information to the created composite weld, you can specify, what parts it connects.

The length of the composite weld in the automatic calculation mode is determined as the sum of all 3D welds that it includes. The properties of a composite weld created from a set of 3D welds are determined by the type specified for this composite weld. The parameters of the source welds are ignored. Please also note that the table of welds generally includes both the source 3D welds and the resulting weld, which is based on them. To avoid duplicates, we recommend using nonstandard 3D welds as the source welds. In this case, those will not be entered in the welds table.

CREATING 3D WELD

A 3D weld is created with the command **W3: Create 3D Weld**:

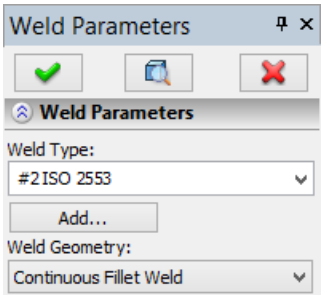
Icon	Ribbon
	Assembly → Assembly → Weld → Weld
Keyboard	Textual Menu
<W3>	Tools > Weld >3D Weld

To create a 3D weld, you need to:

- ✓ define type of the weld being created;
- ✓ select the method of creating the weld;
- ✓ select the source geometrical objects for the weld (depends on the weld creation method);
- ✓ define the weld length (if necessary);
- ✓ define additional weld parameters (the size of weld, the contour type or any other parameters depending on the weld type);
- ✓ confirm the weld creation.

Selecting Type of 3D Weld

The type of the weld being created is selected in the command's properties window, in the **Weld Type** drop-down list. The list contains all types of welds created in the current T-FLEX CAD document.



If the required weld type is not yet created, you can use the button **[Add...]**. Clicking this button starts the command of the new weld type creation. When the type is created, the system returns to the weld creation command. The created weld type will be automatically selected.

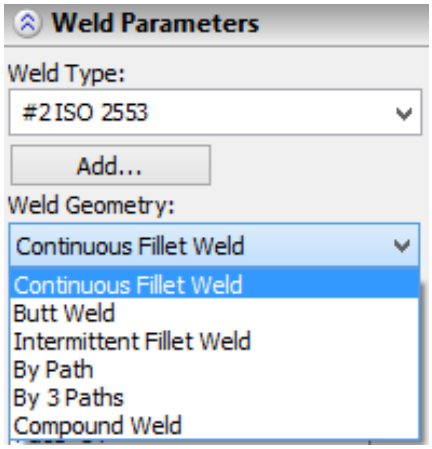
If no weld types are defined in the current document at the time of starting the **W3: Create 3D Weld** command, then the weld type creation command will be launched automatically.

> How to create a new weld type is described in the "Weld" chapter of the "Fundamentals. Two-Dimensional Design" manual.

Besides the weld types that the user created, the "Weld Type" list contains an additional option **Non Standard**. It is used to create nonstandard 3D welds.

Selecting the Method of Creating a 3D Weld

You select a method of creating a 3D weld using the **Weld Type** drop-down list in the command's properties window.



To create a 3D weld on model faces, you need to select one of the following entries in the list – **Continuous Fillet Weld** (creation of a fillet weld), **Intermittent Fillet Weld** (creation of an intermittent fillet weld), **Butt Weld** (creation of a butt weld).

Welds by one guide or three guides can be created by selecting, from the list, the options **By Curve** and **By Three Paths**, respectively.

To create a 3D weld based on existing entities of the 3D model (except faces) you shall select the option **Compound Weld**.

> The "Compound Weld" option is not available in the list, if the "Weld Type" field specifies "Non Standard".

Selecting the Source Geometrical Objects of a Weld. Advanced Weld Parameters

The source geometrical objects selection for the weld being created, as well as additional weld parameters, depend on the method of creating this weld. Therefore, we will review different methods of creating a 3D weld separately.

Welds on faces. Continuous and intermittent fillet welds

As you select the "Continuous Fillet Weld" or "Intermittent Fillet Weld" item in the list, the following options become available in the command's automenu to select the two sets of faces being welded:

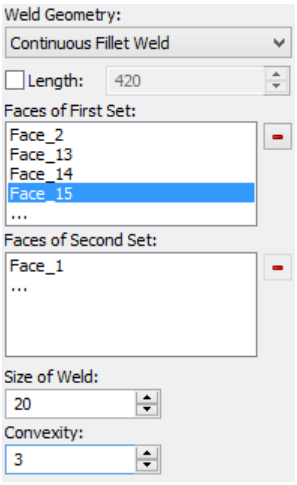
	<1>	Select Faces of First Set
	<2>	Select Faces of Second Set

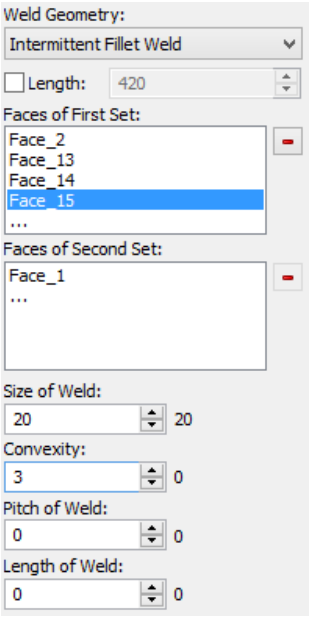
The selected faces are entered in the lists **"Faces of First Set"** and **"Faces of Second Set"**. To remove some face from a selected set, pick it in the respective list by clicking, and then click the button .

The size of weld can be defined using the same-name parameter in the properties window.

The contour type of the weld defined by parameter **"Convexity"**. This parameter define the value of maximum deviation of the curved face line of the weld from the straight line. A positive deviation value means convexity of the weld, whereas negative – the concavity. Zero value corresponds to the flat contour.

In the case of an intermittent fillet weld, you can additionally specify **pitch** and **length of weld**.





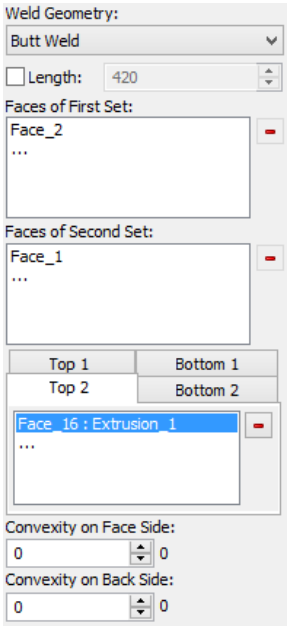
Welds on faces. Butt Weld

The two sets of welded faces in the butt weld are selected using the same options and , as in the fillet welds. Additionally, there will be the options in the automenu to select the top and bottom faces of the bodies being welded:

	<T>	Select Top Faces of First Set
	<B>	Select Bottom Faces of First Set
	<Y>	Select Top Faces of Second Set
	<N>	Select Bottom Faces of Second Set

The selected faces are entered in the lists **Top 1**, **Top 2**, **Bottom 1**, **Bottom 2**. To remove some face from the selection, pick it in the respective list by clicking, and then click the button .

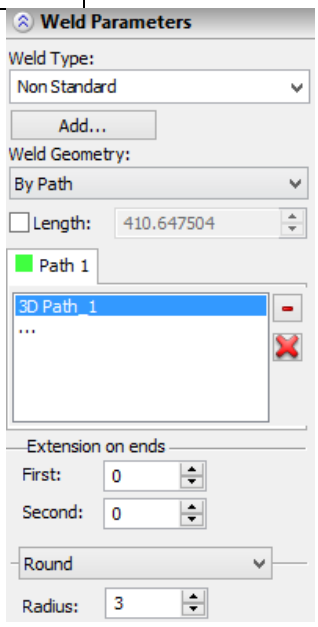
In the case of the butt weld, you can also specify **Convexity on Face Side** (that is, the contur type of the weld on the side of the top faces) and **Convexity on Back Side** (the the contur type of the weld on the side of the bottom faces) in the properties window.



Weld by Created by Guide

For creating the weld by a guide, the user should select the option "By Path" found in the list "Weld type". After that in the automenu the option for selecting the guide will be available:

	<T>	Select 1st Guide

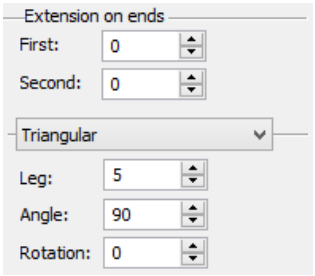


The pull down list of this option contains the filters for selection of 3D objects – loops, edges, 3D paths, 3D profiles, etc. The objects selected as the elements of the guide are put into the list **Path 1** in the command's properties window. To cancel selection of a 3D object, select it in the list with the help of and press the button .

If you want to cancel selection of the guide entirely, press the button in the properties window or use the following option of the automenu:

	<D>	Cancel Selection of 1st Guide

If the selected guide is not long enough, it is possible to increase the weld's length by specifying the magnitude of the extensions. This can be done with the help of parameters of the group **End Extensions** found in the command's properties window.



Parameters' group found in the lower part of the section **Weld** of the properties window allows the user to select the type of the weld's profile: **Round** (circle) or **Triangular**. When selecting the **circular** profile it is necessary to indicate its **radius**.

For **triangular** profile you need to specify the length of the lateral side of the triangle (parameter **Size of Weld**), angle between the sides of the triangle (**Angle**) and the angle of rotation with respect to the direction of the profile orientation (**Rotation**).

The direction of the triangular profile orientation can be specified with the help of the following option of the automenu:

	<R>	Select guide for defining profile orientation

To cancel selected direction, use the following option:

	<U>	Cancel selection of guide for defining profile orientation

Weld Created by Three Guides

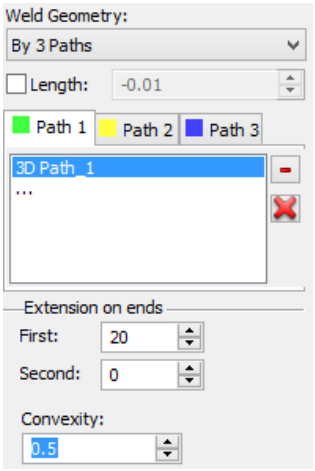
For creating the weld by three guides, the user should select the option "By Three Paths" found in the list "Weld type". After that in the automenu the following options for selecting the guides will be available:

	<T>	Select First Guide Curve
	<D>	Cancel Selection of First Guide Curve
	<W>	Select Second Guide Curve
	<K>	Cancel Selection of Second Guide Curve
	<A>	Select Third Guide Curve
	<M>	Cancel Selection of Third Guide Curve

Selected objects are put into the list **Path 1**, **Path 2**, **Path 3** in the command's properties window.

The length of the created weld is equal to the length of the first guide. You can increase the length by specifying the magnitude of extensions at the ends of the path with the help of the parameters of the group **End extensions**.

In the dialog of the properties window for the weld created by three guides, it is also possible to specify the contur type of the weld (**Convexity**).



Welds on edges, 3D paths, 3D profiles, operations, and 3D welds

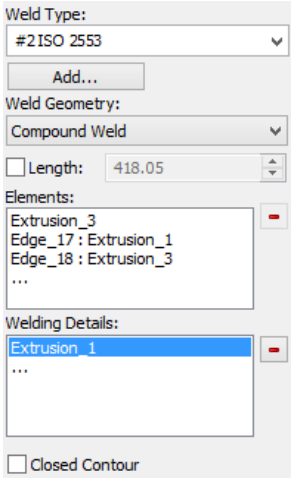
To create a weld on edges, 3D paths, 3D profiles, operations and other existing 3D welds, you need to set the option **Compound Weld** in the "Weld Type" list. After that, the options to select the 3D model source objects will become available in the automenu:

	<C>	Select weld elements

The drop-down list of this operation contains the filters to select 3D objects – 3D welds, edges, 3D paths, 3D profiles, and operations.

The objects selected as the members of the 3D weld are entered in the "**Elements**" list in the command's properties window. To remove some 3D object from the selected, you need to pick it in this list by clicking, and then click the button .

The parts that will be connected by this created weld can be specified with the option:



	<D>	Selected Weldment Parts

You can additionally specify that the composed weld makes a closed loop, by setting the flag **Closed Contour**.

Defining Weld Length

The method of calculating the weld length is determined by the state of the **Length** flag. When the flag is off, the weld length is calculated by the system automatically. When the flag is enabled, the length value is defined by the user in the input box next to the flag.

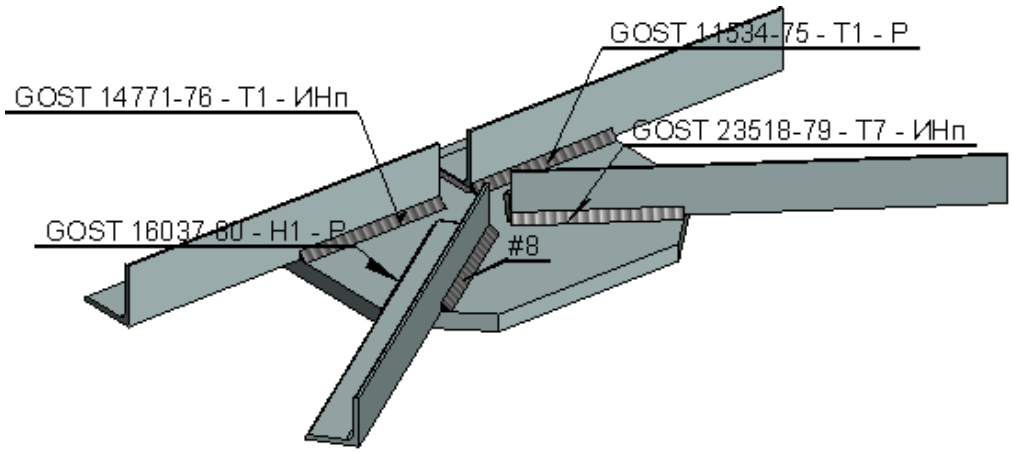
Creating 3D Weld Annotation

A 3D weld annotation is created by the same command as an annotation for a 2D weld:

Icon	Ribbon
	Assembly → Assembly → Weld → Weld Annotation
Keyboard	**Textual Menu**
<WN>	Tools > Weld > Weld Annotation

Working with this command is described in the "Weld" chapter of the "Fundamentals. Two-Dimensional Design" manual.

A 3D weld annotation is a 3D annotation, whose text is automatically generated based on the weld parameters.



The 3D weld annotation can also be applied on a 2D projection as a 2D annotation, when it is selected within the command of creating the annotations for the projection line of the 3D weld.

Creating Weld Table

How to create a weld table is described in the "Weld" chapter of the "Fundamentals. Two-Dimensional Design" manual. Here, we will just remind that the table of welds is created using the command **WT: Create Weld Table**:

Keyboard	Textual Menu	Icon
<WT>	Tools > Weld > Weld Table	

By using this command you can create the table of welds in the current T-FLEX CAD document, or export it into an MS Excel document.

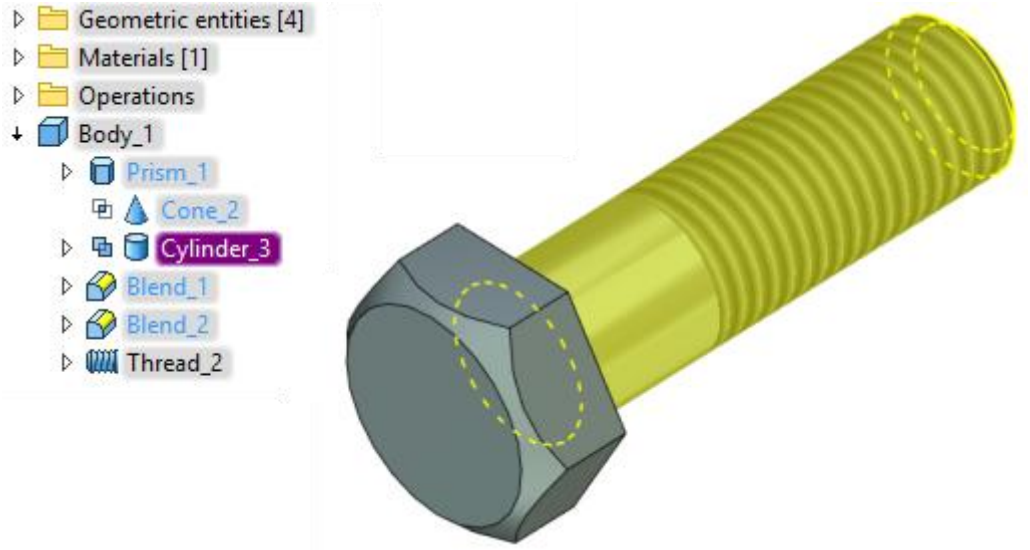
All 2D welds and the standard 3D welds are entered in the welds table.

PRIMITIVES

MAIN CONCEPTS AND CAPABILITIES

Commands united in the **Primitives** group allows to create basic geometric bodies: **Box, Cylinder, Cone, Sphere, Torus, Prism** and **Pyramid** without additional construction elements creation.

You can specify location and general dimensions for primitives and apply Boolean operation for them immediately. This allows you to quickly create models based on primitives.

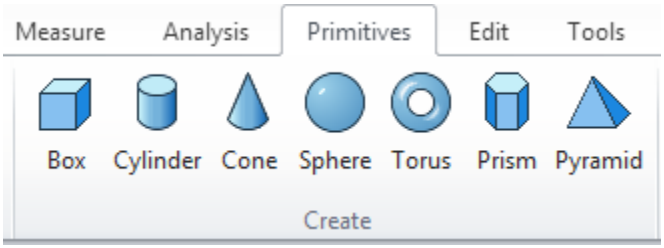


CREATING OPERATION

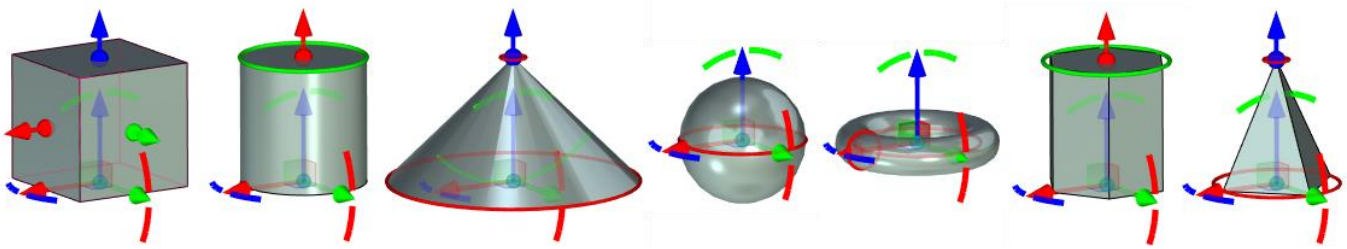
A drop-down list of commands for primitives' creation can be found in the **Primitives** group:

Icon	Ribbon
	3D Model → Special → Primitives
Keyboard	Textual Menu
	Operation > Primitives

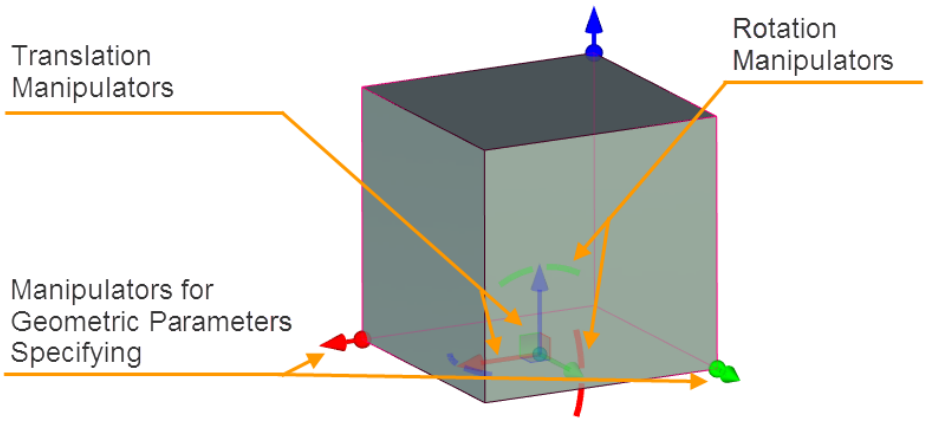
There is also an additional Ribbon tab for primitives:



When you select a command from the list the corresponding body appears in the 3D scene.

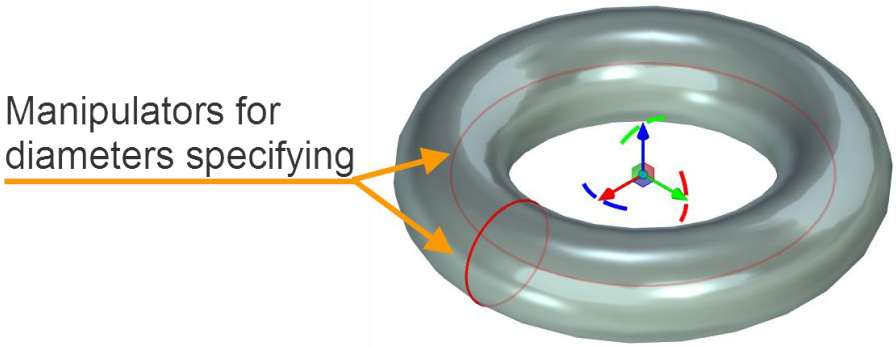


Translation and rotation manipulators and manipulators for geometric parameters specifying are available for the bodies.



All transformations that are perfored using translation and rotation manipulators are displayed in the properties window in the **Transformations** section.

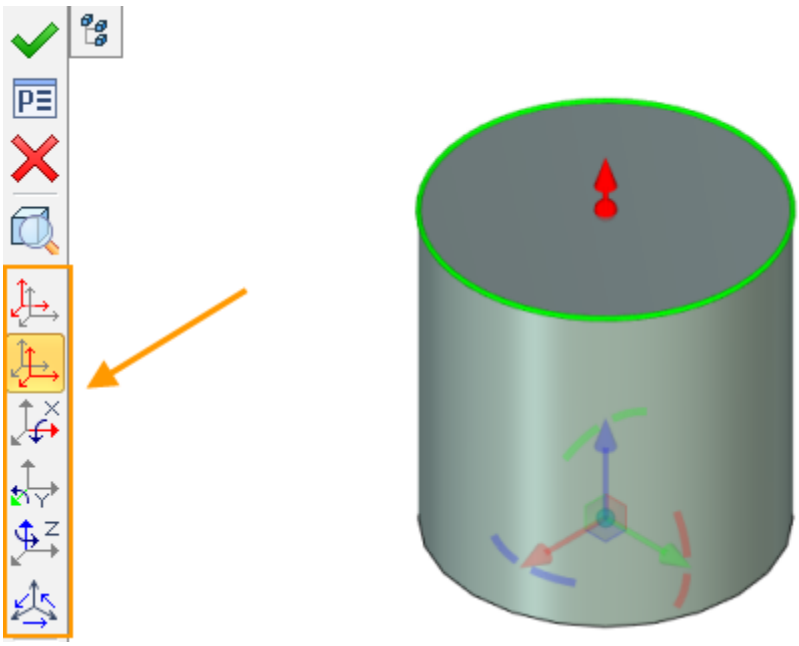
Special manipulators are used to change diameters of rotation bodies.



OPERATION OPTIONS

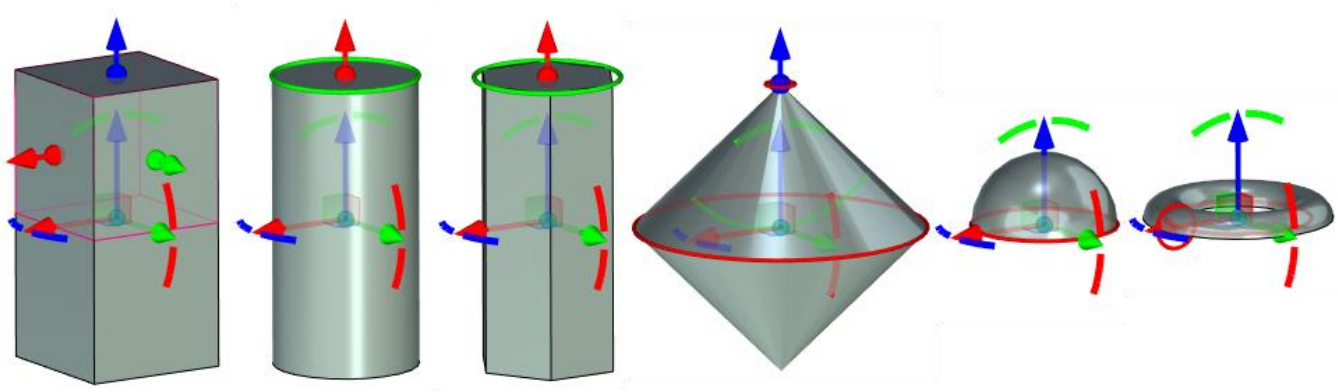
Commands for LCS creation and transformation are available in the automenu of each primitive type.

Source LCA is associated with the primitive and allows you to set its position in the scene. Source LCS can be merged with target LCS, which already exists in the scene. Their axes will be combined.

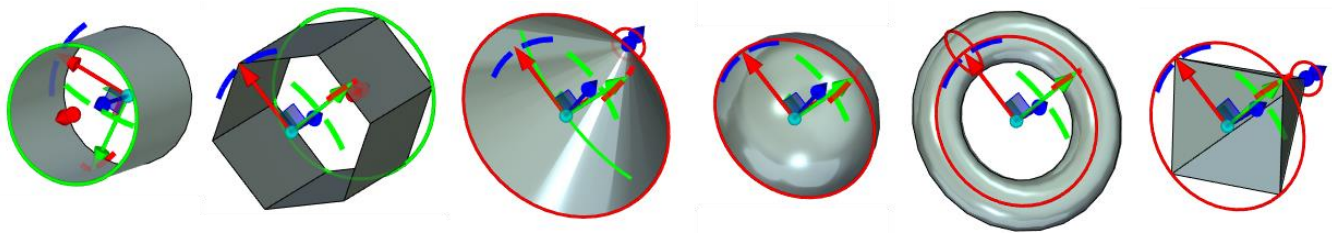


More information about LCS transformation and manipulators for translation and rotation can be found in "Translations" chapter.

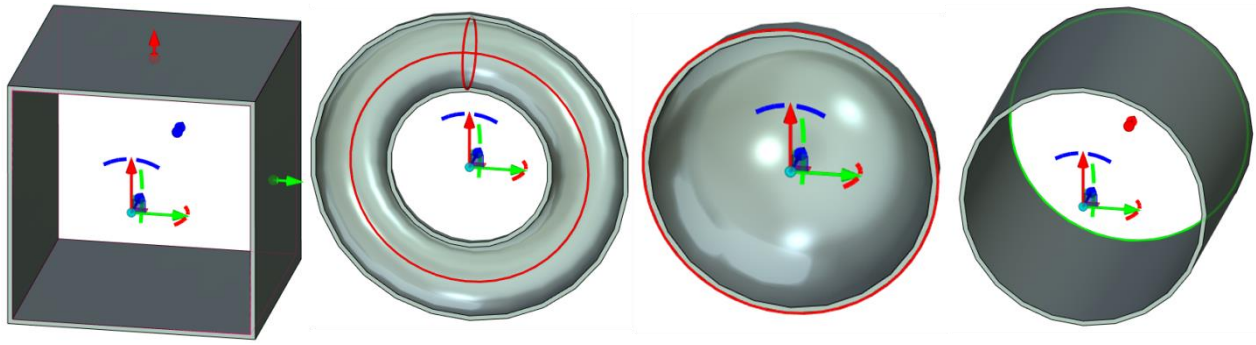
Symmetry option exists in automenu for all commands from **Primitive** group, except **Pyramid** command. A single body that consists of two symmetric parts is created after its activation. Additional symmetry operations are not performed in this case. Source coordinate system of the primitive is moved to the center of the body. The option is active for **Sphere** and **Torus** by default.



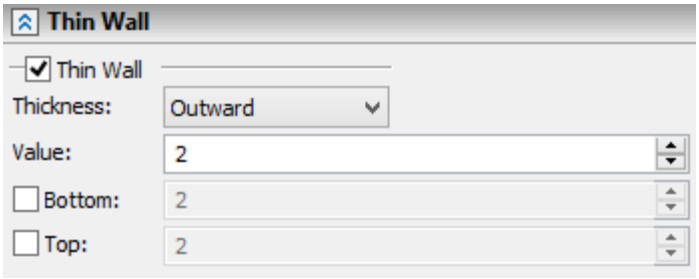
A **Thin wall** option is available for all commands from **Primitive** group. Bases of the bodies are removed and side faces are left.

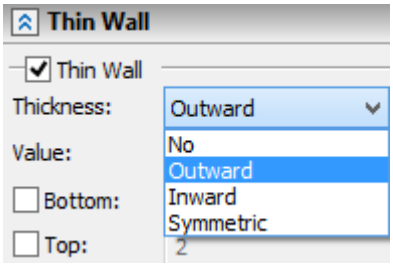


Possibility to create thin wall element is available for **Box, Sphere**, **Cylinder** and **Torus**.



Thin wall element parameters can be set in **Thin Wall** section of **Properties** window.



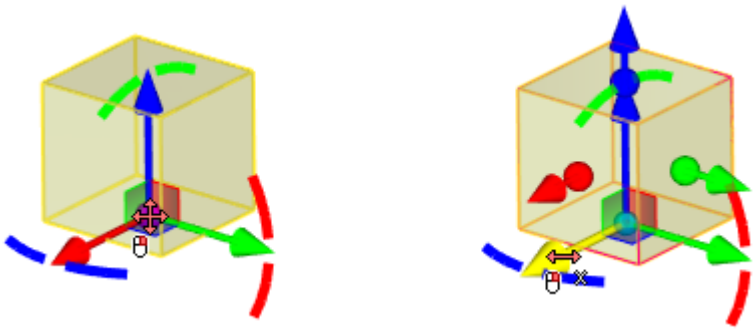


Bottom and top of thin wall element can be set for **Box** and **Cylinder** primitives.

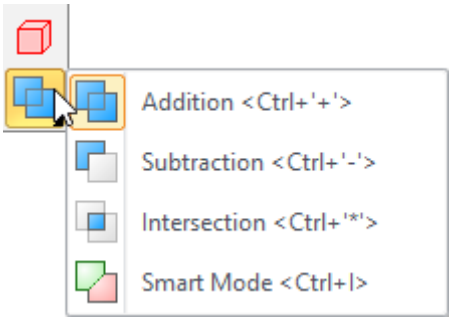
> More information about thin wall elements can be found in “Extrusion” chapter.

You can change way of primitive translation in the scene using **Move** option. If the option is enabled, the primitive follows the cursor. If the option is disabled, you need to use draggers for primitive translation.

	<8>	Move



Options for Boolean operation creation are also available in the automenu.



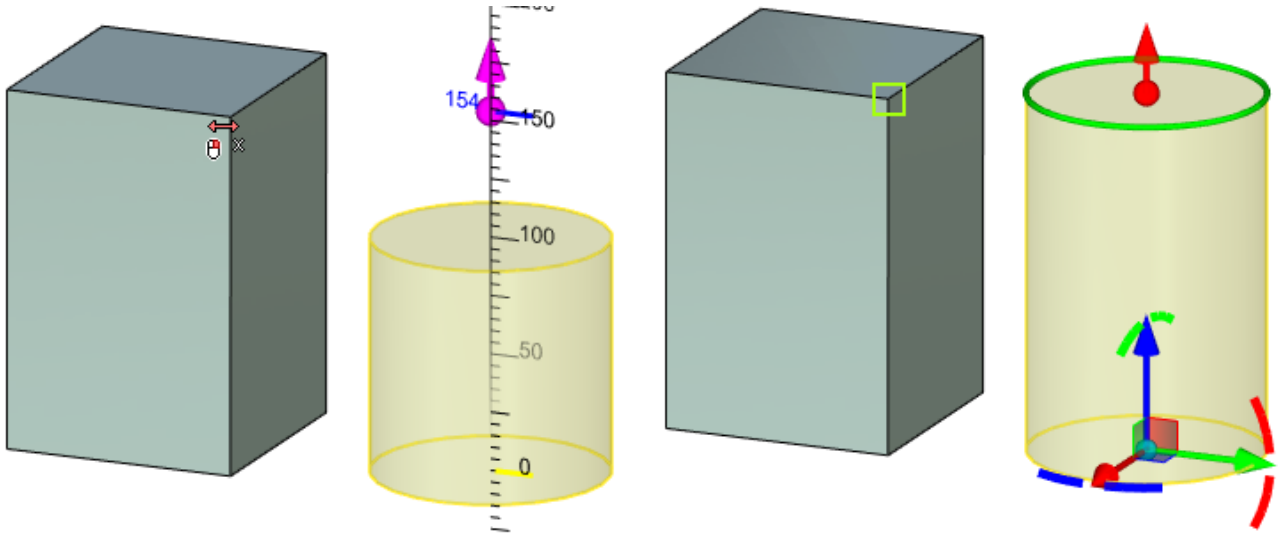
More information about Boolean operations can be found in “Boolean operation” chapter.

Additional Features

Link Dimensions to Existing Geometry

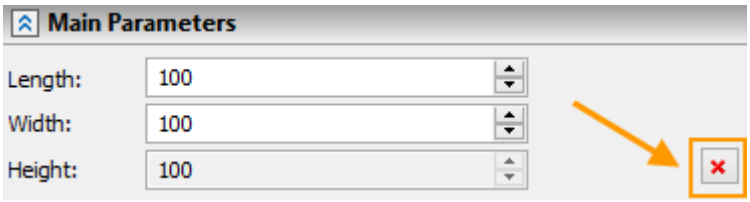
There is possibility to link primitives’ dimensions to the existing geometry.

For this purpose use you need to on the corresponding manipulator of the parameter and to select a point on the existing geometry using.



A primitive will transform according to the changes in the linked body.

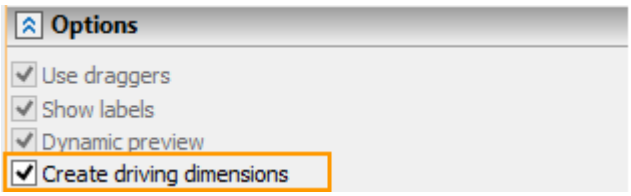
You can break a link using special option. The option appears near the linked parameter.



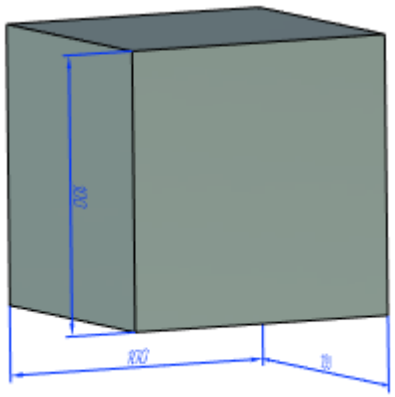
You cannot create a link if a source coordinate system of the primitive has been changed using **Select or Create source LCS** option. Warning appears in this case.

Create Driving Dimensions

You can create driving dimensions for primitive using option **Create driving dimensions** in the **Options** section.



Driving dimensions can be created for **box**, **cylinder** and **prism** primitives.



PRIMITIVES GROUP OPERATIONS

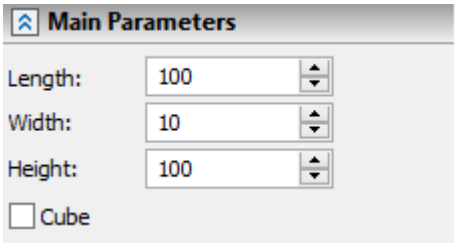
Unique set of parameters is displayed in the **Properties** window for each operation from **Primitives** group

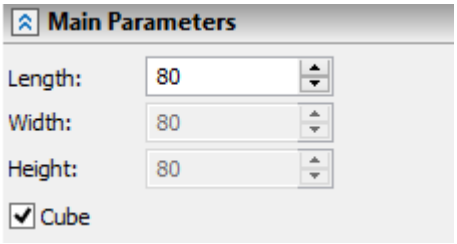
When you change primitive dimensions using 3D manipulators in the scene, corresponding parameters are also changed in **Properties** window. Value of changed dimension is displayed in the top of the window upon manipulator movement.

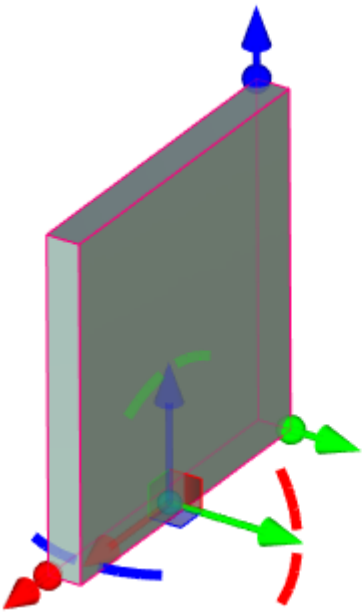
Box

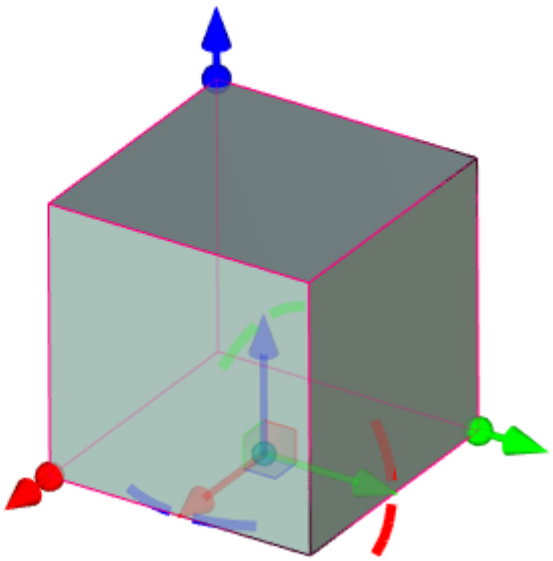
You can set the following box parameters: **Length, Width, Height**.

When the **Cube** flag is set only **Length** parameter field can be changed. You can specify length of all cube sides here.



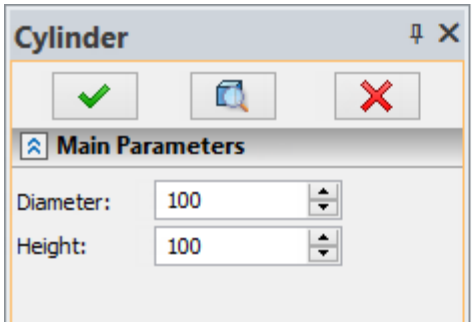


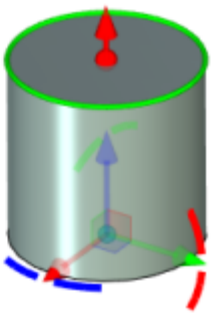




Cylinder

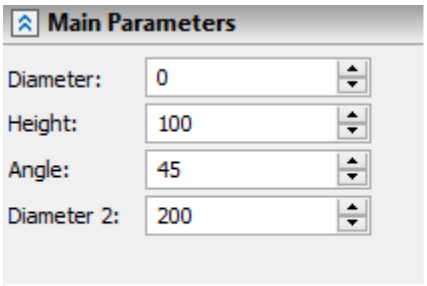
You can specify **Diameter** and **Height** for the cylinder.

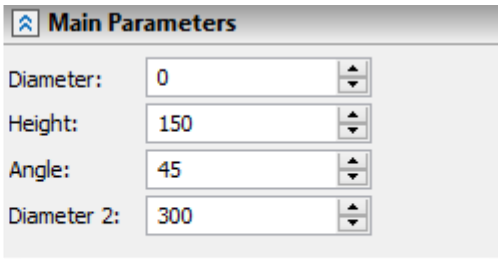


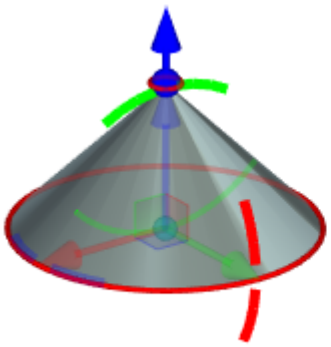


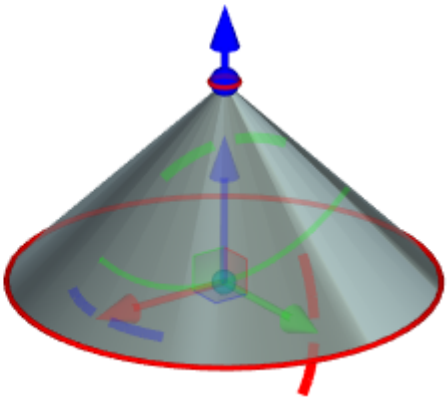
Cone

It is available to specify **Diameter**, **Height, Angle** and **Diameter 2** for the cone. **Diameter** defines cone apex diameter. **Angle** and **Diameter 2** values depends on each other. Angle value depends on height.



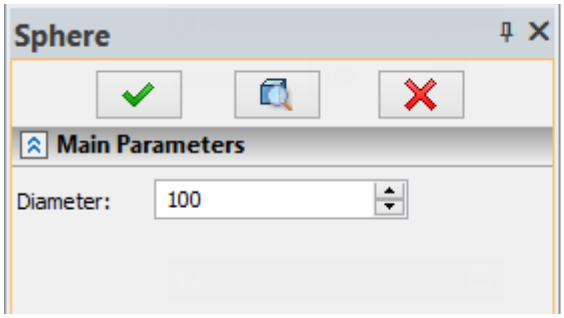


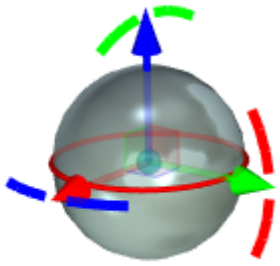




Sphere

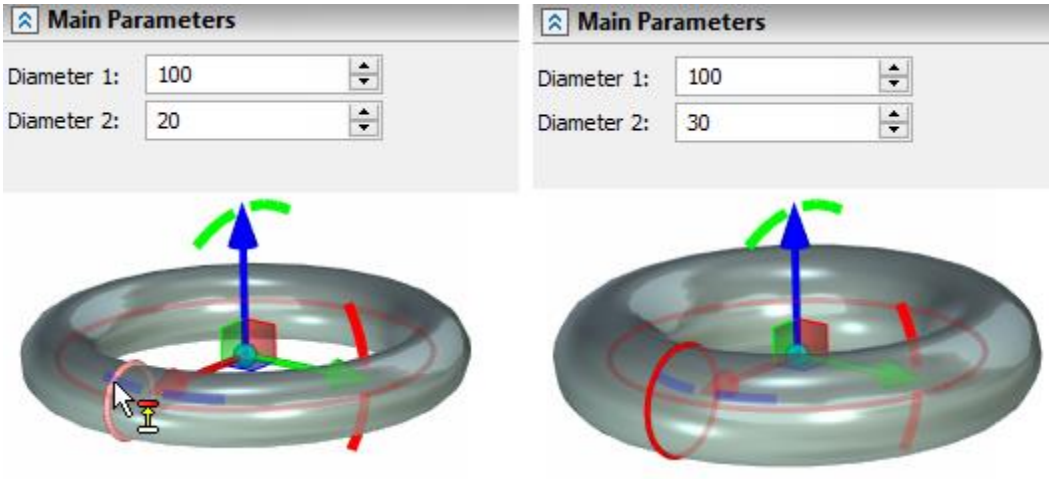
You can specify diameter for the sphere.





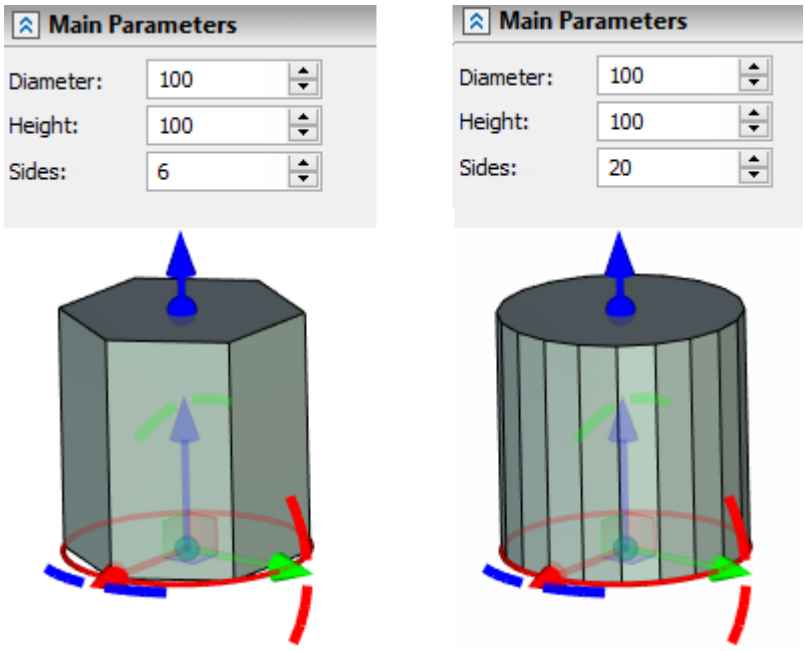
Torus

It is available to specify two diameters for the torus.



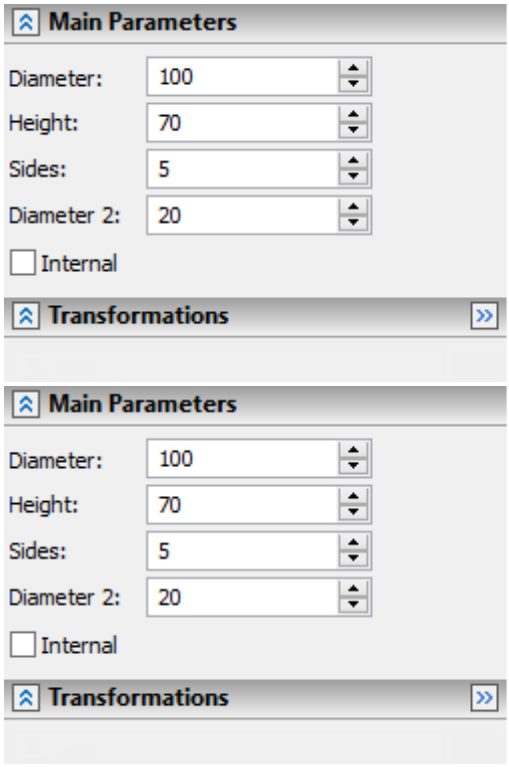
Prism

You can specify **Diameter**, **Height** and **Sides** for the prism. The maximum number of sides is 100.



Pyramid

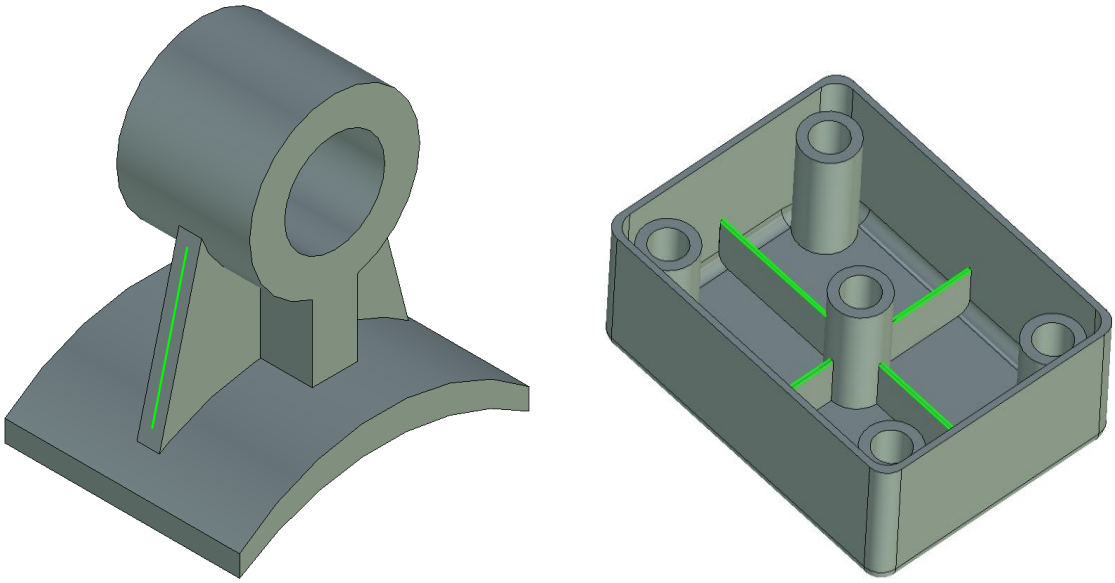
You can specify base diameter, apex diameter, height and number of sides for a pyramid. The maximum number of sides is 100.



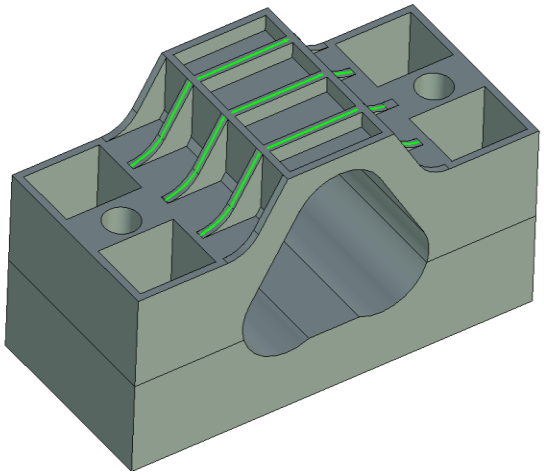
Rib

A rib is an element that is obtained by extension of a contour in the given direction until intersection occurs with the faces of the initial body.

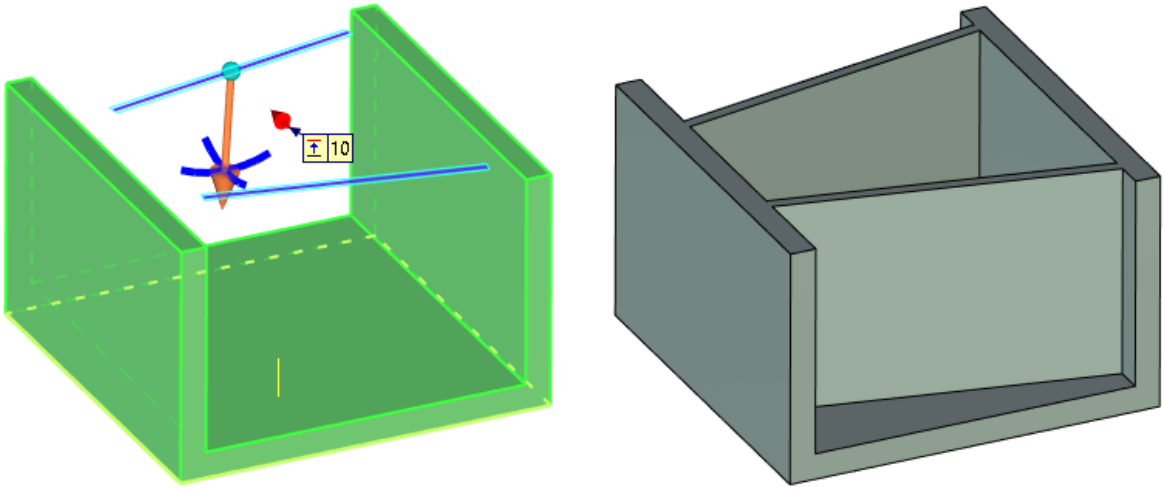
Ribs can be created on the basis of closed as well as open contours. The result of execution of this operation is a solid body obtained with a Boolean operation of union of created rib with the initial body.



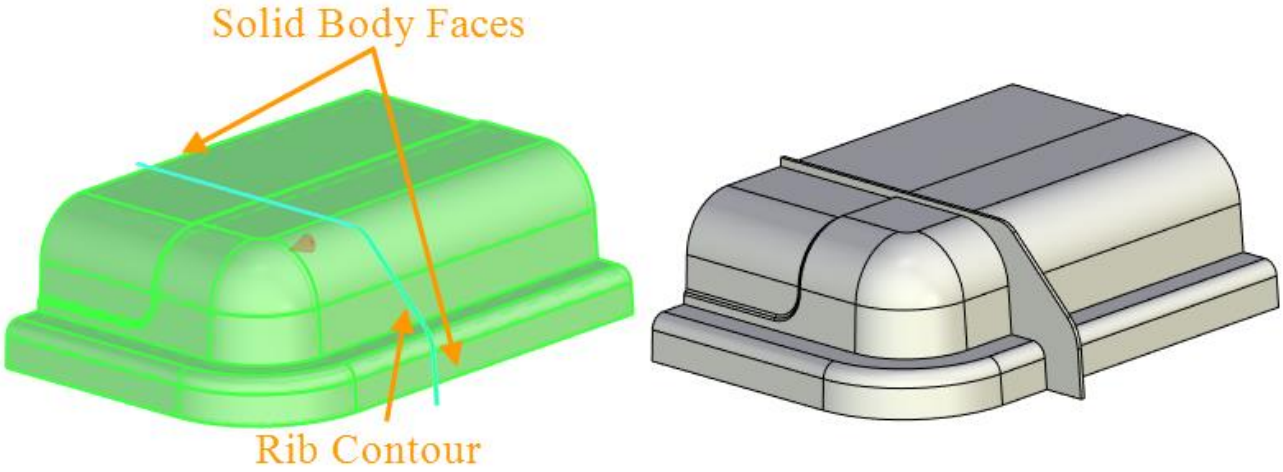
The user specifies direction of extension of the initial contour and its thickness. After selection of several contours a grid of ribs can be created. In this case the thickness for several objects is specified immediately for the entire grid.



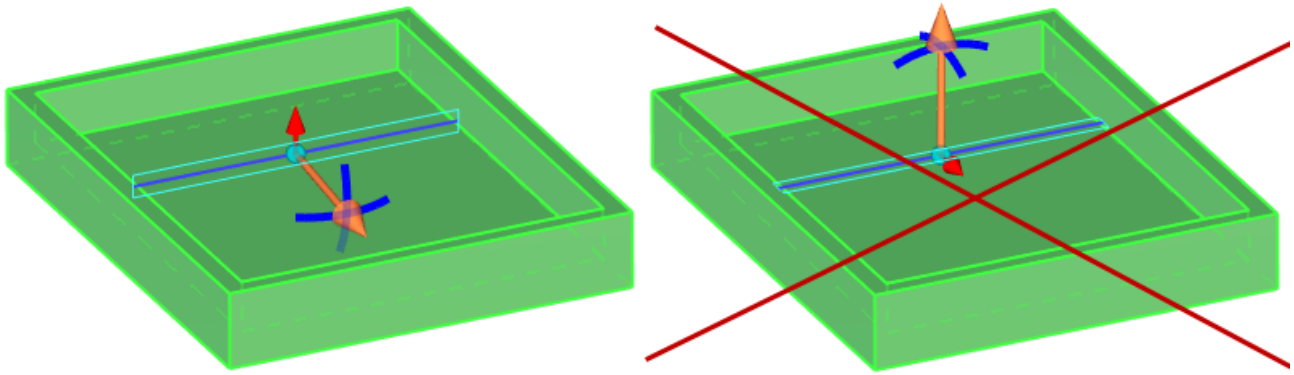
Contour may consist of several parts. In this case one direction to solid body is specified for all parts of the contour.



It is not necessary to limit rib contour with solid body walls. In most cases system can automatically determine the boundaries of the created edge. On the picture below the rib is limited with planes of the solid body faces.



Note that if borders can't be automatically determined the construction of the rib will not be possible.



CREATION OF RIB

Before construction of a rib, it is required to create a contour that will define the shape of this rib.

To create a rib we use the **3ST: Create rib** command. This command can be invoked via:

Icon	Ribbon
	3D Model → Modify → Rib
Keyboard	Textual Menu
<3ST>	Operations > Rib

To create the operation, it is required to execute the following sequence of actions:

- ✓ Select the source body;
- ✓ Select forming contour;
- ✓ Select direction of extrusion of the rib;
- ✓ Specify thickness of rib;
- ✓ Specify additional capabilities of the operation (not a necessary step);
- ✓ Confirm creation of operation.

Selection of Source Body

To create a rib it is firstly required to select the source body. After invoking this command the following option will automatically be activated in the automenu:

	<S>	Select body

Selection of contour

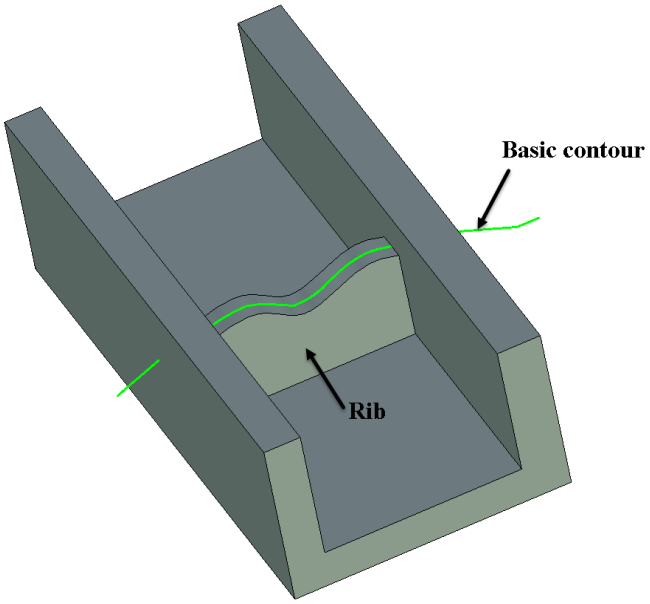
After selection of the source body the option of selection of the contour that defines the form of a rib being created is activated.

	<C>	Select forming contour

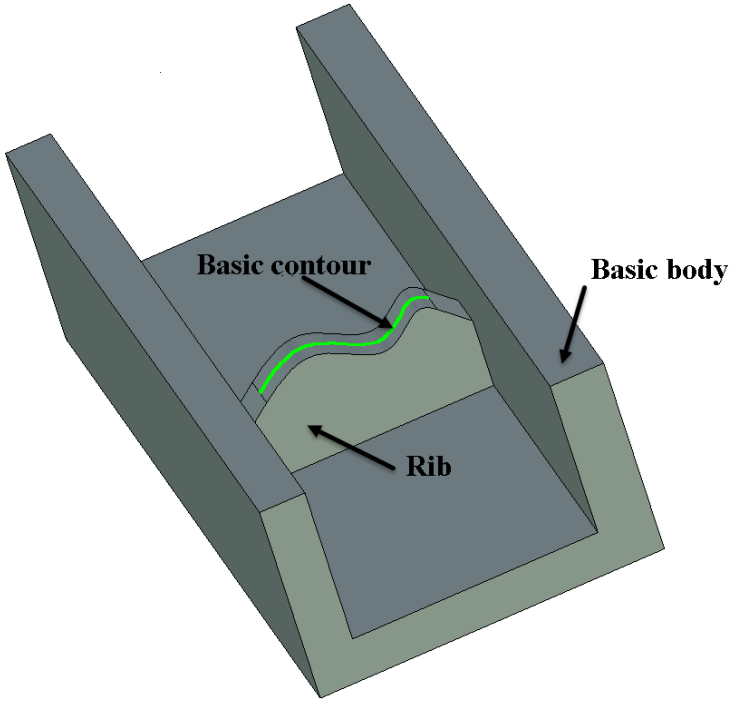
As a contour we can select 3D profiles, faces, edges, paths, loops. If there is only one profile in the scene, it will be selected automatically.

After the first contour has been selected this option remains active, and if necessary you can select several contours.

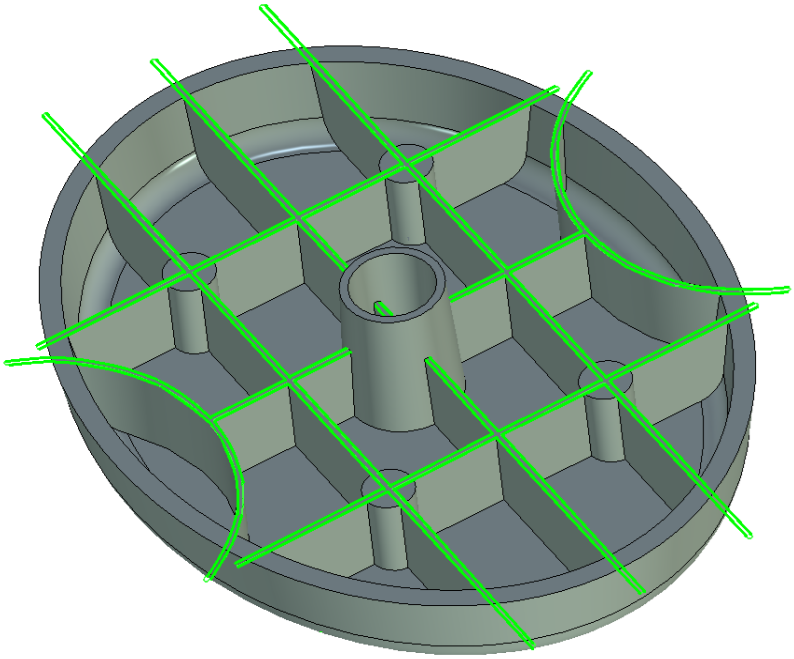
When contour extends beyond the limits of the source body it is cut to the required size.



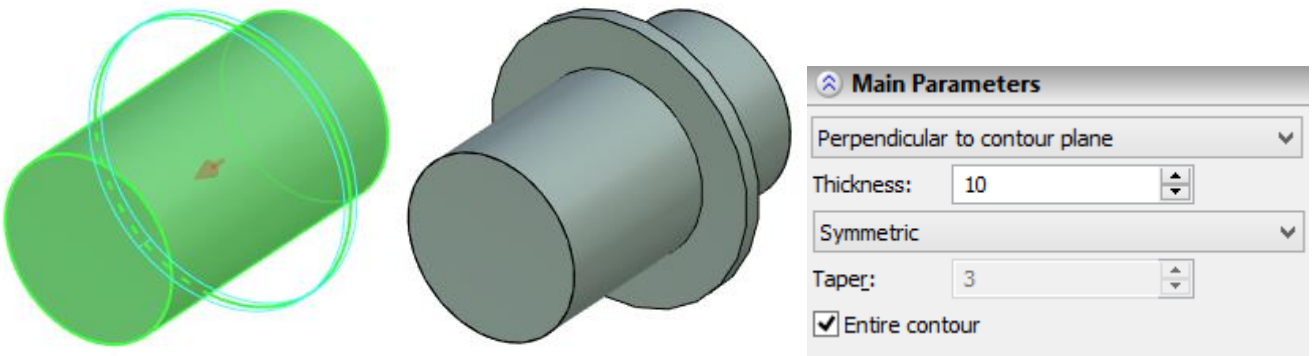
If the contour is open, it is extended until intersection occurs with the model, and the entire space between the surfaces of the part and the given contour will be filled with the material of the given thickness.



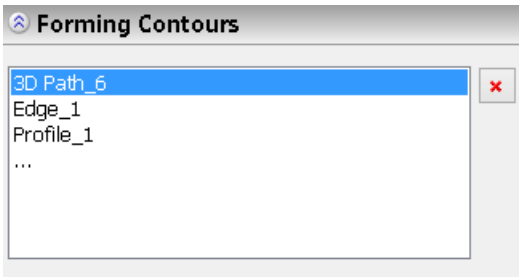
Contour, which consist of several parts, may create two ribs and more.



Entire contour. When flag is set, rib can be created outside of the body using the closed contour.



All of the selected contours are displayed in the «**Forming contours**» tab in the command's properties window.



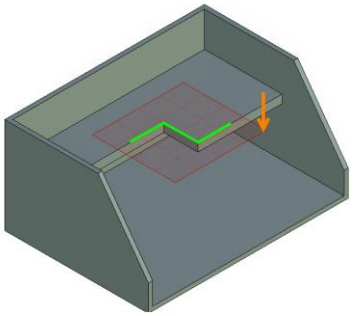
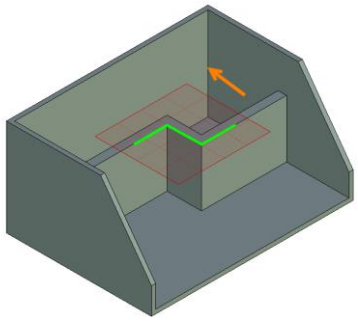
The button removes selected contour from the list.

Direction of Extension of Rib

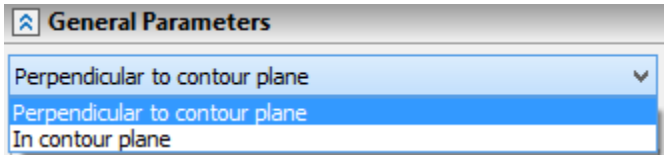
Extension of a contour can be carried out in two directions:

In the plane of contour

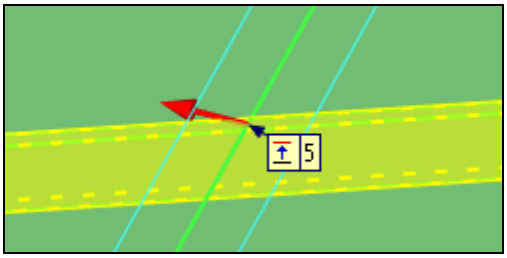
Perpendicular to the plane of contour

Direction	Position and shape of rib	Result of construction
Perpendicular to plane of contour	Surface of a rib is formed by extension of contour in the direction perpendicular to its plane	
In the plane of contour	Surface of a rib is formed by extension of contour in the direction of its plane.	

To choose the method of contour's extension it is required to use the «**Main parameters**» tab of the command's properties window.



Direction of contour extension is displayed in the scene as arrow- manipulator.



Contour Direction for a Straight Segment

If forming contour is specified by a straight segment, its positioning plane is determined automatically by the following rule. If segment presents part of a profile, system will use plane of the profile creation. If segment presents part of a path, positioning plane is defined by rotating the segment in global coordinate system to the nearest plane the segment coincides.

Contour extension direction is created according to the determined positioning plane.

To select contour extension direction manually, you need to use option:

	<D>	Select contour direction

Next you need to select an auxiliary element. If you select object that specifies plane – face, workplane, etc., then the direction will be a normal to this plane. Alternatively, you can select object that specifies direction – edge, coordinate axis, etc.

Specified contour extension direction will affect all selected contours defined by the straight segments.

To cancel direction selection use option:

	<K>	Cancel direction selection

Direction to Solid

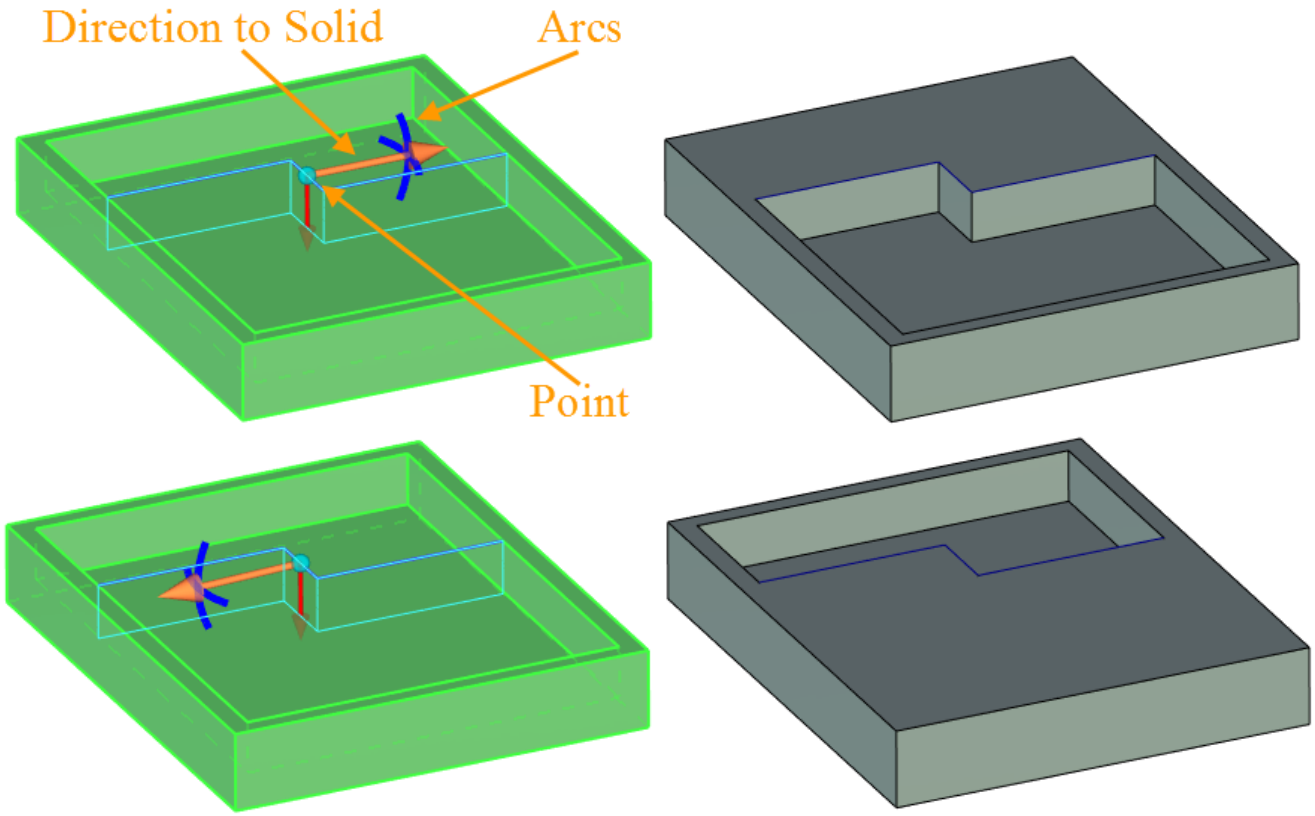
In addition to contour extension direction, you can specify direction to solid. It determines which part of the half-space, created by contour, will be transformed into the rib.

Special manipulator is used to specify direction to solid. It indicates direction inside the rib body from the point located on the contour. If contour is open, the point is located in the center of the contour line. For closed contour, the point position is chosen arbitrarily. For convenience you can move this point along the contour.

Direction to solid can be defined individually for each contour. To change direction you need to move the manipulator, pulling one of its arcs.

The manipulator allows you to set the direction to solid manually. Thus, system does not need to determine correct direction automatically.

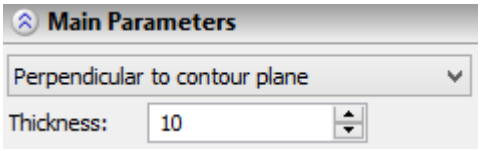
After the operation creation, all space between the walls of the source body and the contour in determined direction is converted into the solid rib of specified thickness.



Thickness of rib

When forming a rib, new solid material is added in the direction perpendicular to contour extension plane.

The value of thickness can be specified in appropriate field of the **Main Parameters** tab of the command's properties window or by means of manipulator.

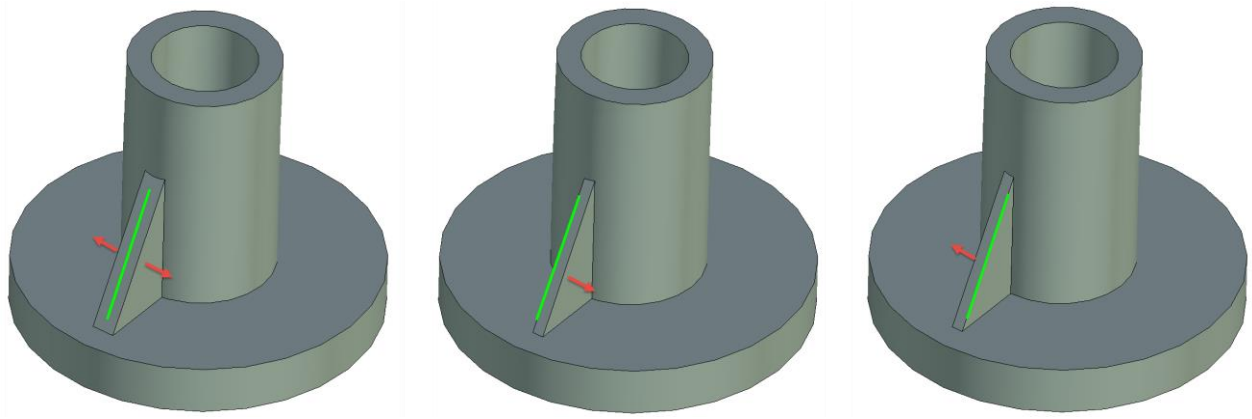


The direction of thickness adherence must also be specified for a rib:

Forward direction. Direction coincides with the direction of the normal to the contour's plane.

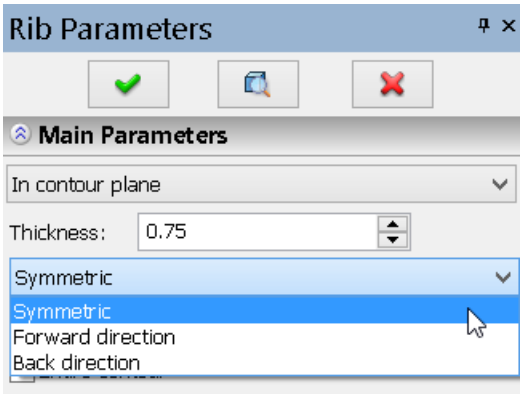
Back direction. Direction is opposite to the direction of the normal to the contour's plane.

Symmetric. Thickness is added to the rib equally in both directions with respect to the plane of the contour.



Symmetric | Forward direction | Back direction

Direction of thickness adherence can be selected from the drop down list on the **Main parameters**» tab:



In case if the specified value of the thickness parameter becomes larger than the value of the contour's length, the corresponding field will be renamed to **height**.

Additional capabilities of operation

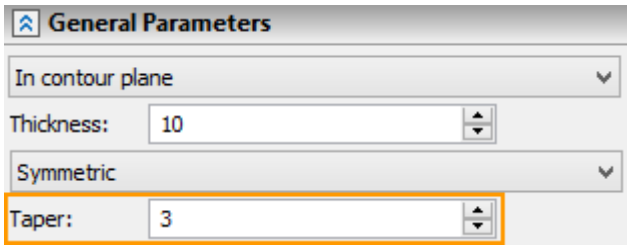
Taper of Rib's Faces. If it is required that the faces of the rib have a taper, the following actions must be executed:

- ✓ Select direction of a taper by using the option:

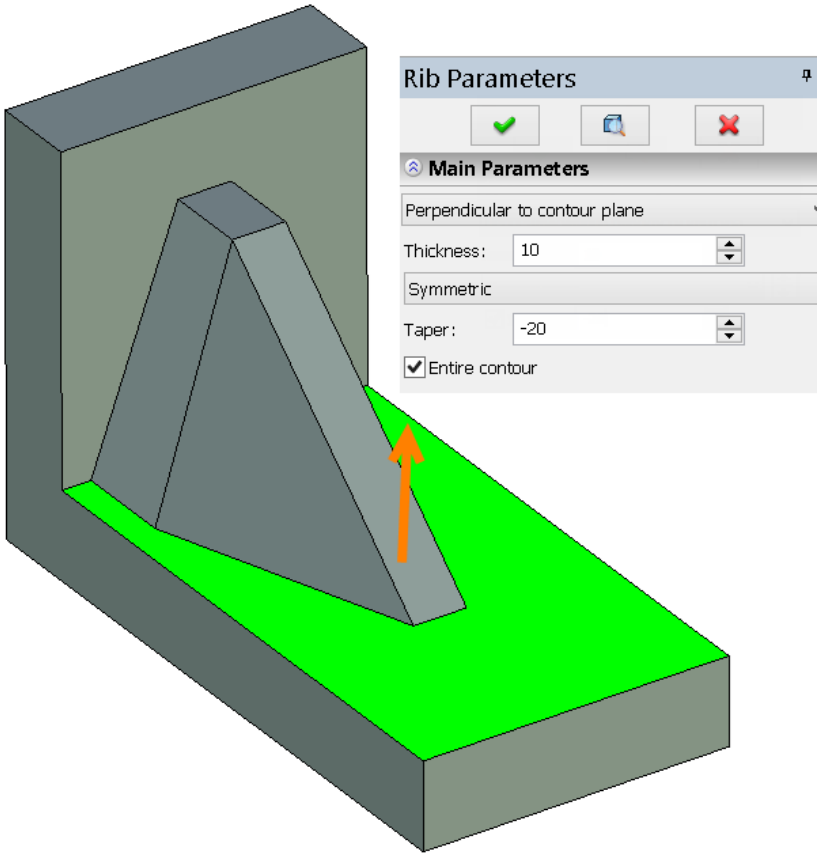
	<T>	Set tapering direction

When this option is enabled the geometric element is selected that specifies the direction of the taper.

- ✓ Enable the **Taper** flag on the **Main parameters** tab of the command's properties window.



✓ Enter the value of the tilt angle in the field next to the flag.

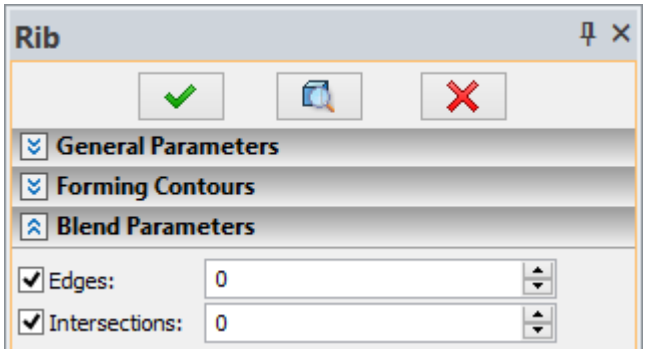


In the example the normal to the selected face serves as a geometric element that specifies the direction of the tilt.

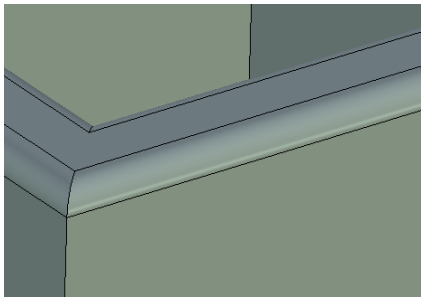
To cancel selection of tapering direction use option:

	<R>	Cancel taper direction

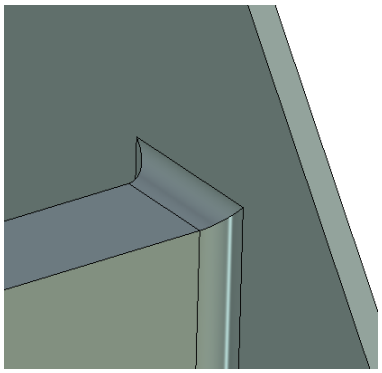
Blending ribs. To specify parameters of ribs blending it is required to use the **Blend parameters** tab in the command's properties window.



To smooth out the side faces of a rib on its ends it is required to enable the **Edges** flag and input the required value of blend radius in the field next to the flag.



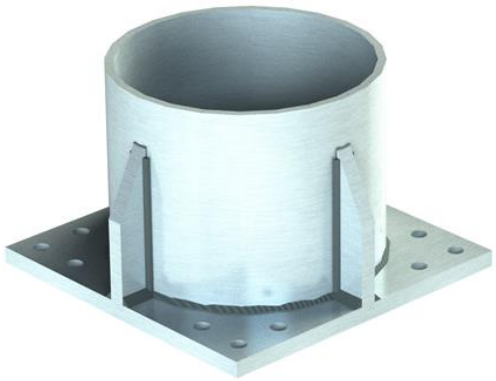
To smooth out intersections of a rib with the source body it is required to enable the **Intersections** flag and input the required value of blend radius in the field next to the flag.



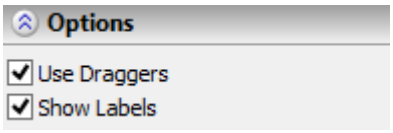
Separate body. Allows to create a rib as a separate body.

The option, in particular, allows you to create welded structures using ribs.

This method may be used for example when modeling welded structure of flange and pipe.

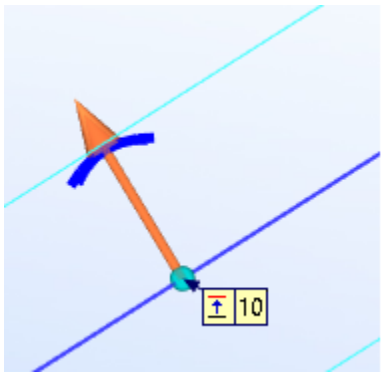


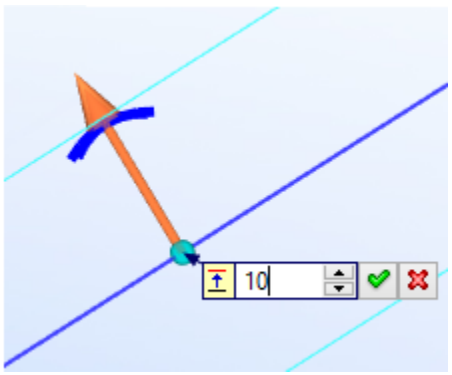
In the **Option** section, you can set options for displaying auxiliary elements.



Option **Use draggers** allows you to enable/disable the display of manipulators that specify the direction to solid and rib thickness.

Option **Show labels** allows you to enable/disable the display of the labels that output rib thickness.





RIB EDITING

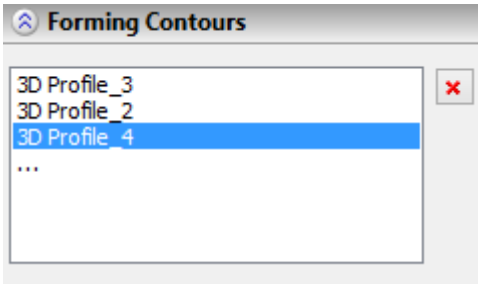
To edit a rib use command:

Keyboard	Textual Menu	Icon
<3EST>	Edit > Operations > Rib	

To reset thickness and direction to solid, you need to select the rib in edit mode, and then use option:

	<F4>	Execute edit command

In the "**Forming Contours**" section of the properties window, select contour and reset its parameters.



More information about this command can be found in the chapter **Editing**.

3D Assemblies

3D Assemblies Creation

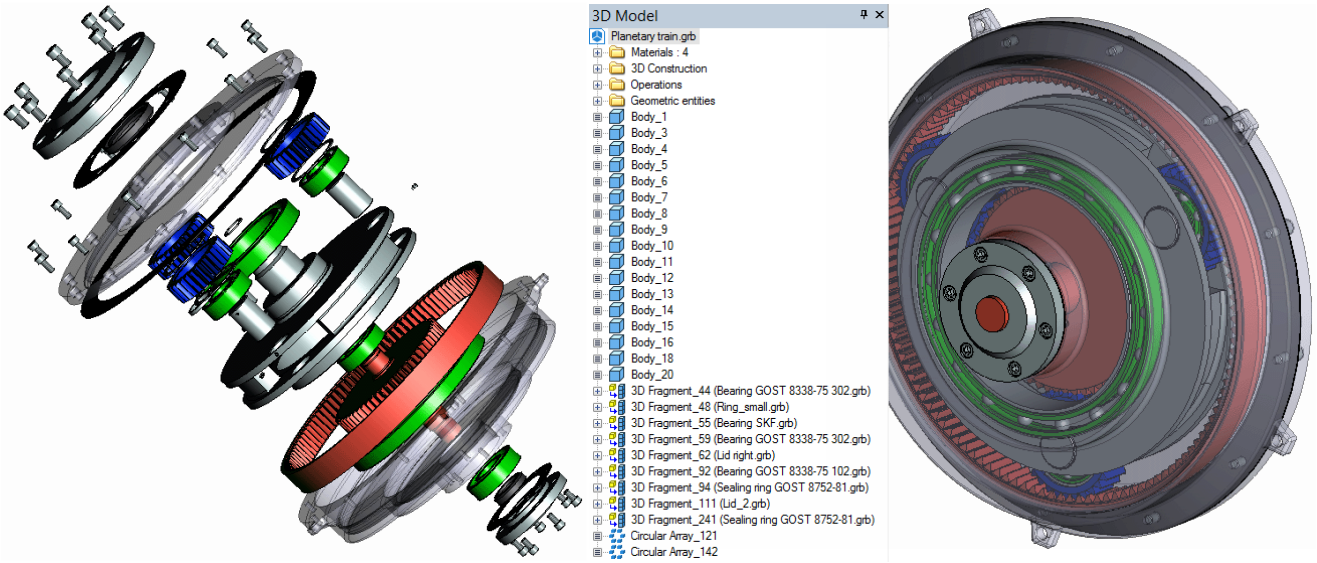
This chapter describes the process of creating three-dimensional assemblies in T-FLEX CAD. To design a three-dimensional assembly, you may need to use various commands and operations. Here we describe in details the techniques of working with 3D Fragments, Parts and Configurations.

General Information

What is a Three-Dimensional Assembly?

A three-dimensional assembly in T-FLEX CAD is a model that incorporates geometry of other 3D models stored in separate files. An assembly component can be a 3D model created in T-FLEX CAD, or a model from another system imported in T-FLEX CAD in one of the supported data exchange formats. To be clear, the engineer may think of an assembly (by its definition) as a model built from elements-operations within one document, rather than composed of separate part files. T-FLEX CAD does support such functionality. However, we will not call such document an assembly, since it does not refer to geometry taken from other documents.

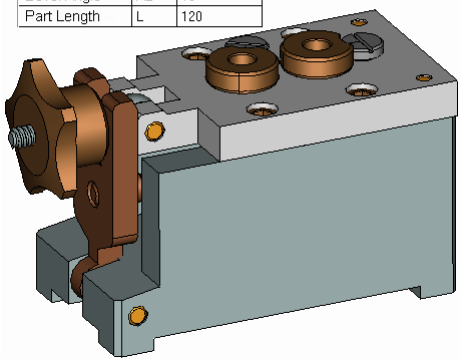
An assembly document always maintains relations with assembly element (part) documents. As you work with an assembly, the system constantly tracks the state of the involved files. In the event of changes in an assembly component file, the system queries the user for updating the data. Each updated component will be regenerated and reloaded into the assembly. An assembly cannot be used without its components, while the each file involved in the assembly can be a fully independent document and, in turn, be an assembly itself. The number of nested levels of subassemblies is unlimited in the system.



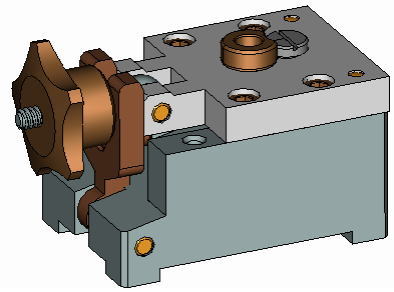
Use of Parametric Models

Any model in T-FLEX CAD can be built in such a way, that it can be modified by using variables. Using such a model as an assembly component brings significant benefits. This is particularly effective, when such model is entered multiple times in one or several assemblies with different sets of external variable values. The same parametric part file can serve several assemblies at once. Parametric components of the assembly can be managed from within the assembly document.

Comment	Name	Value
Part Hight	H	70
Part Width	B	40
Diameter	D	16
Bevel Angle	AL	10
Part Length	L	120



Comment	Name	Value
Part Hight	H	50
Part Width	B	40
Diameter	D	10
Bevel Angle	AL	12
Part Length	L	90



Creating BOMs

Each assembly component may contain data within for automatic creation of various kinds of reports and Bills of Materials (BOMs). However, this chapter will not concentrate on creating BOMs. Detailed information on this topic can be found in the chapter “Bill of Materials”.

Body Intersection Check

One of the typical tasks that require creation of an assembly is checking the model for assembling compatibility, checking all bodies of the assembly against mutual penetrations. T-FLEX CAD provides special tools for solving such tasks. You can start checking two selected bodies or all bodies in the model for intersections, using such tools. You can also measure particular geometrical elements or relations. For detailed information on this topic, refer to the chapter “Model Geometry Analysis”.

Assembly Design

There are two main techniques to creating an assembly distinguished by the way of creating their components.

In many cases, it is convenient to take an already prepared model and insert it in the assembly by defining its position in the assembly. This assembly design technique is conventionally called **“bottom up”**. In this framework, the components of a T-FLEX assembly are called **3D fragments**. This approach is convenient for designing complex assemblies or standardized mechanisms made to large extent of standard parts. The inherent advantages of this approach are as follows.

- ✓ Designing parts in a separate file allows simplifying and subdividing the overall design process of a complex product.
- ✓ The same 3D fragments are available for reuse in different assemblies and for including into libraries of frequently used parametric 3D fragments.
- ✓ 3D fragments can be created with the provision for automatic positioning in the space based on 2D fragments and the workplane (so-called "layout").

Nevertheless, it is often easier and clearer to design a part once you have the idea about its position in the assembly, and its size, based on geometrical elements of other parts in the assembly. We will call this design technique **top-down**, while the assembly components, whose geometrical base is within that same assembly - **Parts**. An assembly model can be created "from scratch", as the engineer defines the general layout of the assembly unit. In the next step, separate elements are identified and refined, either in separate files or in the assembly context. This approach in some cases simplifies defining element attachments to each other, providing parametric relations between those. Use of the "top-down" technique sometimes allows avoiding use of external variables. The part parameters in this case are defined by references. Changes at low levels of the hierarchy of the related parts trigger automatic adjustment of all child parts in the assembly. Besides, the values of the original parameters of the model can be derived directly from the assembly context, based on the reference elements. For example, one can make holes in the cap based on the holes in the body. In this way, the engineer deals with elements belonging to the fragments, while seeing the part surroundings and having access to elements belonging to the assembly. This mode will further be referred to as **working in the assembly mode**.

The inherent difference in the two approaches does not prohibit combining both when creating assembly documents.

Assembly Creation from 3D Fragments. "Bottom-up" Design

How 3D Fragments Work

In the approach "from part to assembly", the assembly is composed of 3D fragments. In this case, it is implied, that the bulk of work on creating the part model is already done, and what is left is just assembling it. The main steps for assembling a 3D fragment are as follows:

1. Select the 3D model file to use as the 3D fragment.
2. Attach the 3D fragment to the assembly model.
3. Define the values of the fragment external variables and parameters.

Next, let's review the main concepts of 3D fragment insertion.

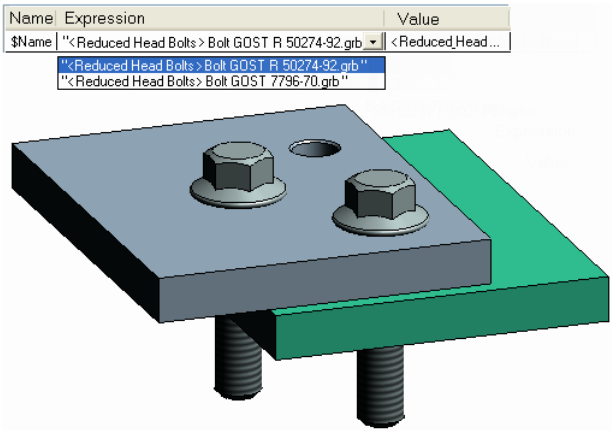
Relation Between Assembly and 3D Fragment File

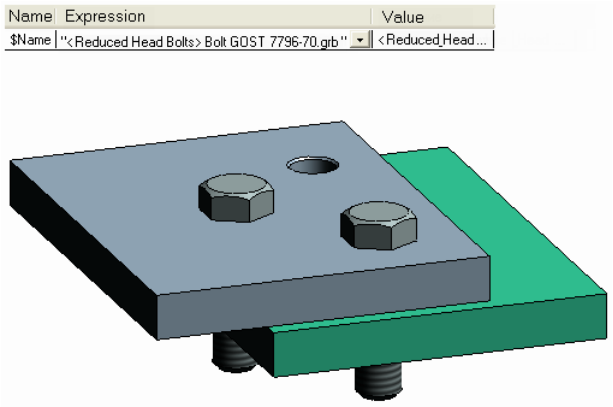
There are two ways of selecting a 3D model file. The first is by explicitly selecting the model file. The second way allows using the existing 2D fragments within the assembly drawing, that have the respective 3D models. This is possible in the case when the 2D fragments were already used in the current document, as, for instance, for initial creation of the drawing assembly.

The system always maintains the relation between the 3D fragment file and the assembly file. The assembly file keeps, among each 3D fragment parameters, the reference to the 3D fragment file. T-FLEX CAD automatically tracks changes in the 3D fragment files. If necessary, it reloads the 3D fragment data into the assembly. To do this, the 3D fragment is searched using the store the reference. The reference path to the fragment file can be full or relative. To provide greater system flexibility, especially when porting models from one file system to another, we recommend using relative paths for the references. When writing the relative paths, the assembly file and its location in the file system are used as the guide. Another way of creating relative references relies on library configurations.

Fragment with Variable Reference

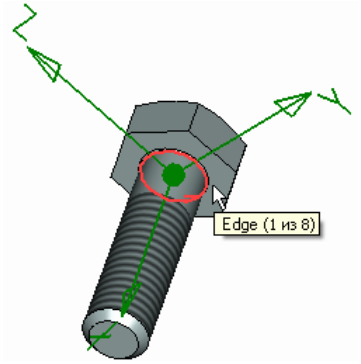
Just like in the case of using 2D fragments, a textual variable can be used as the reference path for 3D fragments being assembled. The value of such variable is the path to the file of the currently used 3D fragment. Depending on some conditions, this value can vary, becoming the reference to a different 3D fragment file. This allows creating assemblies with changing elements. There is one limitation on use of 3D fragments. All 3D fragments used in this scheme must have the original coordinate systems with the same name or be defined in the world coordinates.



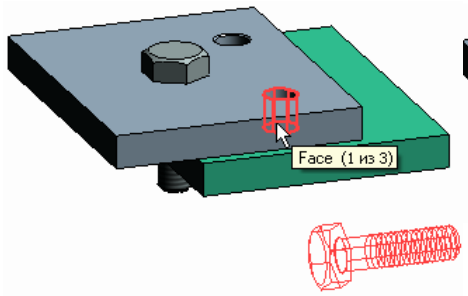


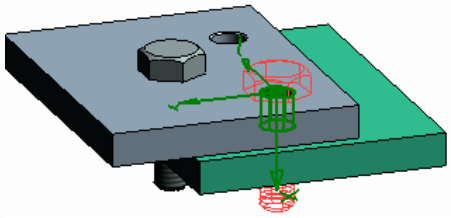
3D Fragment Attachment

To insert a 3D fragment model in the assembly model, you need to decide on the attachment. Attaching a 3D fragment means defining two coordinate systems – the source and the target one. One of the systems belongs to the 3D fragment, while the other - to the assembly. The 3D fragments geometry positioning is fully defined in the space upon placing the source coordinate system over the target one. The source and the target coordinate systems are usually represented by assigning or explicitly creating a Local Coordinate System (LCS) at the time of assembling the 3D fragment. If no LCS is specified, then the World Coordinate System is used instead.



Create source LCS by selecting edge

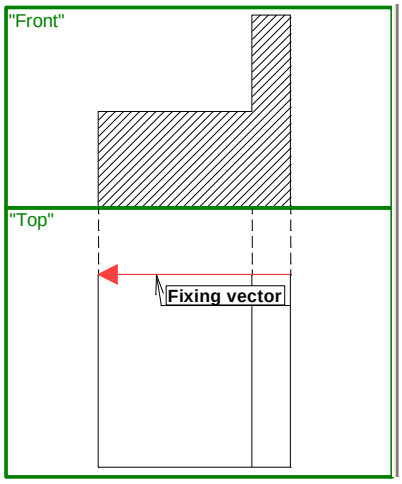


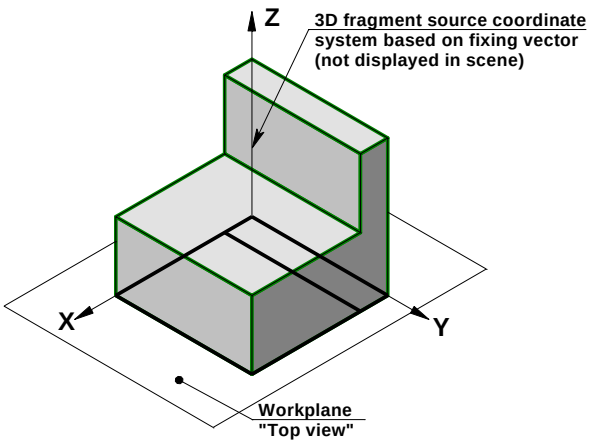


Create target LCS by selecting cylindrical face

In complicated cases, when fixing by coordinate systems cannot provide correct positioning of a 3D fragment, we recommend using Mates mechanism (see below the detailed description of the Mates mechanism).

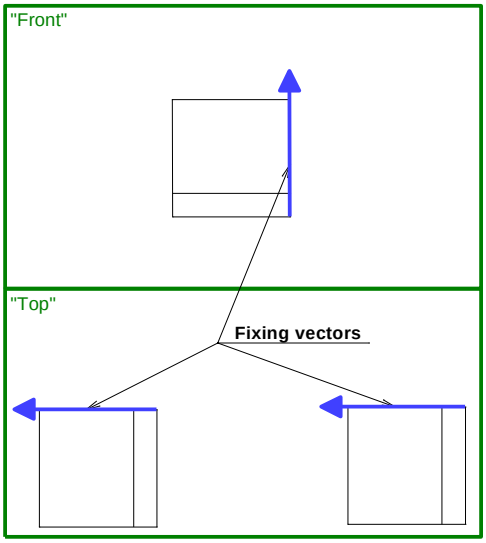
There is yet another way of defining the position of the 3D fragment based on the 2D fragment. This is a technique called “layout”. In this case, the 3D fragment orientation is done according to the 2D fragment fixing vector. The fixing vector must reference a workplane. This results in two coordinate systems. The source system is defined based on the fixing vector and its related workplane in the fragment document. The target coordinate system will be defined based on the final position of the fixing vector in the page of the assembly drawing and the selected workplane.

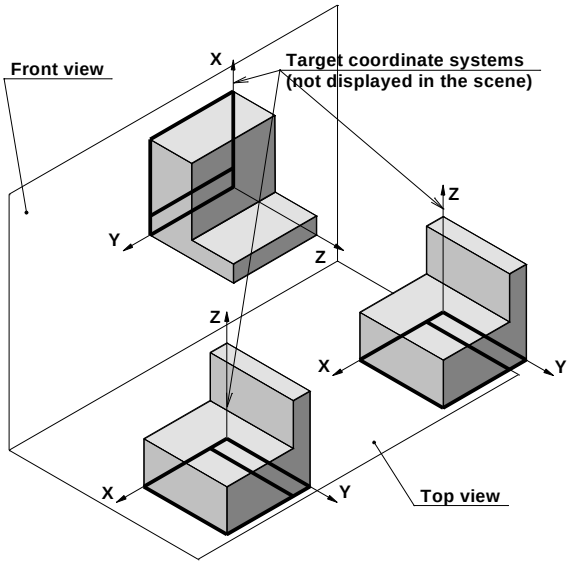




The X-axis of the coordinate system coincides with the fixing vector and lies in the workplane. The Z-axis is normal to the workplane. The Y-axis is the cross product of the two. In this way, no new LCS is created for attachment. The three-dimensional data for the 3D fragment are loaded from the same 2D fragment file. This means, the 2D fragment is used for creating the plan (the drawing), while the 3D fragment - for building the three-dimensional model. This technique is convenient in some special design cases, when you need to place several three-dimensional objects on one level (plane). This can be, for instance, the task of placing furniture in premises or equipment in a workshop. The three-dimensional objects placing

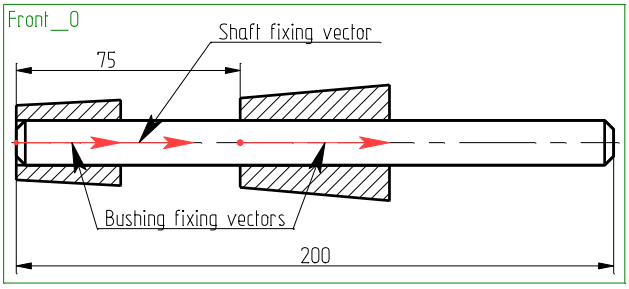
can be controlled in this case directly from the drawing by moving the respective 2D fragments in the plan.

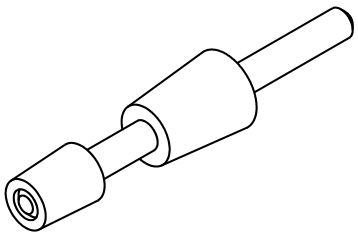




If the 2D fragment is created on a page without workplane, then the system will automatically perform the transformation for carrying the X-axis over the page of the selected workplane. Meanwhile, the fixing vector can be thought of as if carried over the page of the workplane with the same coordinates in the page as it had in the original page of the document. Further constructions are done as usual.

This capability can also be successfully used when creating common assembly drawings. The following diagram shows a trivial model based on a two-dimensional assembly, whose fragments were attached using the fixing vectors. No coordinate systems for attaching 3D fragments were created in the fragment files. Neither were target coordinate systems created in the assembly drawing. The model was created by using the 3D layout technique. Try to realize the steps of building this model independently, in order to check your understanding of this paragraph.





This drawing can be found in the folder *"\Examples\3D Assemblies\3D Fragments\Assembly 3.grb"*.

Additional transformations

Every 3D fragment can use **Additional transformations** as an auxiliary tool for making fragment location more precise or, in some cases, even as an independent positioning type. These transformations represent themselves translations along the coordinate axes of the insertion coordinate system and rotations about these axes. The value of any additional transformation can be defined via constants, variables or expressions as well as by snapping to geometric objects of a 3D model.

3D Fragment Variables

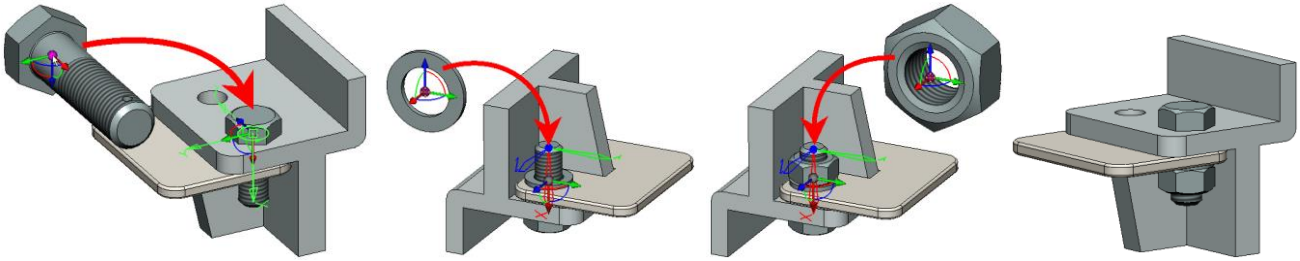
Parametric relations of a 3D fragment are controlled by its external variables. When regenerating a fragment, the required geometrical relations and data are loaded from the original file first, then the values of the external variables are copied from those stored in the assembly for this fragment, and, after that, the regeneration starts. Thus, the original file stays unchanged. The same assembly document may have several instances of the same 3D fragment with unique sets of values of the external variables. For example, the same bolt can be used in the assembly with different diameter and length.

Using 3D connectors

3D connector is a special type of LCS which lets us automatically link external variables for jointed parametric elements of assembled 3D models. Connectors significantly simplify parts positioning and fitting of parameters upon design of the assemblies. When a 3D fragment is attached to a 3D connector, the values of the variables are automatically transferred from the connector to the fragment model, which is followed by model regeneration. Preliminary image of a 3D fragment immediately takes the sizes corresponding to the values of the 3D connector variables.

For example, a nut M10 needs to be put on a bolt M10. When the bolt of size M10 is inserted into the assembly, its connector automatically remembers this diameter. Upon insertion of the nut, a required connector for attaching the nut is automatically chosen at the moment when the bolt elements are being selected. For a variable taking over the nut diameter, instructions are given – ask the connector the value of the variable. If the names coincide, the bolt connector passes the value M10 to the nut. Thus, right after connector selection, location, parameters and image of the nut are brought in accordance with the required bolt.

A user can create 3D connectors independently while preparing the 3D fragment model (See section "**Local Coordinate Systems**"). The elements of service libraries and standard parts, supplied with the system (Fasteners, Holes etc.) already have embedded 3D connectors.



Insertion rules, which remain active upon selection of the given connector in the assembly for attaching the fragment, can be defined for a 3D connector. The insertion rule is a mandatory additional way of defining fragment location by moving it along one of the coordinate axes or by rotating it with respect to the axis (see **Additional transformations**).

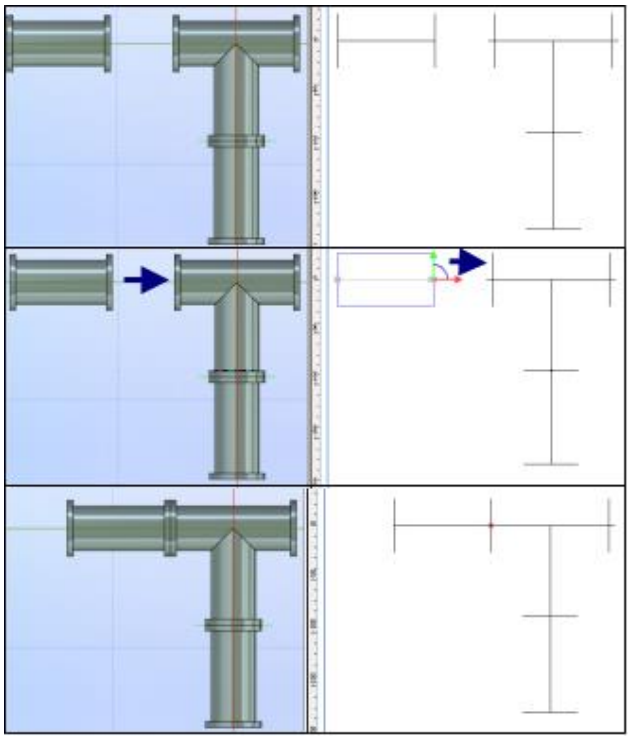
External geometrical parameters of a 3D fragment

External parameters of a 3D fragment can be not only external variables, but also geometrical elements (a solid, 3D node, 3D path, 3D profile, workplane, LCS, vertex, edge, face, loop). Such 3D fragment is called **adaptive**. Normally, such 3D fragments are most often used for direct modeling (grooving, drilling nonstandard holes and similar elements), rather than for creating assemblies; therefore, all information related to handling adaptive 3D fragments is separated into a dedicated chapter in this book (see the chapter "Adaptive 3D Fragments").

Using 2D-fragment as «projection» for 3D fragment

It is possible to insert 2D fragments that present projection of corresponding 3D fragment. This mechanism of operating with assemblies is useful when designing plans to deploy industrial equipment. It helps you to create drawings of floor plans, plans of rooms, etc.

To create a 2D fragment as projection of the existing 3D fragment you have first to specify workplane when creating a 2D fragment. This workplane helps to position future 2D fragment correctly. Then after selecting 3D fragment that will be used as source for the new 2D fragment you need to define the type of presentation for this 2D fragment. This may be done in «2D Projection» dialog where you can chose either type of projection or fixing vector associated with some predesigned 2D drawing. In case of fixing vector it must follow condition of correspondence to the workplane selected on the first step. In case of projection it will correspond to direction defined by the workplane. As result you will get the 2D fragment that will match source 3D fragment. Location of both fragments (2D and 3D) will be interdependent.



Relation between 2D and 3D fragments

When inserting a new three-dimensional fragment, the system always creates the respective 2D fragment referencing the same file. There is always an inherent relation between the 2D and 3D fragments. Both fragments use the same data related to variable values, as well as some settings – the file name, the way of including in a BOM, the status settings. Usually, the 2D fragment is not necessary when creating a 3D assembly model; therefore, the system hides it by default. The 2D fragment can be selected only by using the dedicated command for editing 2D fragments. Using this command may be convenient, for example, when you need to modify a bulk of references to fragment files, since the command provides convenient tools for simultaneous management of lists of the fragments. A 3D fragment document may contain the two-dimensional drawing along with the three-dimensional geometry. This drawing can be used for automatic creation of the assembly drawing, by following certain rules. In this way, the image of each 2D fragment will be put in the first page of the assembly document with the same page coordinates as those of the image in the original file.

3D Fragment Management

The command for inserting a 3D fragment is **3F: Insert 3D Fragment**. The command is called as:

Icon	Ribbon
	Assembly → Assembly → 3D Fragment
Keyboard	Textual Menu
<3F>	Operation > Insert 3D Fragment

To insert a 3D fragment into an assembly, you need to do the following steps:

Select 3D fragment.

The selection is done by one of the following ways:

- ✓ insert a new 3D fragment by selecting a file;
- ✓ select an already inserted 2D fragment and use its three-dimensional model;
- ✓ if the symbol of the model file is dragged from the Windows Explorer or from the service window of the T-FLEX CAD libraries to the 3D window, the 3D fragment insertion command will be started automatically and the given file will be selected for insertion.

Select or Create Source LCS for the fragment. Selection can be done automatically if the 3D fragment has exactly one or a specially designated LCS.

Select or create target coordinate system in the assembly.

Define the values of the fragment external parameters.

This step is skipped when using an existing 2D fragment or if the 3D fragment does not have external variables.

The values of the variables can be set automatically upon attaching 3D fragment to the connector of another 3D fragment.

Define fragment parameters (optional).

Finish input.

3D fragment selection

3D fragment can be selected with the help of the option:

	<F>	Select File

The fragment is selected using the standard File Open dialog. The fragment file can be selected either from a folder on the disk or from a library.

In the transparent mode the following option is available:

	<A>	Select 2D Fragment

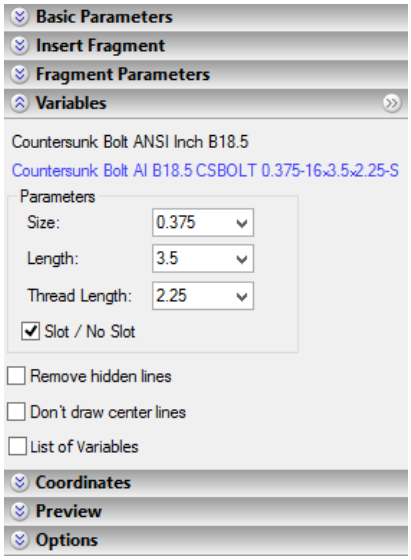
At this moment it is enough to select existing 3D fragment or a file from a library opened in the **Library Explorer**, and the system chooses an indicated element for insertion as the new 3D fragment. Upon selection of an existing 2D fragment in a 2D window the system will use its three-dimensional model.

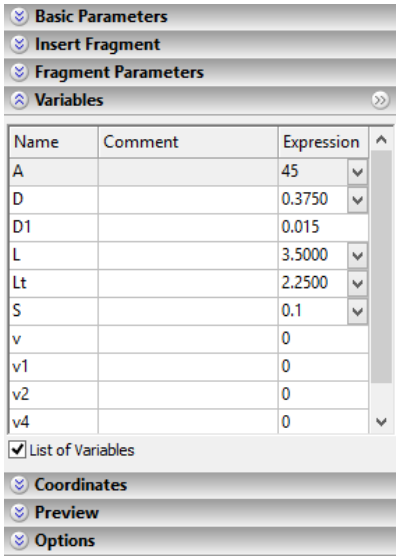
After insertion of a 3D fragment, the path to its file will be stored in the operation properties. The path can be complete relative to the file of the assembly or relative to the library being used.

In some cases, the system may automatically record the relative path to a reference with respect to the assembly file. Detailed information on the format of the relative links (references) can be found in the chapter "Assembly Drawing".

External Variable Value Definition

Upon selecting the file, the properties window is filled with parameters of the 3D fragment. One of its groups deals with external variables. The external variables can be displayed as a list – just as in the variable editor. If a custom dialog was created for managing external variables, it will be automatically built in the 3D fragment properties window. If necessary, you can switch between the two modes of displaying the external variables. This is done by toggling the flag **"List of Variables"** in the properties window.





The information about external variables is displayed as long as the 3D fragment creation command is active. You can modify at any time the values copied from the fragment file. To update the 3D window for the changes in the properties window, use the provided button .

If you find it inconvenient to modify the values of the external variables in the property window (as, for instance, because of the large size of the custom dialog), you can put the group of variables into a separate window and enter changes there. Calling the separate window is done by using a special button at the upper right corner of the group.

> If the 3D fragment model does not have any external variables, then, upon executing the 3D fragment insertion command, no tools are displayed for managing external variables.

When a 3D fragment is attached to a connector, the values of the fragment variables can be assigned automatically. (See section «*Using 3D connectors*».)

Parametric Relation Definition

Defining relations between parts is the main task for designing parametric assemblies. It demands from the engineer understanding of the T-FLEX CAD tools for maintaining such relations. The first tool that you need to master is the variable editor. The variable editor itself or similar environment is used by the system in multiple occasions, as, for instance, when defining the values of the fragment external variables. Working with the editor, variables, functions and such is described in the chapter “Variables”.

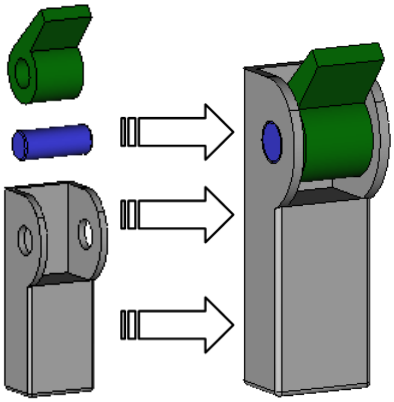
Here, let's review a simple showcase for relating part parameters in order to control them from the assembly model. Here we will skip the description of the steps that are not related to the parametric assembly creation (the description of creating part files, attachment, etc.).

Consider the task – to create a switchboard that includes the mounting plate and a set of switches. The switches can be of two types. The types of the switches differ in some dimensions, however, can be composed from the same parametric parts. The switch lever must snap to two positions “On/Off”. The switch itself is made of the following separate parts - the body, the pin and the lever. These parts are parametric, as they rely on the necessary external variables. By using those we can create a parametric model of the switch, whose external parameters define the switch type and the lever position. We need to create a switchboard assembly by using this parametric model of the switch.

Name	Comment	Value
R	Lever Rradius	R
r	Pin Rradius	r

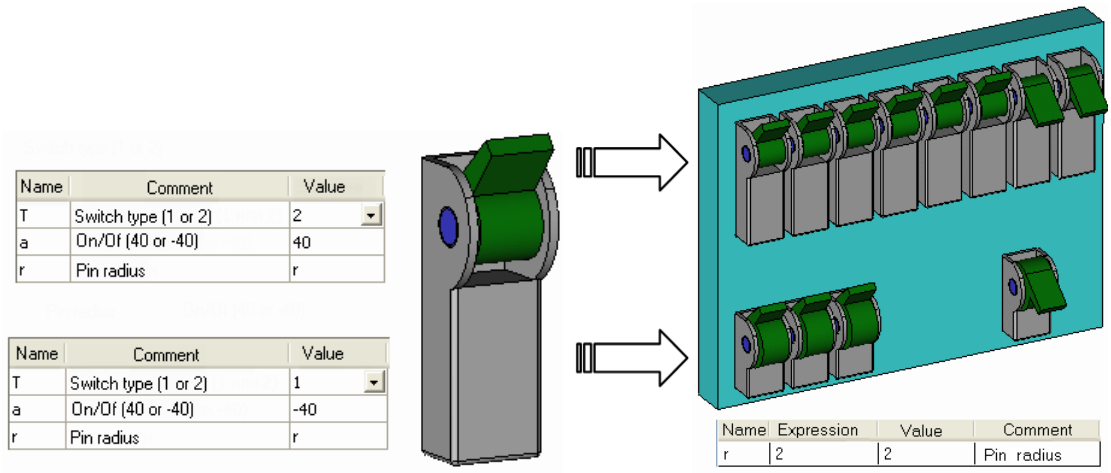
Name	Comment	Value
r	Pin Rradius	r

Name	Comment	Value
r	Pin Rradius	r
h	Body Height	h



		Name	Expression	Value	Comment
1	✓	T	2	2	Switch type (1 or 2)
2	✓	a	40	40	On/Of (40 or -40)
3		R	T==1?6:4	4	Lever outer radius
4		h	T==1?20:30	30	Body height
5	✓	r	2	2	Pin radius

The left-hand side diagram shows the parts of the switch. Tables to the left of each part contain the respective external variables. Originally, the tables appeared in the respective dialogs defining external variables for each 3D fragment being inserted. You can see that the respective assembly variables were entered as the values of the fragment external parameters (which is the switch file at this step). The diagram in the right-hand side shows the switch assembly and the table as appears in the variable editor. You can see on the diagram how the switch parameters are managed: the variables <R> and <h> change depending on the variable <T> value that defines the switch type. The former two define geometrical characteristics of the contributing parts. The variable <a> is responsible for the lever position and defines the model property “On/Off”. The external variable <r> will relay the pin radius value to the next assembly.



The next step is building the assembly (the switchboard) from the created parametric model of the switch. The left-hand side diagram shows the model of the switch and the two tables of its external variables with different values assigned when composing the assemblies. You can see from the diagram, that the variables <T> and <a> are assigned numerical values, while the variable <r> is related, instead of a value, to the variable <r> of the assembly. This is how the parametric assembly is built. The variable editor of the switchboard provides control over the value of the switch pin radius. Other switch parameters are left fixed. To modify those in the future, you could call the command for editing 3D fragment variables.

When connectors are used, parametric relations between two parts can be established directly, skipping the step of creating the common variable in the assembly. At the same time, the variables of the new part, attached to the connector of another part already inserted into the assembly, will depend only on the values of the first part's variables (see *"Using 3D connectors"*).

Selection/Creation of the Source Coordinate System

The automatic sequence of the system steps for attaching a 3D fragment depends on whether predefined local coordinate systems existed in the 3D fragment model, and, if so, how many:

- If there is exactly one LCS in the 3D fragment model, and its properties state that it can be used for attaching the 3D fragment, it will be automatically selected by the system as the source coordinates. The system will then proceed to the next step - defining the target coordinate system. If there are more than one LCS in the 3D fragment document, but only one of them is specifically designated as the source coordinates, the system behavior will be the same. (Refer to the topic "Preparing document for use as a 3D fragment" for details.)
- If a 3D fragment model contains several LCS all suitable for fixing the 3D fragment, then T-FLEX CAD picks up the default fixing (by the global coordinate system) and proceeds to defining the target coordinate system. To select a coordinate system, you need to switch to the special mode of selecting/creating the source coordinate system.

- If no local coordinate systems exist in the 3D fragment model, then T-FLEX CAD picks the default fixing (by the global coordinate system) and proceeds to defining the target coordinate system. To create a new source coordinate system, you need to switch to the special mode of selecting/creating the source coordinate system.

Mode of Selecting/Creating Source Coordinate System is operated in a separate 3D window, in which the 3D fragment model is loaded. This 3D window is launched forced from the 3D fragment creation command by the option:

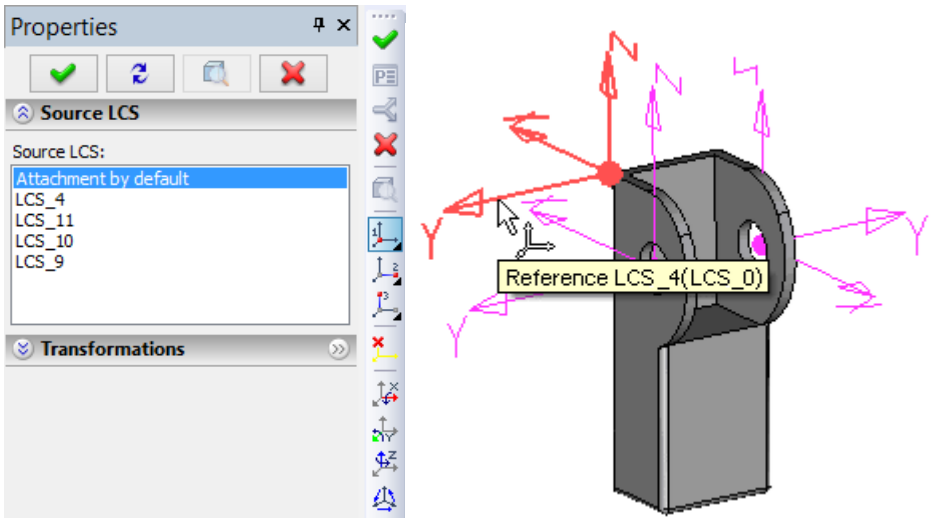
	<S>	Select or Create the Source LCS

After opening the new window, in addition to standard options, the following options can be initially found in the automenu:

	<1>	Select 3D node or vertex as LCS origin
	<2>	Select point to Define Direction of X-axis
	<3>	Select point to Define Direction of Y-axis
	<C>	Reset target LCS
	<A>	Rotate LCS around X-axis by 90°
	<O>	Rotate LCS around Y-axis by 90°
	<Z>	Rotate LCS around Z-axis by 90°
	<9>	Change LCS axes orientation

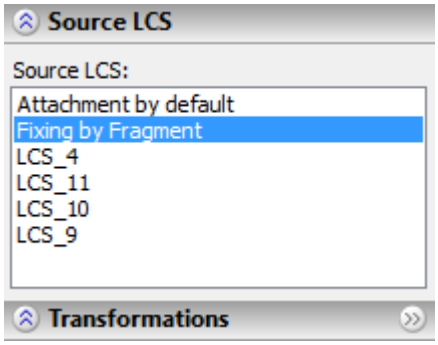
When the option is selected the user is returned to the assembly and the mode of the target LCS specification is enabled. The option is used for transitioning to the actions related to specification of adaptive fragment's geometric parameters (more detailed description can be found in the **Adaptive 3D fragments** chapter).

When the option is enabled selection of the existing LCS is carried out in 3D window or from the list in the properties window.



The system allows creating a new LCS based on the 3D fragment's geometry. To activate the mode of creating new source LCS, the first element for determining the origin of the coordinate system has to be selected in a 3D model. At the same time, it is possible to select any appropriate elements of the model in accordance with the active filters of the selector on the system toolbar. After choosing the origin of the new source LCS, the remaining options for defining LCS will appear in the automenu. Creation of the coordinate system is carried out by the same principles as in the specialized command for creating LCS **3O: Construct coordinate system**. Similar options and tools are used in both cases. The center and direction of the LCS axes are prescribed. In addition, the options for cyclical rotation and rotation of axes by 90 degrees are available. In this chapter we will omit detailed description of the LCS creation process.

When creating a new LCS, keep in mind that it is not created in the 3D fragment file. Rather, it belongs to the current "3D fragment" operation and is stored in the assembly document. The created source coordinate system cannot be displayed in the assembly. To modify it, you need to call the 3D fragment editing command and enter the mode of selecting/creating the source LCS.



When the new source LCS is being created, the new line *«Fixing by Fragment»* will appear in the Source LCS list for the fragment. Also, in the fragment properties window there is going to be a temporary section **Coordinate System Parameters** which enables to control objects selection during LCS creation.

To exit the mode of selecting/creating the source LCS, use the button

Selection/Creation of the Target Coordinate System

The mode of selecting or creating the target coordinate system is turned on by the automenu option:

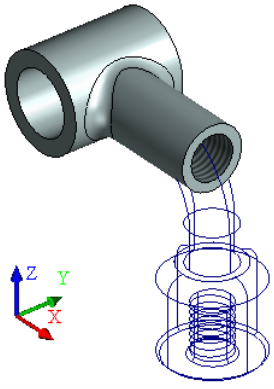
	<2>	Select or create target LCS

In most cases this option is activated automatically after choosing the fragment file and after exiting the mode of creating the source LCS. If a suitable LCS is found in the assembly model then the user actions amount to selecting the LCS in the assembly, to which the fragment source coordinate system should be snapped. If no LCS is selected in the assembly, then the assembly world coordinates will be used as the target coordinate system.

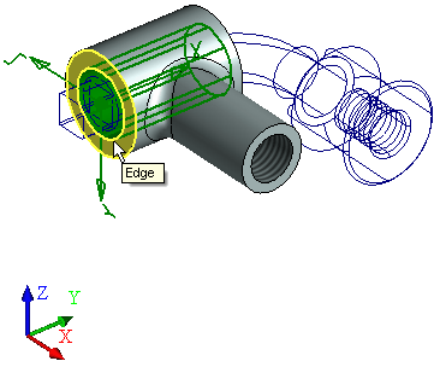
One can also create a new target coordinate system based on 3D model elements. The target coordinate system is constructed using the same tools as those for constructing the source one. The main difference is in that the all actions are performed in the main 3D window for the current assembly. The 3D fragment model is positioned in the space according to the properties of the target coordinate system being defined.

The options for defining elements of the new target coordinate system appear after selecting the origin point of the LCS. Upon selecting the origin, you can proceed with orienting X and Y axes.

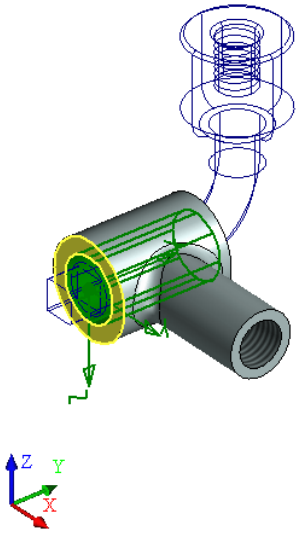
In case you couldn't orient the LCS as desired from the beginning, you can use the tools for coordinate system redefinition. Those allow quickly rotating an existing LCS around its axes by 90 degrees ( ), cyclically reassign the axes of the coordinate system with the fixed origin (pressing) switches the axes assignment).



Fragment position in absolute coordinates



Fragment position after selecting an edge for target LCS



Fragment position after using the tool of rotating X-axis by 90°.

To reset all data currently used for defining the target coordinate system, use the automenu option:

	<C>	Reset target LCS

Choosing 3D connectors

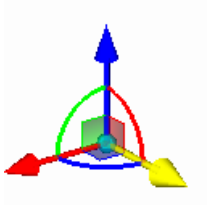
3D connector can be chosen in the mode of selecting or creating target coordinate system activated automatically right after entering the fragment insertion command (option). In this mode it is

necessary to make sure that the selector's filter for choosing connectors is activated on the system toolbar.

For searching connector on some part, bring the cursor to one of the elements associated with the connector. In standard elements, for example, in bolts, the screw-threaded faces of the bolt shaft can be chosen as an associated element. Already upon bringing the cursor to such element, the connector is highlighted. Press for choosing connector.

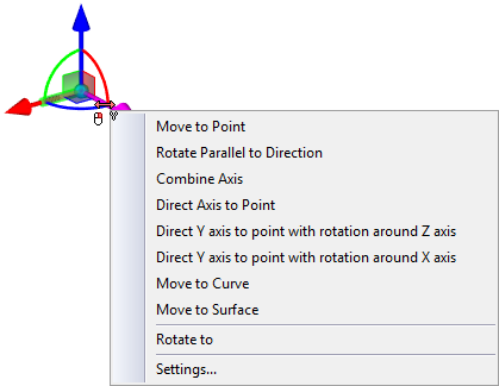
Assigning additional transformations

Controlling additional transformations is carried out with the help of a dragger in the 3D window, group **Transformations** of the properties window and the **Transformations Groups** dialog of the **Main transformations** section of the properties window. The dragger is always present upon insertion of a 3D fragment, if the option **Use Draggers** is turned on in the properties window.



The dragger enables to define an unlimited number of translational transformations along the indicated axes, in-plane shifts or rotations of selected axes by a required angle in the required plane. The dragger represents itself an image of the coordinate system. The coordinate axes are connected to each other by color arcs; adjacent planes are connected with each other via conditional small color squares, the dragger center is marked by a grey ball. Different parts of the dragger are the active elements intended for defining certain transformations:

- the dragger axes are intended for defining translations;
- the dragger arcs are used for defining rotations;
- the images of coordinate planes are used for translation in the planes **XY, XZ, YZ**;
- the dragger center is used to manually enter the mode of dynamic snapping.

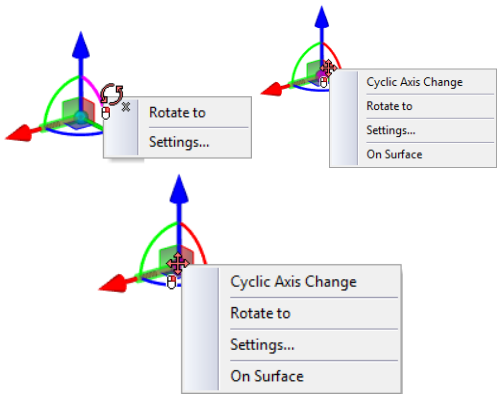


Upon bringing the cursor to different active parts of the dragger the cursor changes its appearance prompting to set the corresponding types of the additional transformations. Next to cursor there is going to be a tooltip providing information about the type of offered transformation. The selected element of the dragger is highlighted with a different color. When a user points at the axis he can move to setting the translation along the chosen axis. Upon pointing at the arc, the system prompts to set rotation in the corresponding plane. At the same time, the closest to the cursor axis of the dragger is also selected so that a user could define a rotation by directing the selected axis towards a specific element of the model.

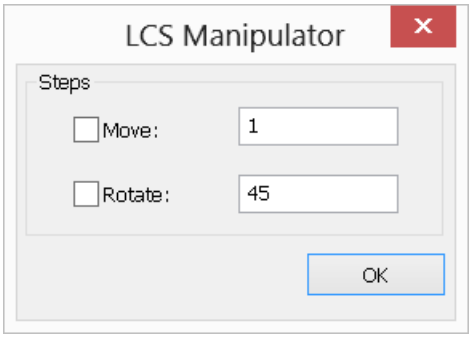
When specifying the coordinate plane's image the user can start defining the shift across the selected plane.

When selecting one of the dragger's elements mentioned above the following commands become available in the context menu:

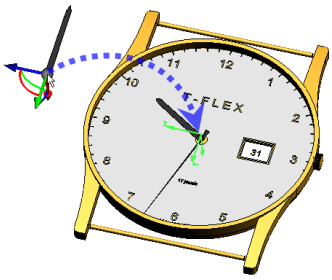
- for translation along the axis;
- for translation in the plane;
- for rotation about three axes;
- for translation along three axes.



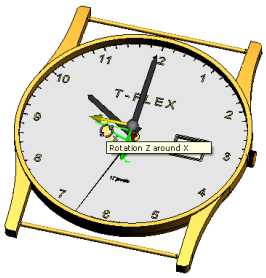
The **Settings** option of the menu defines the step in the change of the distance and angle upon translations and rotations of the dragger.



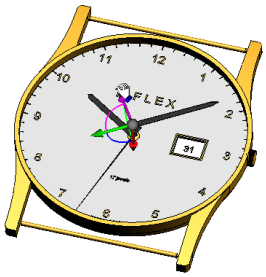
For defining transformation with the help of the dragger a user has to choose an active element of the dragger by pressing . Then, without releasing the mouse button a user can move the cursor. The transformation will have been set upon releasing the mouse button.



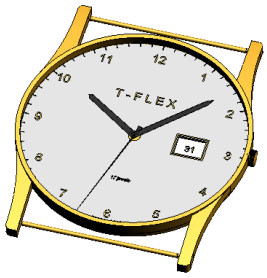
Begin with inserting a fragment (the minutes arm) by selecting the edge on the arm pin to define the target LCS (of the arm fragment position)



Turn the arm. To do this, select the desired element of the dragger and depress the left mouse button



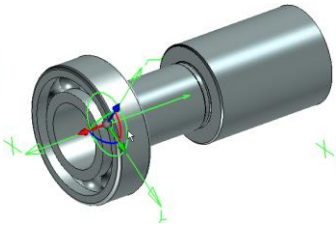
Drag the mouse whilst holding the mouse button down. Turn the fragment about the axis



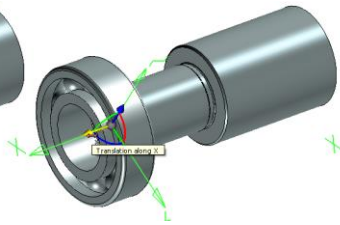
Complete the transformation definition by releasing the mouse button

Also, after dragger's selection it is possible to release the mouse button, shift the cursor and choose a new element (indicate a new point in space) for fixing transformation by pressing twice.

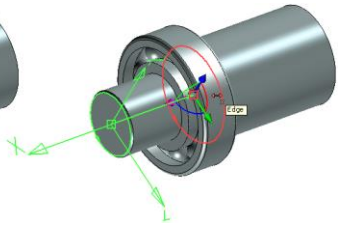
Start inserting the fragment (the bearing)



Select the round edge to define the target coordinate system (the fragment's position)



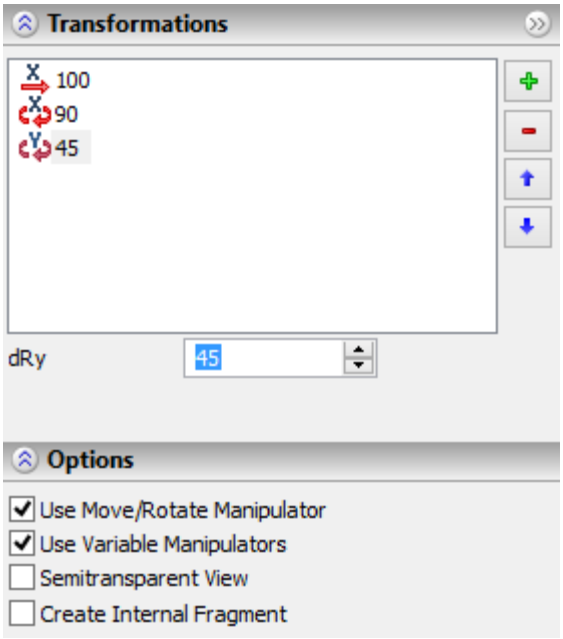
Click to select the dragger's axis to define the translation along the axis



Select the round edge to define the final point of the translation

Click at dragger center signifies transition to operations for modifying the target coordinate system. Simultaneously, the **mode of dynamic snapping** gets activated. In this mode, the fragment image is attached to the cursor. Upon bringing the cursor to the elements of the model, the fragment image will be immediately put in a position showing the location of the fragment in case the element is selected. In this mode it is possible to click at the free spot in space or choose an element of the model for attaching. In the first case, new additional transformations for placing the fragment at the spot of a click are created. After choosing the new element of the model for attaching, all previously defined transformations may be deleted.

Group «**Transformations**» in the properties window can be used for controlling, correcting numerical values, creating or deleting transformations of a 3D fragment. In the special field the list of all additional transformations, defined at the moment for a 3D fragment, is shown. Each transformation is marked by a special symbol, denoting its type (translation or rotation about some axis). If the value of the transformation is set via a constant, then this value is shown next to the symbol. If the transformation was attached to the element of the model, then the name of the element is indicated next to the symbol. The elements of the list are shown in the order in which there were created. Several successive transformations of similar type are automatically summed up.

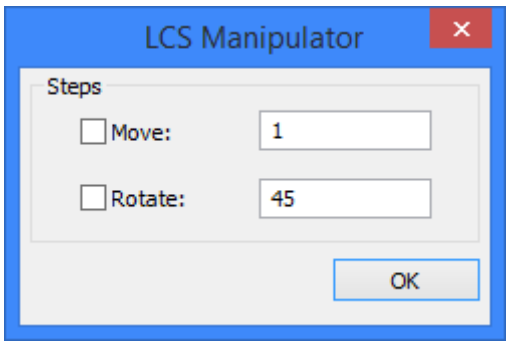


Below the main list of transformations there is a field for entering and editing the values.

To the right of the transformation list there are graphics buttons. The button is intended for adding new transformation. When it is pressed, the list of available transformations, from which a desired type is offered for selection, will appear. After choosing the type in the list of transformations, a new line will appear. In the field for entering the values, one can put the value of the transformation. For saving transformation press **[Enter]**. Moreover, for assigning the values it is possible to shift to the 3D window and use the dragger, which is moved in accordance with the chosen transformation type.

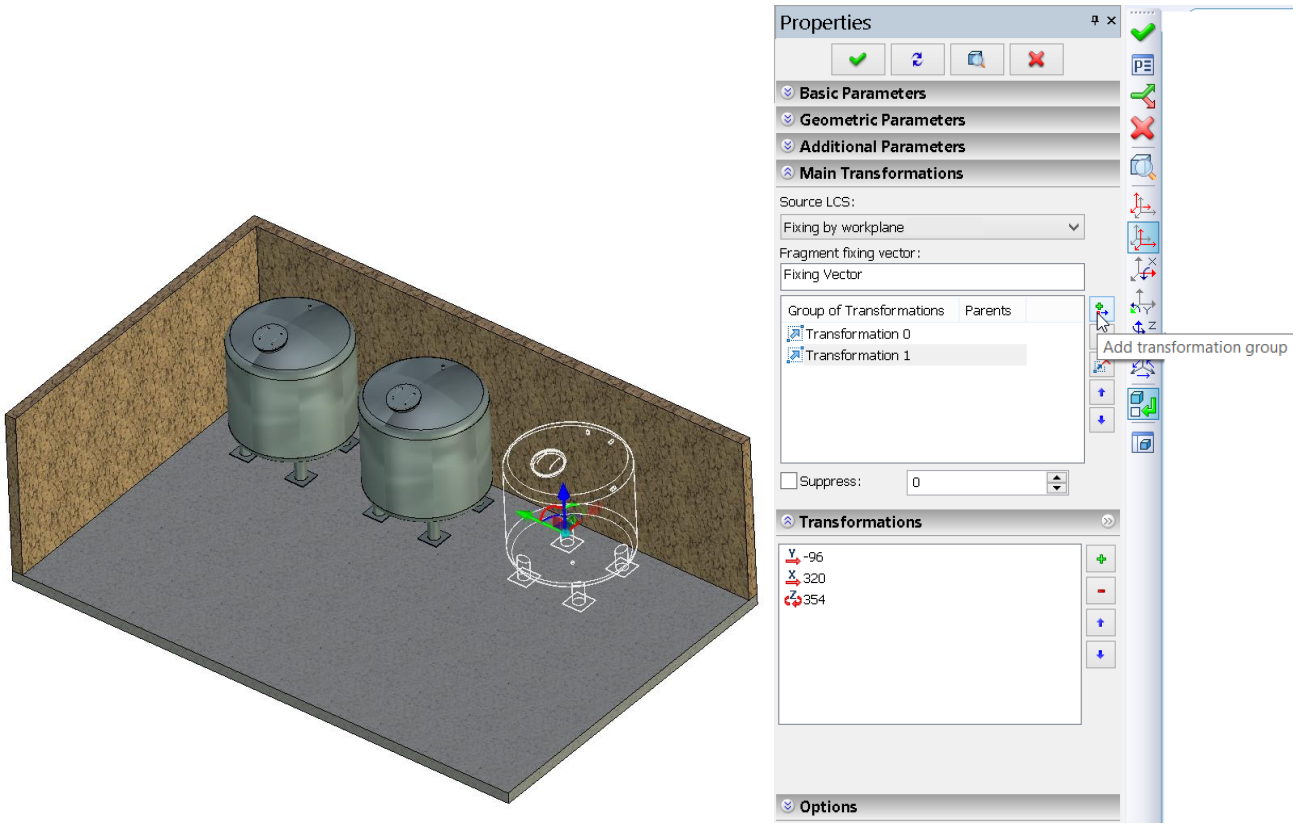
Upon pressing the button the last transformation in the list is removed.

When pressing the buttons the order of transformations executions will change.



The user has also a capability of creating individual transformations groups for 3D fragments. Classifying transformations into groups allows the user to create simultaneously several variants for the 3D fragment locations in the assembly.

For example, let us suppose there is a variant of the room's 3D layout. It is required that the fragment (barrel) change its location depending on the location of walls and addition of new objects to the layout.

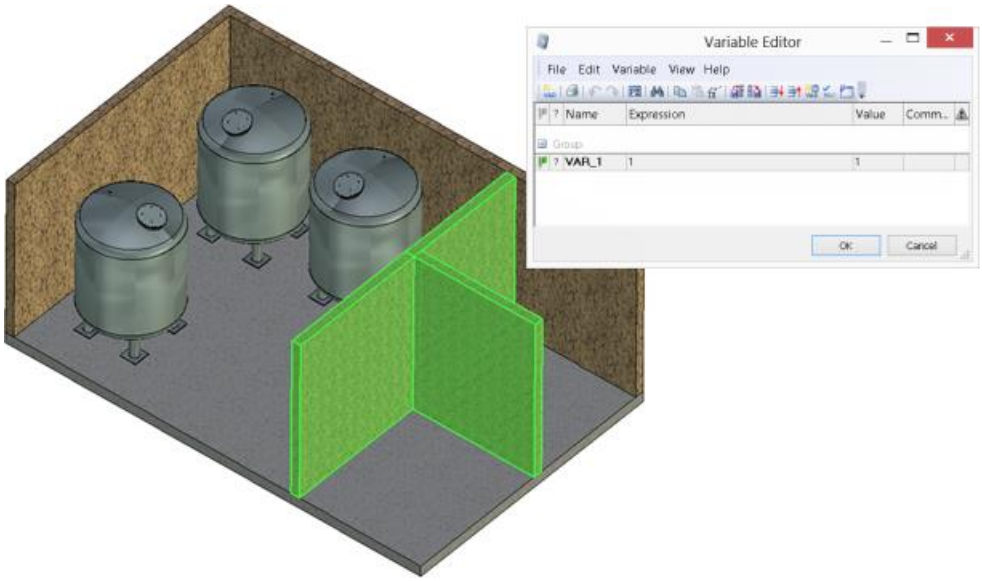


To do so, in the properties window of the 3D fragment (barrel) in the **Main Transformations** section we find the **Transformations Groups** dialog. With the help of the button we add the transformations group, namely we specify the values for translations along the X and Y axes and rotation around the Z-axis.

> It is important to remember that by default all transformations for a 3D fragment are created in a single transformations group.

Each transformations group has it name. The elements of the list are shown in the order that correspond to their creation.

To control visibility of various locations of the fragment there is the **Suppress** flag. Next to the flag there is a field where the numbers can be entered or the numeric variables can be written down. If the value different from zero is entered into the field, the transformations group becomes suppressed. To make the work in the assembly more convenient let us specify the visibility of the fragment's locations with the help of the variable.



After specifying several positions of the fragment (barrel) with the help of the transformations group we obtained the capability of manipulation of its translation in the easiest and fastest way.

In this case for displaying the changes in the transformations in the 3D window it is required to use the 3D model update button .

The graphics buttons are located to the right of the list of transformations group.

The button allows us to add a new group to the transformations group.

When pressing the button the last group of transformations in the list is removed.

The button allows us to remove all transformations groups.

When pressing the buttons the order of execution of transformations groups changes.

> Upon attaching a fragment to the connector, the system can automatically move to defining additional transformation, if the corresponding instructions are set in the connector (see chapter *"Local Coordinate Systems"*).

After insertion 3D fragment all prescribed additional transformations are recorded in the common transformations list of the 3D fragment. A separate record is generated for each additional transformation in the transformations list of the 3D fragment.

Fragment updating

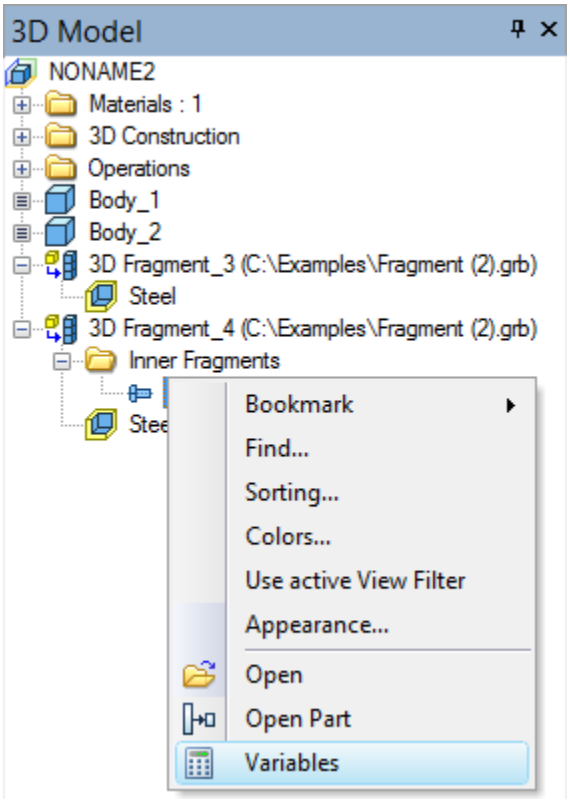
Update 3D fragment automatically in case of changed parameters. If option is set, a fragment will be automatically updated when its variables or adaptive parameters are changed.

Also you can update fragment manually by pressing **update** button in the upper part of the **Properties** window.

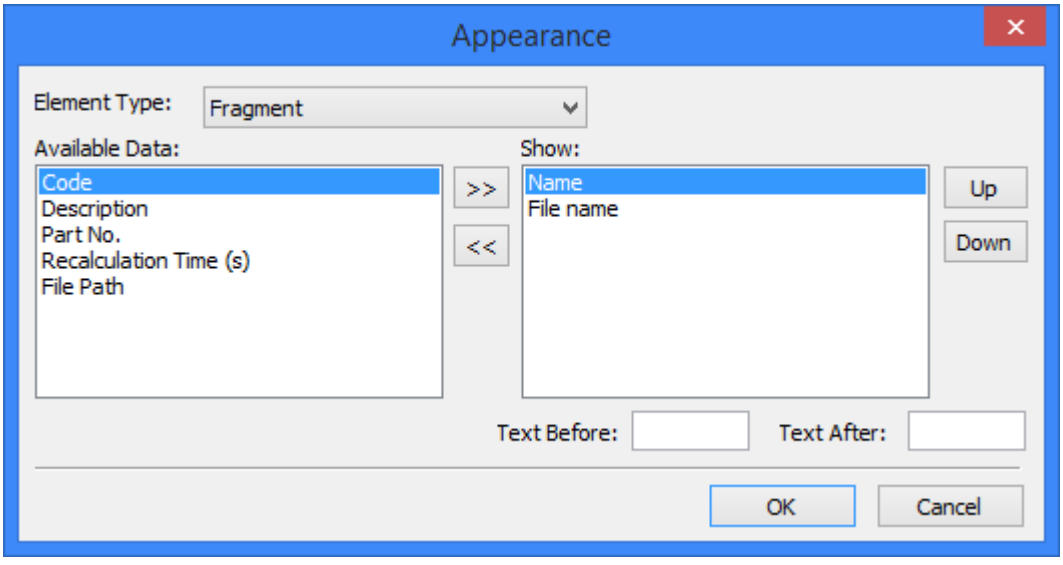
Working with 3D Model Structure

In the course of working with an assembly, you may need to access auxiliary information about the state of 3D fragments. This can be querying their position in the assembly tree, reviewing existing errors, etc. Various tools are provided for this purpose. We will review those tools within the current topic.

3D fragments and Parts are displayed in the "3D model" window. This window shows the complete hierarchy of the assembly model. It allows finding the place of each 3D fragment in the model structure among other 3D operations and construction elements. 3D fragments can be displayed at the root of the model structure, along with Bodies and Arrays. If a 3D fragment is an assembly itself, containing nested 3D fragments, the structure of all its nested elements is reflected in the general 3D model tree as well. To ensure the full support of handling the nested 3D fragments, you need to bring into the editing environment the subassembly that hosts the fragment of interest.



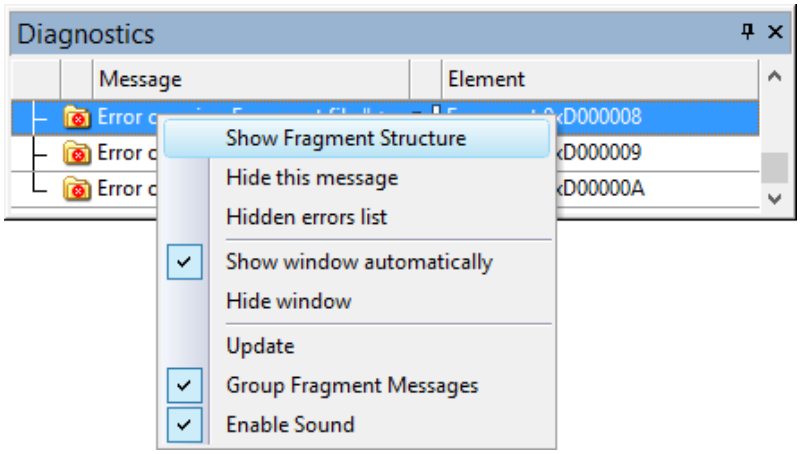
In the window **3D Model** for 3D fragments, besides the name of the file and the operation, various additional information can be shown, if needed, – file path, identifier, memory size allocated for mesh, memory usage for geometry, recalculation time, the name and notation for specification. The amount of displayed information can be set in the dialog box of the command **Appearance**. This command can be called from the context menu by pressing in the window **3D model**.

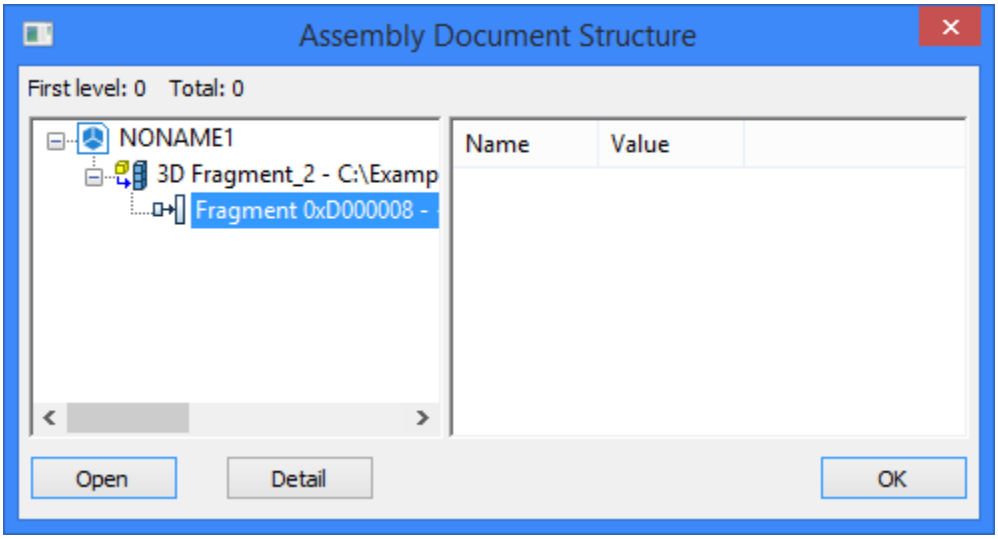


If errors are detected in the model, the affected elements are marked in the model tree with the special symbols:

- ✓ If an error was detected within a 3D fragment model or its coordinate systems, a mark is displayed.
- ✓ The 3D fragments, whose files were not found by the specified reference, are marked by a red cross. 3D Fragment_7 (Handle.grb)
- ✓ The suppressed 3D fragments are marked by a blue cross. 3D Fragment_7 (Handle.grb)

Detailed information about all errors is displayed in the "Diagnostics" window. Right clicking over a message brings up the context menu. If the message refers to a 3D fragment of the first level, then you have access to all commands for handling that 3D fragment. If the message reports an error within nested fragments, then the only command for handling the fragment is **Show fragment structure**.





The fragment, within which the error was identified, is automatically highlighted, showing the branch of the assembly tree that includes the failing fragment. The right pane of the window displays the information about variables used in the fragment: their names, values and comments. To fix the error in

the fragment file, you can open the file by pressing the button *[Open]*. For example, the error in the fragment model can be caused by an invalid variable value. To open the fragment with the current variable values, use the button *[Detail]*.

Normally, the assembly tree displays only its contributing 3D fragments. To view the full assembly structure, use the following dedicated command:

Icon	Ribbon
	Tools → Special Data → Structure
Keyboard	Textual Menu
<SS>	Tools > Special Data > Structure

Detailed information about this command is available in the chapter "Saving Textual Drawing Information".

Layout Creation

To create a 3D fragment using the layout technique, use the following option in the main automenu of the 3D fragment insertion command:

	<G>	Create 3D Fragments by Workplane (Layout)

Upon calling this option, the mode of selecting 2D fragments is activated, as indicated by the glyph attached to the pointer and the pushed icon in the automenu. Now, you need to select a 2D fragment or a set of 2D fragments to use for creating the 3D layout.

Upon selecting 2D fragments, specify the workplane on which the 3D fragments will be placed. The workplane for the 3D fragment orientation can be defined anew, regardless of the workplane that was hosting the 2D fragment. The workplane selection is done by the option:

	<W>	Select Workplane to set view direction of projection

The selection can be done either in the 2D or in the 3D window, provided, that the display of the workplane in the 3D window is turned on.

Upon confirming the selection by pressing , the 3D layout will be displayed in the 3D scene, created based on the 2D fragments.

If errors occur when creating a layout, for example, if the vector of the 2D fragment was not associated with the workplane, the system positions the 3D fragment in the absolute coordinates without a

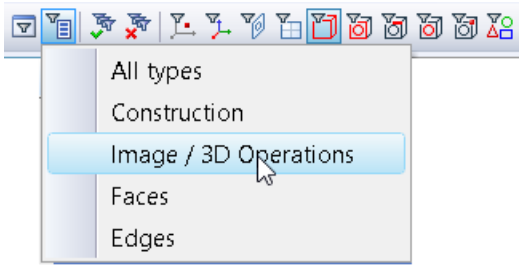
possibility for attaching to an LCS until the fixing vector is associated with the workplane in the fragment document. Besides, a warning is displayed, "Can not find associative Workplane in Fragment document".

There is a capability of controlling location of the 3D fragment directly in the 3D window domain. To change location of the 3D fragment in the 3D window domain it is required to use the option in the automenu of the 3D fragment insertion command:

	<L>	Layout mode

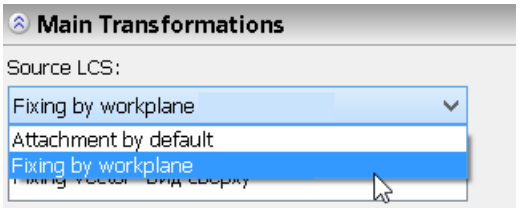
When the layout mode is enabled only the axes of the workplane are displayed on the dragger. All transformations with 3D will automatically be transferred onto the 2D drawing after pressing . Transformations with translation of 3D fragments are displayed on 2D **only with the layout mode enabled**.

It is also possible to edit location of a 3D fragment when choosing configuration of the selector **View /3D operation Elements**.



The layout mode is used only for 3D fragments with fixing vector.

For the 3D fragments for which a 2D fragment has a fixing vector, when choosing the source coordinate system there is an option of fixing by the workplane.



Orientation of the source coordinate system of the fragment snapped to the workplane coincides with the orientation of the fixing vector on 2D.

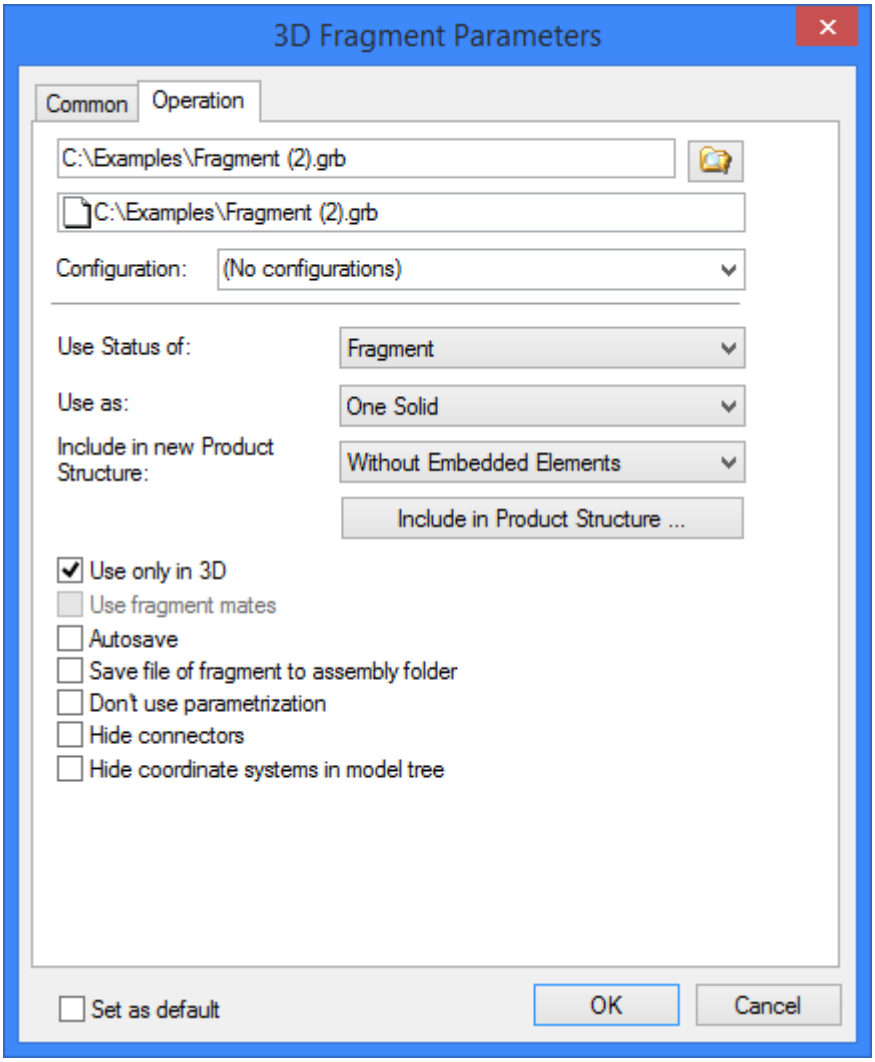
3D Fragment Parameters

Detailed setting of many parameters for a 3D fragment can be accomplished at any moment in the parameters dialog box. It is called with the help of the option of the command for creating and editing the fragment, and, also, from the context menu by selecting operation:

	<P>	Set entity parameters

The dialog contains three tabs. The tab **[General]** contains common parameters for all 3D elements.

Most of the parameters on the tab "Operation" are already described in the chapter "**Assembly Drawing**" of the two-dimensional design manual, as those are used when inserting 2D fragments. Exception is the following two parameters.



Configurations can be used for defining the values of the external variables in 3D fragments. For this purpose, the properties window provides a control for selecting the current configuration. The user can select any existing configuration, or the setting "(No configuration)".

Use as. This parameter can take one of the two values: "One Solid" or "Separate Solids". With the value "Separate Solids" used, the additional operation "Divide Solid" is applied for dividing the fragment into the separate bodies. This is necessary in the cases when the 3D fragment itself is made of several bodies,

while just one of them should be used, for instance, in a Boolean operation with another assembly element. It will be impossible to select a particular body of the 3D fragment without dividing into separate bodies. Instead, you will be always getting the whole operation "3D Fragment" selected as one object. The operation "Divide Solid" can also be done manually after inserting the 3D fragment, with the same result.

Use only in 3D. This flag allows turning off the display of the two-dimensional representation of the fragment in the 2D view window.

Use Fragment Mates. When a different assembly unit, in which the mates have been created and the degrees of freedom have already been defined, is inserted into the assembly, there is a possibility to transfer the entire mechanism into the new assembly by using only operation of insertion of a 3D fragment. To insert such fragment-mechanism into the assembly, the option of using the fragment «**As Separated Bodies**» and the option «**Use Fragment Mates**» have to be turned on simultaneously.

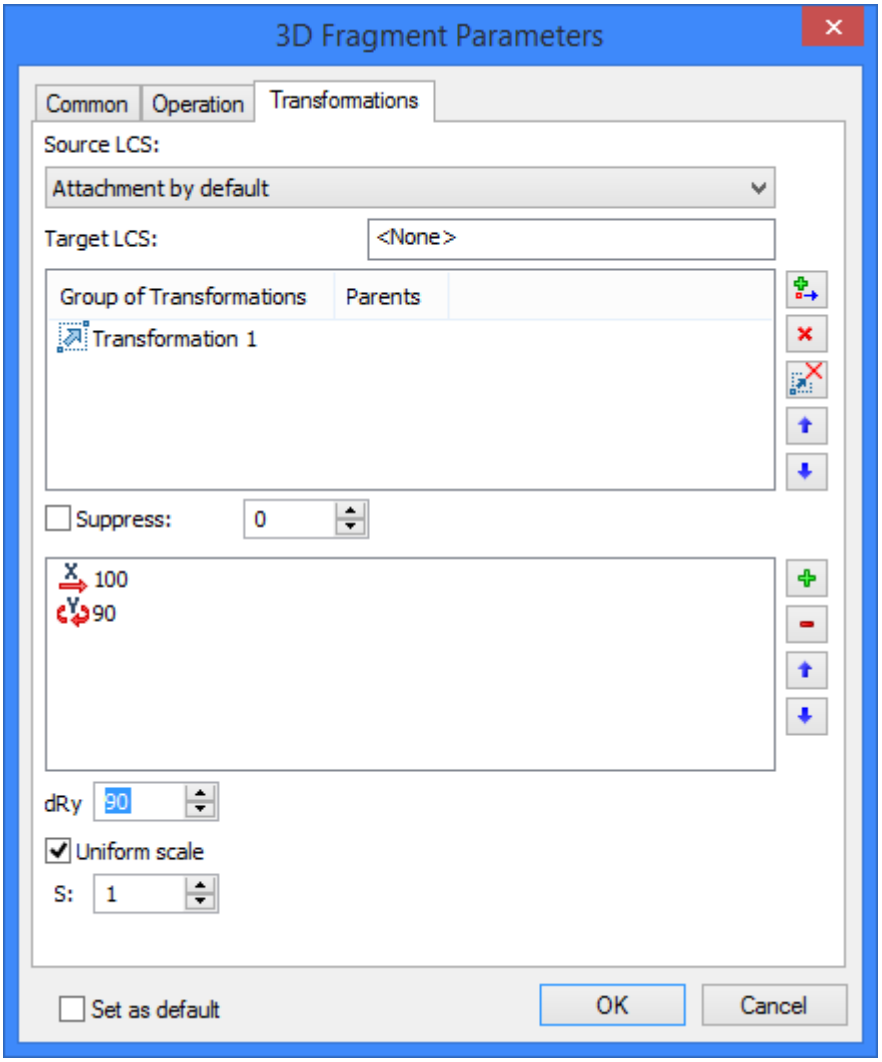
Autosave. When changing the values of external variables of the fragment in the assembly the fragment's file is resaved according to the new values. The fragment's file is updated at the moment of saving the assembly file. This flag can be conveniently used for obtaining detail drawings. If projections are used in the file of the fragment, then for their recalculation during autosaving the mode of automatic update must be selected in the projection's properties.

Save file of fragment to assembly folder. If this flag is enabled, then next to the assembly file the folder is created, which has the name of the assembly file, and the fragment's file is saved into the folder. The relative path for the fragment is created.

Don't use parametrization. When this flag is enabled, external variables of the fragment cannot be changed. The "Variables" window is not invoked from the context menu of the fragment and does not appear during the fragment editing.

Hide connectors. When this flag is enabled connectors of the fragment are not shown in 3D scene and in the tree of a 3D model.

Hide coordinate systems in model tree. When this flag is enabled external coordinate systems of the fragment are displayed in 3D scene, but are not shown in the tree of a 3D model.



Via the tab **[Transformations]**, it is possible to work with list of all geometric transformations which have been applied to the given operation of a 3D fragment. Among those, there could be:

- additional transformations defined upon creating or editing 3D fragment;
- transformations, assigned in the command **"3EG: Transform Element"** (working with this command is described in the section Assigning "Transformations" to 3D Entities»);
- transformations for the «Exploded View» mode;
- specific service transformations for performing the functions of attaching a 3D fragment.

Any transformation can be applied either permanently or only in the special mode of exploded view for the assembly. This is controlled by the corresponding flag in the settings of each transformation.

3D fragments and Parts can be subject to a special type of a transformation called "Attachment...". Instead of somehow transforming the body of a 3D fragment in the space, this type of transformation merely defines the attachment by the coordinate systems. This transformation is introduced

automatically by the system upon inserting each 3D fragment. Its name depends on the attachment type of the 3D fragment:

- **"Fix by Fragment LCS"** - is the 3D fragment attachment transformation in which the fragment LCS was used as the source coordinates.
- **"Fix by Assembly LCS"** - is the 3D fragment attachment transformation in which the source coordinate system was created on the fly in the transparent mode when inserting the 3D fragment.
- **"Fix by Workplane"** - is the 3D fragment attachment transformation in which the attachment is defined by the 2D fragment (layout).

When suppressing the transformation defining a 3D fragment attachment, the fragment model is positioned in the scene by snapping the 3D fragment world coordinate system to the assembly world coordinates.

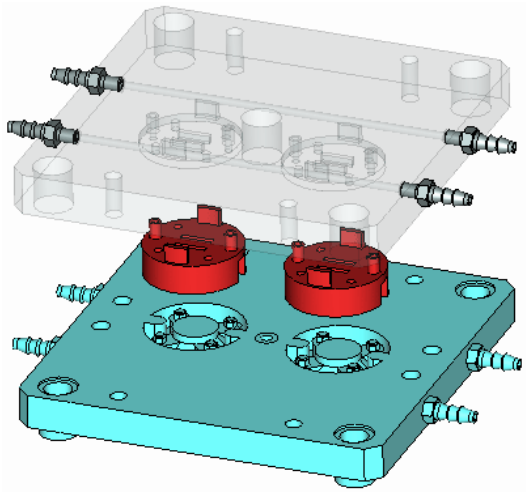
To disable any transformation, set the flag **Suppress** in its properties. The input box next to the flag check box allows entering numerical values or non-textual variables. The transformation is considered suppressed whenever the entered expression is not equal 0.

3D fragments possess another special type of transformation, providing for the automatic 3D fragment exploding data copying from the properties of its source coordinate system. The respective chapter about local coordinates reads: if you plan to use a given LCS as the source coordinates for attaching a 3D fragment, then you can define, among its properties, the necessary translation/rotation for future exploding. This transformation is called **Source LCS Transformation**. It only becomes available, when the source LCS of the 3D fragment actually has those properties defined. Such transformation cannot be modified in any way.

If the exploding transformation was not defined for the source coordinate system, then the system may try to automatically assign some transformations at the time of the 3D fragment creation. The numerical values of the offsets will be automatically calculated based on the overall dimensions of the 3D fragment model. A new transformation named "Transformation_<No.>" will be added to the list of existing transformations at this time. This transformation will have the flag **Only for exploded view** set. You can turn off the system's automatic appending of the exploding transformations upon 3D fragment creation. The setting is accessible in the system options on the tab "3D" (the command **Customize > Options...**, the parameter **Explode assembly (3D Fragments) automatically**).

A special flag **Apply exploded view transformation of base elements** allows exploding a given 3D fragment with respect to its fixing point, accounting for exploding transformation of the base elements. That means, turning on the flag additionally moves the fixing point of this 3D fragment per the transformation of the base element.

Creating Boolean Operation Based on 3D Fragment



Just like other operations, 3D fragments can be used in Boolean operations. For example, you can subtract the body of a 3D fragment from another part. If necessary, you can define a property in the 3D fragment document, that will specify what type of the Boolean operation will be automatically performed upon inserting this document into an assembly (see the topic "Preparing a T-FLEX CAD document for use as a 3D fragment"). If such property is defined, then, upon inserting the 3D fragment, you need to select one of the bodies in the assembly. If the assembly consists of a single body, it is selected automatically. This body will be used as the target body in the Boolean operation. The tool body will be one of the bodies of the 3D fragment, or the fragment altogether, depending on the property setting.

The option can be useful, for example, when inserting holes into the body of a part, or in mold design, when subtracting the part body from the block yields the stamp or master form.

The following options are used for selecting the target body of the Boolean and for undoing the selection:

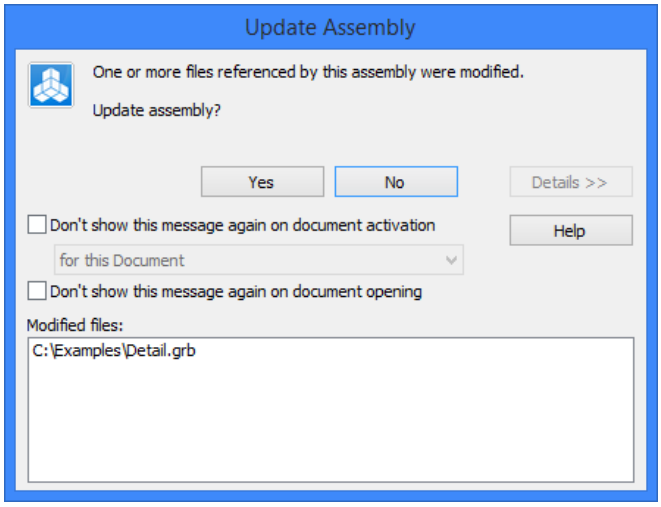
	<B>	Select 1st body
	<D>	Cancel operand selection

Reference Update

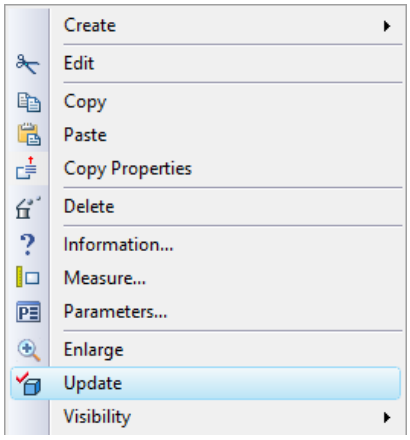
Let us explain the use of the option **Update fragment data**.

When inserting a 3D fragment, the system internally creates a copy of the fragment three-dimensional model in the assembly drawing. Upon a new regeneration of the assembly, the system no longer loads the 3D fragment geometry data; rather, it uses the saved data. To maintain the assembly up to date, the system constantly tracks the file. The latter changes if the fragment file was edited and saved. If files changed, the system instantly offers updating the references to the documents, that is, loading new geometry data of the modified 3D fragments.

Pressing the button **[Yes]** refreshes the data of all modified fragments. If the assembly contains a significant number of fragments, the updating procedure may take significant time, even though only one fragment actually required update. Therefore, you can press the button **[No]** instead, and then select a specific fragment and refresh only its respective data.



Local update of a selected 3D fragment is possible with the command **Update** of the context menu coming up upon right clicking over the 3D fragment operation.



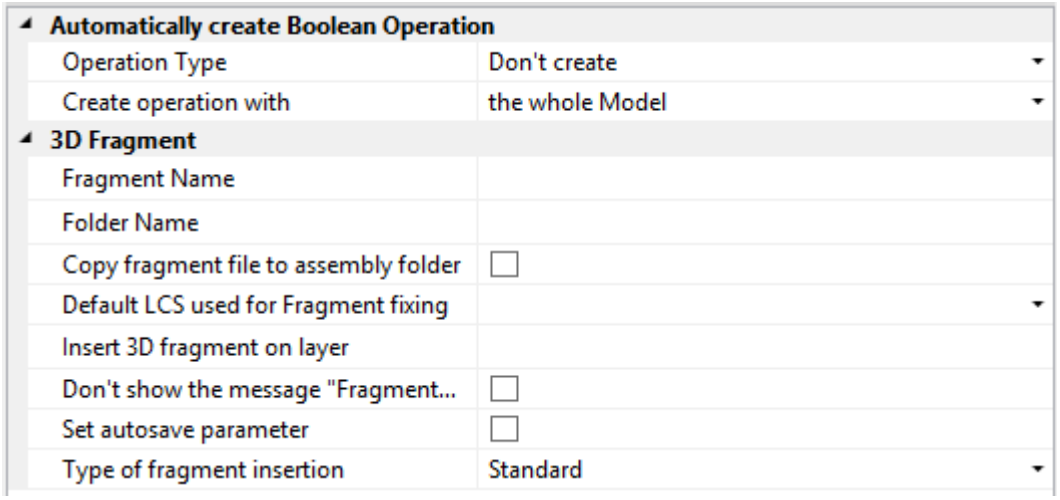
Updating of all modified fragments can be forced by the command **File > Fragment > Refresh Files**. However, if some of the 3D fragment files were not modified, then the 3D fragments are regenerated neither upon the full regeneration nor upon refreshing the 3D fragment references. To guarantee the regeneration of the full assembly with all its member 3D fragments, use the file **converter** with the parameter set, "Don't Check Documents Versions (Resave All)". The detailed information on working with the converter is provided in the chapter "Converting Documents Created in Earlier Versions of T-FLEX CAD".

Preparing a T-FLEX CAD Document for Use as a 3D Fragment

Any T-FLEX CAD document can be customized to simplify in certain circumstances its usage in assemblies as a 3D fragment.

Customizing Document Status

3D fragment tab in **ST: Set Model Parameters** command provides possibility to specify **default LCS used for fragment fixing**, specify **type of insertion**, set **parameters for automatic Boolean operation.**



Automatically create Boolean Operation

When inserting a 3D fragment, one can automatically create a Boolean operation of a specified type. The target body will be the selected body in the assembly, while the tool body - one of the bodies of the 3D fragment, or all fragment bodies at once.

Setting up automatic creation of Boolean operation is performed with the help of several parameters:

Operation Type. This parameter defines the type of the Boolean operation (addition, subtraction, intersection) that will be created upon inserting the fragment in the assembly. The type can be selected from the list.

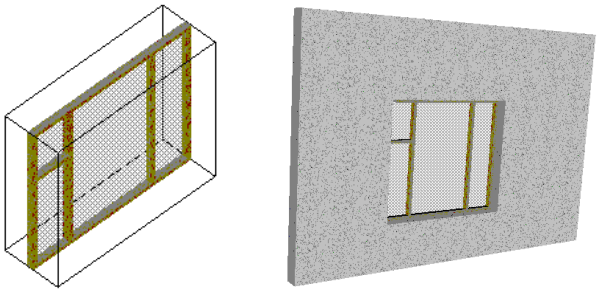
Create operation with:

Use the whole Model. With this option checked, the whole 3D model of the fragment will be used as the target body of the Boolean operation.

Use single operation. Selecting this option makes the Boolean use just a single operation within the fragment**. The desired operation can be selected from the list of all existing operations.**

In the case when a body is used in the Boolean, that is not visible when working with the document of the 3D fragment, it is possible to place it in a special layer marked as "**Visible only when model is used as a Fragment**".

Let's review an example of automatic creation of a Boolean operation on inserting a fragment. Suppose, there is a 3D model of a building wall, and we need to insert a window in it as a fragment. To do so, let's create the window model in a special way. Create a tool block as a dummy for cutting the opening in the wall for inserting the window. Then call the "**Insertion of 3D Model as Fragment**" dialog box (see above). Set the operation type – **Subtraction**, **Use single operation**. From the list of operations select the one that created the tool block.



3D Fragment

Fragment name. This parameter defines the name with which the fragment will be displayed in the assembly's model tree.

Folder Name. This parameter specifies the name of the folder into which the fragment will be placed in the assembly's model tree.

> Fragment's name and folder's name are read from the fragment's file only upon creation of the new fragment. If the values of these parameters are changed, then upon update of the fragment in the assembly the name and the folder will remain the same.

Type of Insertion. From the drop-down list one of the options can be selected:

- **Standard**. Upon insertion into the assembly the translation of the fragment is possible only with the help of the manipulator of LCS (dynamic translation is disabled);
- **Fastener**. When inserted into the assembly the fragment dynamically moves after the cursor of the mouse;
- **3D Placement**. This method of insertion is used for quick creation of the arrangement in 3D scene. In the fragment's file special attachments to the floor, walls, ceiling, horizontal surfaces must be created. These attachments are defined by means of connectors with the specific parameters;
- **By points**. A fragment can be fixed to the selected in the 3D scene points upon insertion. Its size changes according to the distance between the points.
- **Smart fragment (macro)**. This method of insertion is used for parametric fragments whose insertion scenario is described in the program (macro) stored directly in the file of the given

fragment or in the external module (DLL). When inserting the file as a fragment the user-specified macro will be executed.

"Default LCS used for Fragment fixing" is selected from the list. The list contains all local coordinate systems of the model that have the flag **"Use for Fragment insertion"** set. When inserting such document as a 3D fragment, the thus defined coordinate system will be automatically offered as the source LCS.

If necessary, the permitted degrees of freedom can be set in the properties of the prepared coordinate system, which will insure the correct behavior of the given 3D fragment in an assembly in the mode of moving mated elements (see the chapter "**Mates and degrees of freedom**").

For uncomplicated parts, which can be conveniently inserted into the assembly in the mode of **dynamic snapping**, the given mode can be turned by default.

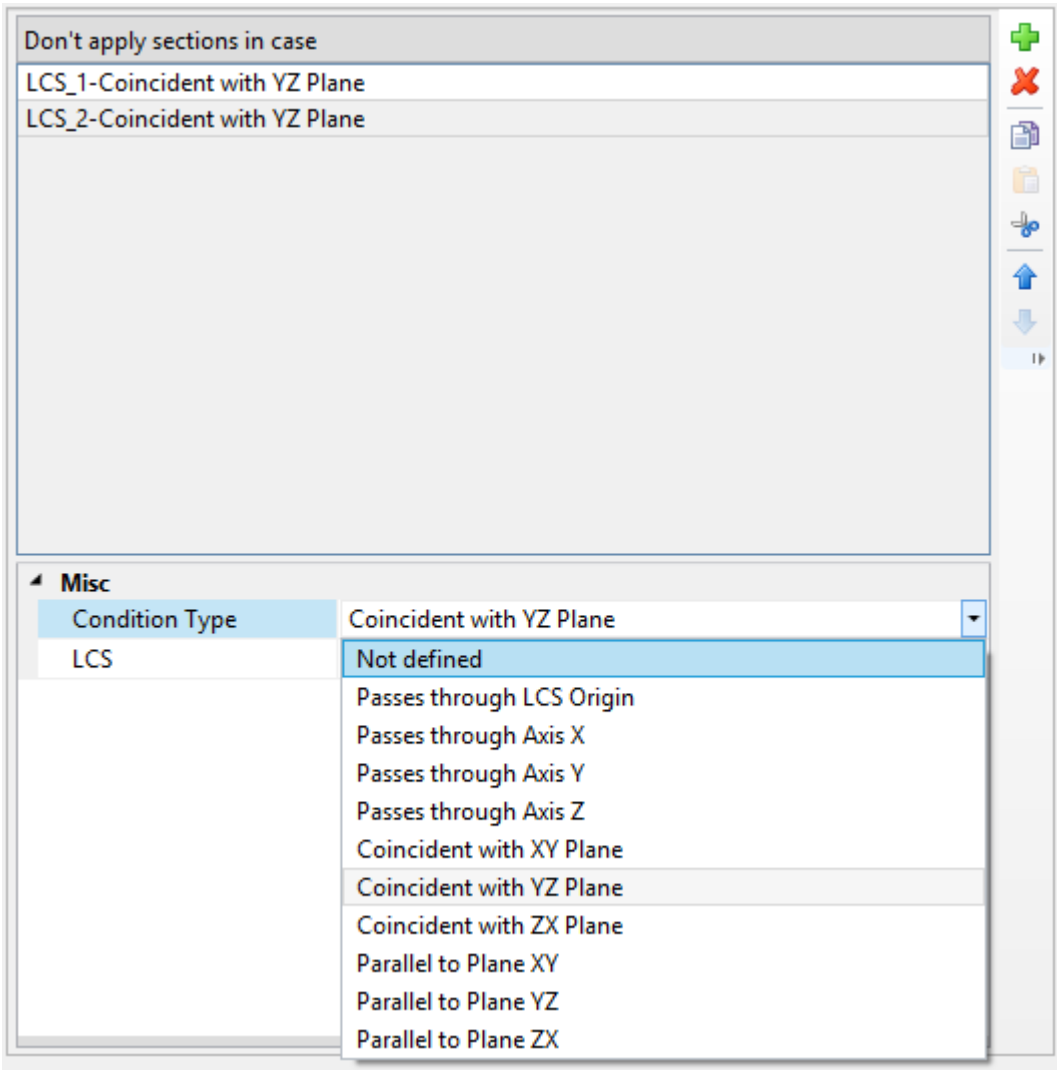
Insert 3D fragment on layer. This parameter defines the name of the layer onto which the fragment will be placed upon its insertion into the assembly. If in the fragment's document the "Insert on layer" parameter is enabled, but upon insertion of the fragment into the assembly this layer does not yet exist, then this layer is created automatically upon request.

Don't show the message "Fragment doesn't have bodies". By default this parameter is disabled. In this case the system does not allow for insertion, into the 3D assembly, of the fragment in which the 3D model is absent or suppressed. The corresponding message in the diagnostics window appears as comments. When this flag is enabled the system allows for insertion of "empty" fragment without issuing any messages.

You can define **options for section views on 2D projections** on the **Sections** tab.

For some models (future components of assemblies) it can be helpful to define special settings for preventing their cutting when used in section views of 2D projections. For example it is known that bolts should not be cut on section views of assembly drawings. This option will help to avoid time consuming manual customization of 2D projection parameters for assemblies that contain such parts.

New condition can be created with the help of button. Section **Application Condition** section will appear with the lists of conditions and LCS names.



Conditions are organized in relation to the alleged source fragment LCS that exists in a document of a future 3D fragment. For example a part may remain uncut if a section plane goes through a specified axis of the source LCS, or coincides with one of the main LCS planes.

Assembly Variable Definition

You can assign an assembly variable counterpart for any external variable of the current document. This can be done in the case, when the external variable drives, for instance, a characteristic parameter (such as, say, diameter), while the assembly variable name has already been defined (for example, D). When assembling this document as a fragment, this assembly variable name will be automatically substituted for the fragment variable in the dialog for defining external variable values (provided that this variable exists in the assembly).

The assembly variable name can be defined in the variable editor. The command is called from the editor textual menu: **Variable > Assembly Variable Name...**. Detailed information on working with the variable editor is available in the chapter "Variables".

"TOP-DOWN" DESIGN

Design Approach

There are two ways of designing a 3D assembly "top-down":

1. By saving already existing solid objects in a separate document.
2. By creating a new Part in the assembly mode.

The first approach allows using already existing solid geometry constructions that can be created by common modeling tools. Saving the body into a separate file can be done, for example, for delivering a set of drawing documentation of the Part. Alternatively, this can be done, if the model is intended for use in this or other assemblies as a 3D fragment.

The "top-down" design technique is quite effective when working with the part in the three-dimensional assembly mode – the second approach. In this mode, you can create new assembly components or modify the existing ones. A Part created in this way can have either associative or non-associative relation with the assembly.

The significant differentiator of the approach is as follows. When working in the assembly mode, you can outsource to the part file not only the assembly operations, but also arbitrary 3D construction elements, such as workplanes, 3D profiles, 3D paths, etc. This property is the main advantage of the assembly-mode design. Thus, for example, you can use assembly edges, 3D nodes or surfaces for defining directions, or use assembly faces, workplanes, etc. for defining the bounding conditions. When creating or editing a part, all system commands are available. You can also use objects already created in the assembly. You can shape up a part without unloading objects from the assembly. Instead, you can reference those as the base data for creating operations or 3D construction elements. In any case, use of assembly objects automatically provides attachment of the part to the assembly.

The "top-down" design approach may not be considered suitable for all cases of designing assemblies. It has a number of shortcomings that somewhat limit its use, such as follows:

- More cumbersome organization, as compared with the approach "from part to assembly";
- Weaker robustness to topological changes in the 3D model;
- Parts are less convenient when using the same 3D model in various assemblies;
- Higher usage of computational resources.

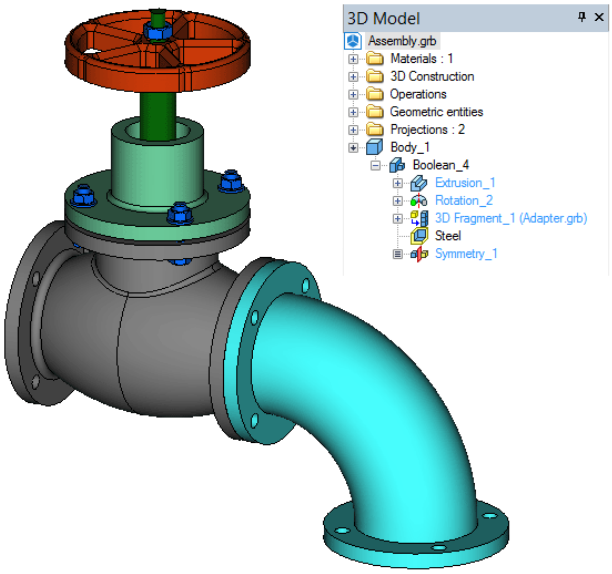
Particulars of Creating and Editing Parts

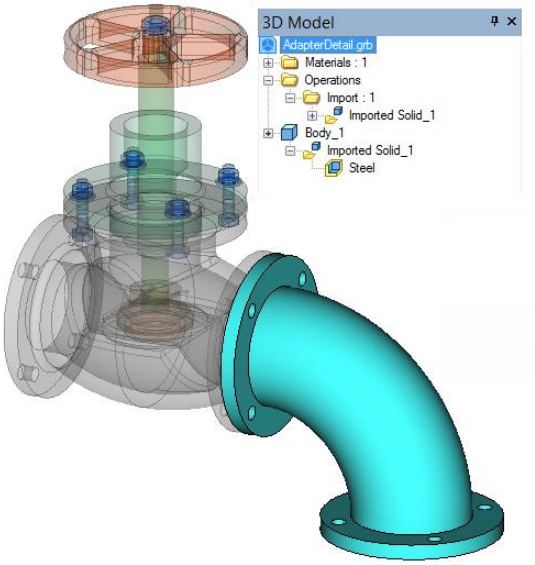
Before you start designing an assembly, you need to think through its composition. Decide, what parametric properties to build in one or another Part; what is the probability of introducing changes in the future, etc. You need to follow certain recommendations and keep in mind the specifics of the

method used in order to achieve the best results in the initial design, as well as to leave open a provision for quick input of changes in the model.

Thus, try to restrict to the Part file most of the steps/operations necessary for building the model, while borrowing from the assembly only the unavoidable data for attaching the Part. This means, you shouldn't always rely on the opportunity of initially creating a most accurate "concept" of the Part directly in the assembly, with later unloading the resulting body of the would-be Part into a separate file. On the contrary, your design should preferably proceed in such a way, that the most of the Part 3D model structure could end up in the Part file. This can be achieved easier by creating the new Part in the assembly mode, when you need, for instance, just edges or faces of the assembly model for the part references and attachment. Following this recommendation will allow, in most cases, modifying the Part without referring to the assembly document. This is particularly important when the introduced changes should affect the Part 3D model structure as the order and composition of the modeling operations change upon adding or deleting operations.

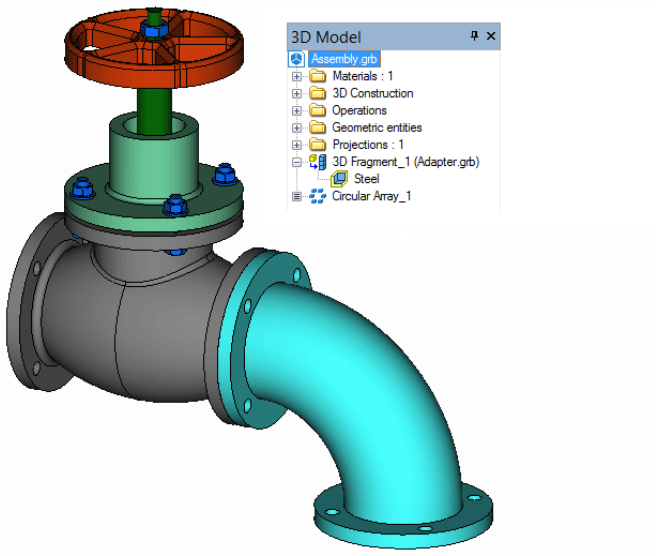
As an example, consider the following diagrams:

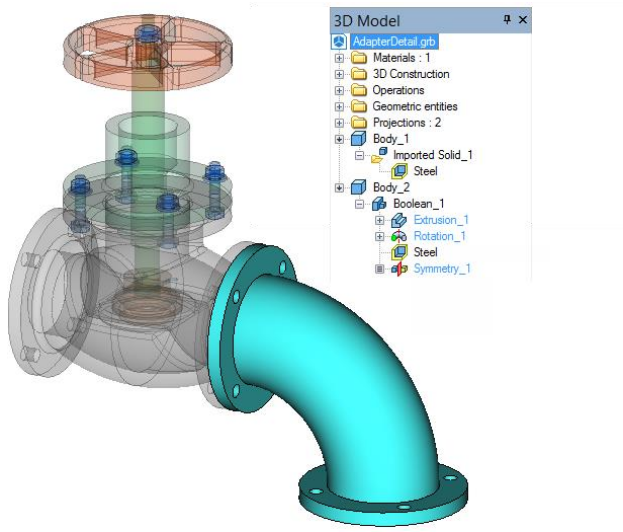




The first two diagrams show the undesirable Part layout. As appears from the assembly structure, the operations were created first that defined the Part body. Those were followed by the operation Part created by unloading the resulted body (Boolean_4). If we try to edit such Part, we will find in its model tree only one external operation unloaded from the assembly. Therefore, to make changes to the original operations of the Part, we will have to load in the assembly. Upon modifying the assembly, we will have to start the procedure of updating the Part file. This order of steps does not promote productivity. However, if the engineer is assured that no changes will be required for the Part in the future, he can proceed with this technique.

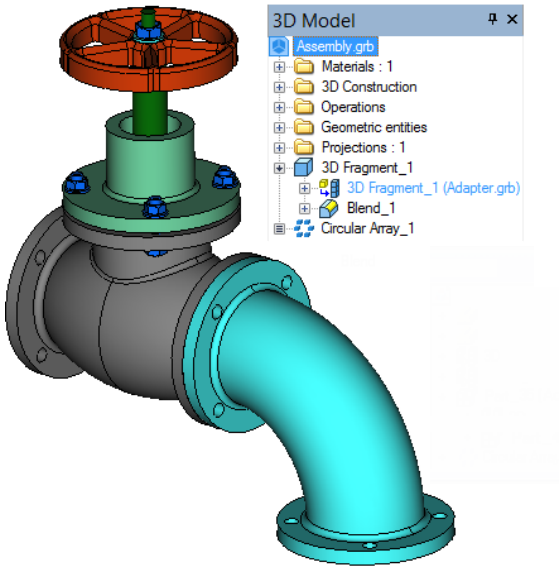
Let's consider another approach:



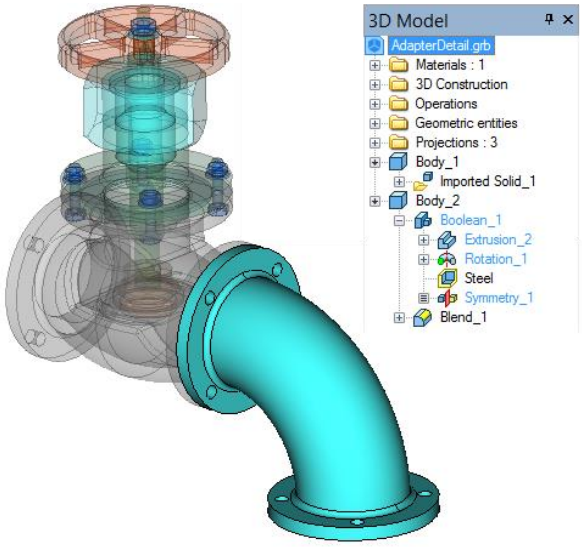


The above diagrams show the recommended approach to the Part creation. In this case, the Part is designed completely in the assembly mode. The flange of the body is used as the geometrical base for attaching the hose. As you can see in the diagram, all operations of the Part body creation are located in the Part file.

Continuing with this example, one can figure out the preferred techniques of introducing changes in the Part geometry. Let's consider a scenario, when we need to "refine" the existing Part by creating, for instance, an edge blend operation in this Part. A common mistake is blending the desired edge directly in the assembly, thus creating the blend operation on top of the operation Part. Proceeding in this way is undesirable, since the blend operation will not reflect on the Part file itself. Preferably, one should perform blending while editing the Part – either in the assembly mode or in a separate file.



Undesired layout

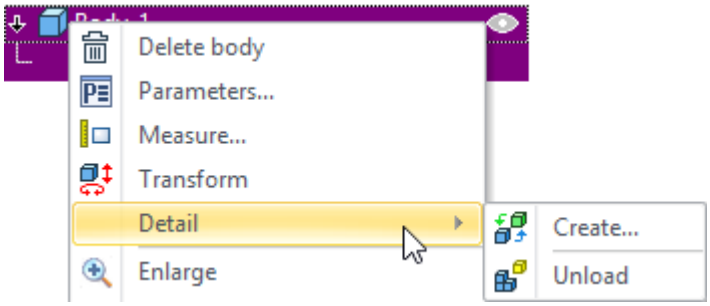


Preferred layout

Detail Creation

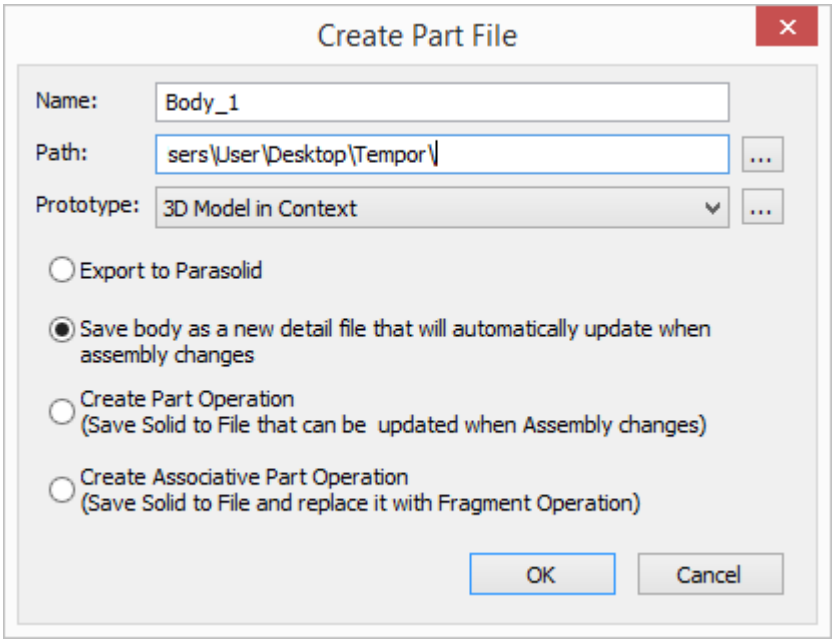
You can unload solid geometry in a separate file using the **Create** and **Unload** operations.

The commands can be activated in context menu in upon selecting any operation in the 3D window or in the **3D Model** window.



Unload

Upon calling the **Unload** command, the system offers selecting the type of relation with the Part. The selection is done in the dialog box that comes up automatically:



You can select a filename in the **Name** field.

You can select a directory for the file in the **Path** field.

Prototype. You can select a prototype for the part. The option allows you to select your own prototype with the required parameters.

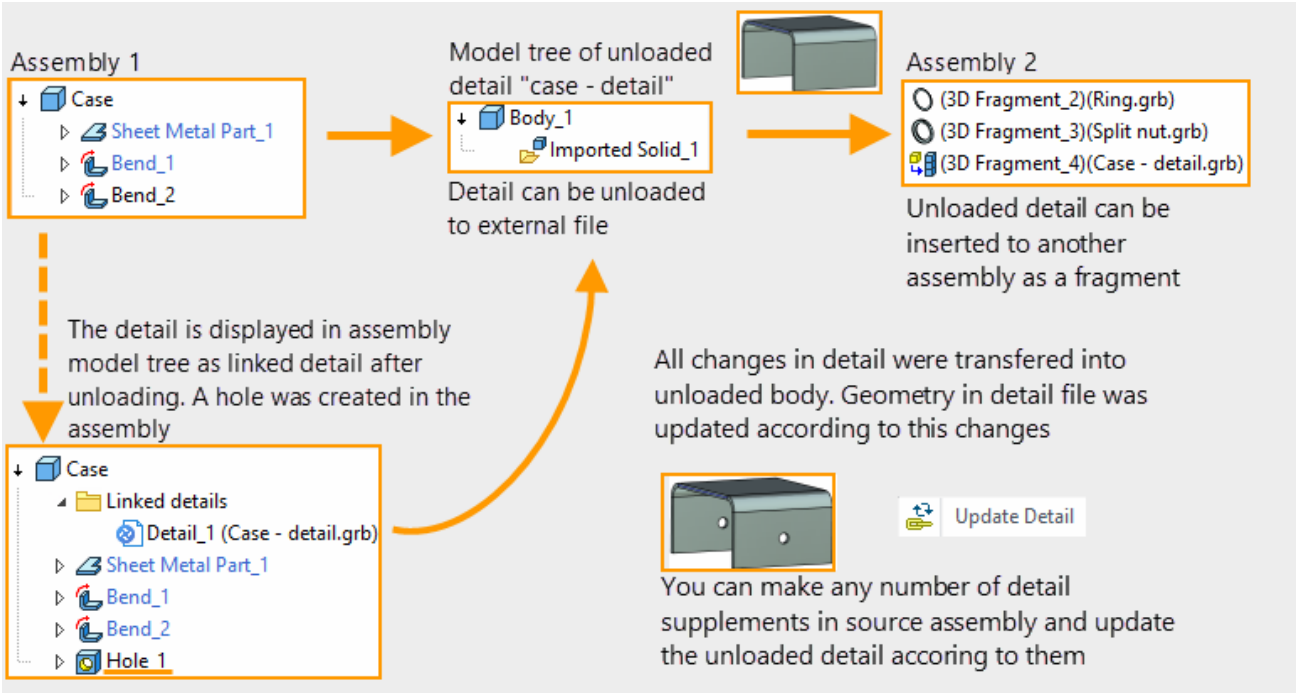
Export (Save Solid to File)

If the item "Export (Save Solid to File)" was selected the dialog for export of the detail in Parasolid format appears. The selected elements are saved in a separate file of Parasolid (*.x_t) format, that will no longer depend on the assembly drawing. The export operation in no way affects the assembly structure.

Save body as a new detail file that will automatically update when assembly changes:

The model with its current geometry is unloaded to a separate GRB file.

An important feature of this method is ability to transfer all model changes from assembly to detail. Detail can be inserted as a fragment into other assemblies.

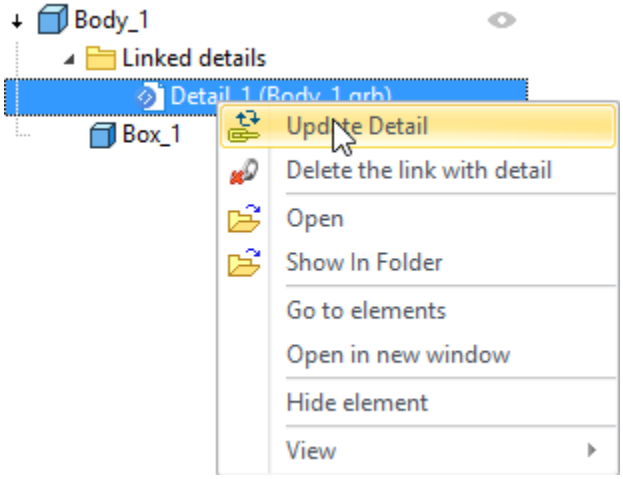


The history of detail creation is not unloaded from the Model Tree.

“Linked details” folder appears in the **3D Model** window after the detail unloading. The folder includes all details that are unloaded for current body or fragment using the current type of relation.

Context menu:

Update detail. Allows to update the unloaded detail according to the detail geometry changes in the assembly.

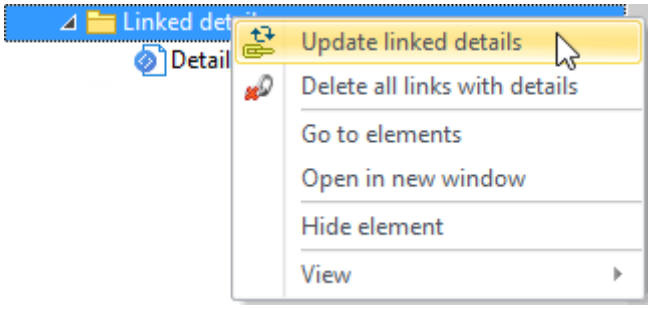


Delete the link with detail. Allows to delete link with the unloaded detail. The detail will not be linked with the assembly and its update will be unavailable.

Open. Allows to open the detail file.

Show in folder. Shows the detail file in Windows explorer.

The **Update linked details** command is available in the context menu. It allows updating details for the selected objects.



The command can be called using text menu:

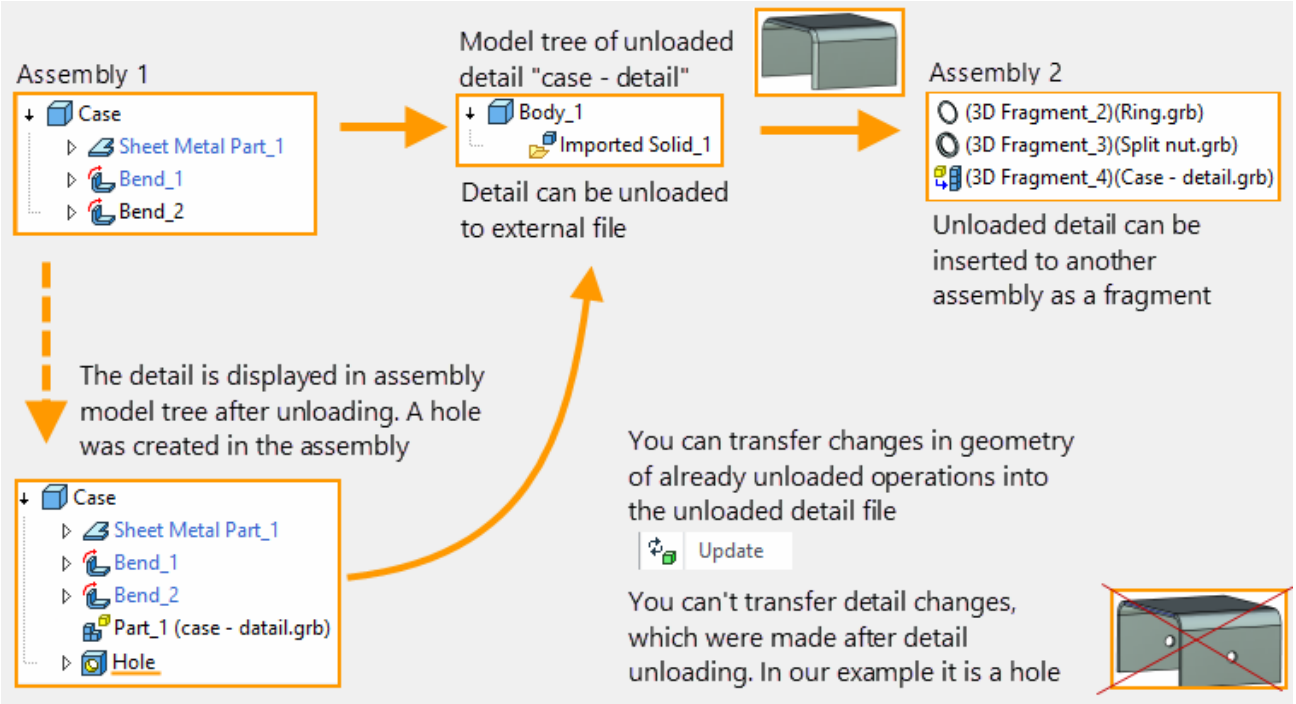
Icon	Ribbon
Keyboard	Textual Menu
	File > Assembly > Update linked details

In this case all details in the scene will be updated.

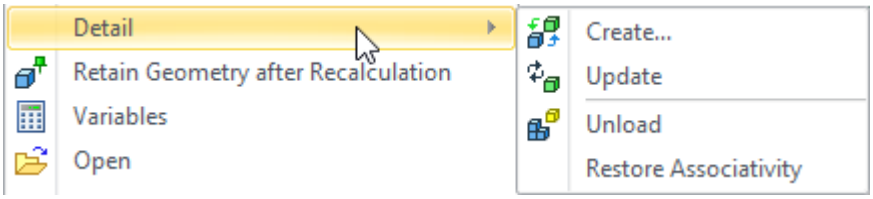
Create Part Operation (Save Solid to File that can be updated when Assembly changes):

The model with its current geometry is unloaded to a separate GRB file.

Detail can be updated when geometry of unloaded operations is changed in the assembly file. However, the operations that were created after detail unloading will not be transferred to detail.



You need to select **Detail** > **Update** from the detail context menu to update the detail according to its geometry changes in the assembly. You can update detail according to the changes in already unloaded geometry.



Restore Associativity. The option allows to unload changes in the detail file into the assembly. Note that errors may occur if there were changes in the detail geometry in the assembly.

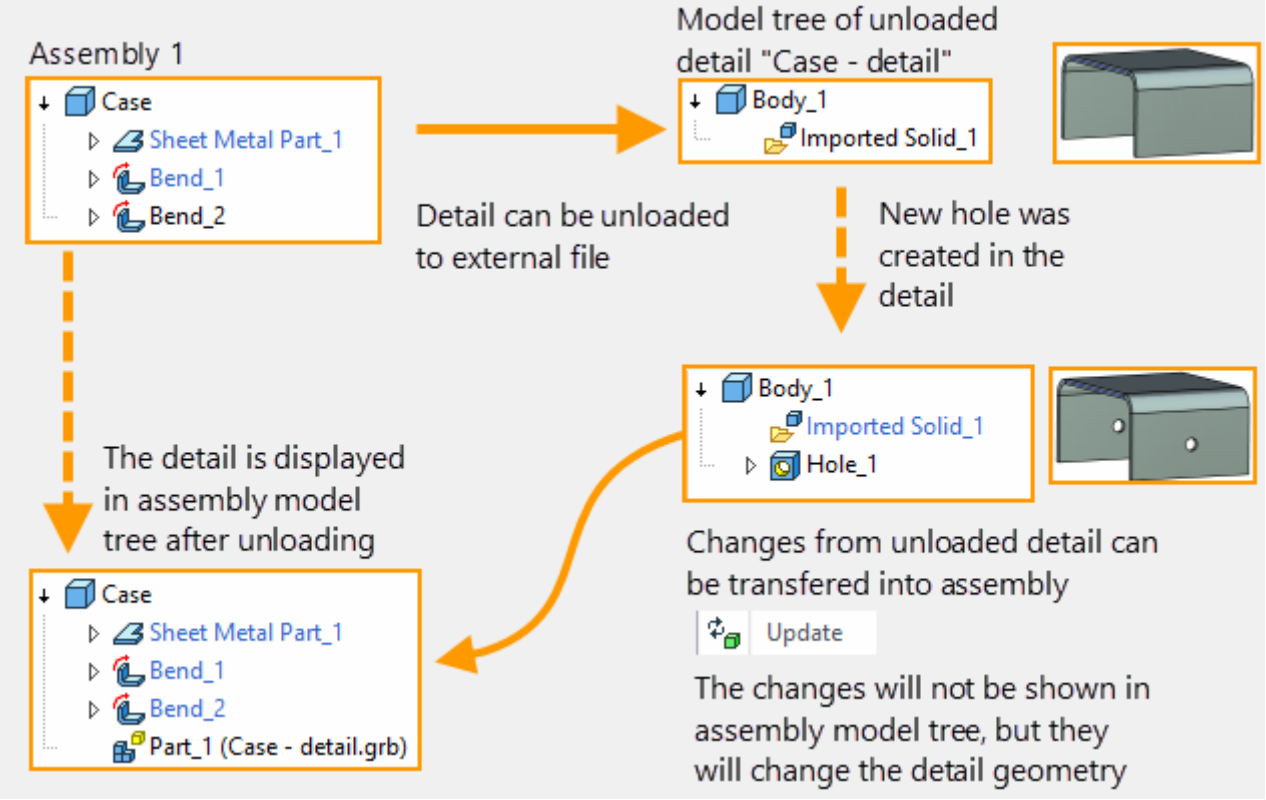
For example, a hole was created in the assembly after the detail unloading, and blend was applied in the detail file. There will be an error because the face for the hole creation was changed.

Create Associative Part Operation (Save Solid to File and replace it with Fragment Operation):

The model with its current geometry is unloaded to a separate GRB file. Meanwhile, the part itself is inserted back into the assembly, using the 3D fragment technique.

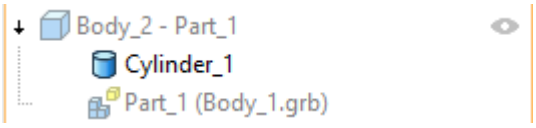
This establishes two-directional relation between the assembly and the part file.

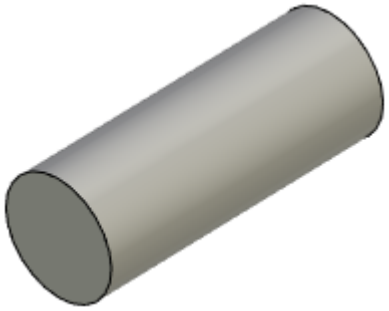
It allows system to update the geometry in the assembly file, if there were any changes in detail. In this case, the detail plays the same role, as the 3D fragment, with the difference that the original data for the detail exists in the assembly document.



Moreover, you can rollback model to the moment of detail unloading and change the detail geometry.

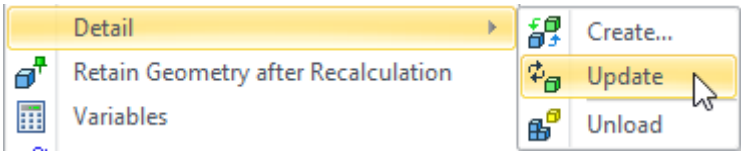
A cylinder body was created in the assembly and unloaded as a detail. Then the rollback to the moment before unloading was performed in the assembly. **Rollback model** option from the context menu Cylinder was used for this purpose.





After the rollback, a hole was created and the rollback was finished.

After finishing of the rollback, you can update the detail and all changes will appear in the detail file.



Create

If the original file of a parametric 3D fragment or Detai differs from its instance currently used in the assembly due to changes in the variables, you may want to save the latter instance in a separate file. This may be necessary, for instance, in order to file a new set of documentation reflecting the current modification. The detail operation creates a new copy of the 3D fragment or body in separate file that can be saved under a different name.

If drawings existed in fragment file that was unloaded as a detail, they will be updated according to the fragment file changes in the assembly.

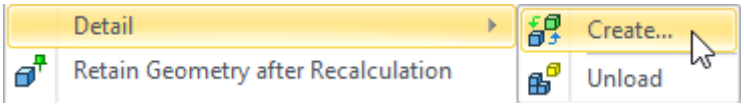
You can unload a detail using one of the following ways:

In the first case the new detail copy does not have any relation with the assembly. It merely provides the user with a modification of the part recalculated with certain variable values and written to a separate file.

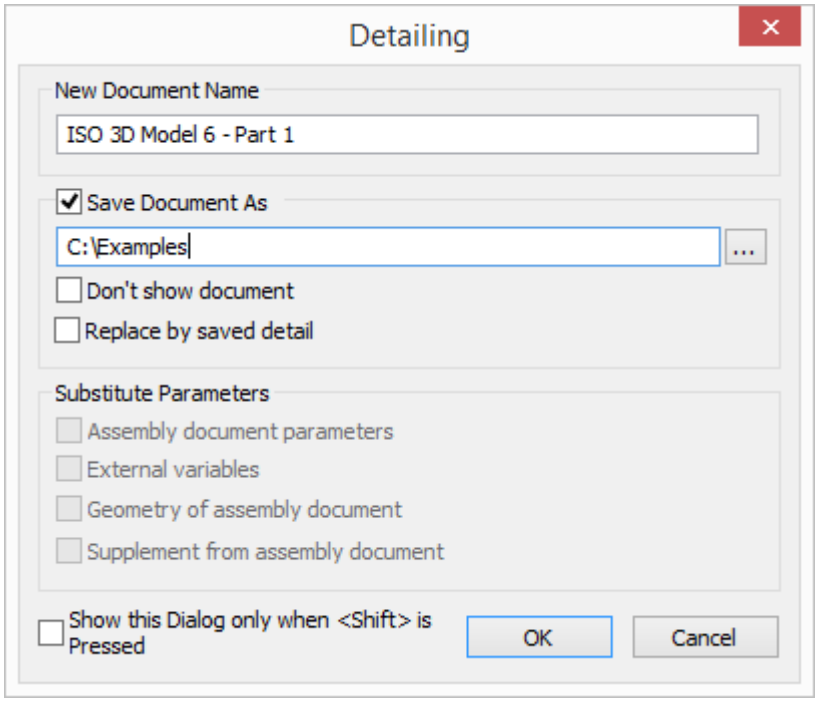
In the second case, fragment, Detail or body in the assembly can be replaced with the unloaded detail. It is important that after this replacement it is possible to transfer all changes from the assembly to the detail.

Command Options

The detail creation command is launched from the context menu corresponding to the selected operation of the 3D fragment, Detail or body, whose detail we want to create.



When performing it, a dialog appears, in which you need to specify the name of the document being created and whether it is required to save it immediately to a file. You can specify derictory for the file.



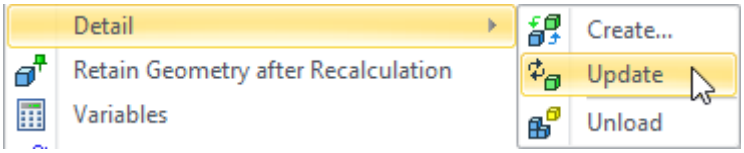
New Document Name. You can specify name for the detail document.

Save Document As. If the flag is not set, the detail file will be opened in T-FLEX CAD, but it will not be saved. It allows to preview the detail.

Don't show document. If the flag is not set, the detail file will be opend in T-FLEX CAD after saving.

Replace by saved detail. If the flag is set the unloaded detail will be linked with the assembly. It allows to update the detail according to the changes in the assembly.

To update the detail use update option from the context menu of the detail in the assembly.



The unloaded detail is a fragment and all its changes can be unloaded to the assembly.

Assembly document parameters. When the option is set all assembly parameters from **ST: Document parameters** command are transfered to the assembly.

External variables. When the option is active all variables from the assembly document are unloaded to the detail

Geometry of assembly document. The option is used for fragments in context and adaptive fragments. It allows to unload geometry based on which the fragment was created.

For example adaptive fragment was inserted into the assembly. If geometry on the basis of which the fragment was created has changed and the changes can be unloaded to the detail. If the flag is not set,

the detail will be created based on the geometry which had existed at the time of inserting the adaptive fragment.

Supplement from assembly document. Allows to unload supplements of the fragment or body from the assembly.

For example, a fragment was added to the assembly. A hole was created in the fragment. The supplement option was enabled upon the hole creation. The supplement can be unloaded to the detail with the fragment.

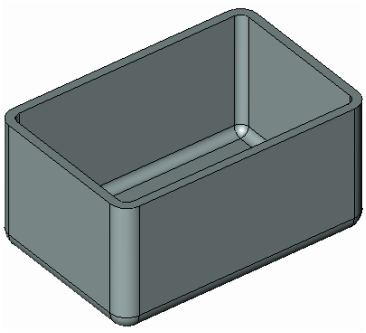
Working in the Assembly Mode (Assembly Context). Creating a Part

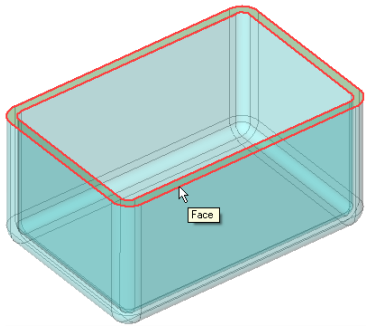
To create a new Part in the assembly mode, call the following command from the **File** menu:

Icon	Ribbon
	Assembly → Assembly → Fragment in context → Create
Keyboard	Textual Menu
<FM>	File > Fragment > Create"

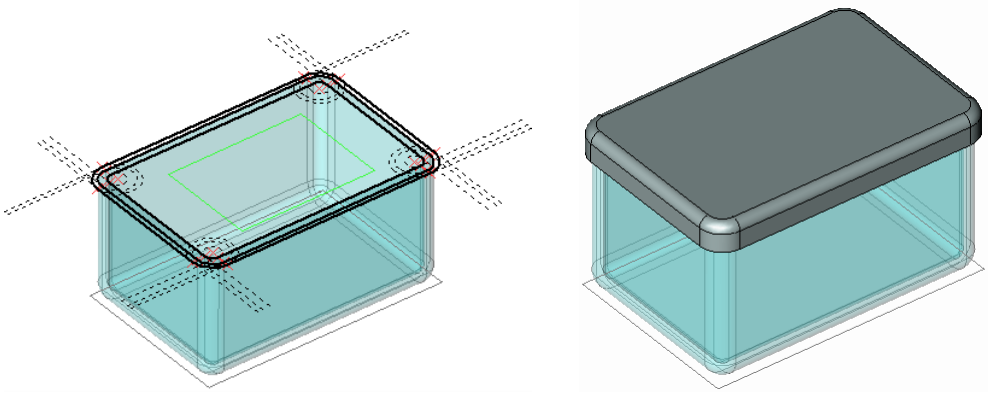
Immediately after calling the command, the system asks the file name for the new Part. The assembly mode turns on thereafter, with all assembly operations becoming transparent. All assembly objects - faces, vertices, 3D construction elements and operations - remain available for selection. From now on, anything created by the usual tools will belong to the Part file.

Suppose, we need to create a top for a box part. To do this, let's call the detail creation command and select the top-end face.



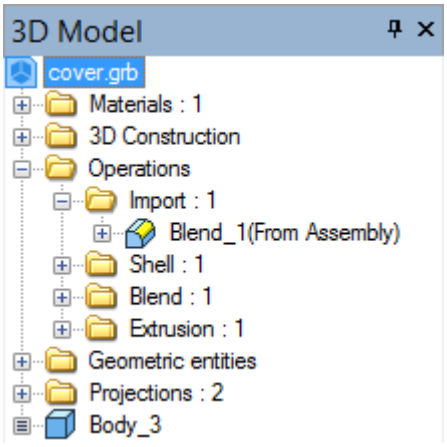


This gives us access to the 2D drafting commands. Create the necessary graphic elements for the 3D profile that will be used for the top creation. Do all steps in the same order as in conventional drafting on the active workplane.



Upon extruding the profile and rounding the edges, we get the ready model of the top.

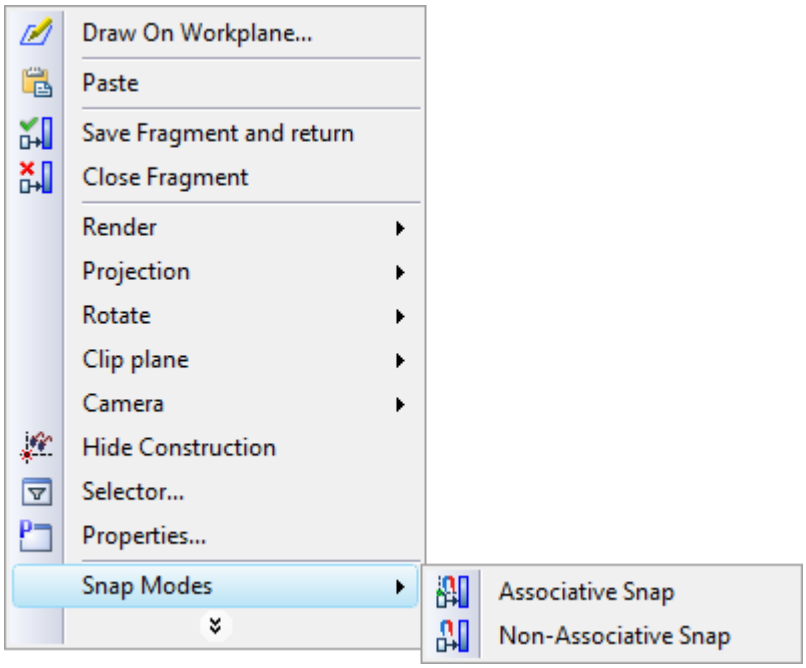
The 3D model tree displays the structure of the Part model. All elements that were unloaded from the assembly model are labeled "(From Assembly)".



> The label that marks in the model tree the elements unloaded from the assembly is appended only when editing the Part in the assembly mode. If you open the Part file separately for editing, then the elements unloaded from the assembly will appear in the model structure merely as the external imported models.

Sometimes, the operations unloaded from the assembly model into the Part do not need to be displayed in the 3D window. Therefore, those can be hidden by specifying an invisible level for them. In such a case, those will not contribute to the Part mass properties either.

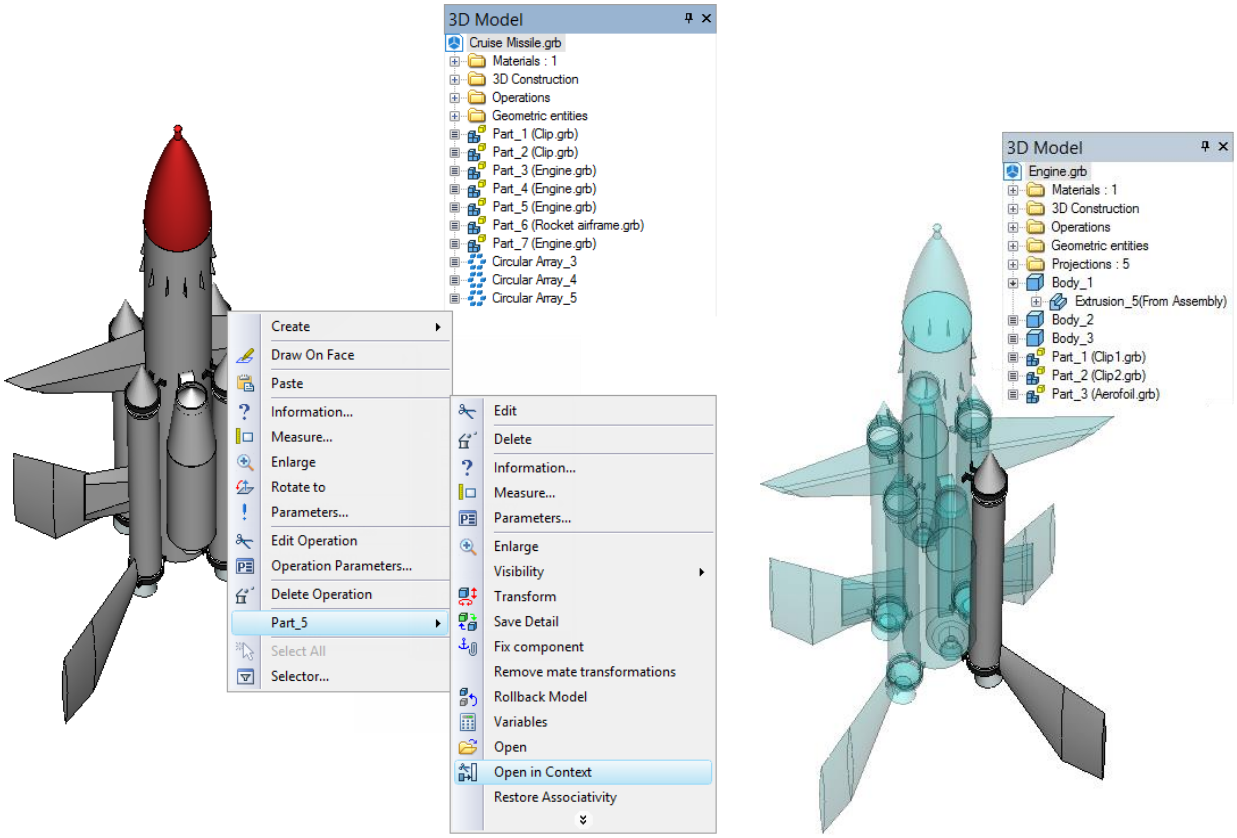
Editing can be finished at any moment either with saving the Part file or by rejecting the changes done since the last Part save. The commands for returning to the assembly document can be accessed in the context menu by right clicking in the 3D window.



The same context menu contains another item with two unfolding sub-items for controlling the attachment mode to the assembly elements. The attachment can be by **associative or non-associative snap**. When creating Part elements in the associative attachment mode, the relation is maintained with the assembly objects. This mode allows propagating any changes from the assembly geometry in the Part model. In the non-associative attachment mode, any assembly object reference is replaced by a non-associative copy of the assembly object, realized as in external operation. This external operation is in no way related to the assembly model. If you don't want this copy to appear in the assembly, hide it by assigning, for instance, an appropriate visibility level.

It is also possible to undo any attachments to the assembly elements, while maintaining the object snapping only to the Part file elements. To turn off any mode of attaching to the assembly elements, you need to pick once again the item of the turned on mode in the context menu in order to undo the snapping.

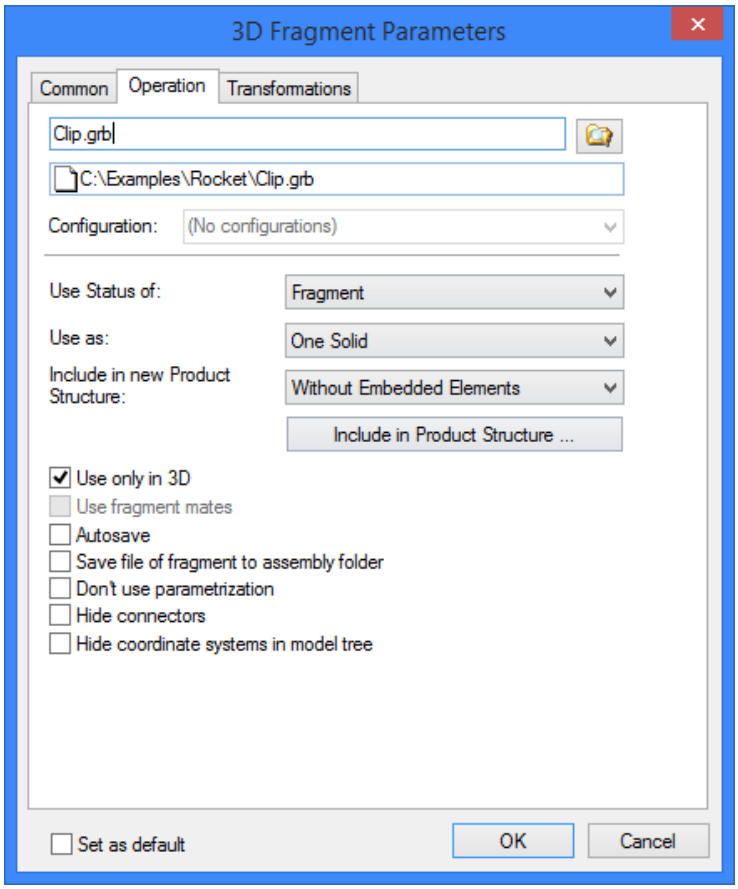
Generally speaking, you can work with a 3D fragment in the assembly model as well as with a Part file. To start editing in the assembly mode, call the command “Edit” from the context menu invoked over the operation “Part” or “3D fragment”.



When editing a 3D fragment in the assembly mode, you can use the assembly elements as references just like when creating/editing a Part. This will not convert the edited "3D Fragment" operation into the "Part", rather, some Part properties will be attributed to the fragment. Thus, for example, use of configurations will be banned, and dependency on the assembly geometry won't be supported. (You need to remember this in order to avoid recursive dependencies.)

Part Properties

The operation "Part" possesses mostly the same parameters as the operation "3D Fragment". In fact, it is a 3D fragment, too, however created by different means. The only item not accessible in the "Operation" tab will be the configuration selection field, as the use of configurations is not allowed for Parts. (Use of configurations is described below.)



If the Part is expected to appear in BOMs, define the necessary data in the same way as for a 3D fragment.

Part transformations are done in the same way as for any other operation that is not using coordinate systems for attachment. For detailed information on this, you can refer to the chapter **Assigning Transformations to 3D Entities**.

Relation between Assembly and Part File

Part file path references are made by the same rules as those of the 3D fragments (see above).

The type of relation between the Part and the assembly is established at the time of the Part creation. As was mentioned earlier, changes can be propagated from the assembly to the Part or vice versa, depending on the relation type. The system would repeatedly unload the original data into the Part file only when forced by the user. All Part files can be updated by the command:

Icon	Ribbon
	Assembly → Assembly → Fragment in context → Refresh Files
Keyboard	Textual Menu
<FG>	File > Fragment > Refresh Files

To update just one selected Part file, call the command “Update”. This command is available in the context menu upon right clicking over the Part.

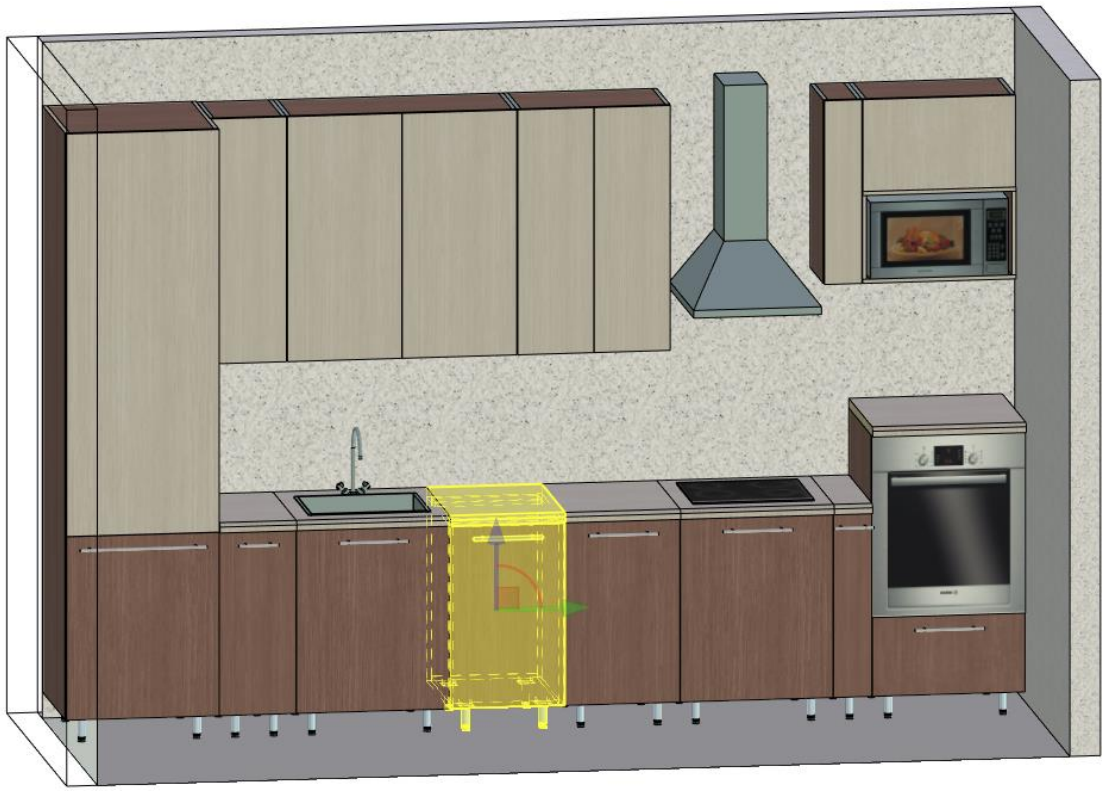
If the reverse relation is established between the Part and the assembly (as in the case of a 3D fragment), then the system tracks changes in the Part files automatically. Everything works as if with 3D fragments. If you need to force reloading of all modified Parts into the assembly, call the command:

Icon	Ribbon
	Assembly → Move Assembly → Component links → Update Links
Keyboard	Textual Menu
<UL>	File > Assembly > Update Links

Reloading the data from one specific Part into the assembly can be done by the context menu command “Update”.

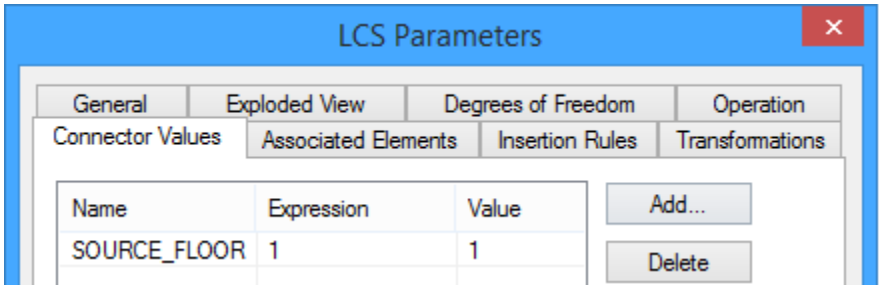
3D Placement

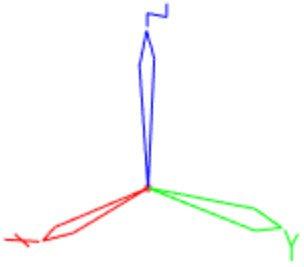
T-FLEX CAD has additional mechanism of 3D fragments positioning – **3D placement**. It allows you to quickly create arrangements in 3D scene by using special attachments to the floor, walls, ceiling, and horizontal surfaces. Attachments are determined with the help of connectors with the specific parameters.



Fastenings for 3D Placement

3D placement fastening – is LCS - connector, which contains the value with special name <prefix>_<type> type, where





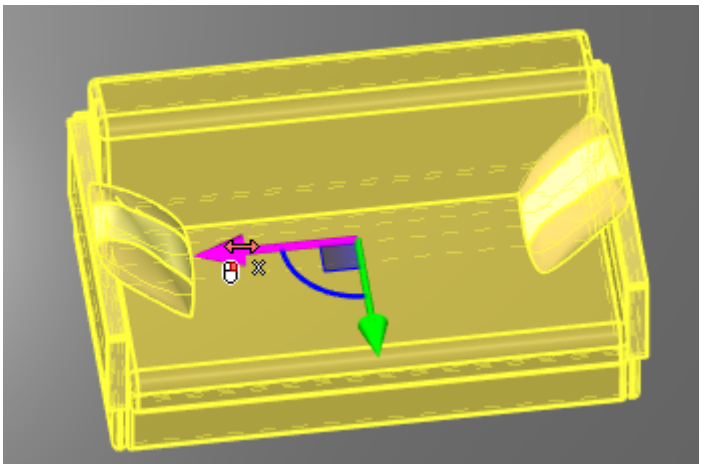
<prefix> substring defines fastening type for the LCS and may take one of the following values:

- SOURCE (SOURCE_<type>) –initial fastening. This type is specified for objects that will be moved in scene (for example, furniture elements).
- TARGET (TARGET_<type>) –target fastening. This type is specified for objects that define attachment planes (for example, wall or floor) or fastenings.

<type> defines type of connectors that interacts with the current LCS. Acceptable versions of <type> depend on the attachment type.

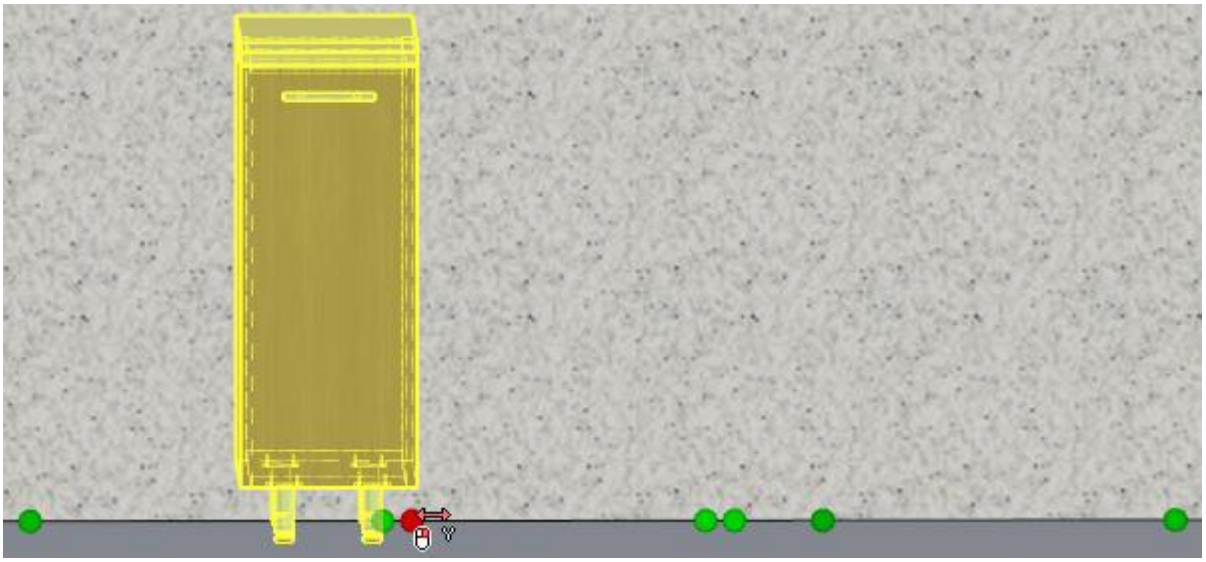
There are two available types of attachments: “LCS –planar face” and “LCS-LCS”.

- “LCS –planar face” – a fragment with the connector SOURCE_ <type> will be moved in the plane of object with connector TARGET_<type> if they have the same <type>.



Sofa moves on the floor plane

- “LCS-LCS” – the connector with SOURCE_ <type> fastening will be matched with one of the existing in assembly connectors with type TARGET_<type>, if they have the same <type>.



Cupboard is attached to one of the “LCS-LCS” fastenings.

More information about attachment types can be found below.

A single connector can contain several <prefix>_<type> values simultaneously. Thus, it can define several fastenings for 3D placement.

SOURCE_RIGHT	1	1
TARGET_LEFT	1	1

SOURCE_<type> fastening will be attached to the TARGET_<type> fastening according to the 3D placement rules, only if fastening is defined in the document of 3D placement object. You can find example of the fastening creation below.

Fastening TARGET_<type> can be specified based on any connector of the current document.

Attachment rules

As it was said above 3D placement objects can be attached according to the "LCS–planar face" and/or "LCS–LCS" rules.

Attachment Rule «LCS–planar face»

There are the following types of the "LCS–planar face" attachments: <type> = {WALL, FLOOR, TABLE, CEILING}. Below are pairs of initial and target fastenings corresponding to these types:

SOURCE_WALL	TARGET_WALL
SOURCE_FLOOR	TARGET_FLOOR
SOURCE_TABLE	TARGET_TABLE
SOURCE_CEILING	TARGET_CEILING

To specify the "LCS–planar face" attachment it is necessary to define the attachment plane and a set of associated faces of the target fastening connector.

The associated faces are highlighted with yellow color.

The attachment plane is specified by the center of target fastening LCS and by the attachment axis.

The X-axis of the target fastening LCS is the attachment axis for the WALL type.

The Z-axis of the LCS of the target fastening is the attachment axis for the FLOOR, TABLE and CEILING types. In addition, the associated faces of the connector must lie in the attachment plane.

Face outside the attachment plane will not be considered in the placement mode.

You must follow particular rules for successful result:

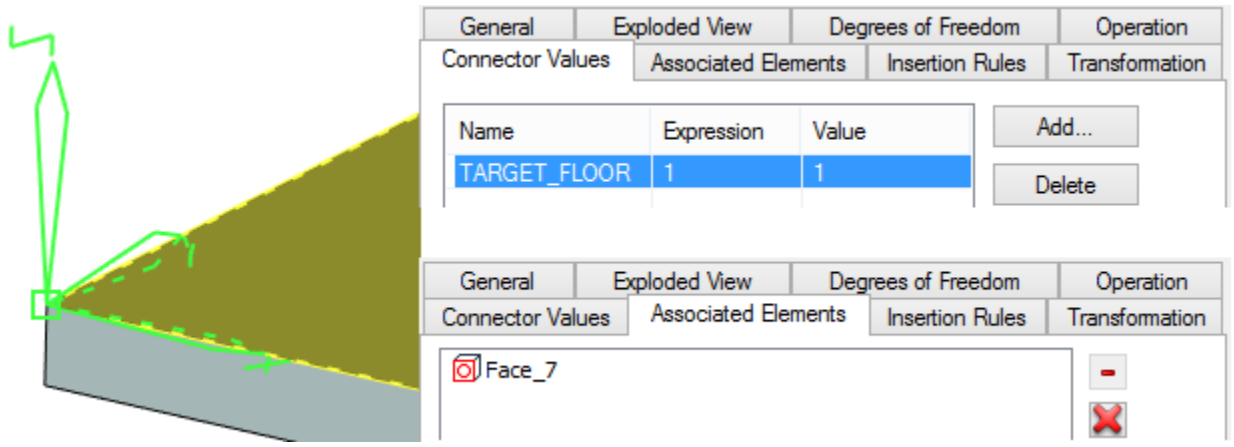
- The Z-axes of all fastenings inside the fragment of 3D placement must be coaxial. The placement object orients itself according to the Z-axis of global coordinate system upon insertion;

– The Z-axes of all other fastenings must be coaxial with the Z-axis of the global coordinate system;

– The initial fastening of every separately taken type “SOURCE_<type>” must be unique in the fragment document for the 3D placement.

Example of Fastening with Target Type Creation

LCS was created for the “Floor” object. Its Z-axis is coaxial the Z-axis of the global coordinate system. The LCS is set as connector.



The attachment of target fastening TARGET_FLOOR is set for the connector. Position of LCS center and direction of Z-axis specify the plane in which the inserted fragment with SOURCE_FLOOR fastening is moved.

The associated face lying in the attachment plane is specified. The face is highlighted with yellow color.

System analyzes all “SOURCE” type connectors in fragment’s document when you insert the fragment into assembly and selects possible attachments. Only the pairs of connectors with the values of the same type can be attached. For example, it can be “SOURCE_WALL” and “TARGET_WALL”.

> If the associated face is not set, the fragment is moved in such way that axis of the LCS of the initial and target fastenings coincide and the center of the LCS of the initial fastening lies in the attachment plane.

Attachment Rule «LCS–LCS»

You can specify any name for <type>, when you use “LCS-LCS” attachment rule. The only exception is that you can’t specify names used for “LCS-planar face” rule given above. For example, you can specify source fastening name SOURCE_POINT.

You can specify several connectors with the same names. In this case, the set of fastenings will be created and the fastening of the inserted fragment will be linked with it.

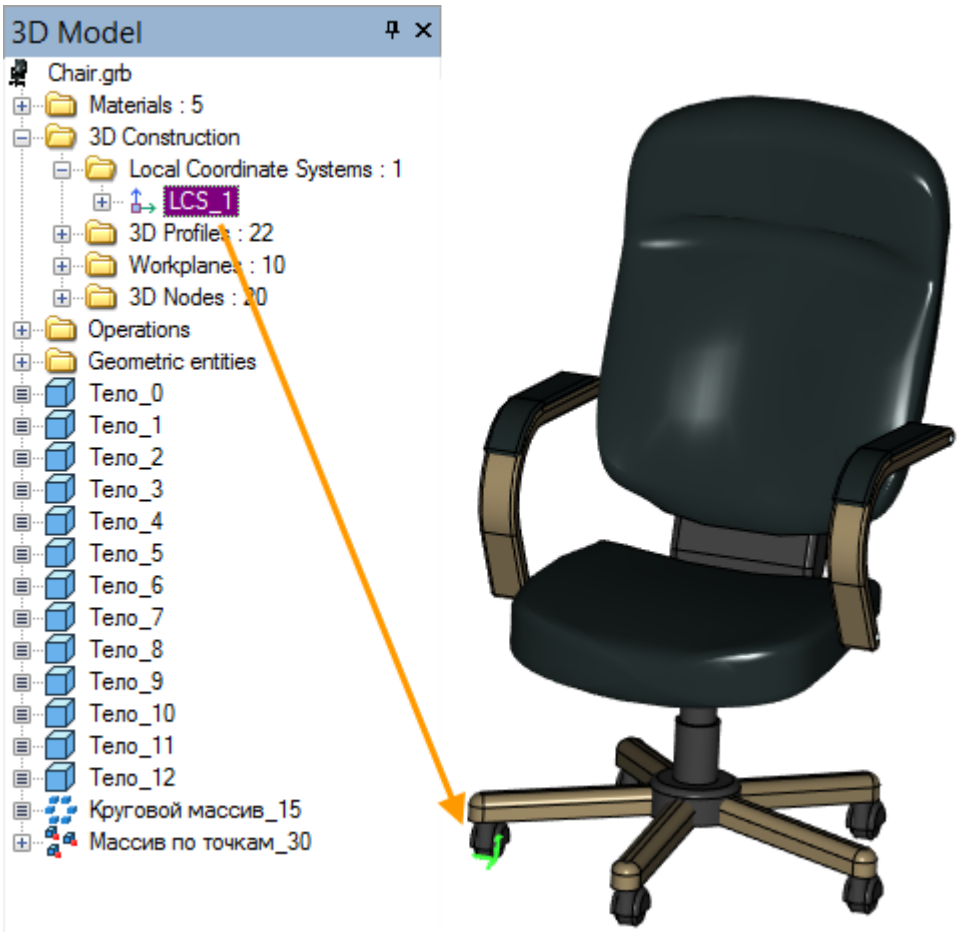
When you insert a fragment with initial fastening, it will be transformed in such a way that the LCS of the initial and target fastenings coincide.

If there are any associated elements specified for fastening, they will be automatically selected when you point the cursor on them.

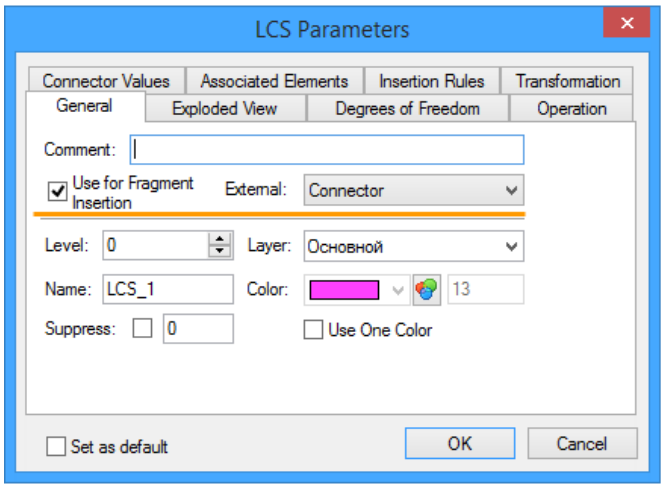
Source Fastening Creation

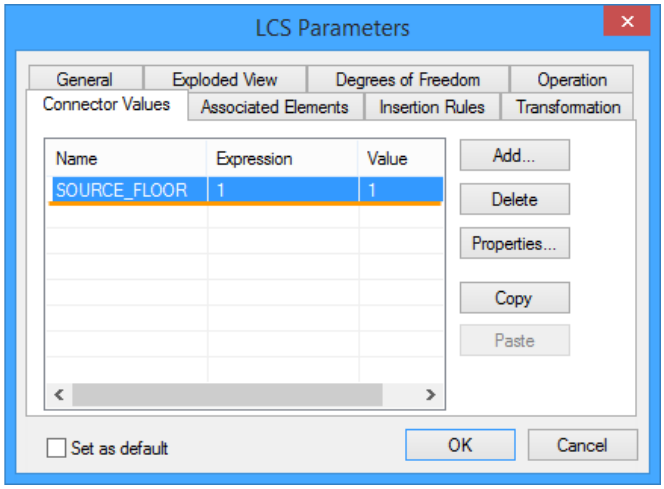
For a fastening creation, you need to follow the next steps:

➔ Create a new LCS. Its Z-axis direction should coincide with the Z-axis of the global coordinate system.



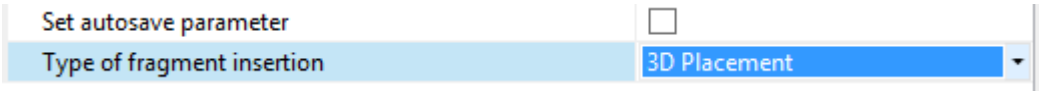
➔ Open **Parameters** window of the created LCS and on the General tab specify it as connector for the fragment insertion.





➔ You need to specify the connector name on the **Connector Values** tab, according to the described above rules. In the example, SOURCE_FLOOR name is specified. SOURCE means that the initial fastening was created for the chair. The **Expression** is set to "1" for all fastenings.

➔ It is necessary to set **Type of fragment insertion – 3D Placement** in the status of the fragment to insert it using the 3D placement mechanism. You need to activate **ST: Set Document Parameters** and open **3D Fragment** tab.



Insert 3D Placement Element

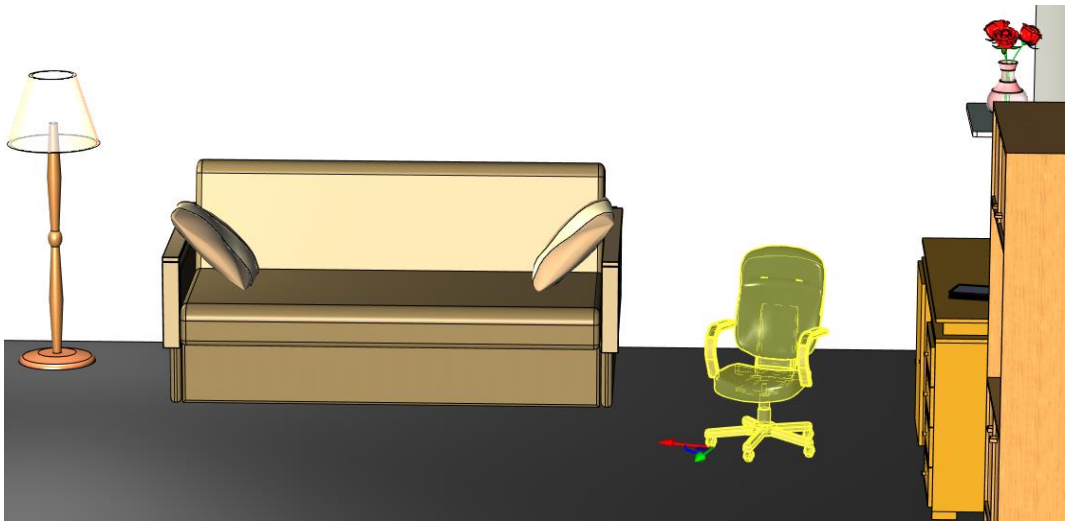
Fragment insertion is performed using command:

Icon	Ribbon
	Assembly → Assembly → 3D Fragment
Keyboard	Textual Menu
<3F>	Operation > Insert 3D Fragment

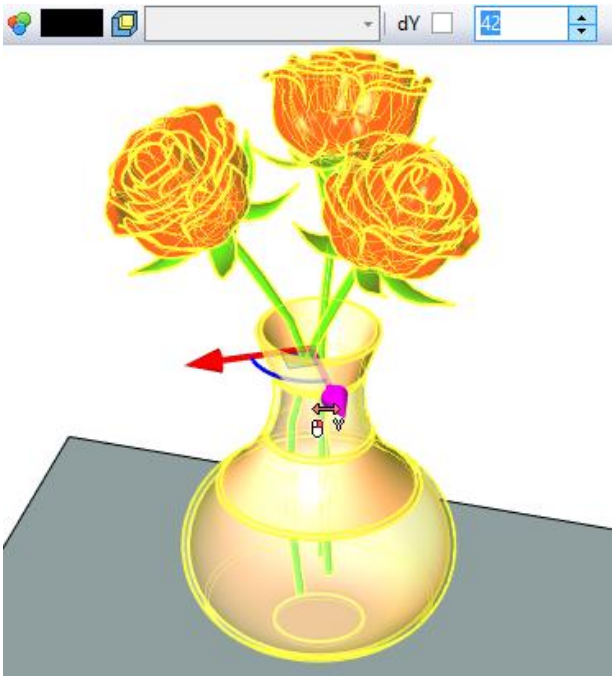
If the fragment insertion type is set to **3D placement,** the **Placement** command activates automatically.

To attach the fragment by the 3D placement method it is required to create source placement fastenings (SOURCE_<type>). Target fastenings should exist in the assembly file (TARGET_<type>). They can be created directly in the assembly document or be raised from 3D fragments that already exist in the assembly.

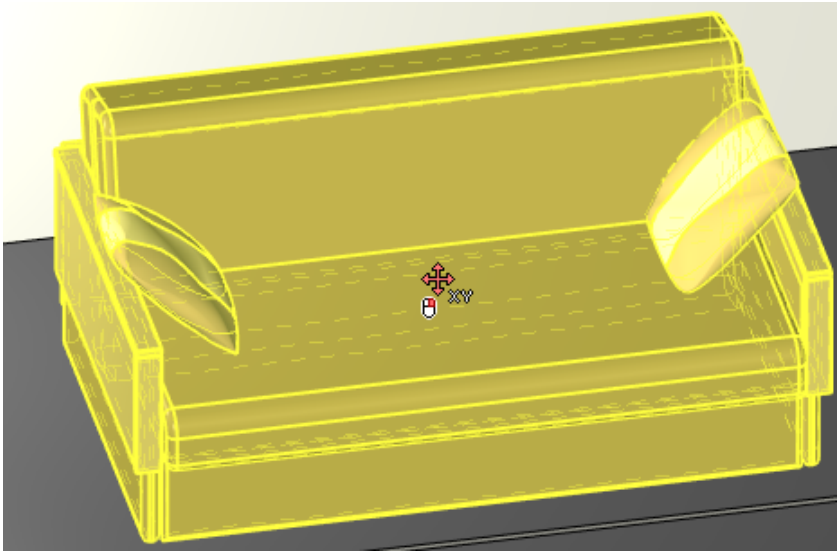
The body intersection checking is dynamically enabled when inserting 3D fragments in the 3D placement mode to ensure correctness of elements placement.



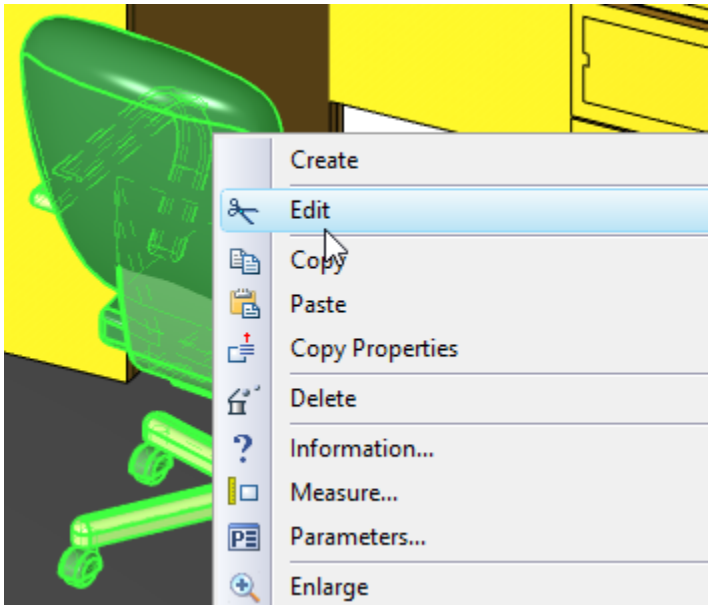
The fragment is moved and rotated using special manipulator, which degrees of freedom depend on the attachment type. 3D placement object with the specified single attachment «LCS–Planar face» can be rotated around axis of the attachment, if this attachment has the type FLOOR or TABLE. For rotation, it is possible to use manipulators or the **Angle of rotation** parameter on the system toolbar.



You can translate fragment along the attachment plane after clicking on it. The fragment will follow cursor according to the attachment rules.



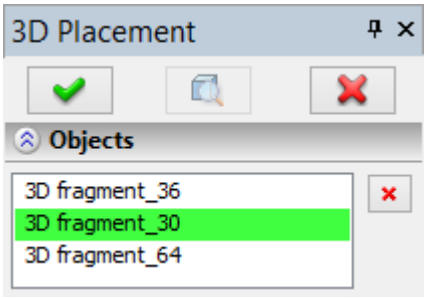
You need to enter its edit mode and to select item **Edit** in the fragment context menu to activate the 3D placement mode for the existing fragment.



3D Placement Properties

The **3D Placement** properties window appears after pressing button **Edit**.

Objects Tab



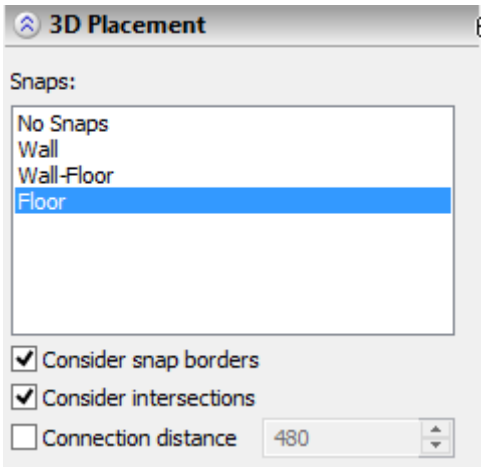
You can specify translations for several fragments in assembly without exiting 3D placement mode. Selected fragments will be added to the **Objects** list in the properties window. The last selected fragment is highlighted with green color.

You can select fragments from the list again, to continue their movement.

All objects movements made in the 3D placement mode are stored consequently. When you press , the last performed movement will be discarded.

Use button to delete a fragment from the list and discard all its changes.

3D Placement Tab



The set of available "LCS–Planar face" attachments is displayed in the **"3D Placement"** list of the properties window. The attachments can be single (for example, "Floor", "Wall") or double ("Wall–Floor").

Consider snap borders. The parameter allows to consider borders of attachment object. In this case, you cannot translate fragment over boarder of the target object. For example, movement of a table will be limited by the floor space.

Consider intersections. When the flag is set, objects can't intersect each other.

Connection distance. The flag specifies distance between two "LCS – LCS" fastenings at which they will be activated.

Automenu Options

There are following options in the automenu:

Option **Apply changes** appears if at least one fragment was moved. The option allows applying all changes to all fragments in the **Objects** list without exiting command. Currently selected fragment stays in the **Objects** list.

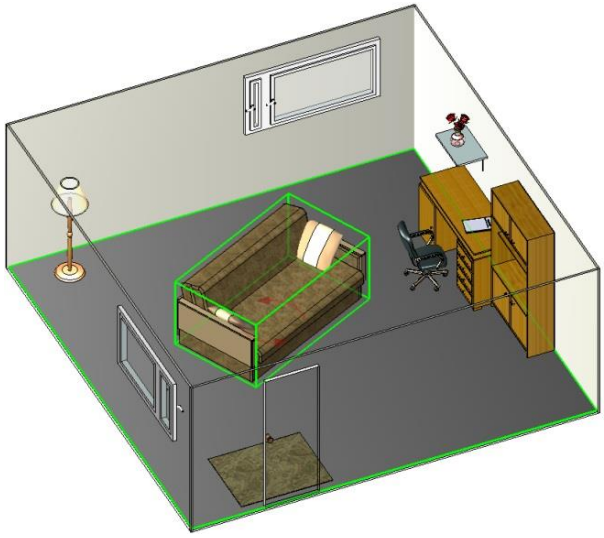
Option allows to change location of a fragment manipulator. It is located in the center of bound box by default. The manipulator is placed according to the LCS position when option is selected.

When the option is active the fragment will be automatically updated after applying changes in its variables.

Positioning of 3D Placement Objects

Single "LCS–Planar face" attachments are specified by parameters of the connector in the3D fragment document. Double attachments are formed by the procedure of insertion of 3D placement object from all possible pairs of single attachments satisfying the condition: double attachments can only be WALL–FLOOR, WALL–TABLE and CEILING-WALL.

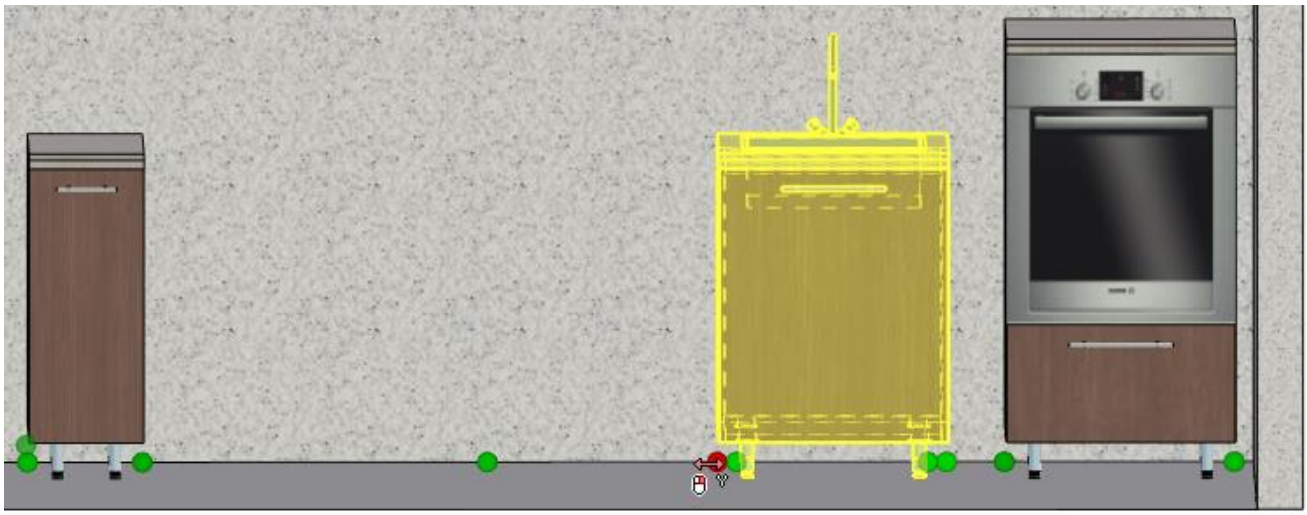
When object is positioned by a single "LCS–Planar face" attachment, it is required to point the cursor of the mouse at the associated face of the target fastening of the desired type. In this case, the contour of the associated face of the target fastening will be highlighted.



It is required to point the cursor at the associated face of the target fastening of the WALL type when positioning an object based on the double attachment,. If the face was found, its contour will be highlighted.

> The presence of the FLOOR or TABLE target fastening is determined by the positioning procedure. If the second attachment is absent, the object will not be positioned based on the double attachment.

Attachments by "LCS–LCS" rule are displayed in the scene only when the attachments by the "LCS–Planar face" rule are not used. In this case the **Without attachment** option is selected for the inserted fragment in the **Attachments** list. All possible target fastenings by the "LCS–LCS" rule are displayed in the scene with the green sphere. Upon selection of one of these spheres the connection between the initial and target fastenings is established, and the selected target fastening will be highlighted with red color.

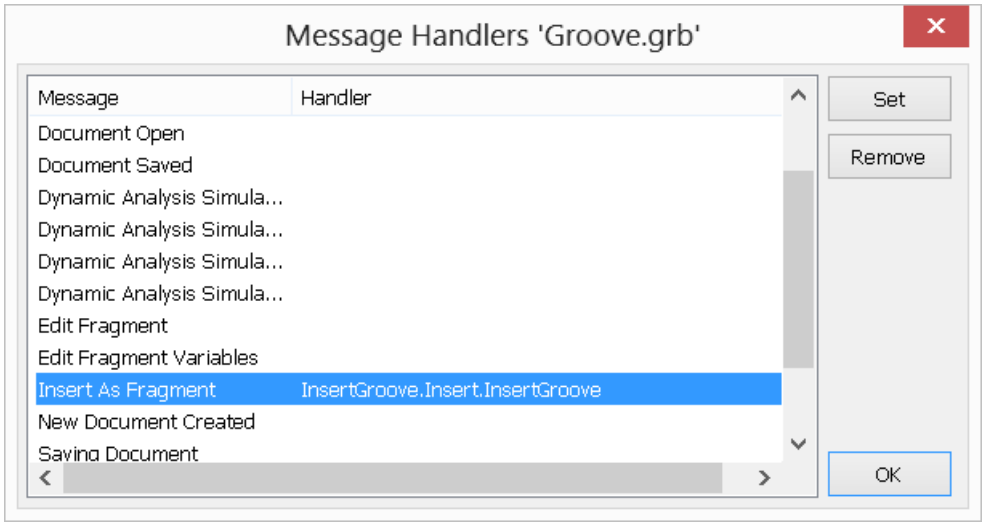


Attachments by the "LCS–LCS" rule can be used indirectly, when attaching a fragment by the "LCS–Planar face" rule. When you move the fragment in 3D scene, system can automatically "grab" the "LCS–LCS" attachment.

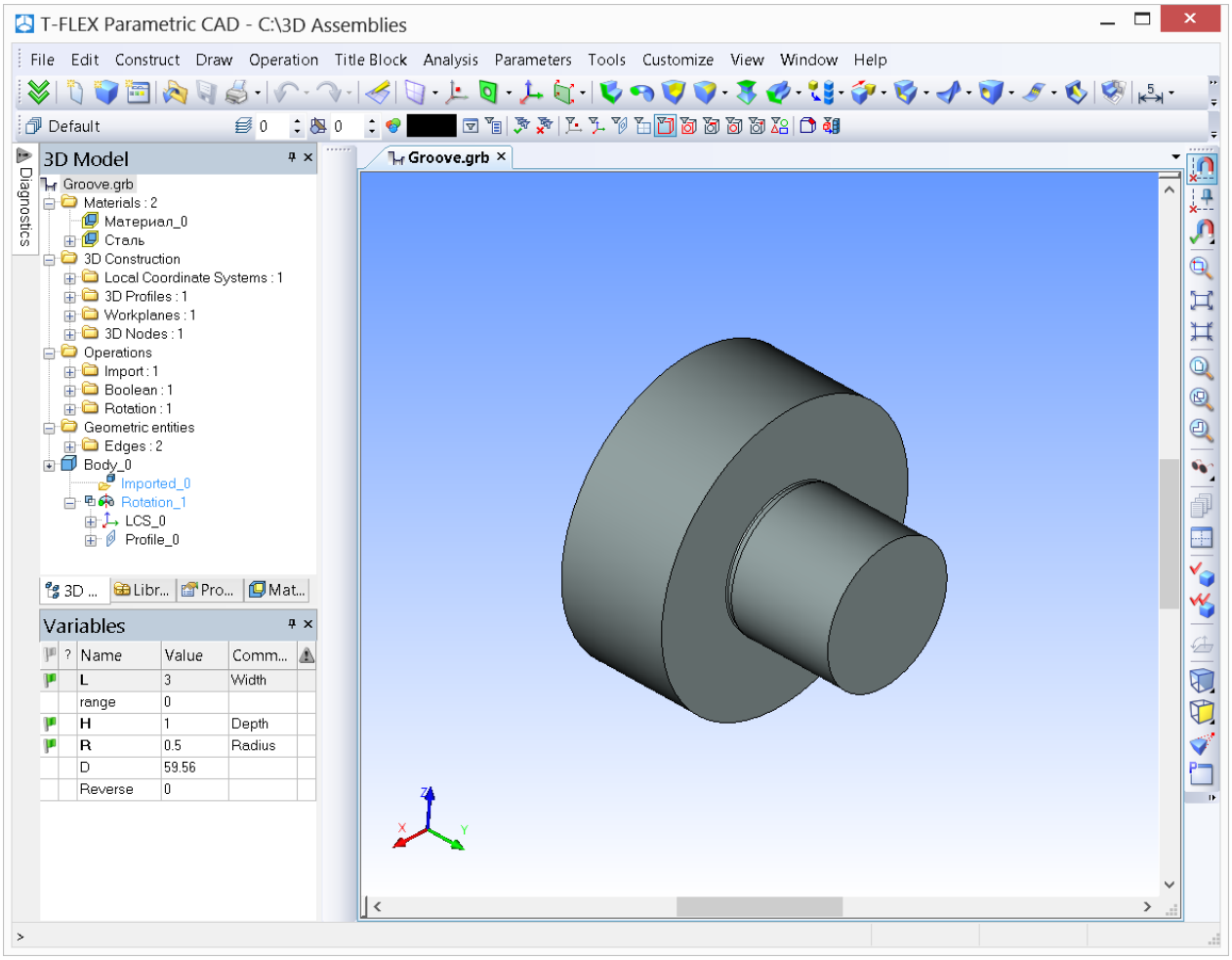
When the fragment with connector value TARGET_<Type> is moved, the related fragment will be moved too, if attachment "LCS" – "LCS" is set between them.

MECHANISM «SMART FRAGMENT»

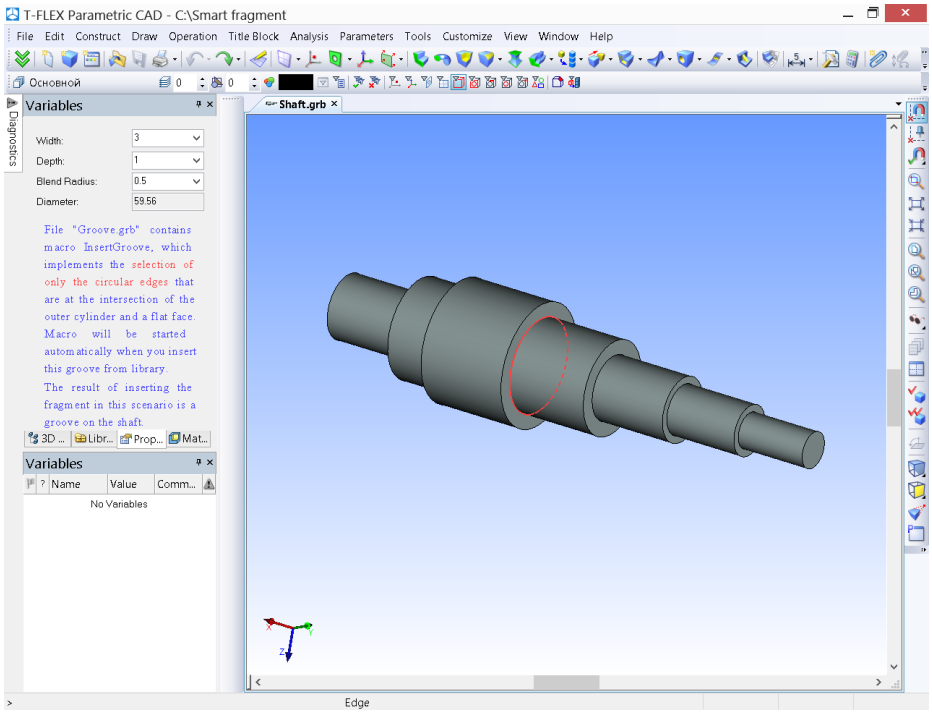
The Smart Fragment mechanism allows us to develop parametric library elements whose insertion scenario is described in the program stored directly in the file of the given library element or in the external module (DLL). Settings of T-FLEX CAD macros and also the program interface (Open API) allow us to describe settings in such a way that upon insertion of the file as a fragment or upon its editing/deletion the user-specified macro will be executed.



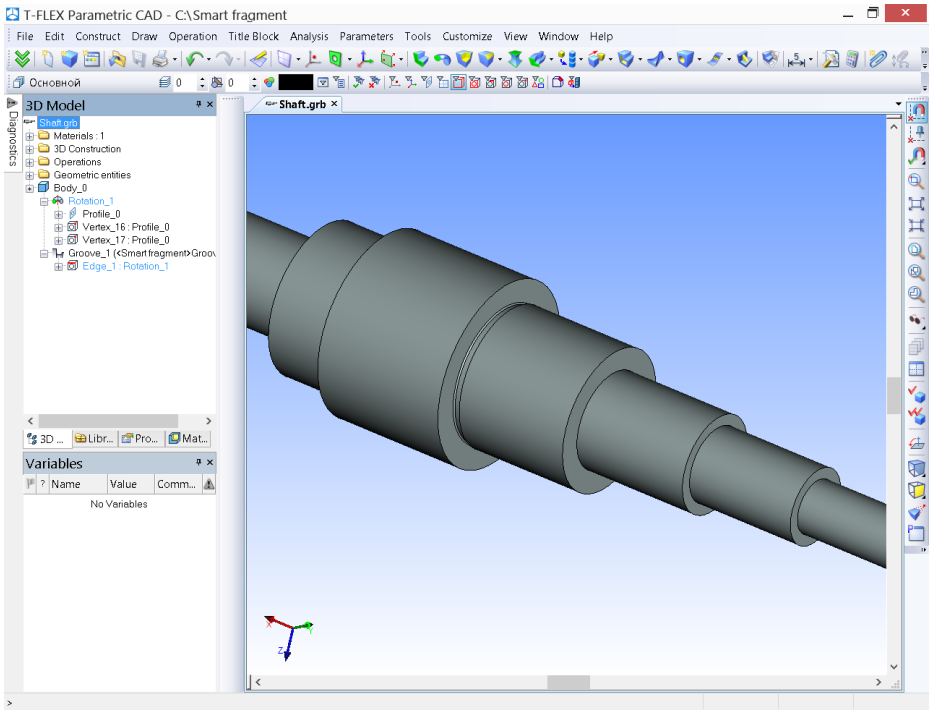
The macro can contain code realizing the required logic of the user's interface on insertion/editing/deletion of the given fragment. For example, the adaptive model «Small groove» was developed that has a set of external variables and adaptive parameter – circular edge which must be substituted upon insertion of the fragment.



The «Groove.grb» file contains the macro InsertGroove which performs selection of exclusively circular edges located at the intersection of the external cylinder and the planar face. Upon insertion of the given groove from the library the macro is launched automatically. Instead of regular command of insertion of a 3D fragment with the full interface, the simplest command is working at this moment that allows us to select the edges satisfying the given condition and to change the values of external variables of the inserted fragment.



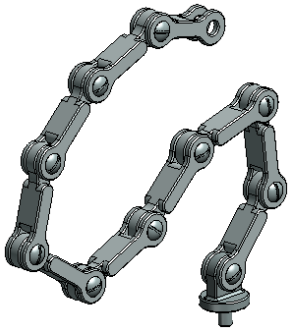
The result of insertion of the fragment by this scenario is the small groove on the shaft.

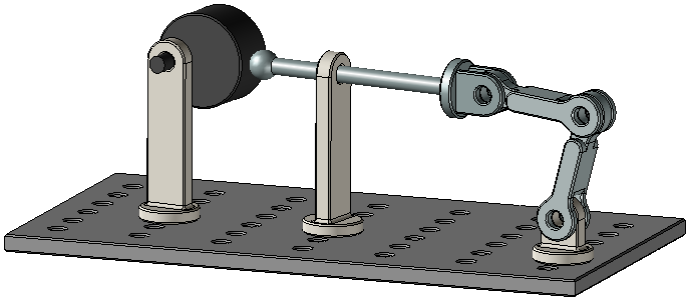


This mechanism allows users-programmers to create the most convenient scenarios of fragments insertion in various applied fields.

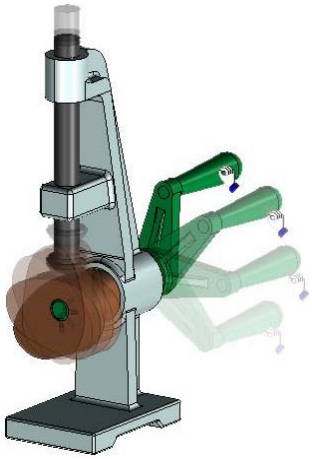
MATES AND DEGREES OF FREEDOM

In the process of designing solid assembly models in T-FLEX CAD, it is often necessary to specify mutual position of parts, which may not always be accomplished by the conventional technique of fixing by LCS adopted at T-FLEX CAD. This is due to the fact that changes in part positions may have to be reciprocal. A given part can simultaneously have multiple contacts with other movable parts. When using the approach based on LCS, recursive dependencies of references may occur in such cases (a part's dependency on itself). Such tasks may occur, for example, when building models of mechanical systems (mechanisms). To handle such tasks, mates are introduced which are the system elements that allow imposing various dependencies on the geometrical objects of two component operations (3D points, axes, curves, planes and surfaces).





A model of a mechanism designed with the use of mates can be animated in the special command, where you move its parts by the mouse.



To fix a 3D fragment using LCS, you can specify additional conditions in the way of the permitted degrees of freedom. If such a 3D fragment is later attached via an LCS to another constrained element, then it will be included in calculations of the constrained mechanism. The method of positioning the selected parts by using mates is to be used as an auxiliary way of fixing together with the use of fixing by local coordinate systems. The use of mates requires additional computational resources for evaluating

solutions. Therefore, we recommend using mates in the cases when other approaches to the described tasks are not suitable for some reasons.

Main Concepts

What is a Mate?

The "Mates" tool serves for mutual attachment of assembly model elements. It allows positioning those according to the specified geometrical conditions. These conditions define the mutual situation of the three-dimensional model objects (such as faces, edges, vertices, characteristic points, axes of surfaces of revolution, etc.) with respect to each other. The system automatically resolves the specified set of mates and situates the objects in the way satisfying the specified conditions. Mates allow precisely positioning the parts of the mechanism being designed with respect to each other. Those allow to impose certain properties in a mechanism's model and determine how its components move and turn with respect to other parts. Combinations of different mates can be used for more precise definition of the restrictions on an assembly element's position with respect to another element. Relations between pairs of components are associative. If one part is moved then another part will move along. For example, if a screw is tied to a hole, then relocation of the hole will cause the screw to follow along.

Mates are imposed on pairs of geometrical objects. Those either connect two components together or fix a body to its environment (with respect to a fixed object). A fixed object is an object with all its degrees of freedom restrained or a one whose position in the space remains still. It is recommended that at least one component be fixed in the space. This creates a "ground" for all other constrained parts and can prevent an unexpected dislocation of a mechanism's components.

Each mate appears as a model object having its place in the model structure. As a fully functional system object, a mate has a name, working properties and is displayed in the 3D model structure. The user can turn off some of the mates in order to temporarily exclude those from the general solution. This allows experimenting with various types of mates without redefining the mechanism interdependencies.

Types of Mates

When defining geometrical ties in the form of mates, you can use only the following geometrical terms: 3D point, axis, curve, plane, surface. In order to specify the geometrical data, the user selects topological elements of the model or construction elements. Use of 3D construction elements is possible under the condition that those are constructed based on an operation body. Otherwise, the second component will turn out to be bound to the still environment.

> Upon fixing by construction elements, you lose the capability of dynamical articulation of the given connection in the constrained mechanism; therefore, if possible, prefer to use the model's topological elements (vertices, edges, faces, etc.)

When creating mates, make sure to understand the difference between the geometrical data used by mates and topological objects used for selection. For example, a flat face has boundaries, while the plane

used in the mate is infinite. Therefore, the created geometrical relation based on a flat face will still be valid beyond the boundaries of this face.

Coincidence

This type of mate insures full coincidence of one geometrical object with another one. The number of remaining degrees of freedom depends on the geometrical type of the objects being joined. For example, aligning two points will create a spherical joint that blocks any translation while allowing any rotation. Aligning a point and a curve will allow elements to rotate, and also to slide along the curve. The following table shows the possible combinations of geometrical elements when using the coincidence:

Coincidence	Point	Axis	Curve (except circle)	Circle	Surface (except conical surface)	Conical surface	Plane	Wire body	Sheet body	Solid body
Point	+	+	+	+	+	+	+	+	+	+
Axis	+	-	-	-	-	-	+	-	-	-
Plane	+	+	-	-	-	-	+	-	-	-
Circle	-	-	-	-	-	+	-	-	-	-
Conical surface	-	-	-	+	-	-	-	-	-	-

A point can be aligned to any geometrical object. If two flat faces are selected, those will lie in one plane. Coincidence of an axis and a plane will insure the axis' placement on the plane of a flat face. The condition of coincidence between a point and a solid insures the point's placement being maintained on the solid's surface.

Upon selecting a circle and a conical surface, matching of the selected surface and the circle is realized in such a way that the plane of the circle is perpendicular to the axis of the cone.

Using the coincidence in combination with other types of mates allows imitating various mechanical connections. For example, coincidence of a pair of 3D points provides for a spherical joint.

Parallelism, Perpendicularity

These types of mates represent a special case of the "angle" mate type. These insure mutual parallelism and perpendicularity of the selected geometrical objects. The following table presents the possible combinations of geometrical elements when creating parallelism or perpendicularity mates:

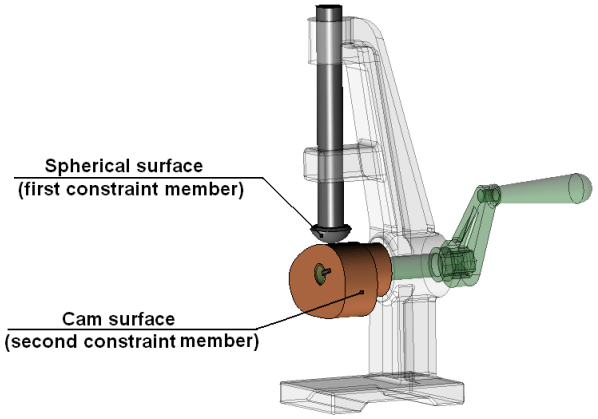
Parallelism, Perpendicularity	Point	Axis	Curve	Surface	Plane	Wire body	Sheet body	Solid body
Axis	-	+	-	-	-	-	-	-
Plane	-	+	-	-	+	-	-	-

Tangency

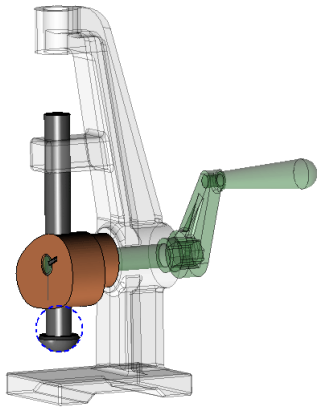
Tangency insures a constant physical contact between two geometrical objects. Depending on the geometry type of the interacting objects, a contact may be occurring in one point (example: a plane and a sphere) or along a line (a plane and a cylinder). This type of mate can be set between two planes, a plane and a surface, two surfaces. Tangency between an arbitrary body and a surface works only for a spherical surface.

Tangency	Point	Axis	Curve	Surface	Plane	Wire body	Sheet body	Solid body
Surface	-	-	-	+	+	+	+	+
Plane	-	-	-	+	+	+	+	+

Sometimes, there are several possible solutions to connecting the selected objects. For example, if a plane and a sphere are selected, then the solution yields two tangency points. In the case of a non-unique solution, the system makes a decision automatically. Normally, the nearest point to the current location of the mechanism is accepted as the solution. To transit to another solution domain attainable by the moving mechanism, use the "dragging" motion mode (see below).



First solution for tangency



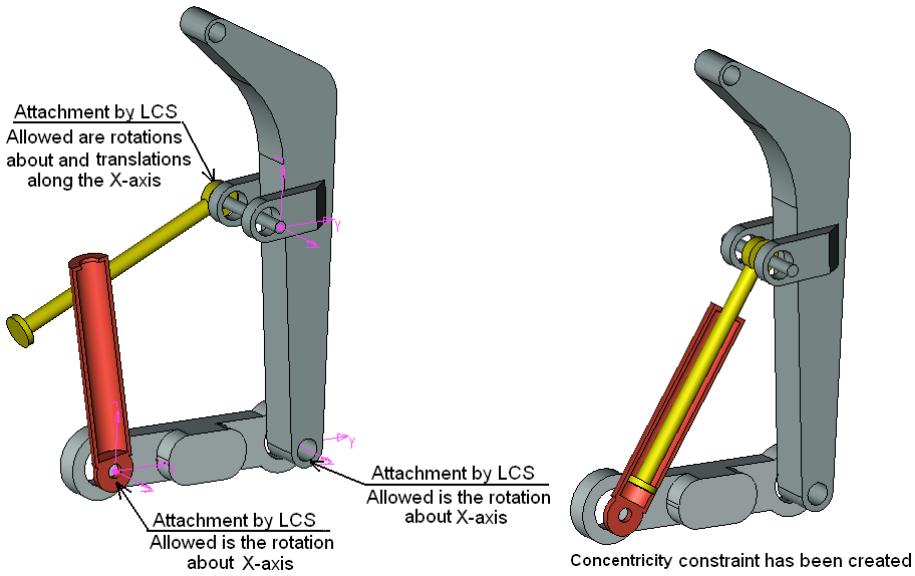
Second solution for tangency

In the case of a non-unique solution, it is recommended to bring the components being constrained into the area of the desired result before creating the mate. Parts of a constrained mechanism can be moved within the command "Move Mated Components". The components that were not yet subjected to mates, can be moved by using the command "Transformation".

Concentricity

"Concentricity" is a special case of the "coincidence" mate type. It insures coincidence of two axes. This type of mate is a most commonly used one. Normally, this mate type is engaged in combination with other types. For example, to snap a part with an axis to a hole, coincidence (of flat end faces) is used quite often in combination with concentricity. To define concentricity, one can select model elements suitable for defining an axis (surfaces of revolution, elliptical edges, straight edges, etc.).

The diagrams show constraining parts of a schematic hydraulic cylinder by concentricity. Other parts of the simple mechanism arc connected by local coordinate systems. For viewing convenience, some parts of the mechanism are sectioned along the axis.



Distance

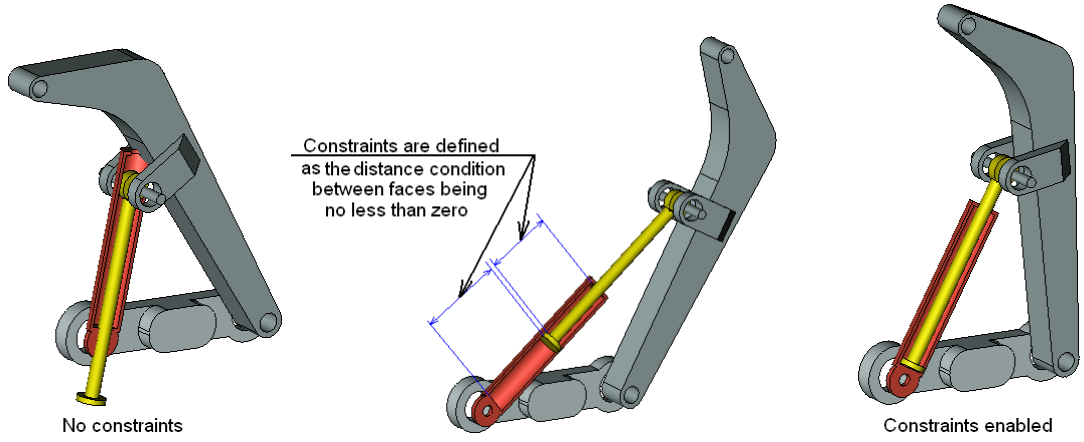
The "distance" mate defines a relation between two objects by fulfilling the condition of maintaining a specified distance between two geometrical objects. One can specify the conditions of types "no greater than", "no less than" or "equal" to the specified value.

The following table presents the possible combinations of geometrical elements when creating the "distance" mate:

Distance	Point	Axis	Curve	Surface	Plane	Wire body	Sheet body	Solid body
Point	+	+	+	+	+	+	+	+
Axis	+	+	-	-	+	-	-	-
Surface	+	-	-	+	-	-	-	-
Plane	+	+	-	+	+	-	-	-

This type of mate can be used, for example, for defining an object's shift or for restricting mutual penetration of faces. To define a restriction on mutual penetration of faces of different parts, you just need to specify the distance condition "no less than" zero. This can be used for mechanisms whose operation relies on physical contact (collision) of its parts.

Continuing with the hydraulic cylinder example, let us show how to use the "distance" mate. The mechanism created at the previous step will operate incorrectly in certain motions, because some parts of the hydraulic cylinder may penetrate each other. This problem is resolved by the "distance" mate. One can specify the condition that the distance between the respective faces of the piston and the cylinder must be greater than or equal to zero. When mates are dragged kinematically, this will yield the correct representation of the mechanism's behavior.



Angle

This mate defines angular distance between two geometrical objects. The turn angle can be used for rotating a component referencing the desired position. Just as in the "distance" case, this mate type supports use of conditions. When defining such connection, normally there are two or more solutions.

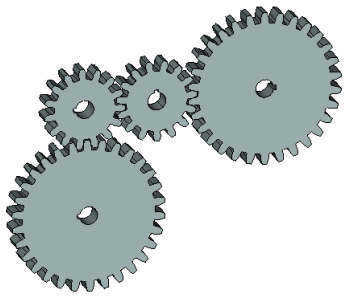
The following table shows the possible combinations of geometrical elements when creating the "angle" mate:

Angle	Point	Axis	Curve	Surface	Plane	Wire body	Sheet body	Solid body
Axis	-	+	-	-	+	-	-	-
Plane	-	+	-	-	+	-	-	-

«Gear-Gear» Transmission

This type of mate serves for creating conditions for the interaction of two bodies spinning about their respective axes. The conditions are specified as a transmission ratio between two spinning objects, that is, the number of revolutions made by one of the components while turning the other one. Upon introducing such a mate, a two-sided relation is formed. This type of mate is convenient for monitoring mechanisms that incorporate dented wheel or belt drives.

> Remember that this type of transmission relation, just like any other transmission mate, merely tells the system the mutual ratio of angular (in this context) velocities. Also note that each component must be fully fixed in its place by the time of creating a transmission mate and be assigned all required degrees of freedom and constraints.



To create such a mate, select two axes of revolution and specify the transmission ratio. In this case, the shape of the dented wheel is not significant for the system, as the wheel dents can be created for «cosmetic» purpose only. The law of the mechanism's movement will be defined solely by the mate conditions.

«Gear-Rack» Transmission

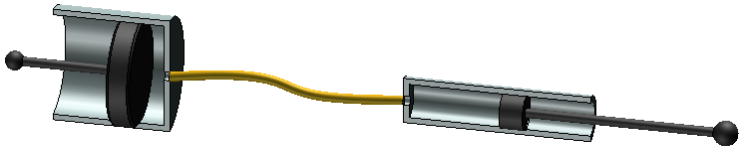
This type of mate serves for creating the conditions for two interacting bodies, in which one component spins, while the other one moves about one or multiple axes. This type of mate allows modeling, for instance, a nut being picked up by a screw, a transmission link of the type «pinion-rack», etc. The mate conditions are defined as the amount of the distance traveled by the movable component along the specified axis while the other component makes one full turn. A two-directional relation results from creating this kind of mate.

To create a mate, you need to define the axis for each of the two components and specify the distance in the model units that one component would travel while the other component rotates by 360 degrees. The first one defines the «pinion» axis, the second one - the «rack» axis.

«Linear Movement» Transmission

If there are two objects in the model moving in their respective axial directions, then you can specify a ratio using the mate being described, that would characterize the mutual displacement of the two objects. With such a relation established, the travel distance of the second component will be calculated by multiplying the first component's travel by the specified ratio. The relation can work in the reverse

direction as well (from the second component to the first one); in this case, the system will use the ratio which is the reciprocal of the specified one. This type of mate helps modeling the mechanisms representing systems of pulleys, hydraulic mechanisms, etc.

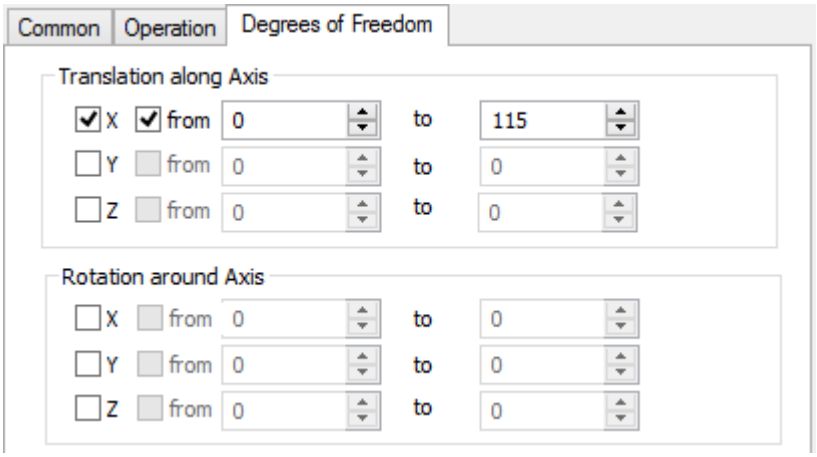


To create a mate, you need to define the axes for each of the two components, and enter the value of the velocities ratio of the moving components in the provided input box.

Attaching 3D Fragment by LCS

A 3D fragment inserted in an assembly using local coordinate systems can also be involved in a 3D model built on mates. To define a 3D fragment's behavior with respect to the fixing LCS, the permitted degrees of freedom are specified in the fragment's parameters. There are total of 6 DOF's – distributed as 3 for translations and as many for rotations with respect to LCS axes. An additional limit can be set for each DOF as a range of values.

One can also specify the degrees of freedom in the parameters of the source LCS in advance, in the fragment's file. Upon inserting a 3D fragment, the conditions are copied from LCS parameters to the parameters of the 3D fragment together with the degrees of freedom and are later accounted for when calculating the mates complex of the developed 3D assembly model.



If the target LCS was not specified at the time of creating a 3D fragment, making the fragment attached "by default", the degrees of freedom management mechanism will not be applicable to such fragment.

When designing complete assemblies of mechanisms using the degrees of freedom and mates, it is important to account for the following point. A 3D fragment being inserted in an assembly by fixing to an LCS may or may not be related to some assembly component. In the latter case, the component behavior may not agree with the expectancy as other assembly components are moved. For example, it

may stay still while its surrounding parts move, or, vice versa, moving it may not induce motion in the neighboring parts.

To have a 3D fragment be related with another component, its target fixing LCS must be built on geometrical objects of that component. Otherwise, it will be attached to the still environment.

The so-called "Aggregate" mechanism is introduced in the system for creating complex assemblies. This mechanism allows creating mates in the model of the 3D fragment first, so that those are automatically accounted for when such a 3D fragment acts in an assembly.

CREATING MATES

To create a mate, use the command:

Icon	Ribbon
	Assembly → Mates → New
Keyboard	Textual Menu
<3CT>	Tools > Mate > Create Mate

To create a new mate, complete the following steps:

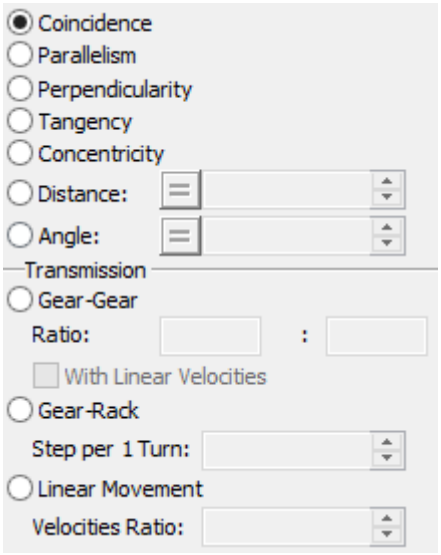
1. Select the mate type.
2. Select the first object.
3. Select the second object.
4. Define parameters of the selected mate type (if necessary).
5. Confirm the selections.

Selecting Mate Type

A mate type selection is done in the properties window. To make a selection, set the radio switch into the desired state.

For "Distance" and "Angle" mates, one can define a condition with respect to the specified value. Pressing the button before the numerical value cyclically switches its condition sign.

For transmission mates, the numerical parameters are defined in the respective input boxes.



Selecting Mate Elements

The modes of selecting the first and second geometrical objects are engaged by the following options:

	<F>	Select first element
	<S>	Select second element

Upon selecting the first element, the system automatically turns on the option for selecting the second element. If it is necessary to return to defining the first element, turn on the respective mode manually.

Depending on the type of the mate being created, different geometrical objects can be selected. Tuning the selection to the desired type of geometrical data is done by the automenu options:

	<V>	Select point
	<A>	Select axis
	<C>	Select curve
	<U>	Select surface
	<L>	Select plane
	<W>	Select wire body
	<H>	Select sheet body
	<B>	Select solid body

For some mates, a number of options may be unavailable or temporarily inaccessible (see the tables of combinations above). Each option turns on a set of filters for selecting objects suitable for defining the required geometrical element.

Selected objects are highlighted. Also highlighted are the operations whose geometrical elements are selected. Simultaneously, the system offers a preliminary choice of solving the mate. According with this, the system moves the elements as to fulfill the mate conditions. If one or both elements already have mates with other bodies or fixed still to the environment, then the new solution is calculated by accounting for all earlier imposed restrictions.

Fixing Mate Member

When creating a mate, one of the two mate members can be fixed in the space. Fixing is done by pressing the button marked by an anchor image, in the properties window.

Upon fixing, the object becomes "frozen" in the 3D space in its current position. It will neither be affected by any new transformations nor by changes in parameters of existing translations. The "*" mark will appear before the name of the fixed operation in the model tree.

Repressing button cancels fixing of the mate component.

It is also possible to fix a body in space outside the command of mate creation. Use **Fix Component** from the context menu of any body (icon) if the current 3D assembly contains any mates or uses degrees of freedom for 3D fragments.

The command **Fix Component** is always available in the context menu of the fixed component. The icon corresponding to this command is pressed if a component is fixed. Applying this command will disable fixing in this case.

Selecting Elements for Defining Transmission Ratios

You need to specify numerical parameters (transmission ratios) when defining transmission mates. An additional capability is provided in the cases of «Gear (angular speeds)» or «Worm (pinion-rack)», which is using geometry of the mated elements to automatically determine the numerical transmission ratio. The options for defining the transmission ratio appear automatically in an additional section of the automenu when selecting the appropriate mate type.

The transmission ratio for the «gear» transmission is calculated as the ratio of the two selected elements' radii. The following options are used in the automenu:

	<G>	Select first element to define transmission ratio

	<D>	Select second element to define transmission ratio
	<Z>	Cancel transmission ratio selection

In the case of the «Worm» mate, the transmission ratio is calculated based on the connecting thread parameters. The following options are used in the automenu:

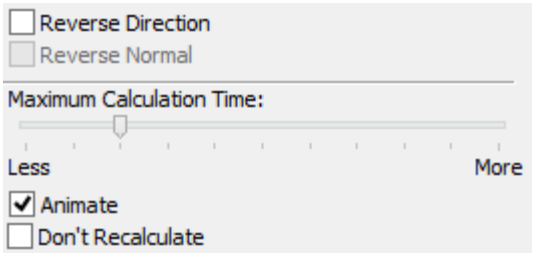
	<T>	Select threaded face to define transmission ratio
	<R>	Select edge or face to define transmission ratio
	<Z>	Cancel transmission ratio selection

Additional Mate Parameters

Reverse Direction. This option rotates the second (or the remaining movable) component being connected by a 180 degrees or moves it into another solution domain. For the transmission mates, this parameter alters the direction of the second component motion.

Reverse Normal. This option works only with the "distance"-type mate between two planes. In addition to the previous option, it allows changing the direction of counting the distance.

Maximum Calculation Time is the parameter controlling the number of passes the system is allowed to use in search for the solution with the specified precision. This limitation is necessary to avoid too long system computations due to an accidental user error, that are possible when searching for solutions in complicated cases. This parameter is set by a slider.



If no solution was found after the current number of iterations, try to increase the allotment. On the other hand, it could be sufficient instead to manually move the components being connected toward the probable solution domain in the command "Move Coinstraint Operations" (see below).

Animate – this parameter lets the system gradually relocate the elements being constrained in order to examine the mate solution behavior.

Don't Recalculate – this parameter suppresses mates calculation at the time of creation. This capability can be used in order to save time when working with large assemblies. This could be useful, for instance, when the user adds several mates one after another. To search for a solution, you will need to do full model recalculation. In this case, note that upon adding several complex mates or simply a too large number of mates, the system might not finally find the correct solution (see the topic «Processing possible erroneous cases while working with mates» below).

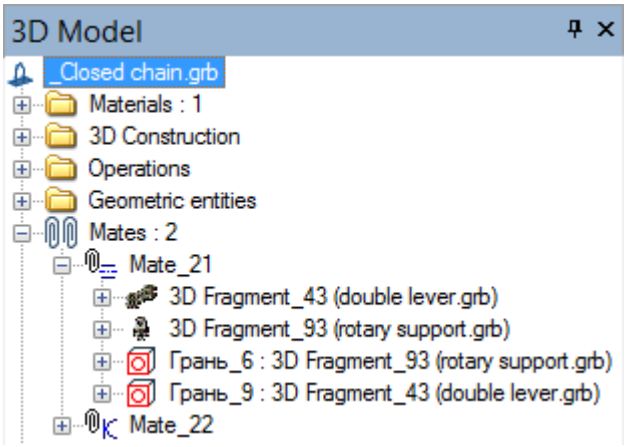
Techniques of Handling Existing Mates

Working with the Model Tree

The created mates are displayed in the "3D model" window. Most of manipulations with existing mates are done via this tool window. A special folder is created in the model tree, in which the system automatically puts all mates. An icon is provided for each mate type that corresponds to the type.

Each mate is preceded by the mark ⊞. Clicking it by will show the objects in the expanding branch of the model tree, on which the mate is based.

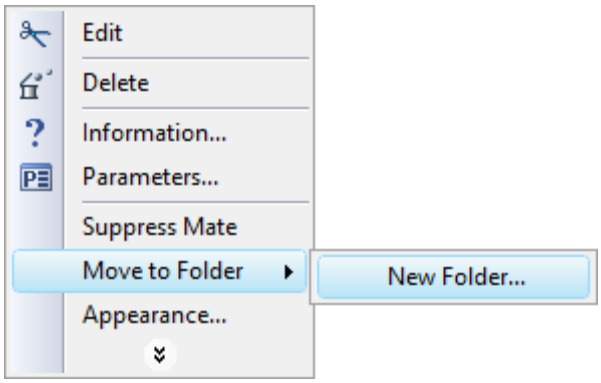
Mates can be additionally grouped in subfolders.



However, only one subfolders nesting level is supported. All mate subfolders are marked with the icon . Each additional folder has a unique name. Grouping by additional folders doesn't affect anything. This is done merely for working convenience when sorting out a large number of mates.

To move a mate to another folder, use the context menu item "Move to folder". The new submenu makes available the commands for creating a new additional mates folder and moving to the main and already existing additional folders.

The rest of commands in this context menu are common across various system elements ("Parameters", "Edit", "Delete", "Information...").



Suppressing Mates

Any mate can be suppressed. The command for suppressing a mate and canceling the suppression is accessed in the context menu by right clicking a mate. It is also possible to impose suppression on a group of mates at once by accessing the command from the context menu on right clicking the mates folder. A suppressed mate is marked with a blue cross in the model tree and is excluded from calculating mates.

Mechanism Motion

To dynamically view the motion of a composed mechanism, use the command:

Icon	Ribbon
	Assembly → Mates → Move
Keyboard	Textual Menu
<3CM>	Tools > Mate > Move Mated Components

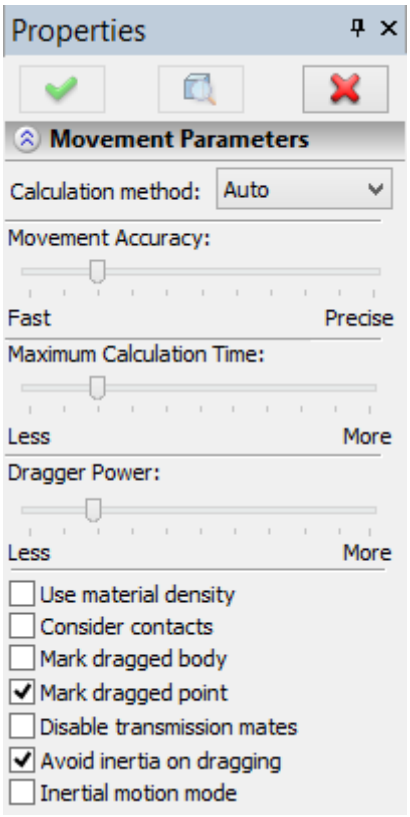
Upon calling the command, you are required to select the desired part of the mechanism for defining motion. Motion is defined by dragging the selected part by the mouse. In this way, the user stimulates an influence on the mechanism as if a force is applied to the selected point on the part in the direction of the mouse pointer in the screen plane. The part of the mechanism starts moving under the influence of the force. The motion is restricted by the specified mates with other parts and the outer environment. The selected part pulls another one, and so on, until the whole mechanism is involved in motion. In this simulation, masses and moments of inertia of the moved components are correctly accounted for.

Any translation and rotation parameters of mated (constrained) objects are recorded in the properties of each mate operation. A special transformation type, namely, “Mating transformation”, is created within the operation’s parameters. All quantifiable properties of this transformation are automatically calculated by the system.

Several important considerations shall be kept in mind when moving mated elements:

1. Exact numerical parameters cannot be specified for a translation.
2. Upon completing the translation command with , the parts cannot be brought back in the exact previous state. If urged, this can be done only by undoing the steps in the current session.
3. When moving the model of the mechanism for which an assembly drawing was furnished, there is a danger of irreversible changes to the original drawing created by projecting. This could happen after refreshing the projection, due to the new positioning of the assembly parts and complications on the way of the exact rollback to the previous state of part positions.

The setup of the motion modes is done in the properties window.



Number of Elements: Maximal/Minimal. When moving constrained elements, it is possible to involve all parts of the composed mechanism or just the minimal chain of mates.

"Calculation method". Sets one of two available methods of calculation – precise or fast. The **precise** method is used for systems consisting up to several tens of joints. The higher the complexity of system, the slower it will run the calculation. For systems including hundreds of joints this calculation may work unsatisfactory slow. In this case you need to use the **fast** method that works with both large and small systems. The **auto** mode allows system to choose calculation method itself, according to the complexity of the system.

The mode of the kinematic dragger is used for moving parts in a mechanism. **This results in smooth motion**. It is a simulation of the effect of applying a force at the point at which the user pulls the part. The motion is generated by accounting for all specified restrictions. As the boundary of the solution domain is reached, the mechanism stops.

Movement Accuracy. This allows reducing the level of calculation precision for faster calculations and for maintaining the constraining conditions during a mechanism's motion.

Maximum calculation time. Sets the number of computational passes allowed to the system for reaching the required computational accuracy during the motion.

Dragger power. This setting adjusts the degree of the mouse impact on the dragging point of the body. This setting controls the applied force and, therefore, how fast the dragged parts will move after the mouse.

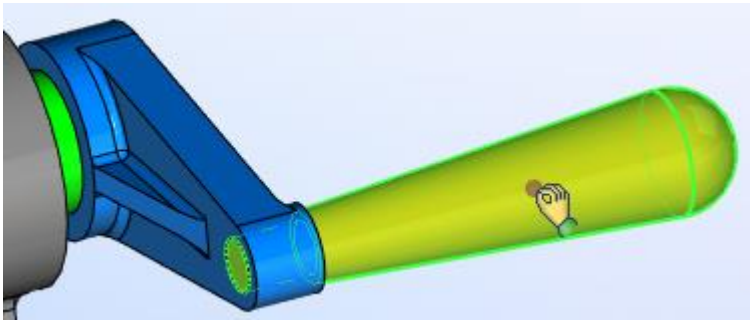
Use material density. For more precise simulation of mechanism motion, a provision is made for accounting for the actual density when calculating mass properties of its parts. Otherwise, the density is assumed the same for all components.

Consider contacts. To improve accuracy of movement simulation there is an opportunity to consider the intersection of objects.

Mark dragged Body. This is an auxiliary parameter for switching On/Off highlighting of the mechanism part being pulled at the user for inducing the motion.

Mark dragged Point. This parameter turns on highlighting of the point which is pulled by the mouse pointer. The point is marked by a cross. This functionality can be used when the body highlight is off.
Disable transmission mates. This option allows disabling transmission mates. Such a capability is convenient for bringing transmission members to the starting mutual configuration (for example, the correct interlocking of wheel dents).
Avoid inertia of dragging. This parameter allows to switch on/off the consideration of the body inertia.
Inertial motion mode. This parameter allows to consider the inertia without its fading over time.

There is a simplified way to change 3D element position in space within the specified degrees of freedom. For activation of this opportunity you need to select the body in the scene using the filter . A small circular manipulator will appear in a second after selection. This manipulator is used for moving the element.

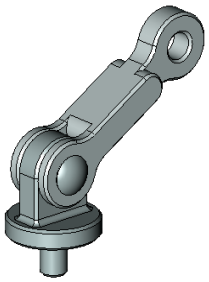


Optimum Length of Chain of Mates

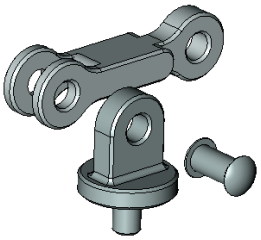
As a mechanism is being created, one part gets "hooked up" to the outer environment, while other parts are linked to each other via mates. In this way, a number of parts subsequently linked to each other can be viewed as a chain of mates. If several parts all connect to one part, the chain of mates forks. When creating mechanisms, pursue the shortest chains possible. Shorter chains are solved faster, and fewer computational errors occur on the way.

Let's review the following typical example – suppose, we need to create a model of a simple mechanism shown on the diagram. The mechanism represents a joint-type connection, joining together three parts – a support, a lever and an axis. The most rational approach in this case will be creating two short chains of mates, rather than one long.

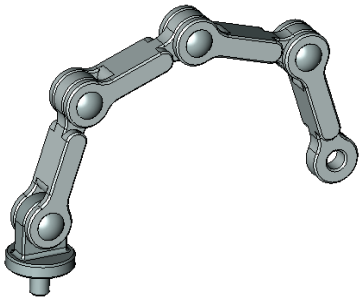
The lever needs to be attached to the support first, using concentricity and coincident flat faces mates. The axis can be attached to any of the parts as well, such as the support, by concentricity and coincidence. This results in two chains of mates: support-lever and support-axis. In this case, it would not be rational to attach, for instance, the axis to the support, and then the lever to the axis, since that would result in a longer chain of mates "support-axis-lever".



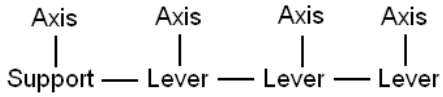
Sample mechanism



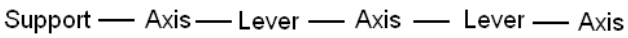
Parts of mechanism



Advancing mechanism (adding new mates)



Rational chain



Undesirable chain

Naturally, this simple example will not exhibit any noticeable slowdown in the system performance; however, the difference becomes substantial in more complex cases.

Recommendations on Combined Use of Fixing by LCS and Mates

In this section, we review the recommended rules of combined use of mates and fixing by LCS when creating assemblies. Fixing parts by mates shall mainly be used together with fixing by LCS. Note also that priority shall be given to the traditional way – fixing fragments by coordinate systems. This approach has a number of advantages over mates.

Firstly, some types of geometrical relations, including the most commonly used (as a single-axis joint), can be solved by one LCS, while in the mate approach it may require bundling two or more mates. In particular, a single-axis joint is defined by two mates (concentricity + coincidence of end faces), or just one LCS (with rotation allowed about one axis). This main advantage has more consequences:

- Overall, the system works faster (the chain of mates becomes shorter, computational errors are reduced, calculations become simpler).
- A simpler model structure results from avoiding extra elements (the mates).
- The model becomes more robust to topological changes (due to minimized use of topological body elements).

So then, when the use of mates is justified? To answer this question, let's review the process of designing an assembly. In regard to the posed question, the designer typically faces 2 tasks.

Task 1: creating an assembly model without a provision for a "movable" mechanism. The user knows in advance what he is developing, and that his model will not have to move upon assembling, that's unnecessary. For example, if we are designing a TV set or a house, it is unlikely that we will need to "animate" those dynamically.

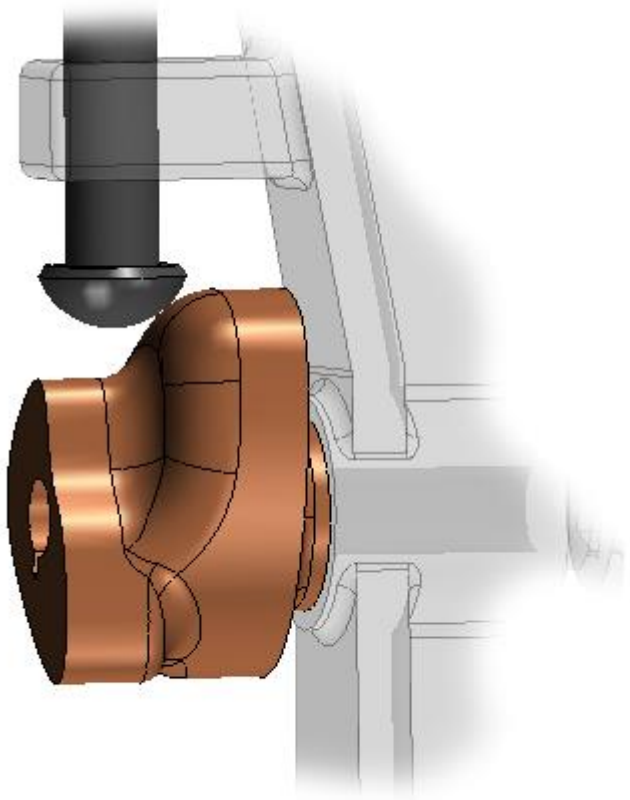
When the user faces such a requirement, then the single main concern of assembling parts is – how to correctly position a part within an assembly. The most effective solution is achieved by the "traditional" approach – inserting 3D fragments using LCS. In addition to overall advantages of this approach, fewer actions are required from the user. A 3D fragment insertion technique takes less time because fewer elements need to be defined, specifically:

1. The source LCS (the selection can be automatic).
2. The fixing position (selection of the target LCS or a geometry unit for its creation is done in the 3D window).

Meanwhile, each new mates would require defining geometrical data for two components, while exact positioning of a component usually requires creation of several mates.

Resort to the use of mates:

- in complex cases of geometrical interactions;



- when assembling a part that is in contact with other parts at several locations.

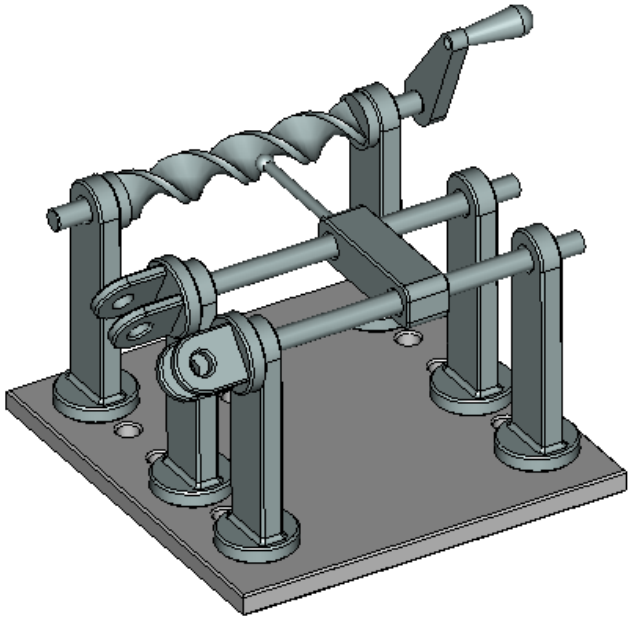
Task 2: building an assembly with a provision for "animating" the developed mechanism in the command for moving constrained elements. In this case, all parts of the assembled mechanism ought to interact correctly with each other. All movable parts shall necessarily have descriptions of the required conditions for calculating the mates arrangement while the mechanism is in motion.

In such a case, we recommend holding on to the rules described below:

- For the parts that are not supposed to change their position with respect to the elements they are attached to, we recommend using LCS with no degrees of freedom. For example, a nut fastened to a bolt shall always be with the bolt and shall not move with respect to the bolt.
- For the parts with varying mutual situation, first use LCS providing degrees of freedom. This will benefit in all main aspects (see above).

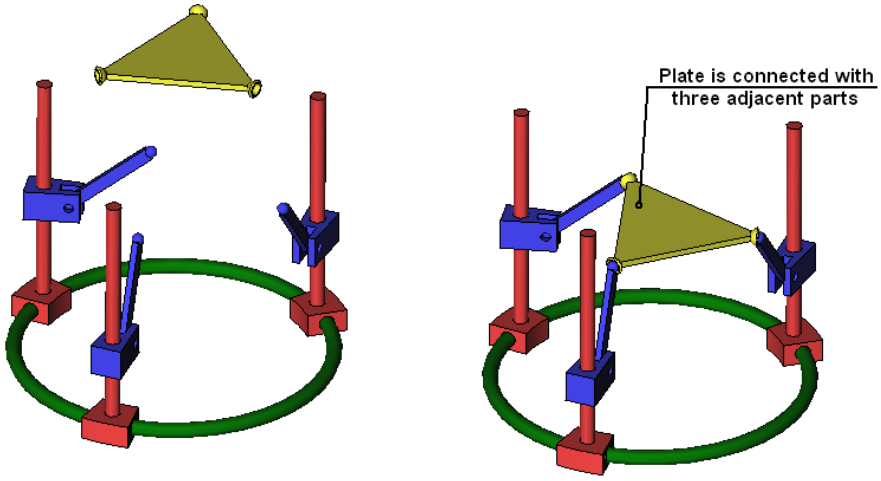
Mates are used for fixing parts in the following cases:

- For parts with complex geometrical associations.



Example of complex interaction (helical surface and pushing rod)

- For parts having more than one contact with several movable adjacent parts.
- For terminating chains of mates. In this way, each chain can be created using an LCS with degrees of freedom.



Three chains of constraints are terminated

- For defining various restrictions, such as a restriction for mutual part penetration. For an example, refer to the hydraulic cylinder where the restriction is specified in the form of the condition “distance=0” (see the topic “Types of mates” above)

Processing Possible Erroneous Cases while Working with Mates

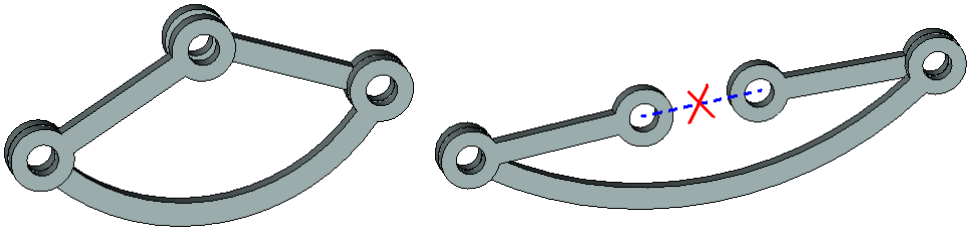
Errors in mates might occur at various stages of handling the model. The system may not always output the detailed reason for such failures; therefore, we will list here the most common erroneous situations

and typical techniques of correcting those. All erroneous situations can be divided into two types – errors occurring at a mate creation, and errors occurring in mates during model recalculation.

The problems occurring at the time of creating mates could be due to two reasons. Firstly, a mate solution may not be found as it simply does not exist. For example, this may happen when the previously imposed geometrical relations in the model are such, that the system cannot relocate the selected components to satisfy the new conditions. In such a case, you need to check up the model. It is possible that excessive, unnecessary relations are present somewhere. If that's the reason, then you could try freeing up some system components or relieve extra geometrical relations. This may often be achieved by suppressing (and later deleting) a certain «obstacle» mate or allowing an additional degree of freedom in the parameters of the respective 3D fragment. Secondly, it could be so that a theoretical solution exists but the system fails to find it. There could be several reasons for that – either too short maximum calculation time is specified, or the system cannot find the solution with the current model components configuration. In such cases, there are combinations of measures helping the system to find the solution. First of all, you could defined the longer maximum calculation time to allow the system run more calculations. Also, before creating a new mate, you should manually move the components of the mechanism to the expected solution area (by using the command of moving mated elements). This would help the system to find the desired solution in complicated cases.

A separate reason for a failure to review is when the user attempts to specify such relations that lead to recursion (self-dependency). Suppose, for example, that two components were initially fixed by LCS. Later, the user wants to impose an additional mate between two components. Meanwhile, the LCS axis of one of the components relies on geometry of the second component for orientation. Moving the second component leads to a change in the first component's position. Satisfying the mate conditions implies that both components move. In such a recursive situation, the system will never be able to find a solution. To escape the trap, you would need to exterminate the reason of the recursion – which in this case means redefining the LCS of the first component to make it independent of the second component.

Errors may occur on an initially correct model after performing the regeneration operation. Mates are calculated by groups (chains). Should an error occur in a group, each mate in this group is marked in the model tree by a red cross. The reasons for regeneration errors could be of two types. It could be so that the model underwent such changes due to which the solution satisfying all conditions is no longer feasible, or the relation between mates and component geometry is broken (as, for example, a missing topology element that was used in a mate). In such a case, restoration of mates is possible under the condition that the parent elements are restored. Otherwise, the mate needs to be redefined.



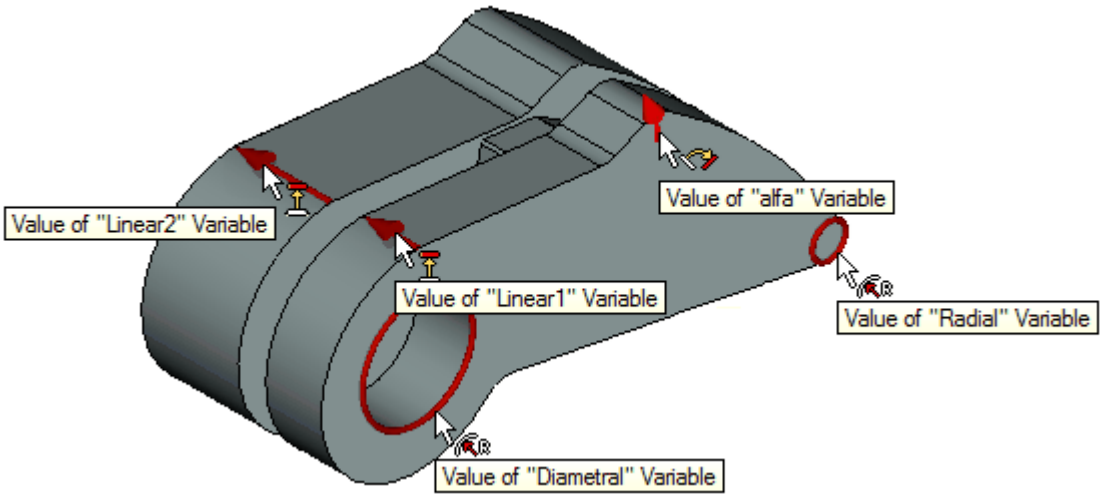
Another example is a correct model yet the system failing to find the desired solution. This may happen, for example, due to insufficient allotted maximum calculation time. Another reason could be the model getting into a trap from which the system cannot escape. In such a case, you could first try to suppress all erroneous mates, and then one by one re-enable them in the model. Simultaneously with that, you could move the components to help the system find the solution. It could also be helpful in such a case to extend the maximum calculation time. This can be done with "Time" flag on the "3D" tab of the **ST: Set Document Parameters** command.

AUXILIARY TOOLS FOR 3D ASSEMBLIES MODELING

MANIPULATORS OF EXTERNAL VARIABLES

General Information

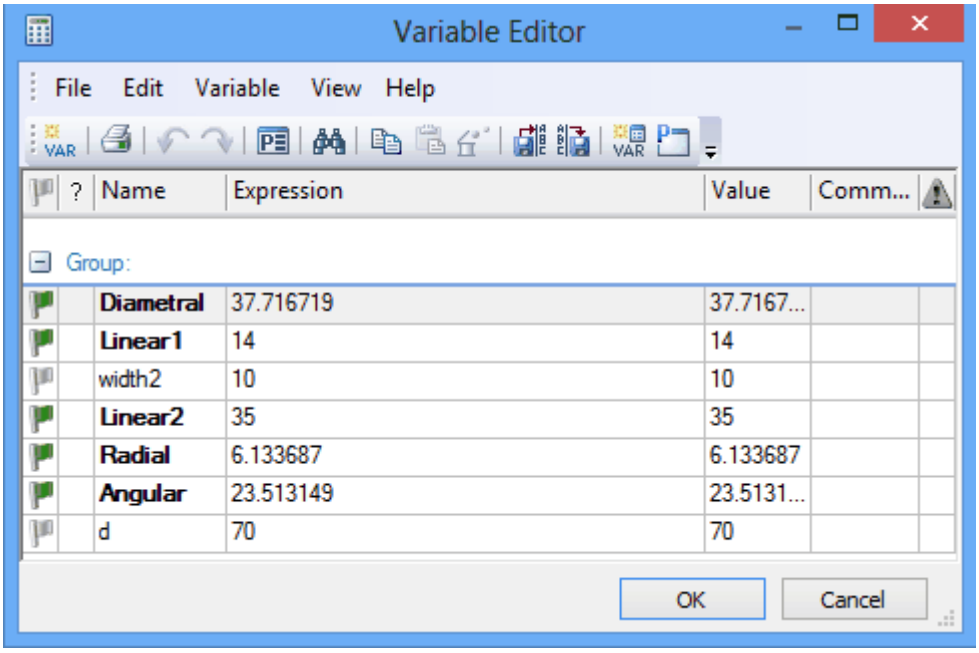
To specify the values for external variables of a 3D model, the user-defined manipulators can be created.



Manipulators of external variables can be used in two ways:

- ·to change parameters of the model in the scene. The command for controlling visibility of manipulators in the scene – **Show/hide manipulators** is added to the object visibility control toolbar in the scene.
- ·to change external variables values on the fragment when inserting the model into the assembly.

  To create manipulators it is required that the current 3D model have external variables that control parameters of the given model.



Creating manipulator

The **Parameters** > **Manipulators** command allows us to create and edit manipulators of external variables:

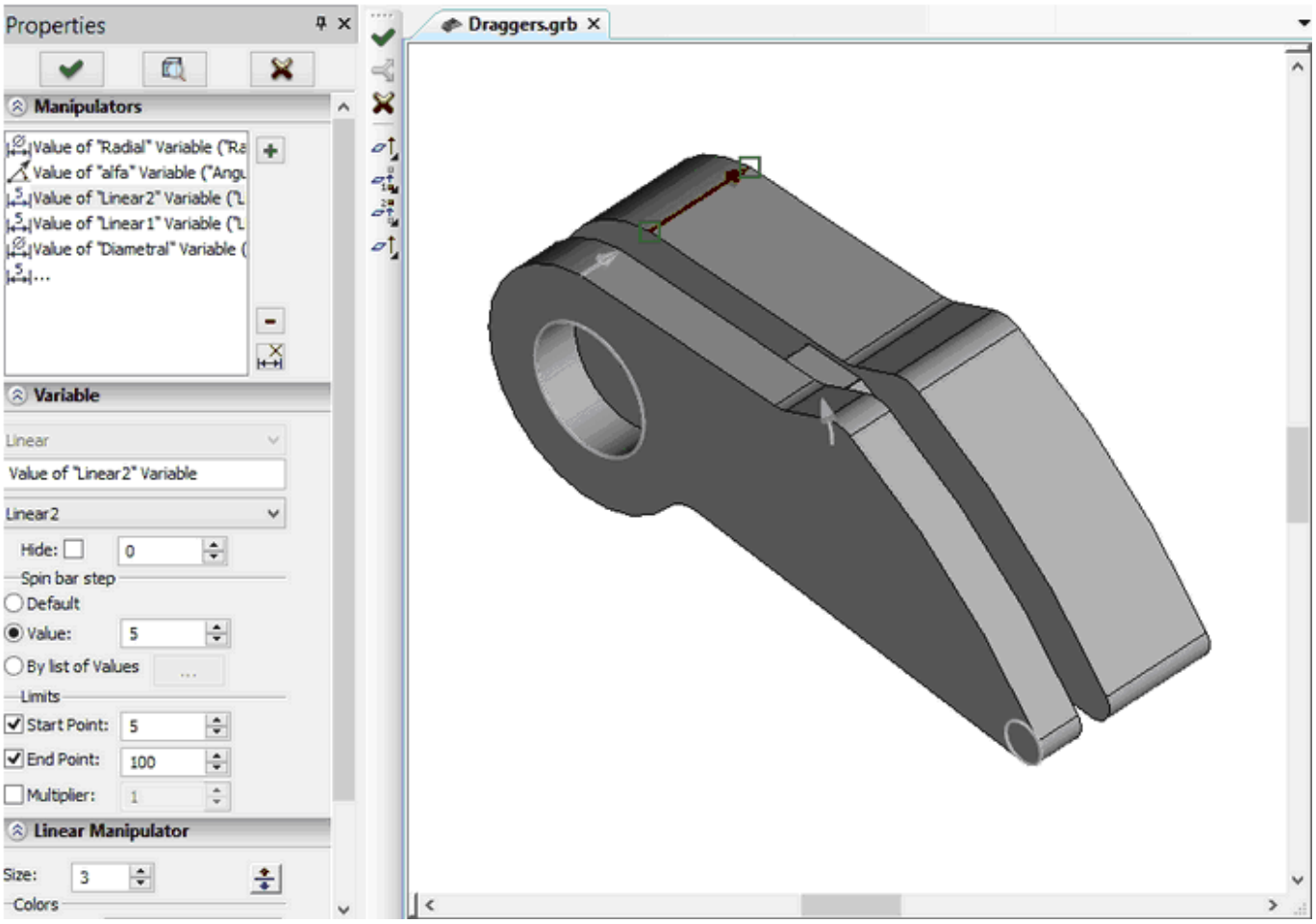
Icon	Ribbon
	Parameters → Tools → Manipulators
Keyboard	Textual Menu
<3CV>	Parameters > Manipulators

In this command it is possible to add new manipulators, remove existing ones, change manipulator's description. Creation of the following types of manipulators is possible:

- ·**linear manipulator** – arrow the length of which is equal to the variable's value;
- ·**radial manipulator** – ring the radius of which is equal to the variable's value;
- ·**diametrical** – ring whose diameter is equal to the variable's value;
- ·**angular manipulator by three points and angular manipulator by axis and initial point** – a segment of a circle whose angular dimension is specified in degrees and equal to the variable's value;
- ·**list** – for external variable the list of admissible values is available. The list can be tied to an arbitrary point on a 3D model. Manipulator of this type is shown as a small ball and upon pressing on this ball the list for variable's value selection will appear.

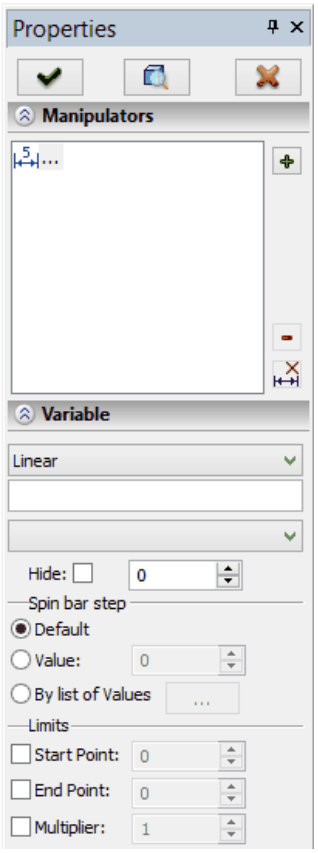
When calling the **Parameters** > **Manipulators** command the list of already existing in the given document manipulators is displayed in the properties window. If there are no manipulators in the document yet, the list will be empty.

Manipulators already created in the document are displayed in 3D window with a grey color. In order to edit the parameters of already existing manipulator, it is required to select it – with the help of in 3D window or in the list of the properties window. The selected manipulator is highlighted in 3D window with a color (by default – red color).



To create a manipulator, the cursor has to be placed over the line in the list, and afterwards parameters of the manipulator must be specified in the fields below:

Section **Variable:**



List for manipulator's type selection: *Linear, Radial, Diametrical, Angular by three points, Angular by axis and initial point, List*.

Field for specifying the manipulator's name. The name (text string) can be typed manually. Also it can be specified automatically based on the name of external variable with which the given manipulator will be linked. For this case, it is required to leave the field empty. After selection of the variable from the list below, the system itself will fill in this field according to the format "Variable's value "Variable_name"".

List of external variables of document – allows us to indicate the external variable whose value will be controlled by the manipulator being created.

Hide. This flag allows us to control manipulator's visibility on the screen. As a value for this parameter it is possible to use the real variable. Manipulator will be hidden if the value of the variable is not equal to zero.

Parameters of the **Scroll step** group specify the step with which the manipulator's value changes:

By default – the step is selected automatically depending on the scene's scale;

Value (the user can specify the exact value for the step);

By list of values (the list is generated by the user).

Parameters of the **Range** group specify the limits for the change in the manipulator's value and scale:

Limit at the beginning. This parameter allows us to specify the minimum admissible value of the manipulator;

Limit at the end. This parameter allows us to specify the maximum admissible value of the manipulator;

Multiplier. With the help of this parameter the coefficient by which the manipulator's value will be multiplied can be specified.

Also in the command's properties window there is additional section with the manipulator's visualization parameters:

Size. Size of the manipulator in 3D scene. This parameter affects only the manipulator's visualization;

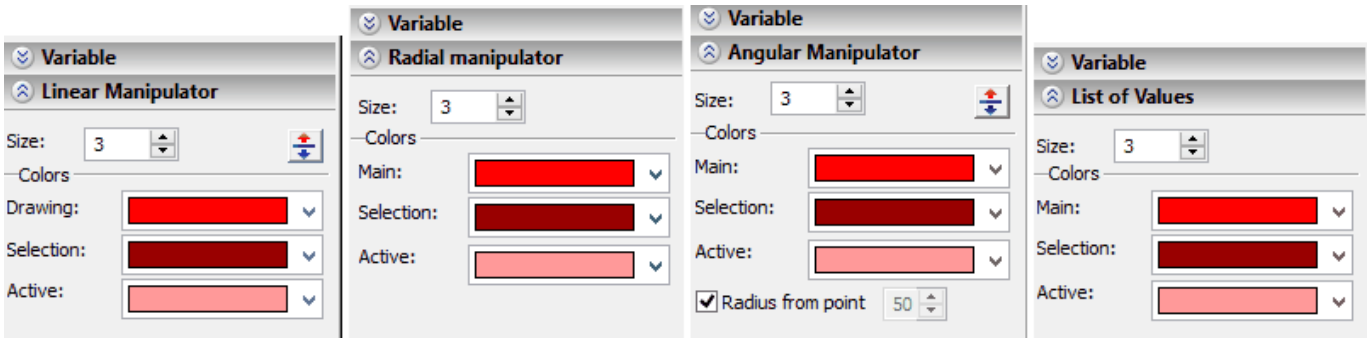
Colors/Draw. The color with which non-active manipulators will be drawn in 3D scene;

Colors/Selection. The color with which the manipulator will be drawn in 3D scene when the cursor comes near the manipulator;

Colors/Active. The color with which the active manipulator will be drawn in 3D scene;

Radius from a point. This parameter is available only for the angular manipulator and specifies the radius of the manipulator's arc. By default this flag is enabled, the radius is determined automatically by the

points specified by the user (when defining manipulator's geometry). When this flag is disabled, the radius value can be specified manually.



Geometrical parameters of the manipulator are specified with the help of the options of the automenu. The set of these options is different for each type of the manipulator.

For a linear manipulator the following options are available:

	<A>	Select axis of manipulator
	<F>	Select start point of manipulator
	<S>	Select end point of manipulator

These options allow us to specify the image of the manipulator in 3D scene by two ways – either to specify the initial and final points of the manipulator's vector or to select a 3D object that is capable of defining the manipulator's vector.

For radial and diametrical manipulators:

	<A>	Select axis of manipulator
	<F>	Select start point of manipulator
	<S>	Select end point of manipulator

These options allow us to specify the axis of the circular manipulator either by specifying two points or by selection of a 3D object capable of defining the axis entirely. The radius of the manipulator will correspond to the value of the variable linked with the manipulator.

For angular manipulator by three points:

	<1>	Specify first point of arc
	<2>	Specify second point of arc
	<3>	Specify third point of arc

The arc of the angular manipulator of the given type is determined by three points: central point of an arc (selected with the help of the option), initial point of the arc (the option) and final point of the arc (the option).

For angular manipulator by axis and initial point:

	<A>	Select axis of manipulator
	<F>	Select start point of manipulator
	<S>	Select end point of manipulator
	<B>	Select initial point of rotation

For the angular manipulator of the given type it is required to specify the manipulator's arc axis – by selection of two points (the and options) or one 3D object (the option). After the axis has been specified the initial point of the manipulator's arc must be specified. The arc length will be determined by the system automatically based on the value of the variable linked with the manipulator.

For manipulator of the type "List":

	<B>	Select point

This option allows us to indicate a 3D point in which manipulator in the form of a small ball will be created.

To complete creation/editing of the manipulator it is required to press after all parameters have been specified.

Using manipulators

When using manipulators, i.e., when changing the values of the variables of the model with the help of the manipulators, additional control element appears in the system toolbar – for specifying precise numeric value of the manipulator.

Manipulators created in 3D model are displayed in the scene of the original document. If necessary, they can be turned off with the help of the **3SM: Turn off manipulators** command. This command is available on the **View** toolbar:

Icon	Ribbon
	View Toolbar → Turn off manipulators
Keyboard	Textual Menu
<3SM>	

CONFIGURATIONS

General Information

Configuration – saved in the document additional version of the model's geometry satisfying several given conditions. Geometry of configurations is not shown in the 3D scene of the document.

By default, the T-FLEX CAD document stores only the geometry corresponding to the last saved state of the 3D model of the given document.

The main domain of application of configurations is for design of 3D assemblies. Geometry saved in the configuration can be inserted into the assembly instead of the actual geometry of the fragment. The use of configurations allows the user to rapidly download the required version of the fragment without additional recalculation and transformation of fragment's model.

Variation is a special configuration type intended to work with the product structures. The variations are used for variant BOMs creation.

More information about variant BOMs can be found in "Product structures, reports and BOMs" chapter.

T-FLEX CAD document can contain any number of configurations/variations. Each of them is assigned the unique name.

Depending on conditions defining the saved configuration, configurations can be divided into 2 types:

Configurations by a total collection of bodies – store the geometry of all bodies of 3D model;

Configurations by optional collection of bodies – store the geometry of an optional collection of bodies corresponding to any operations and Bodies from the model's tree.

If the document contains external variables, in each configuration a collection of values of external variables is also stored, for which the geometry of configuration was calculated.

Configurations can be also created in the 2D documents. In this case, configuration is capable to store only collection of values of document's external variables. Such configurations can be used while working with 2D fragments for quick specification of values of fragment's variables.

Using Configurations

When inserting a document with configurations as a 3D fragment, as usual, it is possible to either specify the values of external variables of fragment required in the assembly (if a fragment has them) or specify one of the configurations saved in the document.

When specifying the values of variables, the system has to recalculate the entire model of the fragment according to the values specified in the assembly. The geometry obtained as a result of recalculation is put into the assembly.

When using configuration (created in the document of a fragment in advance), recalculation of the fragment's model is not carried out, - the geometry stored in the configuration is inserted right away. Geometry inserted into the assembly contains only the bodies which were saved in the configuration.

Since there is no recalculation of the fragment's model upon insertion of configurations, the time spent on assembly construction is significantly reduced. This is especially evident when inserting the fragments containing quite complicated constructions and the large number of operations. The possibility to control the structure of the geometry saved in the configuration provides even wider capabilities for design of 3D assemblies.

Mechanisms, purpose and operation with the first and second types of configurations are considerably different, thus, a separate description of each is required.

Configurations by a total collection of bodies

When creating configuration by a total collection of bodies the user specifies only the values of those external variables of the document to which geometry of the configuration has to correspond. Geometry of all final bodies of the 3D model is saved in the configuration.

The use of configurations of this type for 3D fragments allows the user to avoid the recalculation of fragment's model in the assembly. To take advantage of this capability, upon the fragment's insertion, instead of entering the values of the variables, the user needs to specify one of the configurations of the fragment. Geometry of the indicated configuration will be downloaded into the assembly.

The system can use the geometry from the configuration of the fragment automatically. This happens when for 3D fragment the user specifies the values of the variables coinciding with the set of values for one of the configurations of the fragment. If this coincidence takes place, then instead of recalculation of the fragment's model, the system downloads the geometry from the corresponding configuration. Note that this rule is valid only for configurations by a total collection of bodies.

Configurations by optional collection of bodies

When creating configuration by an optional collection of bodies, in addition to a collection of values of external variables, any arbitrary set of operations and Bodies from the model's tree of the document is specified. The geometry that will be stored in the configuration represents a collection of all selected operations/bodies. When using the current document as a fragment, such configuration will be inserted as a multi-body fragment.

> Multi-Body fragments can be automatically divided into several separate bodies, if, in the properties of the fragment in the assembly, the parameter "Use" (tab "Operation") is assigned the value "As separate bodies".

Insertion of configuration by an optional collection of bodies into the assembly can be carried out only by explicit specification of the name of the required configuration in the fragment's properties.

Configurations by an optional collection of bodies can be used in cases requiring different representation of the model's geometry in the fragment and in the assembly. For example, inside the assembly they allow displaying the bodies corresponding to the operations in the middle of the model's tree of the fragment. Example – the parts with completed finishing of welded joints. Finishing of welded joints requires creation of a large number of additional operations complicating the model and increasing the recalculation time. In the 3D model of the part itself, the finishing of welded joints is necessary, but in the assembly it is often unnecessary and significantly burdens the assembly model. In such situation in the fragment it is possible to create configuration containing the part's geometry without finishing of welded joints. The list of bodies stored in this configuration will consist from one operation which precedes the operations simulating the finishing of welded joints. If, upon the insertion of the fragment into the assembly, the user selects this configuration, the part will be displayed in the assembly without finishing of welded joints.

Configurations can be also used for fragments whose structure must be modified from one assembly to another. For example, assume that the fragment's model contains a collection of bodies, but it is required to include only some of them into different assemblies, and, moreover, in different combinations. In this situation, in the fragment's document the user can create several configurations with different collection of stored operations, and download them into the assemblies as needed.

As another example of the use of configurations consider the following: often, in the assembly the user needs to create a hole adjusted for a non-standard part, and then insert, as a fragment, the part itself. To do that, in the document of the fragment-part the user can create configuration storing a simplified state of the model (operation from the middle of the model's tree) and a final view of the part (operation from the top of the tree or a Body). When inserting such configuration into the assembly with the use of fragments' partitioning into the bodies one of the inserted Bodies can be used for creating a hole (with the help of Boolean operation of subtraction), another Body – for obtaining the fragment-part itself.

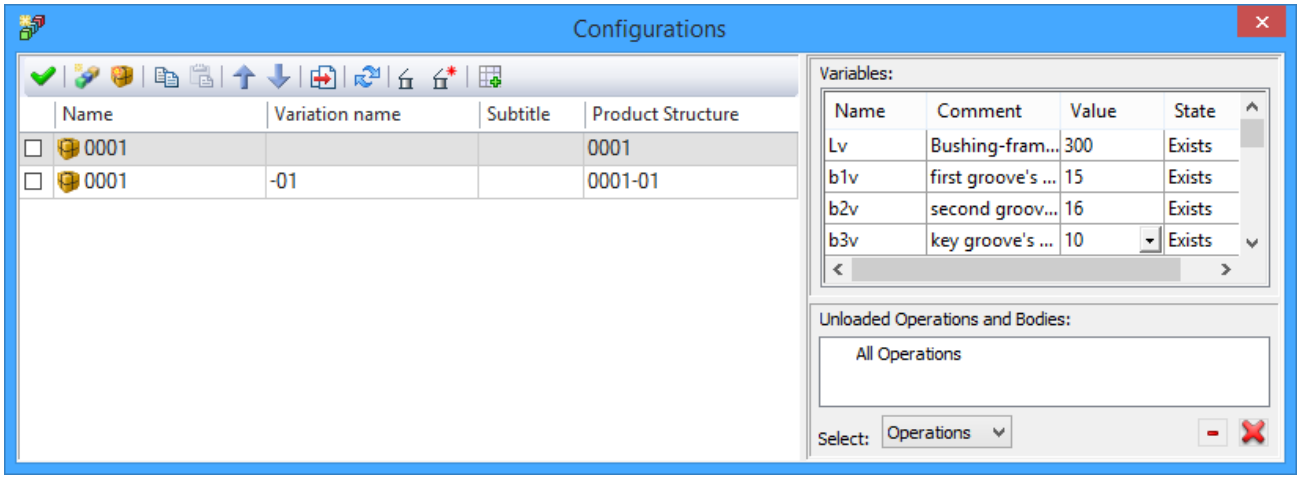
Creating and Editing Configurations

For creating and editing configurations the command **FCE: Edit Model Configurations** is used:

Icon	Ribbon
	Parameters → Tools → Configurations and variations
Keyboard	Textual Menu
<FCE>	Parameters > Model Configurations and variations

This command can be unavailable if the current document was not saved.

Calling the command brings up the dialog box for managing configurations of the current model:



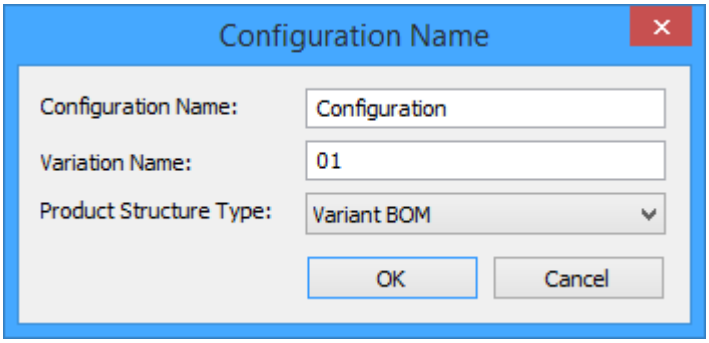
The pane on the left-hand side displays the list of the existing configurations of the current model. To start working with some of the existing configurations, you need to select it in the list. If the document

does not yet contain any configurations, the list will be empty.

Option **Create configuration** is used to create a new configuration. Option **Create variation** is used to create a new variation. A new name may be assigned for the new configuration or variation.

You can specify a product structure type for the new variation. It is recommended to select Variant BOM if you plan to create a variant BOM table subsequently.

If the field "Part No." is filled, it will be set as configuration/variation name by default. If it is empty, the current document name will be used.



The product structure will be automatically created after the variation saving. Its name matches the source name.

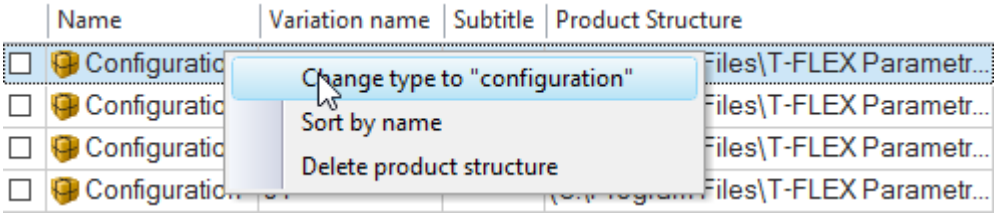
For already created configurations/variations, you can change the name and, if necessary, specify the subtitle by editing the corresponding cell in the table.

The set of variable values that corresponds to the current state of the model is assigned for a new configuration/variation, by default.

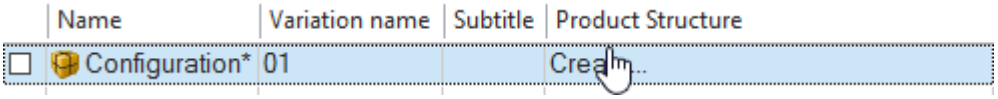
There are following options in the context menu:

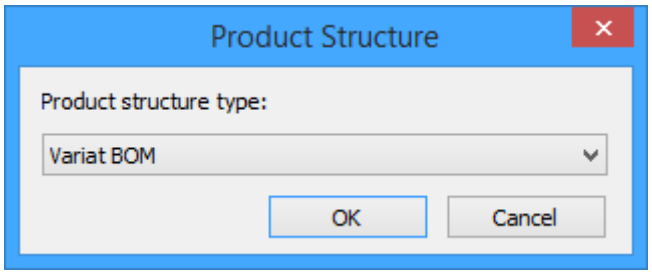
Change type to **configuration/variation** allows to switch type of currently selected element.

Sort by name allows to activate sorting by Name column.

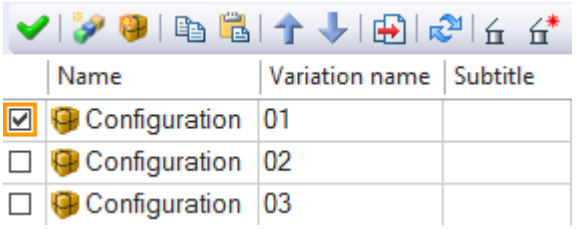


Delete product structure deletes product structure for the currently selected element. You can reassign the product structure in the **Product Structure** window by clicking "Create..."





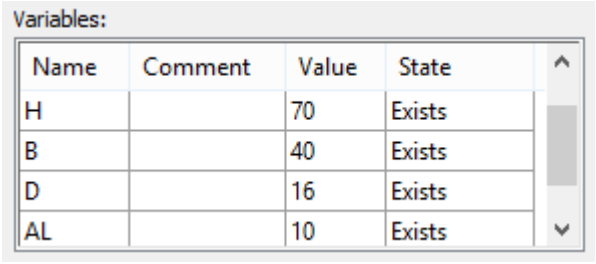
Base variation. The flag to the left from the name is used for the base variation selection.



The properties of product structure corresponding to the base variation are used for data display in the **Product Structure** window in the Apply product structure representation mode. If the flag is not set for any variation, the first variation in the list is considered to be the base.

For selected configuration, the list of external variables values is shown in the table Variables in the upper right pane of the command's window.

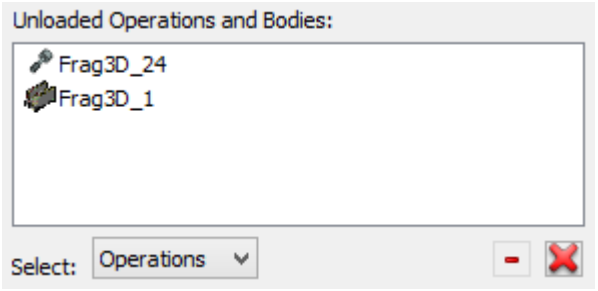
The values of variables can be modified in the table. Upon modifying any external variable, the earlier saved configuration becomes outdated and is marked by an asterisk.

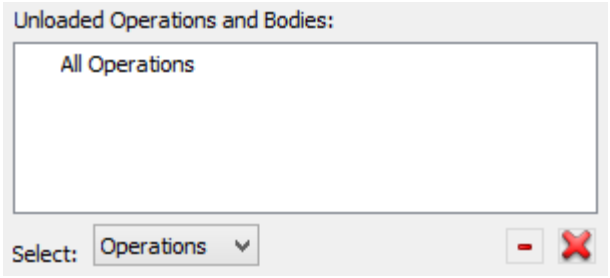


You need to save changes using option to make them actual. Some external variables can be deleted, and some can be created when you edit the model.

The "State" column displays information about each variable in configuration. It shows if a variable exists in the model and if the model variable exists in the configuration.

At the lower right pane, there is a list of operations and Bodies of the model, stored in the selected configuration.





"**All operations**" value is displayed in the list after creating configuration. If the configuration creation is completed at this stage, configuration by a total collection of bodies will be created. Uploaded operations and bodies configuration by an optional collection of bodies is created when you add

elements to the list. For filling in the list, you need to set a desired filter in parameter Select: Bodies or operations. After that, it is possible to select any Bodies or operation in the 3D scene or in the tree of the 3D model.

Buttons and allow you to edit the list of bodies in the configuration:

- – removes operation/Body selected in the list;
- – removes the entire contents of the list (the list content is replaced with "All operations").

The following graphic buttons are used for managing configurations as well:

Apply changes – saves changes and new configurations/variations.

Copy/Paste - allows to copy or create a new configuration/variation based on the existing one. You need to specify name for the new configuration. The variables values and the product structure type are copied from the source object.

Move up/Move down - allows to change elements order in the list. The order is important only for the variations. Their order in the variant reports matches the order in the list.

Load - regenerates model with a variable values defined in the selected configuration. This command allows viewing geometry of each configuration.

Update - assigns current values to the external model variables in the current configuration/variation.

Delete - deletes the selected configuration/variation.

Delete all – deletes all configurations/variations.

Create new variations table – creates variation table of the model.

> You can find more information about variation tables in "Drawing title block" chapter.

EXPLODED VIEW

T-FLEX CAD provides a special command for exploding 3D models made of several operations (normally, this is an assembly 3D model). The exploding transformations can be defined in the parameters of any three-dimensional operation. The operations that require an LCS for creation, such as 3D fragments, 3D copies, etc., are positioned according to the transformations defining translations (offsets) and rotations with respect to the attachment LCS. Other operations can be transformed relative to world coordinates or the center of the outline box. This is defined in the transformation settings. More details on transformations are provided in the chapter "Assigning "Transformations" to 3D Entities".

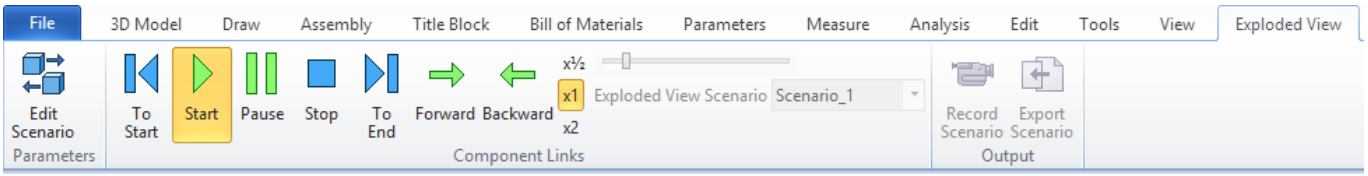
Command Execution

The exploding mode is turned on and off by the command:

Icon	Ribbon
	Tools → Animation → Exploded View
Keyboard	Textual Menu
<3VX>	Tools > Exploded View

The exploding mode is off by default.

After activation, the "Explode animation" toolbar appears.



More information about it can be found in section "Exploded view animation".

Calling the command transforms the current parameters that define positions of the operations. The parameters on the **[Transformation]** tab define the permanent transformations, as well as the exploding mode transformations. When the exploding mode is turned on, the transformations are applied that were defined for the exploding mode.

The exploding mode can be used when you need to create a 2D projection of the product assembly taken apart. All necessary settings are collected in the 2D projection parameters.

More information can be found in the chapter "2D Projections. Creating 2D Drawings of the 3D Models".

Exploded view scenario

The command is used for setting or editing exploded view parameters. You can perform the following steps in **Exploded view scenario** command:

- Create and edit exploded view scenarios. It is possible to create several exploded view scenarios.
- Create your own scenario of fragment exploded view as part of the general scenario.
- Define animation parameters.
- Control bodies visibility on each stage of exploded view scenario.

Several exploded view scenarios can be created for an assembly. For example, assembly parts exploded view, assembly sequence, etc. Each scenario may consist of several stages. Stage includes sequence of actions. Action is an execution of individual transformation – translation or rotation of selected elements group. Group composition for various actions can be different.

To call the command:

Icon	Ribbon
	Tools → Animation → Exploded View Scenario
Keyboard	Textual Menu
<3TE>	Tools > Exploded View Scenario

Creation of Exploded View Scenario

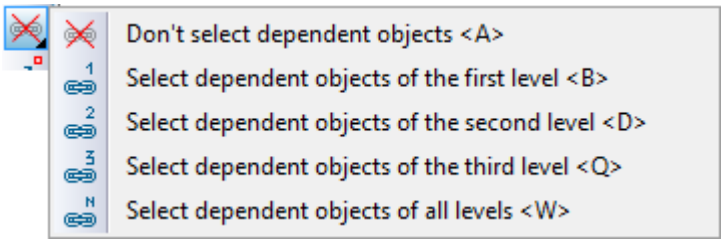
After activation you need to select objects which will be used for explosion view. For this purpose use options:

	<O>	Select object for individual transformation
	<M>	Select objects for transformation

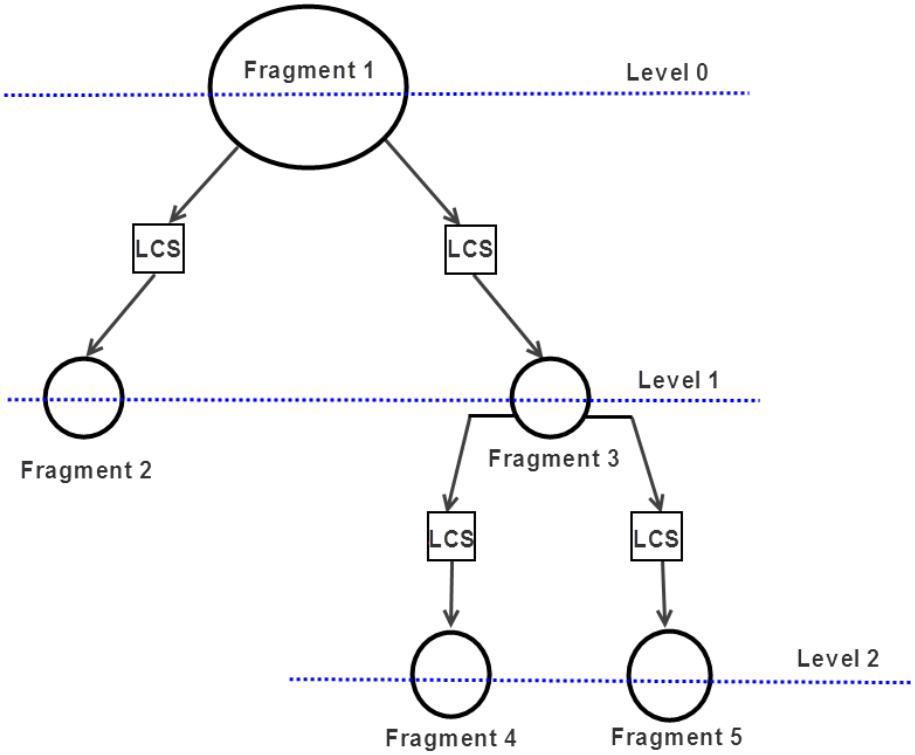
When is selected it is possible to select only one object for transformation. You can select several elements using <Shift>.

Upon selection of it is possible to simultaneously select several objects for transformation.

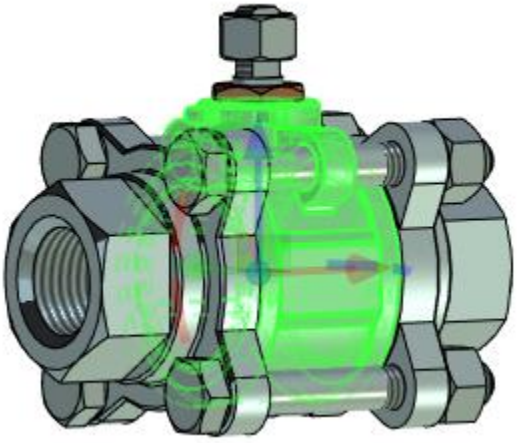
There is the possibility to choose dependent objects. For this purpose use special option in automenu:



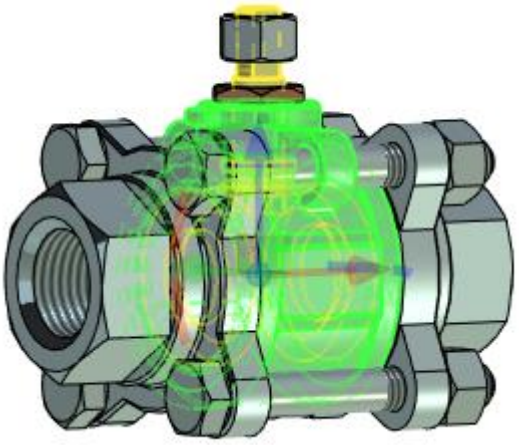
Levels of dependence mean hierarchical dependence of assembly elements. The zero level corresponds to the fragment that is selected in the assembly. The first level corresponds to all the fragments in the assembly that depends on the first one.



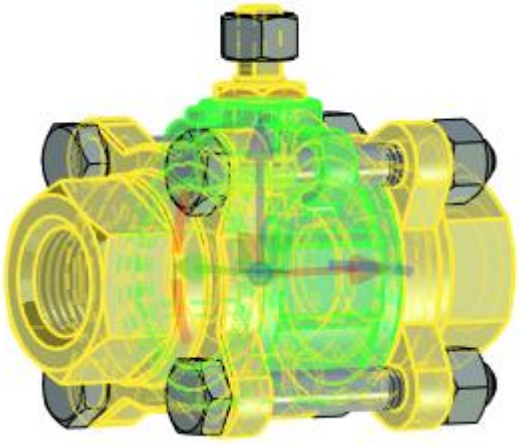
The option **Don't select dependent objects** is active by default, that's why only one fragment is selected in the scene.



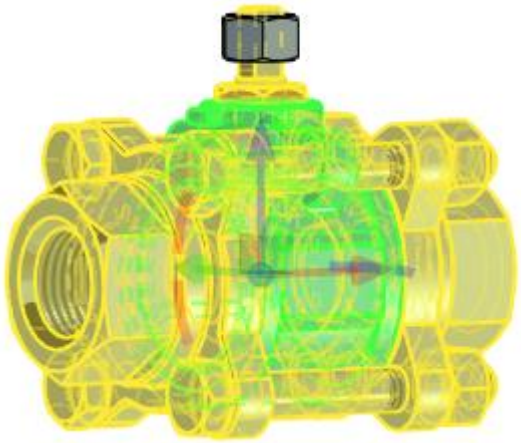
If you activate option **Select dependent objects of the first level** and select the same fragment, all the fragments that are directly dependent on the fragment will be selected.



If you activate option **Select dependent objects of the second level** , the fragments dependent on the first level fragment will be also selected.

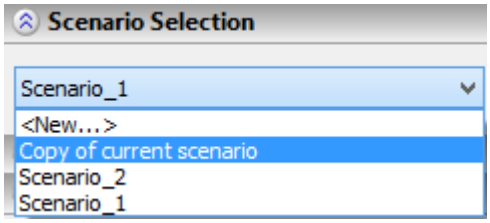


If you activate option **Select dependent objects of all levels** , all elements that directly or indirectly depends on the first selected fragment will be selected.



Using of the option is convenient in case of working with complex assemblies, for example, when you need to translate object together with its dependent objects.

Creation of new or edition of existing explosion view scenario is made in the section **Scenario Selection.**

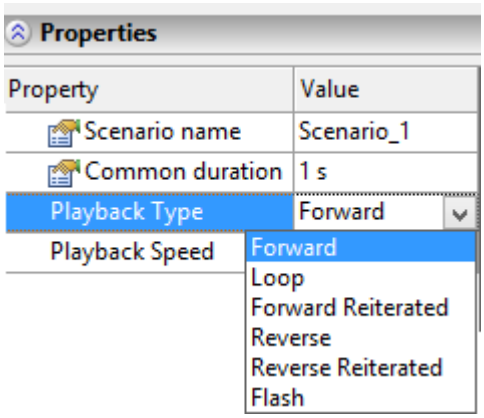


You can create a copy of an already existing script.

> You cannot create a new script, if work with the current one is not finished.

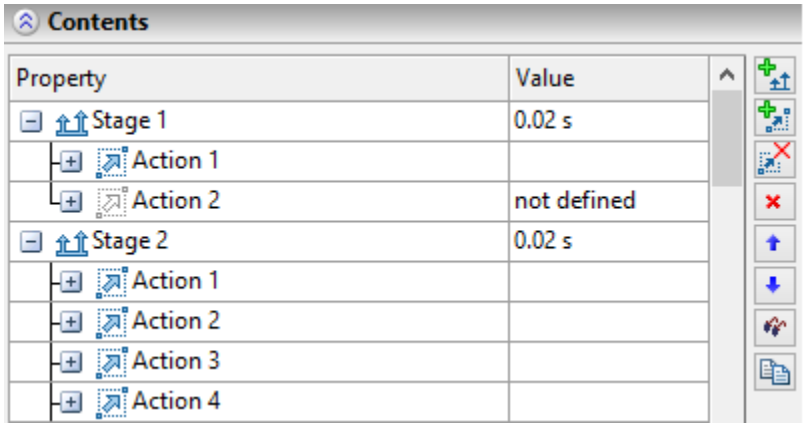
The following objects can be selected as objects of the scenario: bodies, fragments, cameras, light sources.

To edit scenario properties use section [Properties].



Using this section, you can edit name of scenario, set duration, type and speed of the exploded view animation.
Each scenario contains stages, divided into actions.
Full sequence of creating assembly from parts and subassemblies can be traced with the help of several stages.

For creating and editing stages of exploded view use **Contents** section.



This dialog box contains information about created stages, actions, and objects, used in stages. Several graphical buttons are located to the right of the transformations list.

Button is used for creation of a new exploded view stage.

When you press button, a new action for selected stage is created.

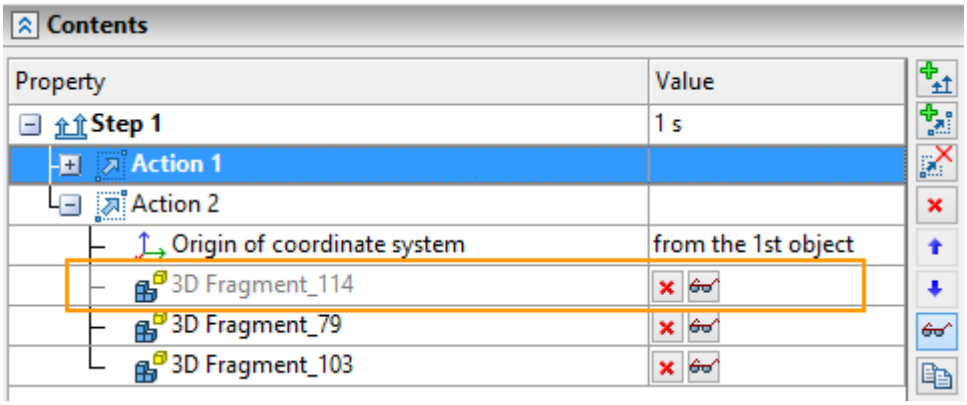
Button is used to delete all transformations of a stage.

Button deletes last exploded view stage in the list. To delete an action use button , located near actions name.

When you press buttons, the sequence of exploded view stages is changing.

With the help of button, you can control stages visibility. When active, icon changes its color . The “Hide” option affects not only the current stage, but also all following stages, until the activation of “show” option. All bodies are visible by default.

There is a possibility to manage visibility and delete individual objects with the help of buttons. Hidden objects are shown in gray in the list.



You can manage objects visibility only in current document.

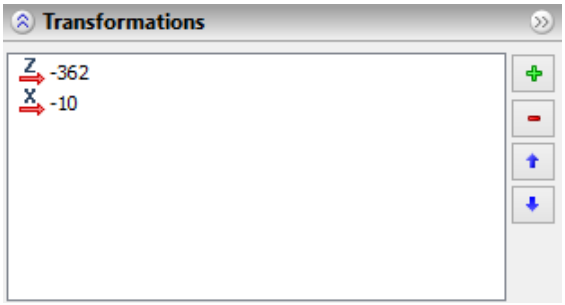
Button allows to copy selected stage. The copied stage is added to the end of the list.

Each action has one object or a group of objects and the chain of transformations for them. As a coordinate system for the chain of transformations you can use local coordinate system or coordinate system of the first group object (in a case of group transformations). A special option from automenu is used for the coordinate system selection:

	<L>	Select LCS for current transformation

In case, when it is not possible right away to orient LCS in a desired way, the functions of additional coordinate system specification can be employed. These functions allow you to quickly rotate the constructed LCS around its axes by 90° (, ,), perform cyclic turning of the axes of the coordinate system around the origin (when pressing the axes change their places).

All transformations are shown in the **Transformations** section of the properties window.



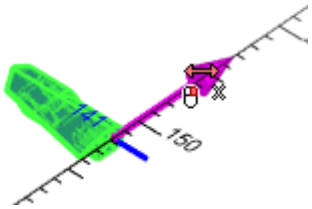
Several graphical buttons are located to the right of the transformations list. Button is used to add a new transformation. You can select required transformation from the transformations list that appears after pressing the button. This this list can be also called from the context menu, after pressing in the “Transformations” window.

After type selection a new string appears in the transformations list. You can specify transformation value in the value input field.

Transformation values can be assigned numerically or using variables and expressions.

When pressing button the last transformation in the list is deleted. Pressing buttons changes sequence of transformations.

To save the transformations you need to press [Enter]. Also, a special manipulator in 3D window can be used to define transformations. This manipulator is moving according to the selected type of transformation.

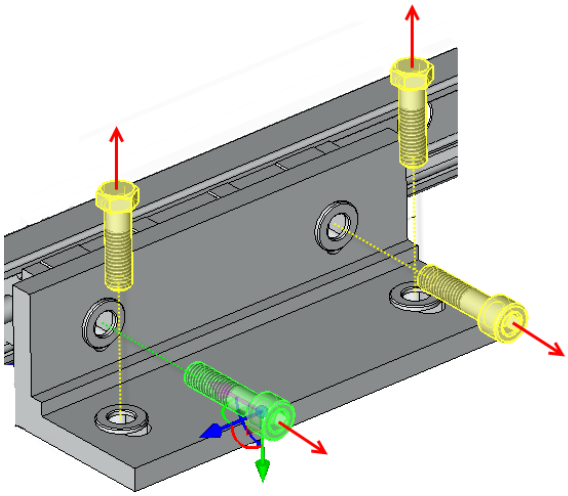


Group transformation can be performed either in the “common” coordinate system, or in own coordinate system of each group member.

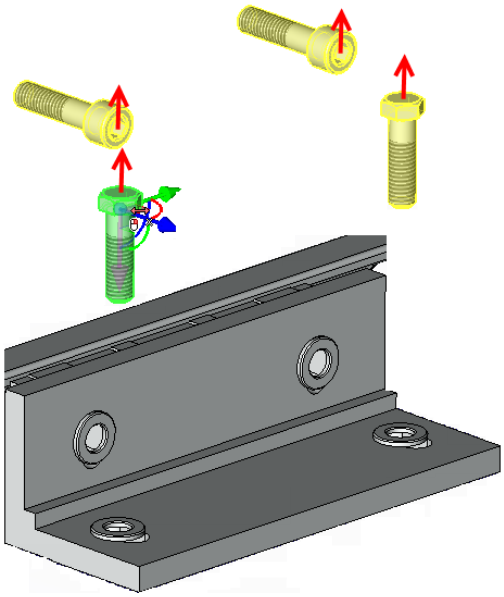
Mode of setting transformations in the common coordinate system is activated by default after selection of several objects for transformation:

	<G>	Set transformations of selected objects in common CS

If this option is switched off, the group transformation is transferred into own coordinate system of each group member. I.e. when you move several fragments, each of them is moving along its own X axis.



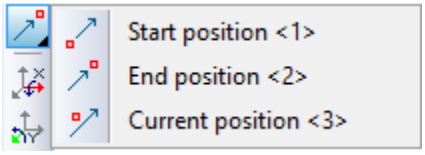
Each bolt is translated in its own coordinate system

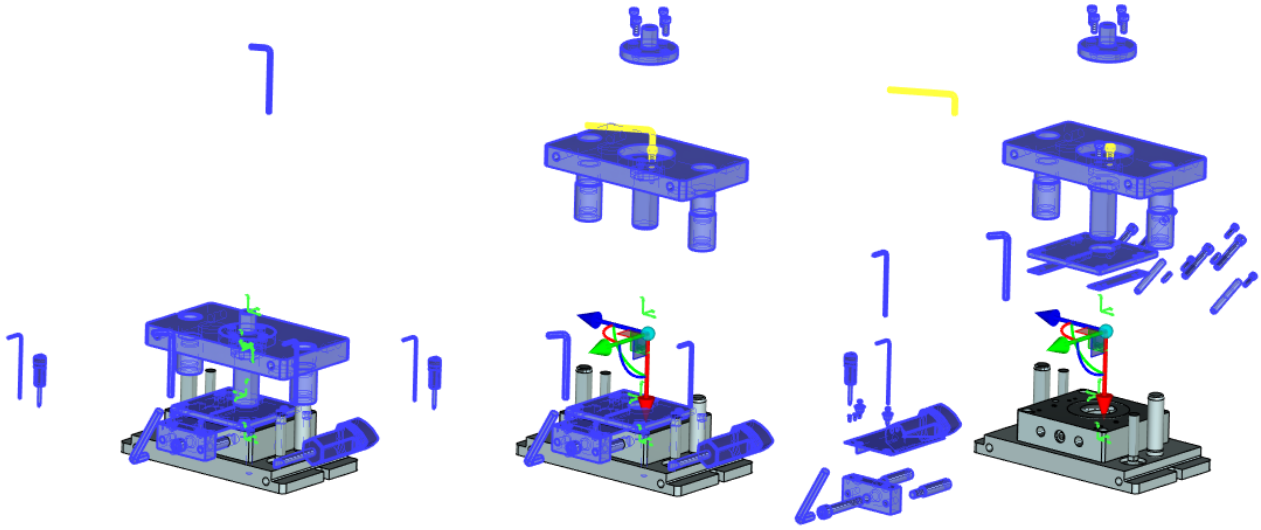


Bolts are moved in the common coordinate system

Thus, usage of the group transformations can significantly reduce time spent on specifying elements transformations.

You can enable display of one of three positions: start, end and current, using options:





Start position

Current position

End position

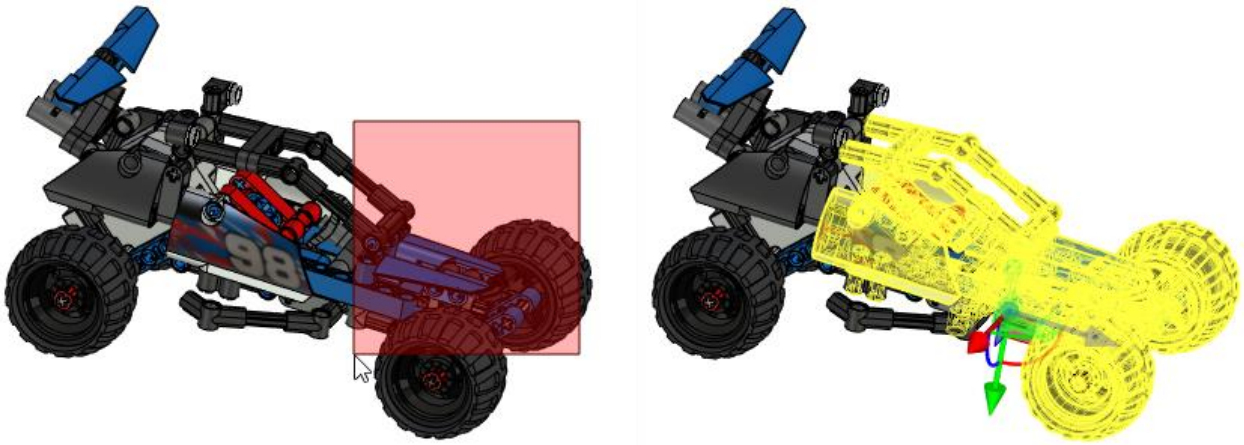
To improve convenience of objects selection use option:

	<N>	Group selection of objects by rectangle

You can select rectangular area, holding . There are two selection conditions:

1. All objects, which are completely covered by rectangular area, will be selected for the transformation. Common manipulator for these elements will appear. The cursor is moving from left to right.

2. All objects that intersect the rectangle will be selected. Common manipulator for these elements will appear. The cursor is moving from right to left.

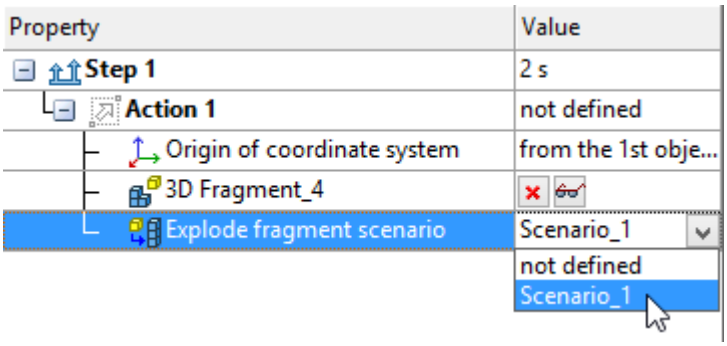


The area color differs for each condition.

Exploded View of Subassembly Elements

If there are subassemblies in the assembly you can move elements of the subassemblies separately. Use one of the following ways for this purpose:

If there is an exploded view scenario in the subassembly document, you can use it in the assembly. You need to select the subassembly fragment in the scene and choose one of its scenarios.

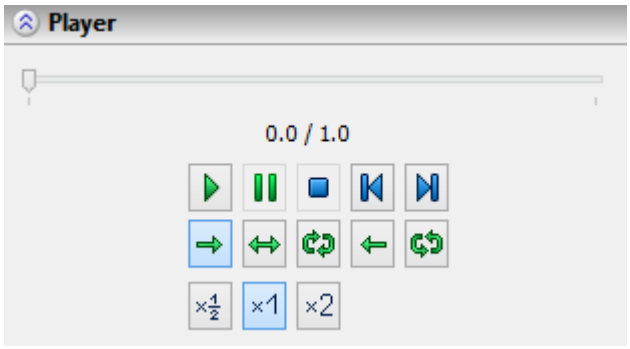


The second way to move subassembly elements separately is to divide subassembly using **3SD: Divide** command. The way is not preferable, because the link between the product structure and subassembly is lost after division. This method can be used if the assembly document is used only for visualization.

Exploded view Animation

In **"Exploded View Scenario"** command, you can create animation of assembly/exploded view. While defining animation you specify position of assembly elements in each period.

The **[Player]** section in properties window is used to manage animation.



All elements movements are displayed on the time scale. Time scale represents horizontal line with slider. Slider indicates the playback time of current stage and total duration of animation. When you move the slider, current animation position changes and model is accordingly updated.

In the lower part of the section there are graphical buttons, which manage animation parameters.

- **Play**. Starts animation.
- **Pause.** Temporary stops the animation at the current stage.
- **Stop.** Stops animation.
- **To Start.** Scrolls animation to the beginning.
- **To End.** Scrolls animation to the end.

Animation starts in one of the following modes:

- **Forward Back.** Plays the animation once from the beginning to the end.

- **Forward Reiterated Playback.** Plays the animation in a circle (from the beginning to the end, from the beginning to the end, and so on) until the button is pressed.

- **Loop Playback.** Plays the animation in a continuous cycle (from the beginning to the end, from the end to the beginning, and so on) until the button is pressed.

- **Reverse Playback.** Plays the animation once from the end to the beginning.

- **Reverse Reiterated Playback.** Plays the animation in a circle (from the end to the beginning, from the end to the beginning and so on), until the button is pressed.

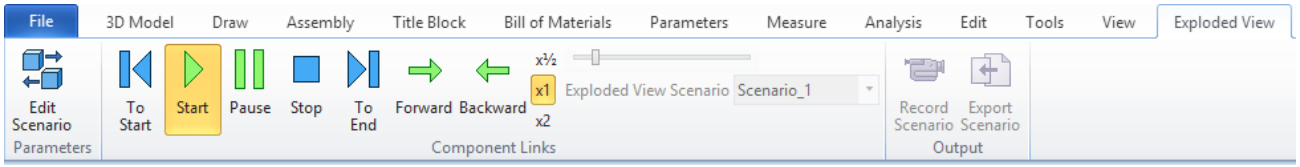
- **Normal playback speed.** Playback speed set by default.

- **Slow down playback.** Playback with half speed.

- **Speed up playback.** Playback with double speed.

The animation mode can be selected either before pressing [**Play**] button or while animation playback.

You can manage animation without activation of **Exploded View Scenario** command. For this purpose, the special tab on the Ribbon is used. It is called using the **3VX: Exploded View** command.



You can activate the command for scenario editing on the tab.

Main options for the exploded view playback are in the **Control** group. You can also select the scenario in the group.

> You can stop animation by pressing <ESC>, model recalculation or reactivation of the **Exploded View** command.

In the **Output** group there are commands for record and export of the scenario.

Animation Record

The **Record Scenario** option allows animation recording in *.avi and *.wmv formats. General scenarios and scenarios by default are supported.

Record Scenario window appears when you select the option. Here you can set the following parameters:

You can select one of three variants in the **Photorealistic view generation** section.

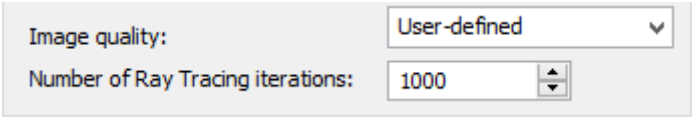
Non-photorealistic rendering – the exploded view scenario is recorded according to the current view in 3D scene.

The following variants are used for photorealistic view creation. Each frame of the video is processed using selected photorealistic view mechanism.

Photorealistic rendering using OptiX – the video is created using NVIDIA OptiX mechanism.

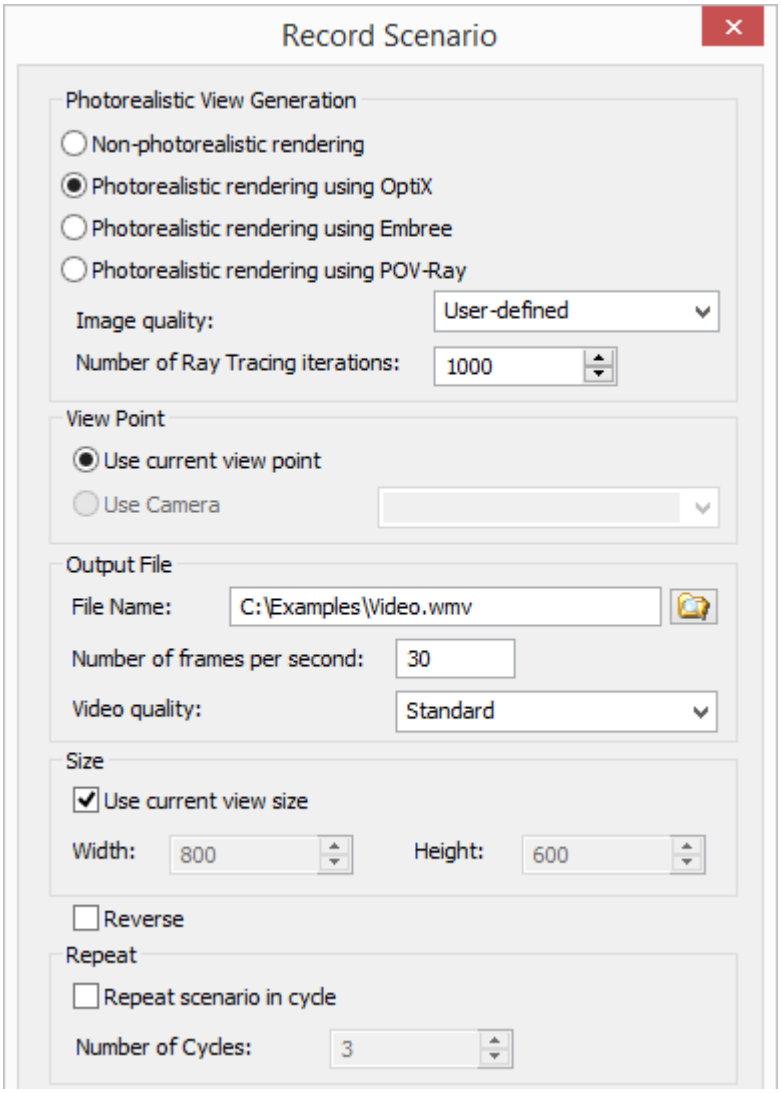
Photorealistic rendering using Embree – the video is created using Embree mechanism.

You can set **Image quality** and **Number of Ray Tracing iterations** for the previous two variants.



Photorealistic rendering using POV-Ray – the video is created u sing the POV-Ray mechanism.

> You can find more information about photorealism in “Photorealistic view” chapter.



You can set view point using **View point** section.

Use current view point. The video is recorded according to the current view point in 3D scene.

Use camera. Video is recorded according to the selected camera from the list. It is available only if there are any cameras in the scene.

You can find more information about cameras in “Cameras” chapter.

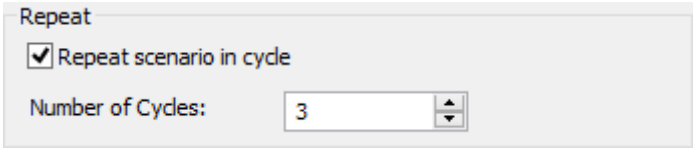
You can set **File name** and **Number of frames per second**. The path to the created file is shown in the **File name** field. You can specify name, file path and file format using button.

Video quality setting is available for the *.wmv format. The video quality and file size are changed according to its value.

If the flag **Use current view size** is active, the video is recorded according to the active view size. Otherwise, you can set size manually.

The **Reverse** flag allows to record exploded view video in the reverse direction to the given in scenario.

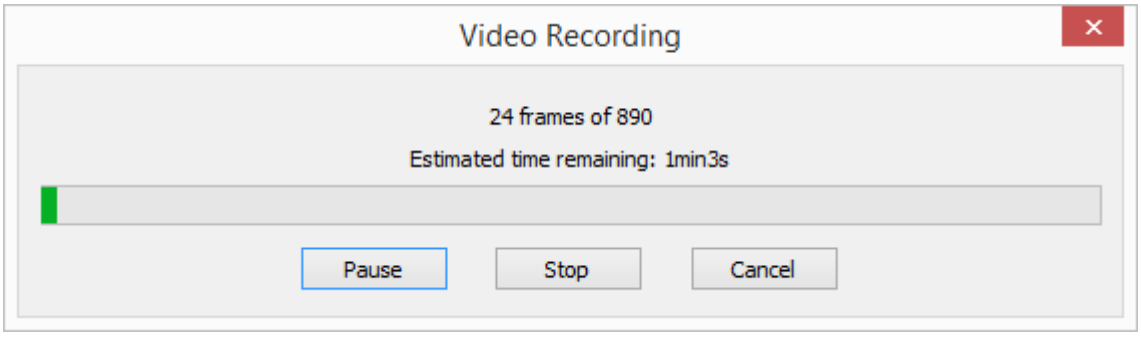
When the flag **Repeat scenario in cycle** is active, you can set Number of cycles. It sets number of exploded view scenario repeats.



The Video recording window appears after the **[OK]** button pressing.

Number of processed frames and time until video creation are shown here.

Using buttons you can pause recording **[Pause]**, stop record on current frame [Stop] or cancel recording **[Cancel]**. When you stop recording, already created part of the video is saved.

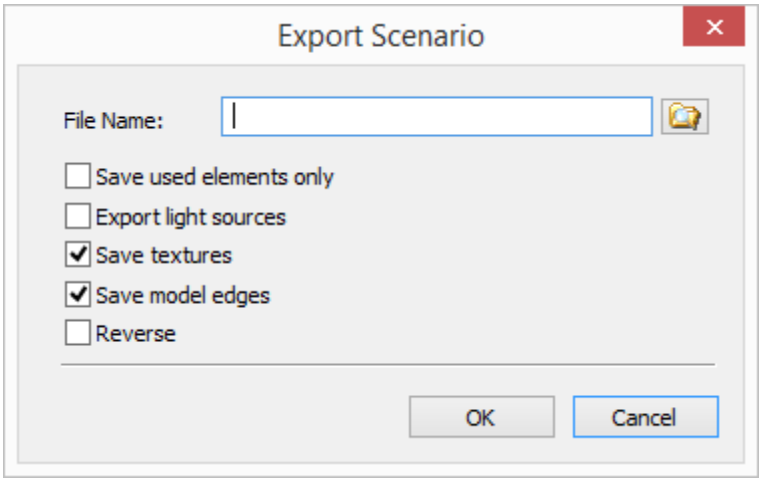


The preview window appears when you use photorealism.

Save Scenario

Save scenario allows to export exploded view scenarios to *.wrl, *.x3d, *.pov, *.u3d, *.pdf formats.

The window **Export Scenario** appears after the command calling.



File path is displayed in the **File Name** field. You can select filename, its path and file format using button .

Save used elements only. All elements that are not used in the scenario will be not included in the file, when the flag is active.

Export light sources. Light sources are included in the file.

Save textures. The objects textures will be saved to the file. Otherwise they are not saved and the material of the object will be displayed instead of them.

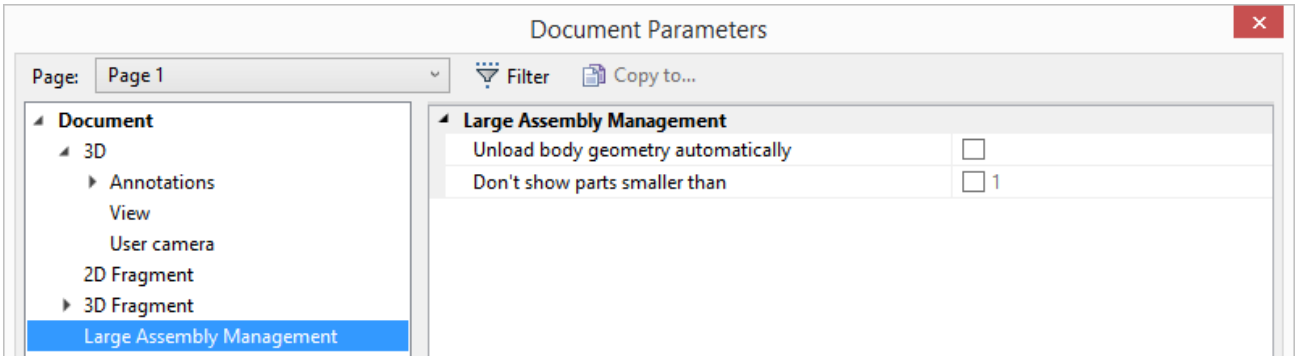
Save model edges. Model edges are saved when the flag is set. The flag is enabled only for *.wrl and *.x3d formats.

Reverse. The scenario will be exported in the reversed direction.

Large Assembly Management Tools

To optimize working with assembly models, the system provides specialized large assembly management tools. Those allow working only with the subassemblies of concern, save the overall used memory and facilitate spinning the 3D scene when it is densely populated with objects. A “large assembly” is a model in which the large assembly mode is activated.

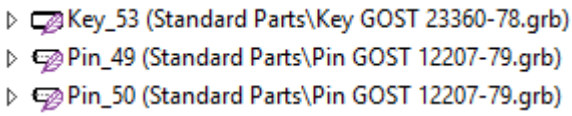
The large assembly mode is activated in the **ST: Set Document Parameters** command on the **Large Assembly Management** tab using **Unload Body Geometry Automatically** parameter:



This mode is turned on automatically. It depends on the model complexity and the computer resource availability.

First of all, the large assembly mode provisions for freeing memory by automatically unloading the detailed information about the assembly objects. Such bodies are displayed seemingly normal, however their edges, vertices, faces, etc. cannot be selected.

Such parts are highlighted in a special color when selected and are marked in the model tree with a special icon (“feather”). The omitted data is loaded automatically as needed.



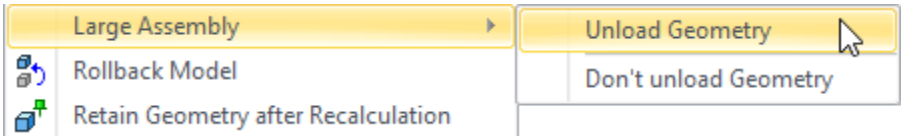
Such a need arises when editing parts, when performing various modifications and geometry analysis, as well as when creating projections.

You can also specify mimimal size of displayed parts.

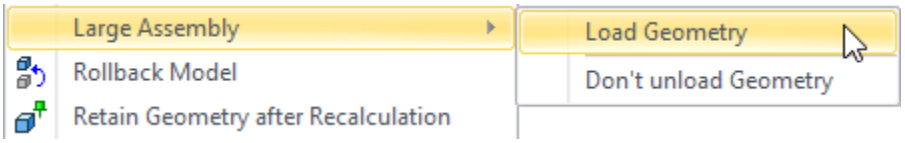
The image detailing level can be set for 3D fragments. One can choose a “Rough”, “Middle” or “Accurate” quality.

When in the large assembly mode, the context menu on selected parts gains special commands that serve to:

Unload Geometry. Force unloading Body geometry from the assembly;



Load Geometry. Force loading Body geometry to the assembly;



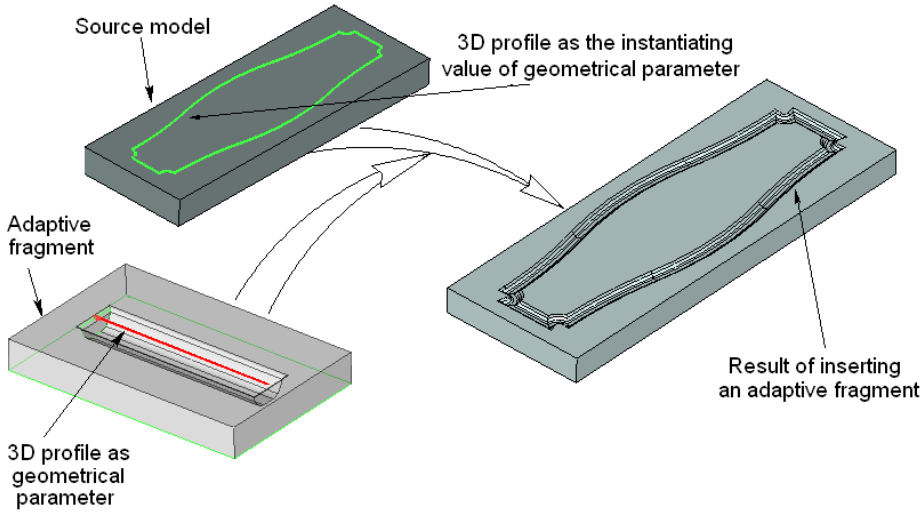
Don’t unload Geometry. When this option is active, the geometry of the concerned body will not be unloaded automatically.

An auxiliary means that helps ease working with large assemblies is the capability of creating 2D projections based on a 3D model located in an external file. This allows separating the model handling process and the drawing creation. An associative relation is still maintained between the drawing and the model.

ADAPTIVE 3D FRAGMENTS

GENERAL INFORMATION

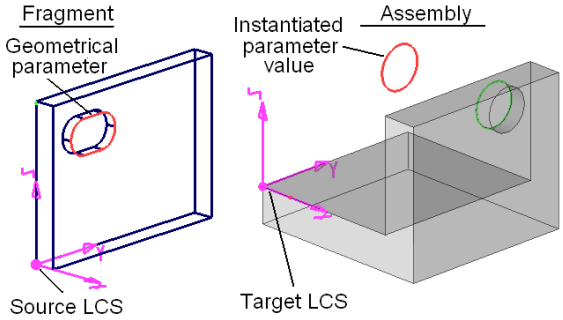
An adaptive fragment in T-FLEX CAD is a 3D fragment with the external parameters being not only external variables, but also geometrical elements. A geometrical parameter can be represented by any model elements (a solid, 3D node, 3D path, 3D profile, workplane, LCS, vertex, edge, face, loop). The idea of the approach is using the substitution geometry of an assembly element instead of the original geometry of the external parameter in the structure of an adaptive 3D fragment when regenerating a particular instance of its model. In this way, the 3D fragment acts as if "adapting" to the objects of the assembly model. For example, adaptive 3D fragments are used in special operations of inserting holes and creating stamped features in the appropriate system commands.



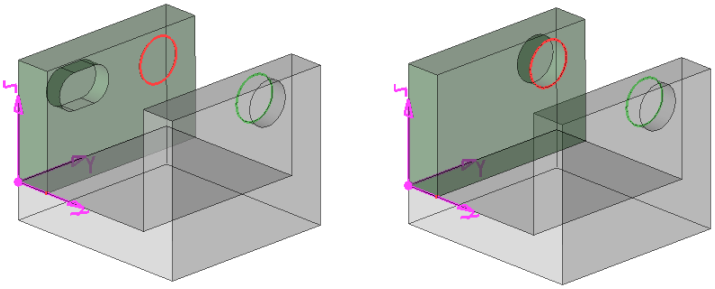
To use a 3D fragment as an adaptive model element, define in advance the set of geometrical parameters that can have relations with objects of an assembly model. Just like a set of external variables, a set of geometrical parameters is defined in the document of the given 3D fragment. The system allows defining geometrical parameters and their values actually at the time of inserting or editing a 3D fragment; however, the created relations will be used in this case only in the current assembly model. In such a case, this information will not be saved in the 3D fragment's document. When inserting an adaptive 3D fragment, the user specifies the values of geometrical parameters. The type of the instantiating object must be the same as the type of the object which is the parameter of the 3D fragment. For example, the instantiating value for a 3D profile can be defined only by a 3D profile.

The use of geometrical parameters for a 3D fragment can be combined with use of external variables and attachment coordinate systems. However, while doing so, keep in mind the specifics of instantiating a parameter object, as the substitution affects both the geometry of the adaptive parameter and its position in the assembly. As we know, the position of a 3D fragment in an assembly is defined by the

source and target coordinate systems. Those coordinate systems can be specified by LCS or defaulted to the global coordinate systems of a 3D fragment and an assembly models.

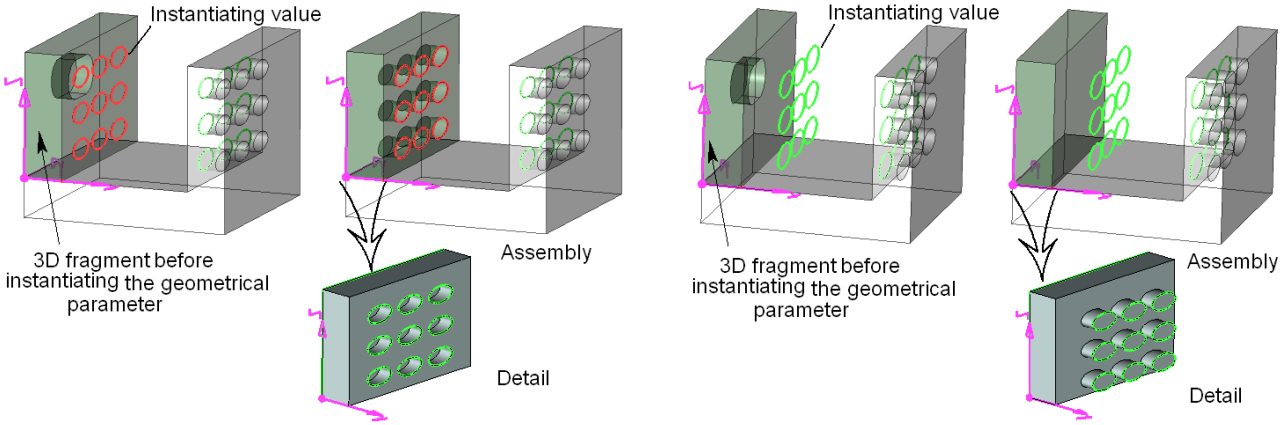


When instantiating the geometrical parameter, the system runs a "reverse" transformation by carrying the object from the assembly to the 3D fragment model, from the target LCS of the assembly towards the source LCS of the 3D fragment.



Upon instantiating the values, an adaptive 3D fragment is regenerated and inserted into the assembly model. The user needs to insure correct orientation of the parameter object instances to have them positioned according to the design intent upon embedding in the 3D fragment model.

On the diagrams below, a 3D profile instance substitutes the original profile defining a hole in a wall of a 3D fragment. According to the intent, the new profile instance will serve for creating an array of holes in a wall of a 3D fragment. Among the two possibilities, in the first case the profile instance will be suitable for creating holes upon embedding in the 3D fragment model, while not so in the second case.



Let's review in details what happens with the 3D fragment model in the second case. Since a different instance of the 3D profile is used in this case, oriented differently with respect to the target LCS, then the result differs in the final position of the profile in the 3D fragment model. The detail view clearly shows that protruding the array of profiles in the second case does not lead to hole creation (to demonstrate that, the second operand of the Boolean subtraction was left in the scene); therefore, the 3D fragment is inserted in the assembly without holes in its wall.

If, upon inserting an adaptive 3D fragment, it is necessary to use the source coordinate system, then make sure that it is created based on 3D fragment model elements (3D nodes, edges, vertices) that are independent of the external geometrical parameters. In other words, the parents of the source coordinate system shall not change upon instantiating adaptive elements.

Setting up Adaptive 3D Fragment

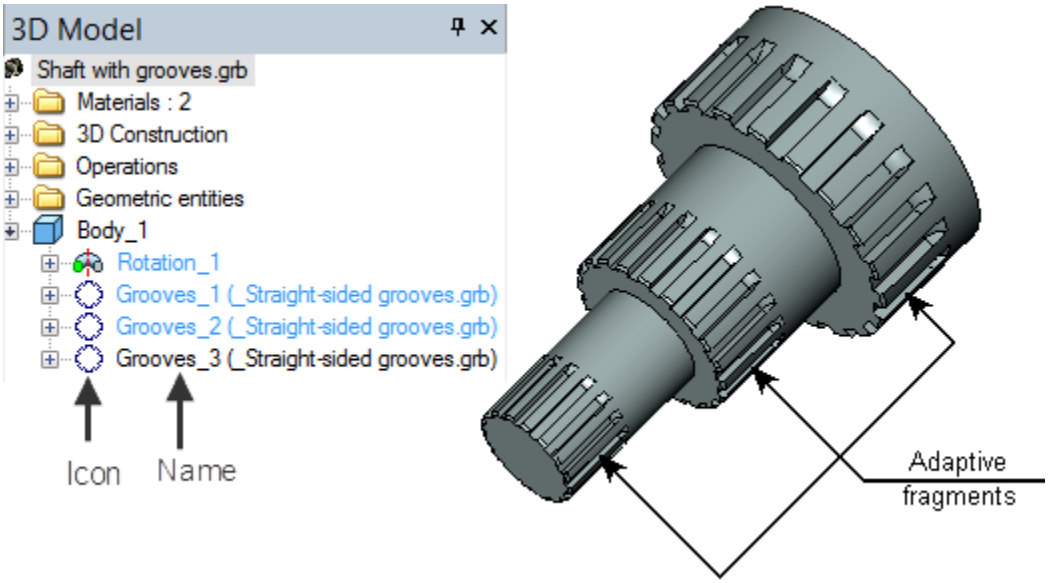
To make a 3D model suitable for use as an adaptive 3D fragment, a set of objects (adaptive elements) needs to be defined, whose geometry will be driven by the assembly objects.

To specify the list of geometrical parameters, use the command **3U: Set Adaptive Fragment Parameters**. The command is called as:

Icon	Ribbon
	3D Model → Advanced → Adaptive fragment
Keyboard	**Textual Menu**
<3U>	Tools > Adaptive Fragment

Upon calling the command, you can fill in the item “Name” in the properties window (optional). The entered string will be the name of the 3D fragment in the model. If the name is not specified, the default “Part_No” is used in the model. The “Comment” item may contain a text explaining the intended uses of the given 3D fragment.

If a fragment document is equipped with a slide, this slide will be used as the icon of the 3D fragment in the model tree of the assembly document. Icons are created in the command **Tools > Special Data > Preview**.

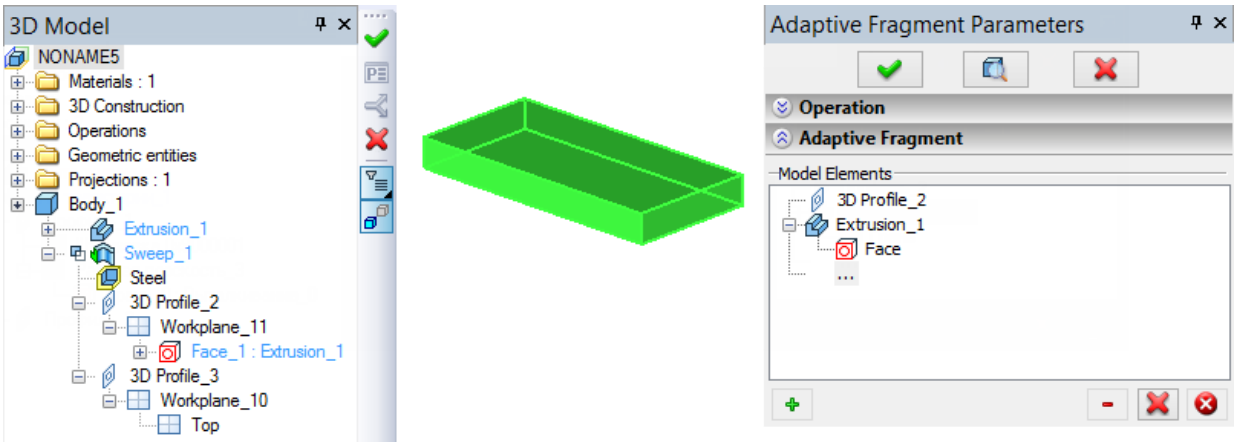


To add a new object to the list of adaptive elements, select it in the 3D view window or in the model tree. The selected element appears in the upper list, “Model Elements”, in the properties window. Upon selecting an object, the system automatically adds to the list the topological elements of that object (edges, vertices, faces) that are referenced by the object’s children.

> The “Model Elements” list provides an intuitive display of interrelations of the selected object with its children and adds a capability of selecting elements that are not at the top level of the model hierarchy as the geometrical parameters.

When creating or editing a 3D fragment in the assembly context (the “Top-down” design flow), the “Model Elements” list is built automatically. The system creates this list based on the objects that are referencing assembly model elements (3D profiles, workplanes, 3D nodes, etc.). If a 3D fragment is intended for insertion in other assembly models, the user can compose the list of external geometrical parameters by using the already built “Model Elements” list.

To give an example of composing a list of external geometrical parameters, let's consider a 3D fragment that defines creation of a stamped feature (see the diagram at the beginning of the chapter). To create a groove in the assembly, all that is necessary is just to specify the body face in which the groove is to be extruded, and the trajectory defining its contour. The first element in the “Model Elements” list will be the 3D profile serving as the base for creating the sweep. Let's designate the “Extrusion_1” as the second element in the list. In this way, the system adds to the list the face of this operation, since the face is the parent element for the “Sweep_2” operation.



The set of object types allowed for selection is defined by the option:

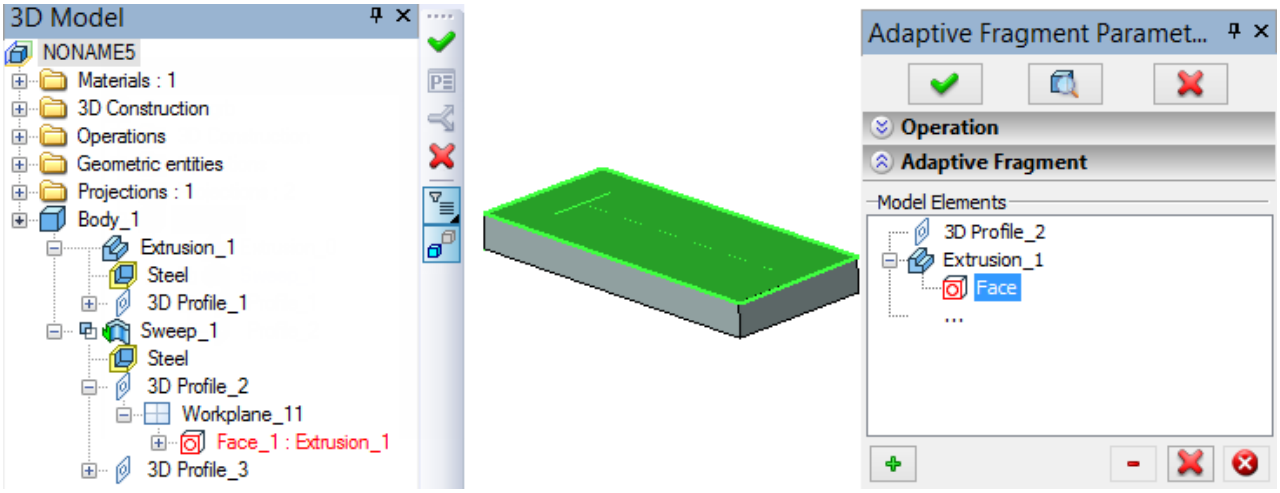
	<V>	Select type of parameter

The drop-down list will help setting up the internal command filters for selecting objects of a certain type: workplane, worksurface, coordinate system, operation, 3D path, 3D node, 3D profile.

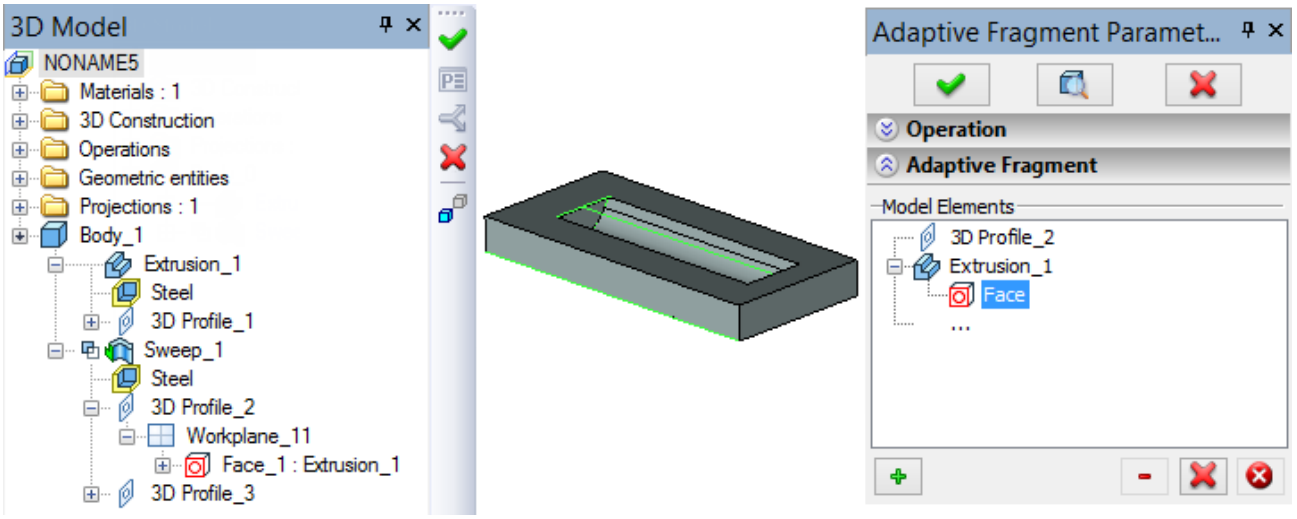
The default active option in the automenu is:

	<M>	Show parameter in Fragment document

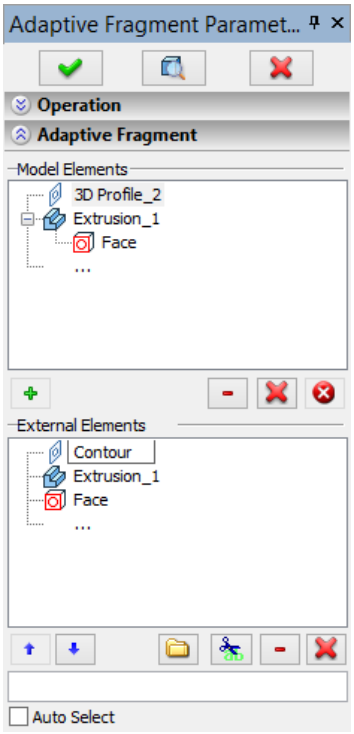
It enables highlighting of the objects that are not at the top of the model tree hierarchy. For example, if the given operation is active, then upon selecting a face of the operation “Extrusion_0” in the list of objects, this face will be highlighted in the model (at the same time, all children of “Extrusion_0” will be temporary removed from the 3D scene).



If the option is not active, then highlighting will only work if the current element in the list is a top-level operation. Therefore, in this particular example, the Extrusion_0 and its face will not be highlighted in the model.



The next required action is composing the "External Elements" list. This list gets the elements from the "Model Elements" list, that will be instantiated upon inserting the 3D fragment into an assembly model.

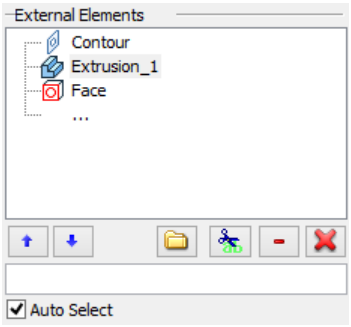


> We recommend putting in the list of external parameters all topological children of the model object, that were entered in the upper list, and instantiate them as values in the assembly.

Suppose, the external parameters list of an adaptive 3D fragment does not contain all topological children of the selected model object, or some of those objects are not instantiated upon inserting this fragment into an assembly. In this case, upon regenerating the 3D fragment model (at the time of inserting it in the assembly) errors will occur when creating the operations (or other objects) that are based on the uninstantiated topological elements.

To avoid a large number of unnecessary dependent elements in the list of geometrical parameters, prefer building the 3D fragment model in such a way that the model elements used as geometrical parameters are at the highest possible level in the model tree structure.

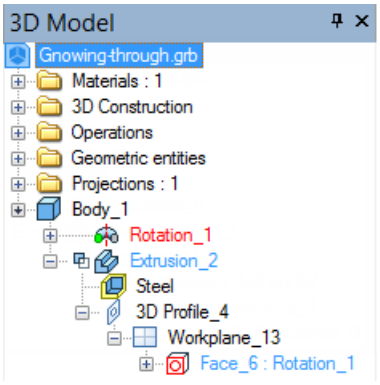
Move elements by selecting in the upper list and pressing the graphic button . You can enter a comment for any element in the list of external parameters. The entered string will pop up as a tooltip in the properties window when selecting the parameter instantiating value in the command **3F: Insert 3D Fragment**. If necessary, an external parameter can be renamed by editing the default name that is the same as the name of the model object. When selecting an element in the list of external parameters, the respective element is automatically highlighted in the list of model objects. This allows easily tracking the renamed model object that was used for creating an external parameter.

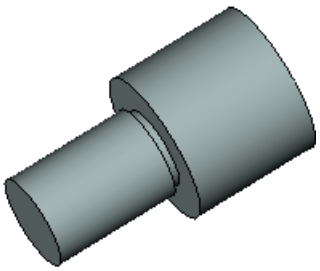


The "Auto Select" flag can be used for objects whose children are external parameters themselves. For instance, in the current example, "Auto Select" can be set for Extrusion_0 in order to reduce the number of items to select.

Example of creating adaptive 3D fragment

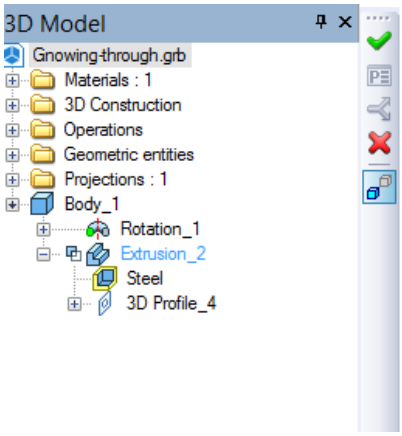
Let's review a simple example – a body in a 3D fragment file created from three operations: rotation, extrusion and a boolean operation. The "Extrusion_1" contour is constructed on a workplane that references an edge of "Rotation_0", extrusion direction – along the normal to the profile plane.

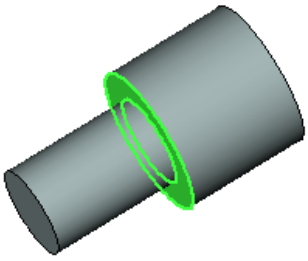


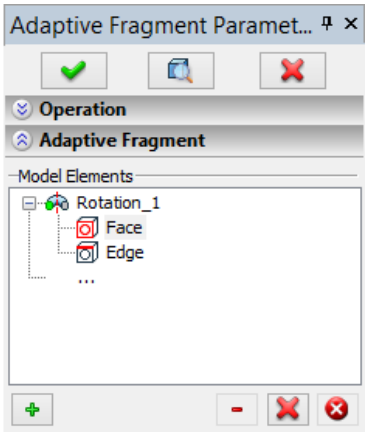


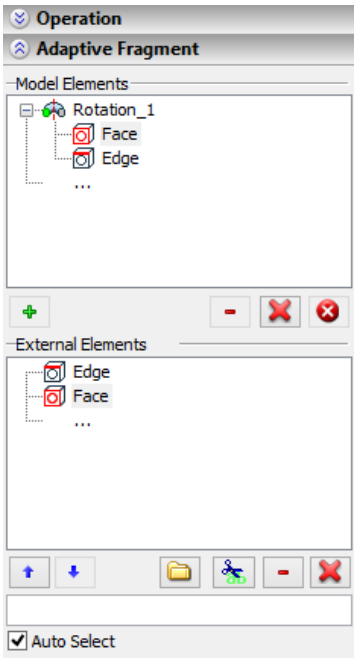
We need to create an adaptive 3D fragment that will be creating a grooving based on a round edge selected on any assembly operation.

To compose the "Model Elements" list, simply select "Rotation_0" in the tree.Its edge and face that are parents of "Extrusion_1" will be added to the list automatically. "Extrusion_1" in this case is the operation that is used for creating the grooving.









The next step will be creating the list of external parameters. It's enough just to move the edge and face into the lower list, while the operation on which the grooving is to be created will be determined by the system automatically. Fill in the comment entries for each element and set the "Auto Select" flag for the face. Let us remind that instantiating upon inserting a 3D fragment is done only for the objects that are entered in the list of external parameters. In a general case, the "Model Elements" list may contain the elements that are not used as the external parameters. If in the current example the edge would have not been moved to the list of external parameters, then it would not be possible to specify what particular edge of the assembly to use for creating the grooving. In other words, the parent would have not been defined for "Workplane_3" in the fragment and, therefore, the "Extrusion_1" operation would have not been created.

The contents of the adaptive elements list can be managed by graphic buttons in the properties window:

- for the "Model Elements" list:

  - **Delete.** Deleting current element;

  - **Delete all.** Deleting all elements;

  - **Delete unused.** Deleting unused elements. This button becomes accessible if the list contains objects deleted from the model. Such objects are marked in the list by the icon .

- for the "External Elements" list:

  - **Up/Down.** Moving current element around the list;

  - **New Folder.** Creating a new folder. The list of external parameters can be organized into a tree structure, by distributing the list elements over different levels or over different folders in one level. When inserting a 3D fragment into a model, its geometrical parameters will be displayed according to the defined structure;

  – **Rename.** Renaming selected external parameter. For each external parameter you can assign the meaningful name;

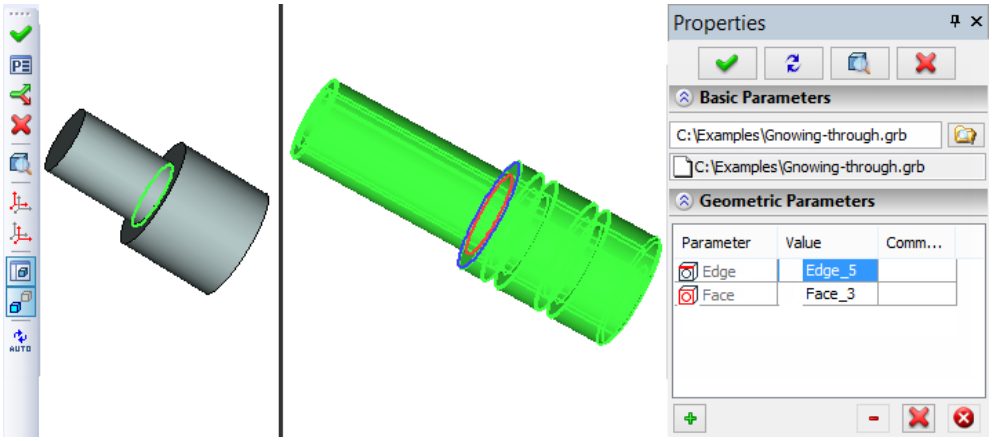
  - **Delete.** Deleting current element;

  - **Delete All.** Deleting all elements.

Elements of the external parameters list can be renamed. To do this, click at an element's name, and then edit the parameter's name.

Inserting Adaptive 3D Fragment, Instantiating Adaptive Elements

When invoking the command **3F: Insert 3D Fragment** for a 3D fragment with defined adaptive elements, the default focus is set on the "Value" entry of the first external parameter in the properties window, in the "Geometric Parameters" section. A bar below the list displays the comment to the current parameter. To define the value, in the 3D view window or in the model tree select an object of the same type as the external geometrical parameter of the 3D fragment.

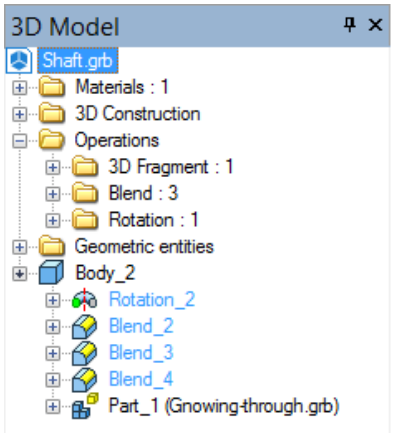


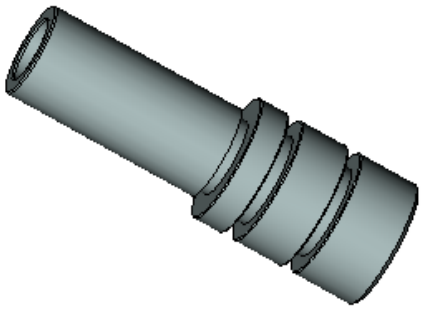
If the 3D fragment selected for inserting has the geometrical parameters preset the following option will be available in the auto menu:

	<R>	Show Geometric Parameters of Adaptive Fragment

It allows viewing the 3D model of the fragment being inserted in an additional window, with the current external parameter highlighted. When selecting an object, the system also picks the instantiating values for those parents of the current parameter that are flagged "Auto Select". The automatic instantiating can be altered by selecting a different model object.

Use the option to complete insertion of a 3D fragment. If errors are encountered in the 3D fragment's model due to the specified values of the geometrical parameters, then the 3D fragment will not appear in the scene and the appropriate messages will be output in the diagnostics window.





DEFINING ADAPTIVE ELEMENTS OF AN EXISTING FRAGMENT

The system supports defining the list of external parameters and their instantiating values for both inserting new fragments and editing existing fragments in an assembly. To add an additional parameter, call the command for editing a 3D fragment. In the "Geometric Parameters" section of the properties window, press the button and select the 3D fragment model's element to become an external parameter. The next step will be either instantiating its value or adding more parameters to the list. The difference of this approach from the steps described in the "Setting up adaptive 3D fragment" section is that such changes in the list are remembered only in the current assembly. When inserting this 3D fragment into another assembly model, the elements added to the list will not be available. Therefore, any 3D fragment of an assembly model can be converted to an adaptive 3D fragment, and the predefined list of geometrical parameters of an existing adaptive 3D fragment can be extended.

2D Projections. Creating 2D Drawings of the 3D Models

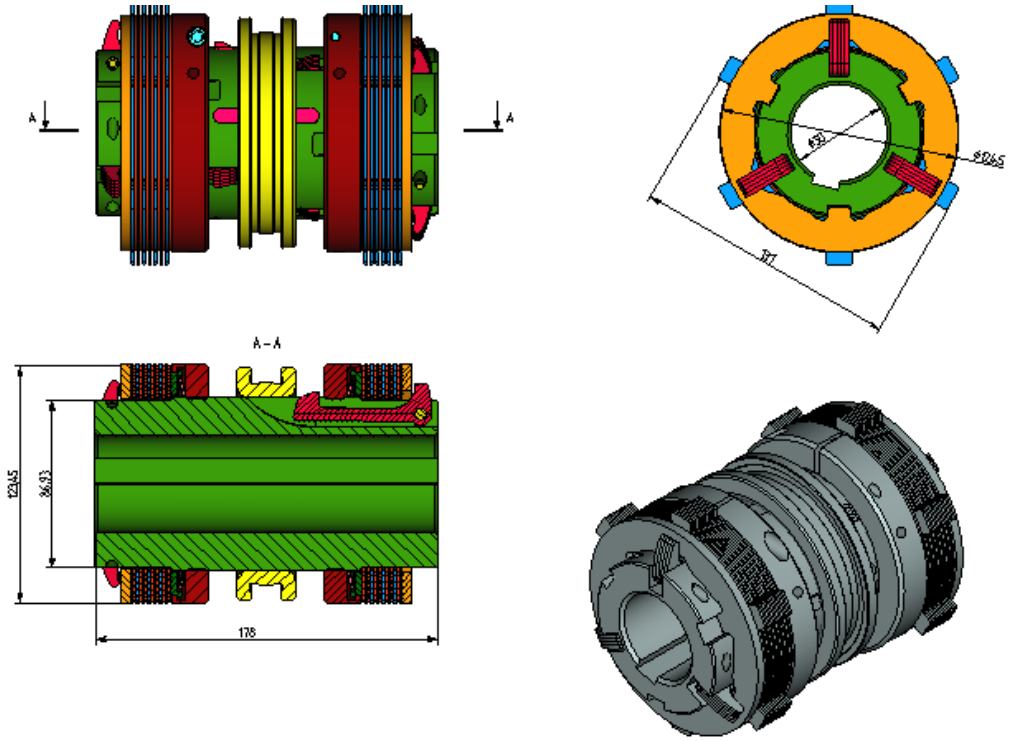
2D Projections. Creating 2D Drawings of 3D Models

A 2D projection is a two-dimensional image located in the 2D window, obtained by projecting a three-dimensional model or its selected portion on the specified plane. Projections can be used for creating the necessary views on a 2D drawing, with their subsequent decorating, as well as for obtaining local views and sections. One can project whole bodies or even groups of bodies, as well as selected elements of the model, such as faces or edges. If you are working with large 3D models, use the method of creating 2D projections in a separate file. Projecting is also actively used in three-dimensional modeling for drawing on active workplanes bound to the topology of three-dimensional objects, for example, to faces or edges.

Types of 2D Projections

T-FLEX CAD allows creating 2D projections of the following types:

- ✓ **Precise (Image Lines)**. The view of the projection of this type represents itself a collection of the graphic lines;
- ✓ **Vector Picture**. The view created on the projection of this type entirely corresponds to the type "Precise (Image Lines)". However, in this case the graphic lines themselves are not created, and the projection is drawn as a picture. This mode allows a user to reduce the 2D projection generation time, and also reduce the memory usage required for creating a large number of graphic lines;
- ✓ **Shading**. The view of the projection is drawn in the form of shading which corresponds to the mode of the same name in the 3D window;
- ✓ **Rendering**. The view of the projection is drawn in the form of rendering which corresponds to the mode of the same name in the 3D window;
- ✓ **Wireframe**. The view of the projection is drawn as a wireframe which corresponds to the mode of the same name in the 3D window. In this mode, the parameter "Remove Hidden Lines" is available depending on which the removal of the hidden lines is carried out or not.



Projections of the types “Precise (Image Lines)”, “Shading”, “Rendering”, “Wireframe” have the following limitations:

- it is not possible to hide (remove) separate elements (lines) of the projections or modify their parameters;
- it is not possible to project 3D construction elements (3D paths, 3D profiles). Only bodies (operations) can be selected for projection.

All types of 2D projections support the creation of the service elements based on the projection types. On the projection of the type “Precise (Image Lines)”, the service elements are attached to the projection lines, for the rest of the projection types – the service elements are attached directly to the elements of the 3D model.

Creating projections of all types is carried out similarly.

CREATING 2D PROJECTION

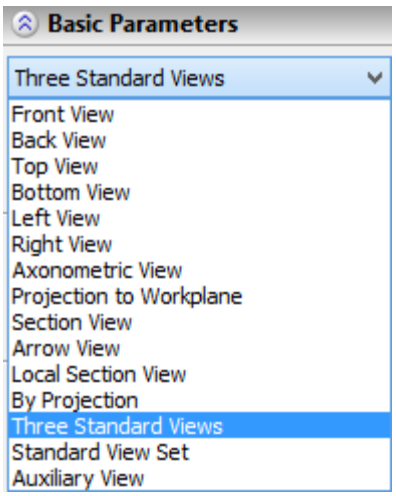
To create a 2D projection, use the command **3J: Create 2D Projection**. Call the command as follows:

Icon	Ribbon
	Draw → 2D Projection
Keyboard	Textual Menu
<3J >	Draw >2D Projection

Upon calling the command, the following actions are available:

	<1>	Create three standard projections
	<2>	Create section view
	<3>	Create additional view
	<4>	Create local section view
	<5>	Create set of standard projections
	<6>	Create standard projection
	<7>	Workplane projection
	<8>	Projection by Projection
	<9>	Arbitrary View
	<F>	Select File with 3D Model to create Projection

Before you begin creating drawings using projections, it is recommended that you determine the format of the drawing, how many pages it will occupy and what will be the scale on each page. All of these parameters are defined on the **Page** tab in the command **Customize > Document parameters**. Each page uses its own settings. The settings can be modified at any time.



You can also select the type of the projection to create in a combo box provided in the properties window

The general algorithm of creating a 2D projection includes several required and optional steps:

Define the projection direction (required) by one of the following ways:

- Using one of the options for creating standard projections (the options , ,).

- Automatically by the system, depending on the section line position, in the case of creating a section or local section view (the option).
- From the direction of the detail view specified by the detailing element (the option).
- Same as the direction of the original projection, when creating a local area view ( ).
- By a workplane (the option).
- By 3D model elements, such as a flat face, using the set of additional options.
- By defining the directional vector of projecting and the rotation angle about this vector in the projection parameters.
- Along the projecting direction of an existing 2D projection (the option).

In most cases, the system attempts to determine the type of the projection being created by the first selected element. For example, if a section is picked, the system will offer creating a cut; picking a hatch prompts a local area view; picking a face – a general view, etc.

Define the attachment point in the 2D window (optional). Normally, you need to specify the location on the drawing via a 2D node or free point at which the projection is to be created. In this case, the projection to be created is shown as a box indicating the projection extents.

Select section to be used on the projection (optional). This step is necessary when creating a local view or section.

Select bodies to subject to the section (optional). This step is necessary when creating a local view or section affecting only selected elements of the model being projected.

Select model elements to project (optional). By default, the whole 3D scene is projected in its current state; however, it is also possible to project separate Bodies, operations or specific topology elements (faces, edges).

Define projection parameters (optional). Some properties, such as scale, can be specified in the "2D Projection Parameters" dialog box or in the properties window.

Select the type of 2D projection being created: "Precise (Image Lines)", "Vector Picture", "Shading", "Rendering", "Wireframe";

Confirm the input (required). As in any 3D command, the element creation must be confirmed upon competing definition of all parameters. This step is possible, when the option becomes accessible in the automenu:

	<Y>	Finish input

Creating Standard Views

The simplest way of quickly obtaining 3D model drawings is creating one of several standard projections (which is often sufficient).

Creating three standard projections

For convenience, the capability of creating the three most common used standard views is separated in the special command, providing for the Elevation ("Front") View, Plan ("Top") View and Side ("Left") View:

	<1>	Create three standard projections

Upon selecting the option, the automenu offers the group of commands for creating the set of standard projections:

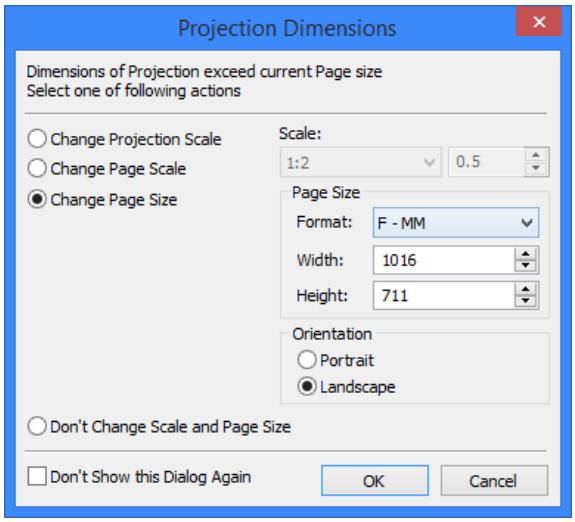
	<M>	Change projection placement
	<K>	Set/Break projection link with Front view
	<E>	Select model elements to project

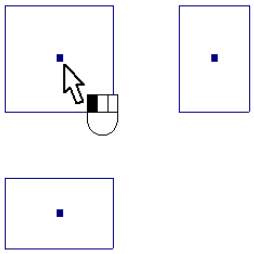
When creating standard views, the automenu has several additional options common for all types of projection (options for creating broken views and handling sections). Working with those will be described separately.

If overall size of a projection exceeds the page size, system will open the dialog box with options for correcting inconsistency between the projection and the page. The dialog box offers the following options:

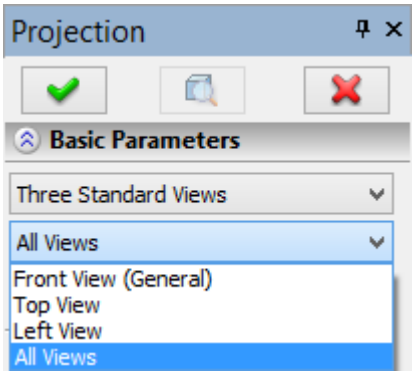
- Change projection scale;
- Change page scale;
- Change page size;
- Don't change Scale and Page Size.

All correctional parameters are calculated automatically. User can agree with them or set custom values.



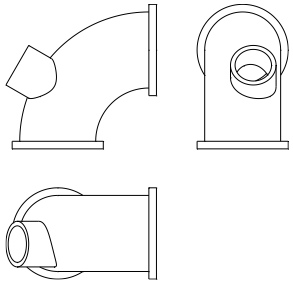


Next, you need to position the projections on the drawing. Three blue frames appear in the 2D window, attached to the pointer. These frames mark the extents of the projections being created. Currently, all three projections are selected. Position a particular view, selected first. View selection is done by clicking once within the blue box (the second click activates the option of positioning the view) or by selecting an item in the combo box in the properties window. After selecting a projection, you need to select the fixing point in the 2D view. If the view positioning option is not yet active, engage it in the automenu. Click, and the projection will be attached to the position of the pointer. The option of positioning a view allows fixing a projection to an unconstrained point (right under the click) or to a 2D node.

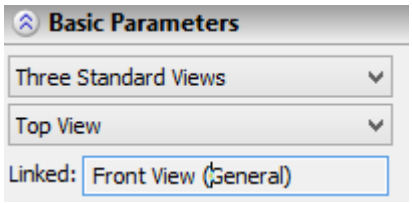


> The mechanism of positioning a projection on the drawing is common to all types of projections; therefore, we will skip its detailed description in the future.

If the front (main) view is being positioned, the rest of the views maintain their position with respect to the main one. When adjacent views are positioned, the projective relation is maintained with respect to the main view. The projection centerpoint snaps to the attachment point. To assign a specific 3D model element as the attachment point, such as a 3D node or vertex, you need to use additional options for creating the projection that are made available by the option (see the section “General case of creating projection”).



By default, the adjacent views maintain the projective relation with the main view. This relation can be broken by the option . The property of being related to the main view is indicated by a special item in the properties window.



Identical parameters are set for all views created simultaneously. In the case when different settings are required for a particular view, select it in the properties window. After that, you can define the specific elements to be projected on this view or select a 3D section to be used on the view. In some cases, this approach to creating views can save time.

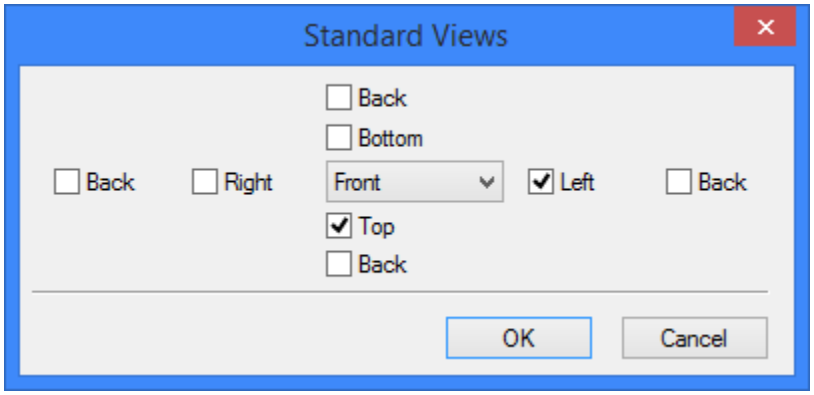
If necessary, one can select model elements to project using the option . Selection of bodies for projecting is described in details below, as this command is common for all types of projections (see the section “Selecting bodies for projecting”).

Creating the set of standard views

This command is different from the previous one in that you can define an arbitrary number of standard views. All other steps are identical. Create a set of standard projections using the option:

	<5>	Create set of standard projections

Upon calling the option, a dialog box appears for defining the projections to obtain.

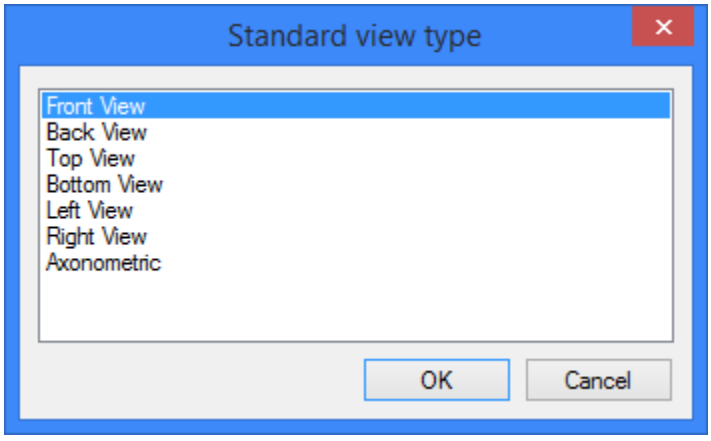


Creating one standard view

Create the one standard view using the option:

	<6>	Create standard projection

In the list of the standard views, select the one to be created. The type "Axonometric View" is added to the list of the standard views. Elements of this list are also present in the properties window, in the overall list of the 2D projection types.



As usual, first, position the projection on the drawing – select the attachment point. In this case, you can attach the view being created to an existing 2D projection. You can define the relation between the view and another projection using the option . Once the option is engaged, select one of the projections.

Such a relation allows creating a "standard" view with respect to any arbitrary projection selected as the main view, that is, the "Front View". The relation can be broken by the option .

After attaching the view to the drawing, you can select elements to be projected using the option .

Creating Additional View

The directional view element is often used for decorating drawings. It can also be used for creating a projection. In this case, the projecting direction will be determined by the arrow and the projection nearest to the arrow. The mode of creating an additional view is turned on by the option:

	<3>	Create additional view

Next, select a directional view element. The nearest to the arrow projection will then be highlighted. The system will use that projection as the main view. The additional view will be constructed relative to the main view. By default, the additional view maintains the relation with the main view. This relation can be broken using the option:

	<K>	Break link of Projection with Front View

The final steps are positioning the projection on the drawing and, if necessary, selecting the model elements to be projected.

Creating Local View or Section

Before beginning with creating a local view or section, have certain elements available. First, you should create in advance either a 3D section (the command **3SE:Construct Section**), or a 2D section on the drawing (the command **SE: Create Section**). Note that the latter element should be constructed on one of the existing projections. Only in this case it can be used for creating sections.

The mode of creating a local view or section is set using the option:

	<2>	Create section view

Next, you need to select a 2D section on the 2D drawing, or a 3D section. The normal to the section plane defines the projection direction. The elements are selected using the options:

	<S>	Create 2D Projection based on 3D Section

	<L>	Select section defining projection

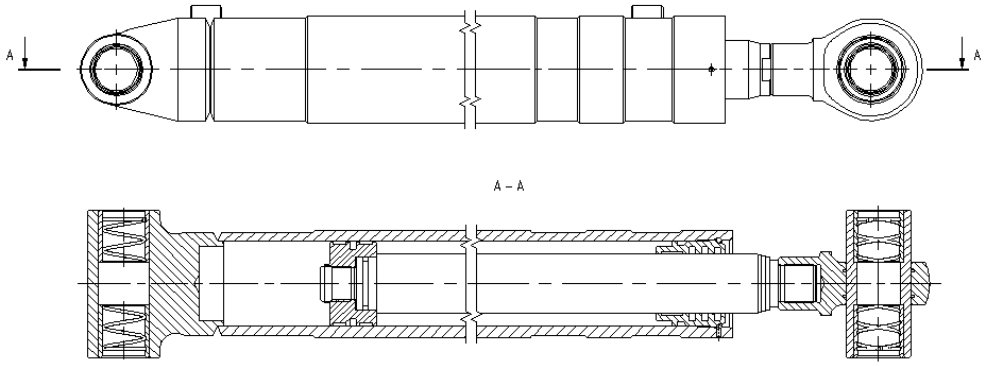
Next, define the projection attachment point in the 2D view using the appropriate options.

Breaking or reinstating the relation between the projection being created and the specified section is done using the option:

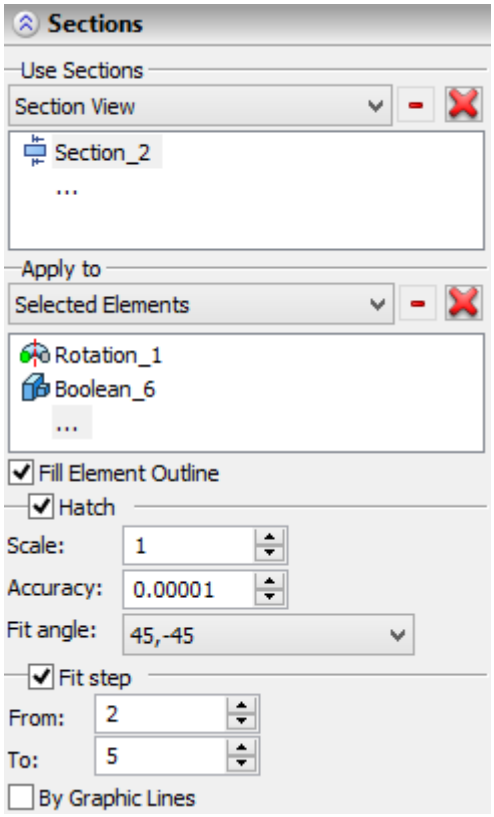
	<K>	Set/Break link with projection

After that, you can select elements for projecting and elements to be sectioned. Selection of elements for projecting is done in a conventional way – by the option . By default, the section is applied to all bodies selected for projecting. However, this is not always required. There is a provision for specifying the list of operations or Bodies to be subjected to the section.

When creating a local view or section on the basis of 2D section view, the presence of a gap on the initial projection is taken into consideration.



The parameters for controlling the sections that are in use are joined in one group and positioned on the tab “Sections” of the properties window. The used sections are displayed in the list “Use Sections”. This list displays the names of 3D sections or IDs of section notations. To select an additional 3D section, place the pointer at an empty line marked by an ellipsis, and then select the desired element in the model space or in the “3D model” window. Alternatively, a section can be selected by the automenu option:



	<Ctrl+S>	Select 3D Sections to apply

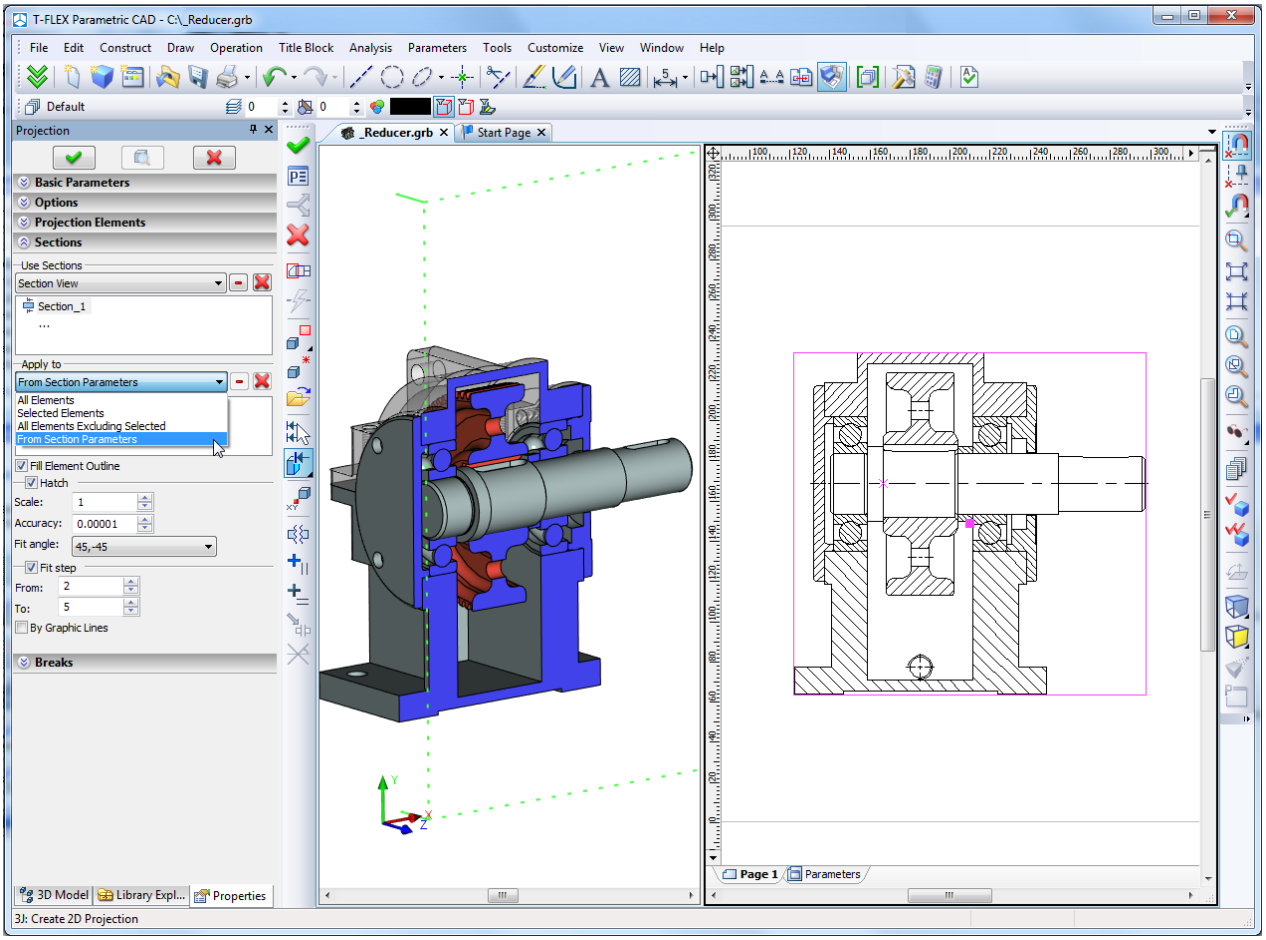
The type of the projection created by using sections can be selected from the combo box: Section View, Only Section Planes, Unfold/Radial Section and Section with Unfolding. Unfolding can be done only if a composite section was used (a one containing multiple segments).

Each of the sections can be applied to all objects being projected at once, or just to the specifically selected objects among those being projected. The list of operations/Bodies subject to the selected section is displayed in the “Use Sections” list. To select elements, use the list in the properties window or the automenu option:

	<Ctrl+O>	Select Operation to which 3D Section will be applied

This option allows selecting not only operations, but also Bodies.

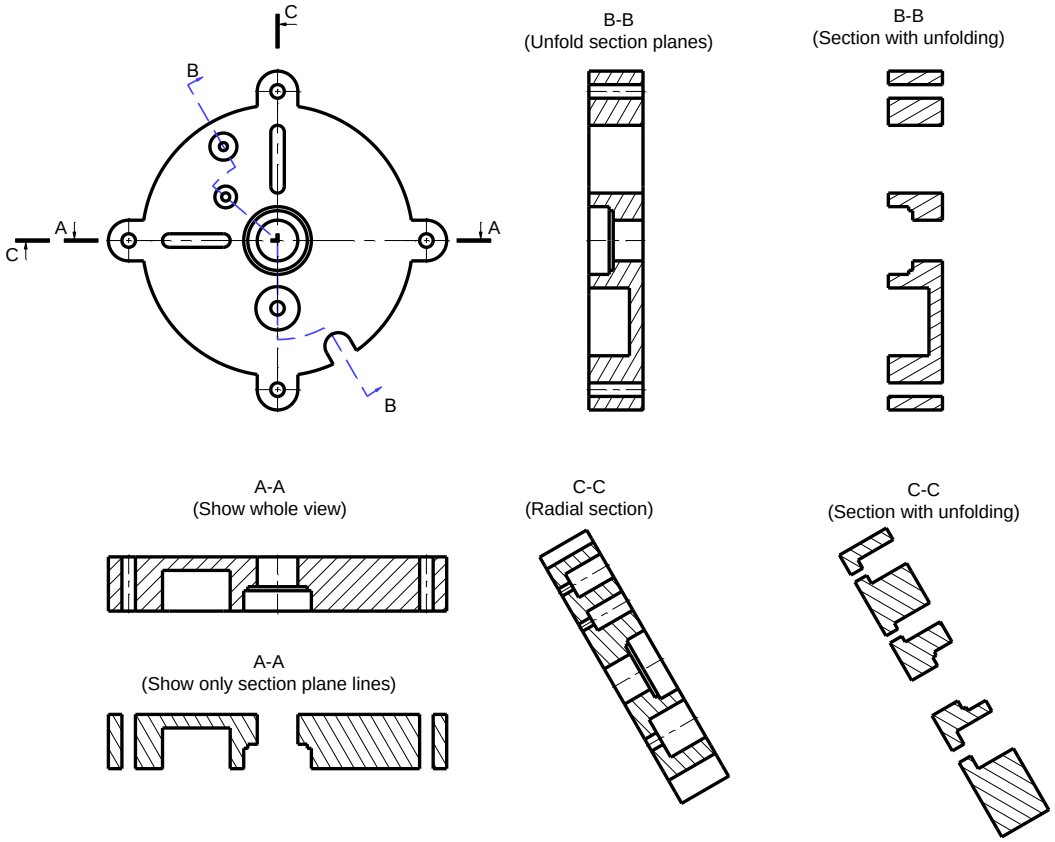
The way of using the list of bodies can be set in the combo box. The available options are “Selected elements”, “All elements excluding selected”, “All elements”, “From section parameters”. The “All elements” option is set by default in the used if the elements list is empty. When choosing the “From section parameters” option the section application rules are taken from the 3D section properties.



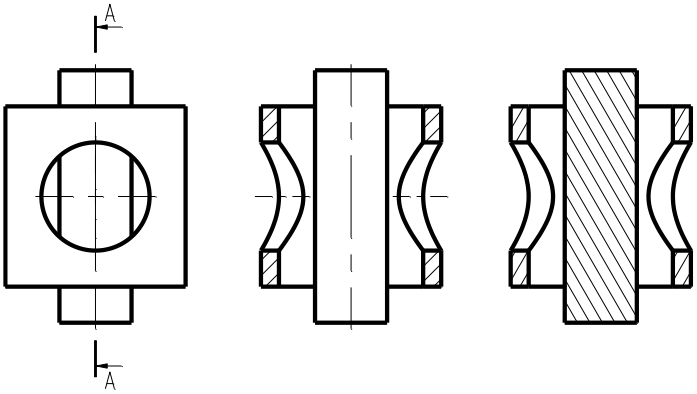
It is possible to specify beforehand that a 3D fragment will NOT be cut by a section when creating a section view of an assembly. This option can be set in **ST: Set Document Parameters** command and is usually used in the libraries of standard parts.

The option “Unfold section planes/Radial section” can work in one of two modes. This depends on what type of multiple-point section is used in the case. If a multiple-point section is composed of all straight segments, the whole section will be fully unfolded. If at least one segment of a multiple-point section is a circular arc, then the radial section mode turns on. In this case, all arcs are skipped when unfolding the section.

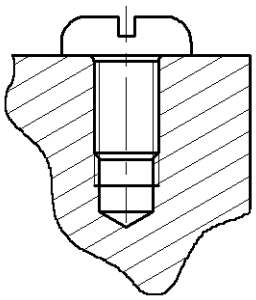
The option “Section with unfolding” always creates full section unfolding, even when the section contains circular arcs.



In the following example, two sections are created, both based on the main view. In the first case, the section was applied to the shroud only, while in the second case – to all bodies.

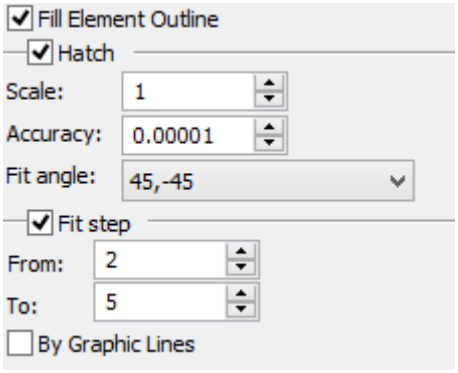


The following additional settings are provided in the sections handling group of parameters:



Fill Element Outline. This setting helps correctly draw threaded connections in which elements penetrate each other (due to the specifics of this type of connection). Additionally, some elements of the threaded connection (for example, the standard elements) can be exempted from section.

Hatch. This parameter defines whether to hatch the section or local view. The precision of defining the hatch contour can be set in the provided box.



Scale. The hatch scale

Accuracy. This is the accuracy of tracing the hatch contour. Adjusting this parameter helps in complicated cases, when the system cannot hatch the contour under the previously specified accuracy.

Fit angle. Sets the choice of automatic hatch angle definition for crosscut parts. The combo box provides two choices: «45, -45» only and «30, 45, 60, -30, -45, -60»

Fit step. Sets the upper and lower limits on the values, within which the system automatically picks up a hatch step. The step size is determined relative to the area of the hatch contour.

By Graphic Lines. *This option is available only for projection of the type "Precise (Image Lines")*. When this flag is on, the hatch on cuts/sections is formed by graphic lines of the projection, when it is off – by faces which appear on the projected body as a result of applying 3D section.

By default this flag is turned off, i.e., the hatches are created by the faces of the section (since the graphic lines on the projection do not always form a closed contour due to loss of accuracy while projecting).

It is recommended to turn on the flag "By graphic lines" if the hatch being created will be used later for construction of 3D profile or local section view.

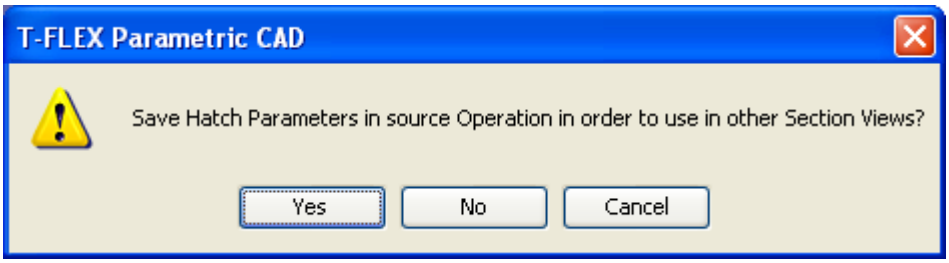
Managing hatch properties in cuts and sections

The drawing standards require that all cut or sectioned parts are hatched in a certain way. If the material the part is made of requires hatching by a special pattern, then the appropriate «pattern» hatch type must be set in the material properties. In all other cases, the material is hatched in the sectioned areas by the repeating lines with a certain slope and step. If multiple bodies are involved in a section, then the hatch slope and step must differ across them. When creating a cut or sectioned for the first time, the system automatically picks different hatch properties (slope angle and step) for different parts. Note also that the hatch parameters that were once picked for a certain part (operation) are automatically

remembered in the model, and the same hatch parameters will be used for the same part in any cuts or sections on other views.

> Those hatch properties are stored together with the original operation and are propagated on child operations, whenever those have the flag set, «Attributes from source operation».

Nevertheless, the user can modify hatch properties in a cut at any time by editing its parameters. Upon confirming changes to hatch properties, the system will query you, offering two choices for the continuation.



If the user responds «Yes», then the specified new parameters of the hatch slope angle and step will be saved in the model and will later be used for this part in all cuts and sections in which the part is involved. If the user responds «No», then the hatch will assume individual settings and will not depend on the operation properties in the future.

> To revert the individually specified hatch properties back to the properties assumed from the 2D projection, one would need to right-click this hatch and in the coming up context menu select the item «Properties from Projection».

Another useful item in the context menu of a cut hatch is **Do not apply section**. Upon calling this command, the operation for which the cut hatch was created will be added to the list of the operations that shall not be subjected to sectioning in the given projection. After that, the projection will be automatically rebuilt.

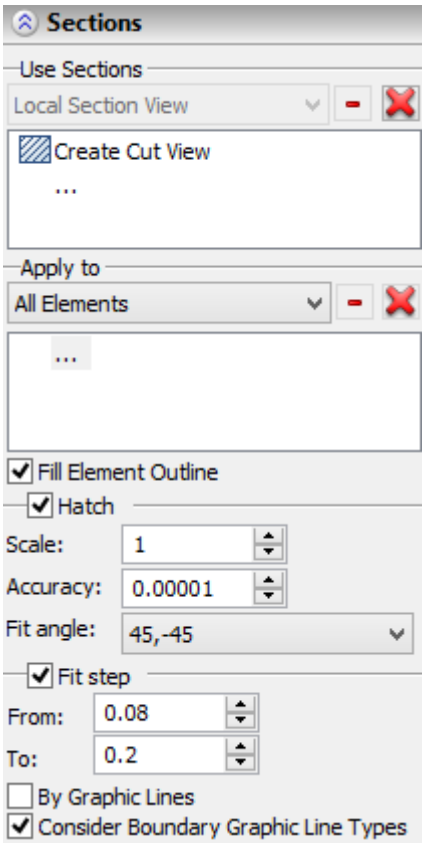
Creating Local Section View

This command allows creating local section views on an existing projection:

	<4>	Create local section view

The boundaries of the local section view are defined by the hatch contour. The latter must be created before creating the local section view. To select the hatch, turn on the option:

	<H>	Select Hatch as boundary of local section view



Upon completing creation of the local section, the hatch becomes invisible, while screening out a portion of the main projection. The lines of the local section will be drawn with higher priority, than those of the hatch and the main projection.

Geometry of the graphic lines bounding the contour of the original hatch can be taken into consideration when defining the boundaries of the local section view. For example, if the hatch contour pattern was set as waves, the boundaries of the local section view will also have waves pattern. A user can toggle on/off the lines type switch found in the parameters of the local section view.

Next, define the position of the section plane. The plane normal automatically coincides with the viewing direction of the main projection – the one used for creating the local section view. Thereafter, position the view plane either by a 3D node or by a 2D node in another projection. For example, if you are creating a local section on the Left View, then the 2D node can be selected in one of the adjacent standard views: Front, Top, Bottom, Back.

The 2D or 3D node can be selected using the options:

	<3>	Select 3D Point, defining section position
	<2>	Select 2D Node on other Projection, defining section position

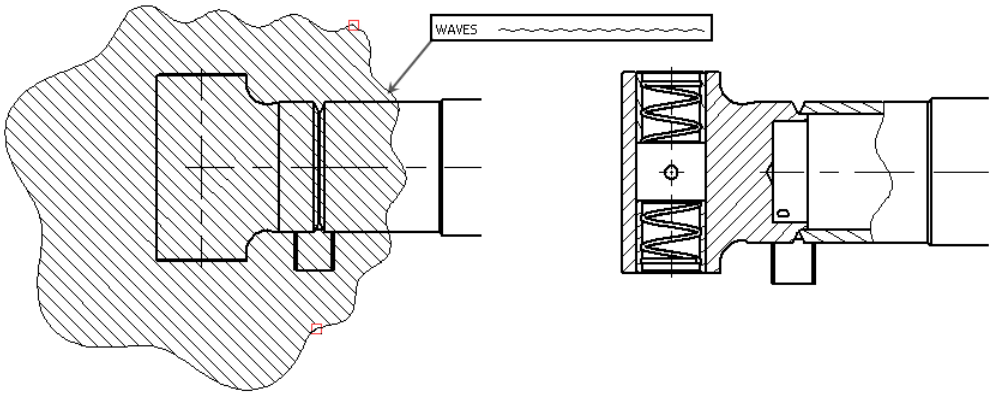
The local section view is automatically tied to the projection on which it was created. However, this tie needs to be broken in some situations, in order to attach the local view to the drawing at a different location. The option to break the tie is:

	<K>	Break link of Projection with Front View

Once the tie is broken, the following option is available for attaching the projection to the drawing (an unconstrained attachment or snapping to a 2D node).

	<M>	Click to select 2D Projection attachment Point

When creating a local section view, we recommend selecting specific model elements to be projected. Selecting the elements that actually fall in the area of the local section view speeds up regeneration of the projection.



Creating Projection on a Workplane

The projection on a workplane is used when working with active workplanes. For example, when creating a new workplane based on a flat face, the selected face can be automatically projected on that workplane. The command of constructing the projection on an active workplane can be called by the icon , located on the main toolbar in the button sets “Workplane” and “Workplane (Sketch)”.

If the workplane is present in the 2D view, sometimes it is convenient to use that workplane for projecting model elements. The following the option is provided for this purpose:

	<7>	Workplane projection

The workplane can be selected either in the 3D window or on the 2D view. The projection will be created in the same page of the drawing as the selected workplane. You do not have to explicitly position the projection on the drawing. The position of the projection is determined automatically, since each workplane has its specific position in the 3D space. The objects are projected on the workplane along its normal direction.

Projecting 3D point on workplane

> The capability of creating a 2D node on a workplane as a projection of the 3D point is not directly related to the 2D projection construction command, nor it has to the T-FLEX CAD “2D projection” element.

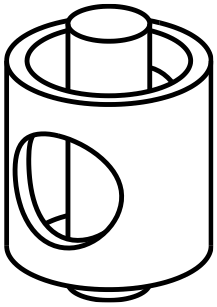
Creating a 2D node as a projection of a 3D point is used when working with active workplanes, often together with the command for constructing a 2D projection on a workplane the command for creating 3D nodes - projections of 3D point saves computational resources of the computer when it is not necessary to create a conventional path 2D projection.

The icon that launches the command of constructing a 2D projection node is located on the main toolbar (in the sets “Workplane” and “Workplane (Sketch)”) and is available only when working with the active workplane. To create a projection node, select a 3D point in the 3D window.

General Case of Creating 2D Projection

The most general approach to creating a 2D projection is supported by the "Arbitrary View" item. Upon turning on the option, the following actions are available:

	<M>	Click to select 2D Projection attachment Point
	<F>	Select Plane to set view direction
	<N>	Select point for superposing with the attachement point
	<K>	Cancel setting of view direction
	<S>	Select 3D Sections to apply
	<E>	Select model elements to project



First, define the position of the projection on the drawing – select the attachment point.

The axonometric projection of the whole scene is created by default in the world coordinates. The projection of the world coordinates origin is snapped to the point on the 2D view selected at the previous step. A 3D point can be selected to define the point in the space for aligning with the fixing point on the projection.

The projecting direction can be changed by one of the following means:

- Select projection plane. The projecting direction will be defined by the normal to this plane. To select a plane, use the option:

	<F>	Select Plane to set view direction

- Define the viewing point in the properties window. It can be defined using coordinates or by selecting one of the standard or custom views.

The coordinates of the view direction can be defined in the properties window. The X, Y and Z are entered in the respective input boxes of the end point of the direction vector. The start point of the vector is the origin of the world coordinate system. A user-defined custom view can be loaded with the help of the button "Load" . A custom view can be saved via the command "**3VP: Set View Parameters**".

Angle is the parameter defining the rotation angle of the 2D projection on the 2D view about the fixing point. The positive values define rotation in the counterclockwise direction.

Defining the coordinates and the rotation angle is possible only after creating an arbitrary view. When creating other types of view with projective relation, such as standard or section view, the coordinates are defined by the system, and so these fields may be blocked.

Creating a 2D projection by an existing projection

To create such a projection, the option is used. Upon selecting this mode, the following options become available in the automenu:

	<V>	Select source Projection to specify Parameters
	<E>	Select model elements to project
	<A>	Select all Solids in current scene
	<F>	Select File with 3D Model to create Projection
	<Ctrl+S>	Select 3D Sections to apply
	<Ctrl+O>	Select Operation to apply 3D Section

First, the source 2D projection is to be specified in order to use its properties for the projection being created. The new 2D projection will match the source one in its position, orientation, and the use of broken views. The rest of the projection parameters can be edited: you can change the list of the objects to be projected and the applied sections, the projecting (mapping) method, available line styles, and hatch parameters.

The newly created 2D projection is automatically assigned the priority which is by five points greater than the source projection. In addition, the flag “Create Outlined Area” is set in the properties of the new 2D projection (see the section “2D projection parameters”).

This method of creating a 2D projection can be used to construct projections of threaded connections where both the “screw” and the “nut” of the connected pair are sectioned. The thread lines of both the “screw” and the "nut" will be overlapped on the projections in which both parts are simultaneously sectioned. To insure the correct appearance of a threaded connection, what one can do is originally create the projection of the sectioned “nut”, and then create another 2D projection based on the first one, where you specify the “screw” only in the “to be projected” list.

Selecting Elements for Projecting

By default, that is, when nothing was selected, the whole scene is assumed selected for projecting. The option “Full scene” projects all operations visible in the scene at the time of regeneration, regardless of the number of the operations that were in existence at the time of creating the projection. New 3D elements cannot be created based on such projections (due to possible topology change or recursion), as prompted by the system warning at the time of creating the projection.

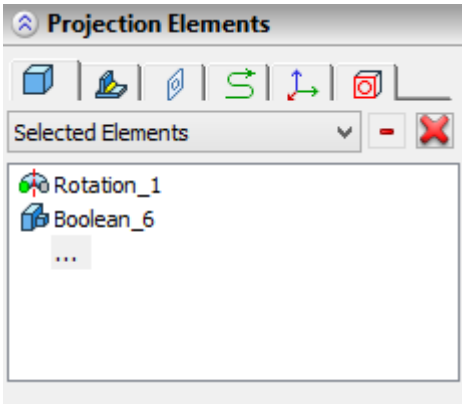
The elements need to be selected in the cases, when only specific elements need to be projected, rather than the whole model (particular operations, Bodies or construction elements/topology elements), or if

the projection is intended for creating 3D elements. Besides, often happens, that some model objects are simply not visible on the projection because those were screened out by other bodies. For example, some parts may be inside a shroud. In this case, it is useful to exclude from projecting the objects that are not visible anyway. This will benefit in saving regeneration time of such a projection.

To select specific model elements, use the option provided in any mode of creating projections:

	<E>	Select model elements to project

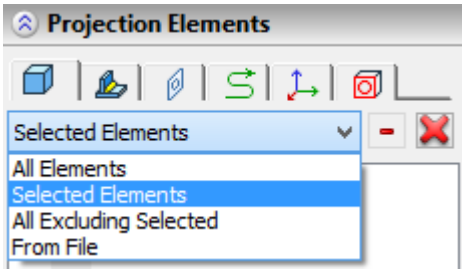
This option has a pulldown list of filters for selecting the desired objects. Allowed for selection are Body, operations, 3D profiles, 3D paths, faces, edges and loops. The set to filter options for selecting objects is duplicated on the system toolbar.



When activating the option the group **Projection Elements** automatically opens in the properties window. This group contains a list into which selected elements are put. The list is subdivided into separate tabs according to the type of selected objects – "Bodies and operations", "Welded joints", "Profiles", "Paths", "Coordinate systems", "Geometric elements".

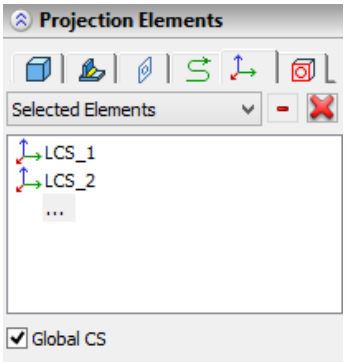
It is also possible to activate the mode of selecting the elements for projection with the help of dialog of properties window. For that, it is sufficient to place the cursor in a free line of the list marked by dots and the option will be turned on automatically.

For each type of the selected elements the user can specify the method of projecting:



- *All elements* – all elements of the given type in the scene will be projected;
- *Selected elements* – only selected elements of the given type will be projected;
- *All except selected* – all elements of the given type that has not been selected will be projected;
- *From file (only for Bodies and operations)* – this option is set if projected operations and Bodies are selected from external file (see Section "Construction of 2D projection from model in external file"). When selecting this method, the option of selecting a file is turned on automatically in the command's automenu
- *Same as in operation (only for welded joints)* – method of projecting welded joints will depend on the method set for operations and Bodies. If "All elements" is set for operations, all welded joints will be projected. If any other method is set for operations, then welded joints are not projected.

By default, for Bodies and operations the mode "All elements" is set, whereas for profiles, paths and coordinate systems – mode "Selected elements". For geometric elements (faces, edges, loops) the method of projection is not specified (only selected elements are always projected).



Projection of coordinate systems has certain specific features. The coordinate systems are projected on top of the entire 3D model without taking account of visibility. Projection lines of the coordinate systems are ignored when determining the boundaries of the projection.

Projections of the coordinate system are formed from three graphic lines with arrows. The color of each line-axis coincides with the color of the corresponding axis of the global coordinate system in the 3D scene. The style of the lines can be modified as for the standard graphic lines of the drawing.

> The length of the coordinate system axes is controlled by the parameter **Size** > **Coordinate System** in the dialog of the command **ST: Set document parameters**, tab "3D".

Projection of global coordinate system is specified separately with the help of **Global CS**. Selected method for projection of LCS does not affect projection of global coordinate system.

The automenu provides a separate option for quick selection of all operations currently present in the 3D scene:

	<A>	Select all Solids in current scene

The difference between the behaviors of the option "Select all Solids in current scene" and that of the mode of projecting the whole scene can be described as follows. The list of all operations selected in the 3D scene is formed at the time of using the option and stays unchanged unless intentionally modified. Therefore, the new bodies added to the scene do not appear in the list of the elements to be projected.

Constructing a 2D Projection Using a Model from an External File

The mode of constructing a projection by a model from an external file is useful when working with complex 3D models and assemblies, in which cases you can significantly save the computational resources by avoiding the 3D model recalculation. When creating a 2D projection, the system takes from the external file only geometrical data saved in this file. It is also possible, if necessary, to compose a single drawing from different 3D models (different versions, individual fragments, etc.).

The option to select an external file to construct a projection is available in the command's automenu:

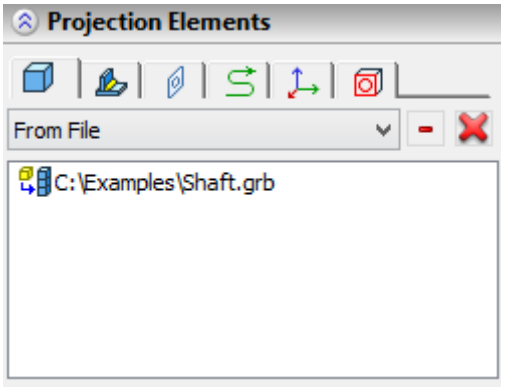
	<F>	Select File with 3D Model to create Projection

If the user calls the command to create a 2D projection, but no 3D model is present in the current document, then at the beginning this will be the only option in the automenu. Upon calling this option, the standard dialog window to select the file will appear. Once you selected a file with the 3D model, all the rest of the options to construct the projection would become available in the automenu. After that,

the projection is constructed as usual. At the same time, there are restrictions on selecting elements to map (project). In this case, it is always the entire 3D model contained in the selected file that will be mapped (projected).

If it is necessary to create several projections based on an external file, you do not have to select it over each time. When constructing related views (a related standard view, cuts and sections, arrow views), the data about the 3D model is automatically accessed from the initial projection.

The information about the used external file is recorded in the 3D model tree next to the 2D projection name. If you right-click on the 2D projection, then you will see in the context menu the command to open the file, which it uses. The name of the file used to construct the 2D projection can be found on the "Elements" tab of the «2D Projection Parameters» window.



The mode of constructing a projection based on the model from external file can be also activated by setting the method of projection “From file” in the group of the properties window “Elements” for Bodies and operations. Simultaneously, the option will be activated automatically.

Creating Broken View on Projection

When creating long parts, it is often necessary to limit the part view to a certain portion to keep the drawing compact. T-FLEX CAD provides a tool for creating projections with broken view of the three-dimensional parts. The broken view can be created for all types of projections, except for the local section view and the unfolding section view.

Pay attention to the following details when working with broken views:

- When attaching a radial dimension or both legs of a linear dimension to lines of such projection or their children, the system automatically adjusts the dimension to obtain the true value.
- If the broken view was created in an already existing projection (when editing it) with some elements attached to it, such as dimensions, then these elements may get lost view to changes in the projection topology after creating the broken view.

If you create a new projection on the basis of the projection with a break, the break from the main projection will be copied to the dependent projection.

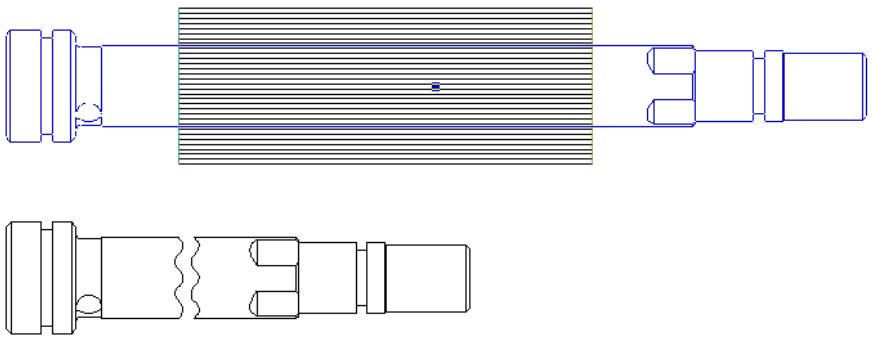
The option for creating the broken view on a projection is available in many modes of creating projections:

	<B>	Add or edit Broken View
	<V>	Add horizontal break
	<H>	Add vertical break
	<R>	Enter angle

The reduction can be performed only in two orthogonal directions. The number of breaks in either direction is unlimited. A break is defined by a rectangular area. Two of its boundaries are determined by the extents of the projection, while the other two are defined by the user using the mouse and the appropriate options.

The rectangular area of the break is hatched by the lines at the time of creating and editing, the lines direction indicating the direction of the reduction (orthogonal to the breaking lines). The preview image of the projection is drawn in blue lines.

The following diagram shows an example of a projection with a broken view.



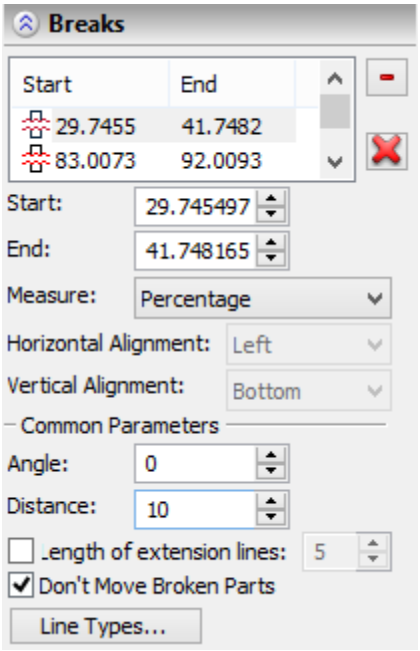
Also additional settings of broken views are done with the help of the properties window.

Each broken view is added to the list of broken views in the properties window. Elements of this list are preceded by the icons marking the break direction (vertical/horizontal). Each broken view has properties defined, that determine the placement of the break lines on the projection. Initially, these properties are predefined automatically, as soon as the user selects a break location by the mouse. If the values are defined by variables then the respective variable name will appear in brackets next to the numerical value.

Breaking lines position can be defined by absolute numerical values (the parameter “In model units”), or in percent (the parameter “In percents”).

Start. This value defines the position of: left boundary - for a horizontal break, or the bottom one - for vertical.

End. This value defines the position of: right boundary - for a horizontal break, or the top one - for vertical.



With the mode "In model units" turned on, the start and end values can be defined by variables.

Distance between the break lines defines the distance between the pieces of the broken projection.

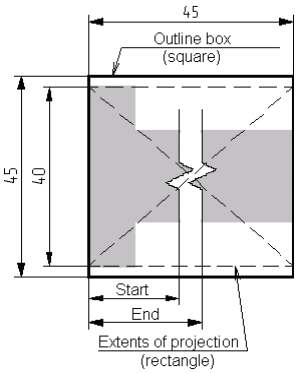
Angle. This parameter defines the angle of rotating the break lines with respect to the vertical/horizontal (according to the direction of the break). The positive direction is counterclockwise.

The angle can be also specified with the help of the option . After enabling this option, the line defining the initial boundary of the break (it is shown with red color) will play a role of a dragger. To specify the angle of rotation, it is enough to bring the cursor to this line, press and move the boundary by a desired angle.

Length of extension line. If the flag is cleared, then the breaking lines on the projection are restricted by the intersection points with the body. Otherwise, the breaking lines extend from one end of the body to the other end and beyond its limits by the "Length of extension line".

Don't move broken parts. When the flag is on, projection lines, remained after applying the breaks, don't change their location.

The following two parameters are intended for controlling the position of the break and its behavior with the model and the projection modifications. Those are available only if the break is defined "In model units".

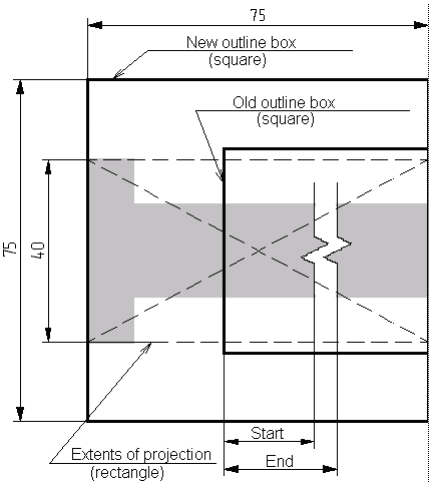


Horizontal binding (Left, Center, Right).

Vertical binding (Bottom, Center, Top).

To describe how these parameters are used, we need to introduce the concept of the "outline box". The center of the outline box is located in the center of the circumscribed rectangle of the projection. The outline box is a square, whose line length is equal to the length of the longer side of this rectangle. It is not displayed on the screen.

The outline box is always are calculated on the projection regeneration. Therefore, a new outline box is generated as the projection dimensions change.

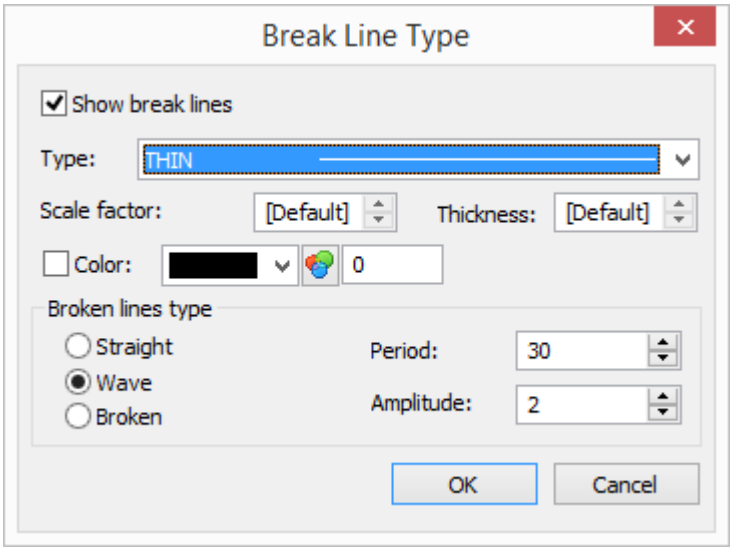


An important fact is the distance defined by the parameters "Start" and "End" is always counted from the lower-left corner of the old outline box.

The parameter values for the horizontal\vertical attachment defined a way of positioning the old outline books with respect to the new one. Suppose, for example, that the right center attachment is specified. This means that, after calculating the new outline box, the right side of the new box will be snapped to the right side of the old box, while the centers of both boxes will be placed on the center line of the part. The breaking lines are always constructed with respect to the lower-left corner of the old outline box.

To set up the properties of the lines of a break's boundaries, call an additional dialog. To launch it, press the button **[Line Types]** in the properties window.

Show break lines. The option allows to hide break lines on the projection.



Broken lines type. The break in the projection can be outlined by lines of different types. This can be a straight-line, a waves, or a polyline. The waves and the polyline are provided the parameters

Amplitude and **Period**. The units are same as the model units.

Keep in mind the specifics of the broken view projection creating mechanism. When generating the projection, the program internally creates a cut according to the specified break type. Therefore, the more complex the breaking line, the longer it takes for regeneration of a broken view projection.

The parameters **Scale factor**, **Thickness and Line Type** in this tab define the style of the breaking lines.

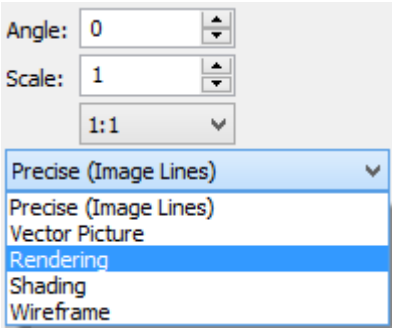
When creating a new broken view, the breaking parameters can be copied from an existing projection.

Upon pressing the button , the system waits for selecting a projection, whose breaking parameters will be copied to the projection being the defined. This is a one-time action. That is, when modifying the breaks of the original projection, the breaks on the projection to which they have been copied are not modified.

Type Selection for Created 2D Projection

The type of a 2D projection being created (regardless of its creation method) is set in the command's properties window, in the group “Basic parameters”.

The projection type is selected from the drop down list.



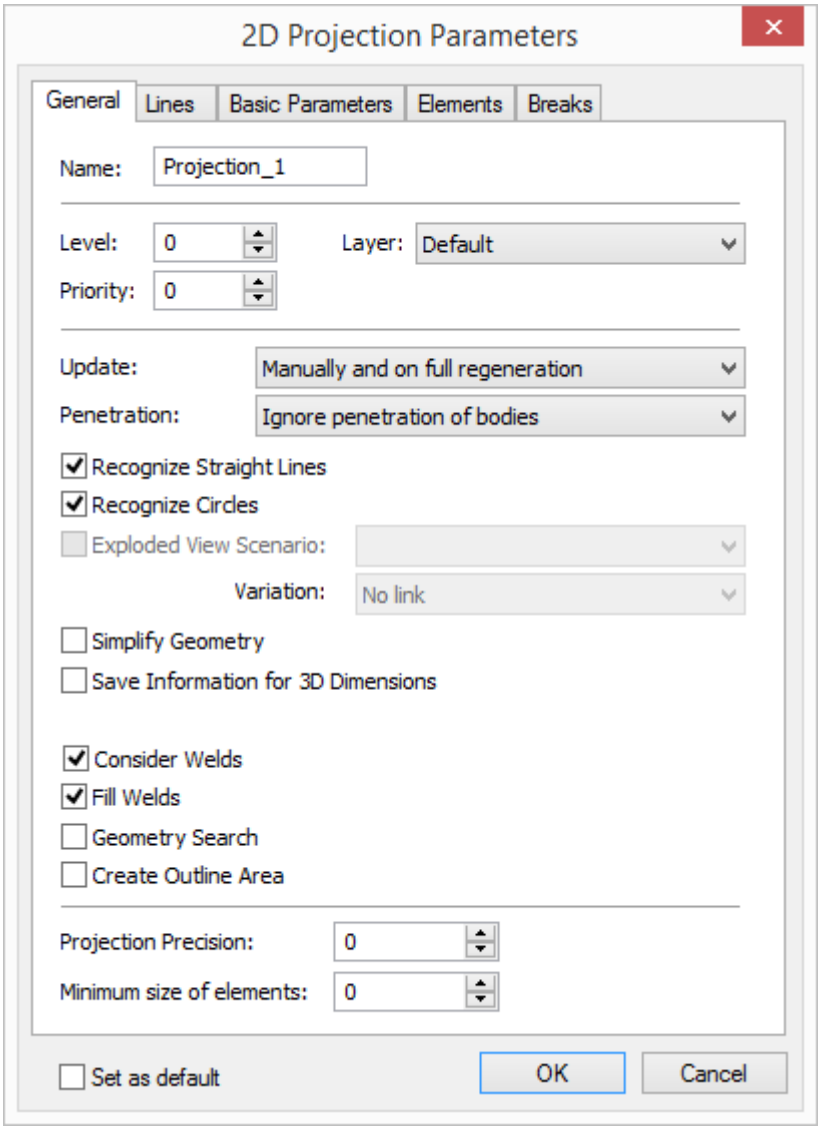
PARAMETERS OF 2D PROJECTION

Some parameters of a 2D projection can be defined in the properties window at creation or editing. Additionally, the parameters dialog box can be accessed at any time from the context menu, or at any time while editing, when the option is accessible:

	<P>	Set entity parameters

“General” tab

Name. The following default names are assigned to 2D projections: Projection_0, Projection_12, etc.



Update. This parameter defines how the projection will be updated.

- Auto (on any changes to model).
- Manually and on full regeneration (when choosing the command "Update 2D Projection" or upon full model regeneration). These settings make sense only when dealing with complex models, whose full regeneration takes considerable time.

Regeneration of “Vector picture” projection, in case of manual regeneration or regeneration on request, is carried out in background mode. When starting up regeneration, 2D projection view on the screen is replaced with a simplified view (similar to the view displayed in the 3D window in the mode “Quick hidden lines removal”). After that, they user can work with the system, for example, detail a drawing based on this projection. In parallel, the accurate regeneration of the projection

will be performed simultaneously. When the regeneration is completed, the simplified picture of the projection is replaced with the results of the accurate calculation.

This approach allows the users to simplify the work with projections having long regeneration time. After creating and editing the projection, the users can continue working with the system right away, not awaiting the completion of projection recalculation.

Penetration.

Ignore penetration of bodies. Bodies intersection is not considered.

Consider penetration of bodies. When projecting a group of mutually intersecting bodies, you can turn on the body intersection analyzer. With the parameter is set, the system will build the projection, taking into the account the intersection of bodies. It will increase the time of calculating the projection.

Define intersection of bodies. Allows you to create accurate isometric projections with a section.

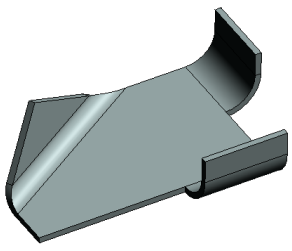
Recognize straight lines. Recognize circles. These parameters turn on special algorithms that recognize straight segments or circles (circular arcs) within some tolerance limits. This is necessary in the cases when complex geometry is being projected. For example, projecting a variable-radius blend creates a polyline. With this option turned on, the polyline is replaced by a circular arc. The resulting arc can now be dimensioned. When these options are used, there is some loss in the projection accuracy. This may sometimes cause problems with automatic hatching of the sections and local views. This is due to rips that might occur in a contour, preventing the contour from hatching. A special option is provided to deal with the problem by controlling the accuracy of searching hatch contours (see the tab "View").

Exploded View Scenario. The option is available only if there is at least one exploded view scenario.

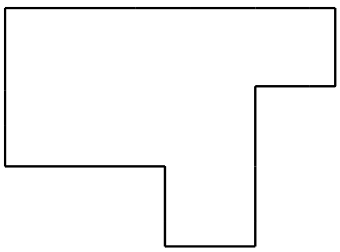
The projection is constructed using transformations from the selected exploded view scenario.

Variation. The option allows to to set link between the model configuration and the projection. Using the option you can create projections for each variation existing in the assemly. To specify a link, select the variation from the drop-down list.

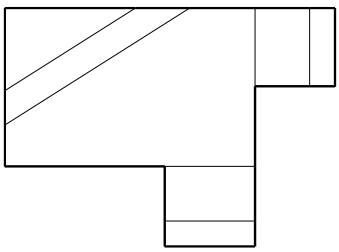
Simplify geometry. When the flag is set, **Simplify model** operation is performed before projections creation. This option is turned off by default, because it increases projections creation time. The option must be turned off when creating a projection of a sheet metal part unbending, for showing smooth edges of unbent areas. Option for showing smooth edges is located on in the **Lines** tab of projection parameters. When the option is active, splines are replaced with more simple geometry. You can create dimensions for it.



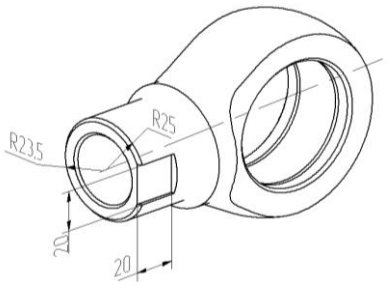
Original part



Simplified projection



Bend areas are shown



A projection with «3D dimensions»

Save information for 3D dimensions. This parameter allows creating "spatial" dimensions on nonstandard and perspective views (such as the axonometric view).

Consider Welds. This option enables the display of weld seams created on the 3D model.

Fill Welds. This parameter enables the mode of filling the side strips of the weld seams with a color.

Geometry Search. If this flag is set on, at recalculation of a 2D projection, the search of service elements on the projection is performed by the geometry of the projection, and not by model IDs.

When searching by geometry, the old location of the original projection element is taken (line, arc, etc.) and within the tolerance the nearest element of the same type is sought for. The identification search is performed regardless of the geometry by the projection elements internal ID formed during the history of the element creation.

It is recommended to use geometric search when the model was subject to significant modifications and the usual search does not produce results (for example, upon updating the projections of the imported geometry after modifying the imported model with an external application).

Create Outline Area. This option constructs an invisible outline hatch on all projected bodies with the priority of one point less than the priority of the projection itself. As a result, you can overlap some projections with other projections.

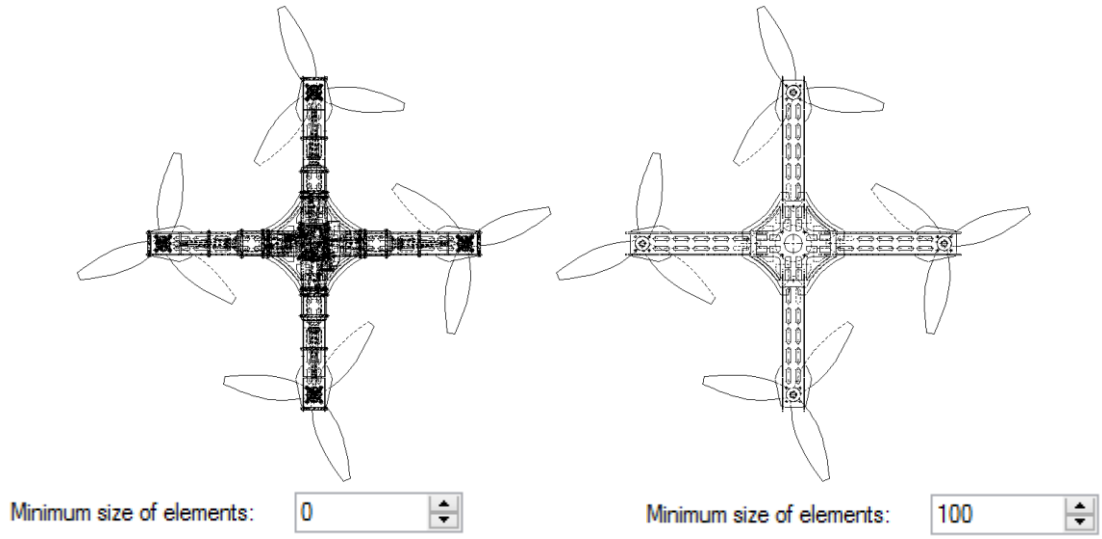
This option doesn't work or works incorrectly in the case when paths, edges, faces, or loops are selected for projecting, or if the projection method without the removal of hidden lines is specified. This option doesn't work for sections either.

On the other hand, this option is automatically enabled for a 2D projection created on the basis of an existing 2D projection.

Projection precision. Defines the accuracy of the projection lines representation. The minimum value of the parameter is 0.000001.

Minimum size of elements. The parameter allows to reduce time of construction and recalculation of projections by omitting small details. In addition such detail exclusion may increase drawing readability in some cases.

The minimum size is set in the current units of model measurement. All bodies with a smaller size won't be shown on the projections.

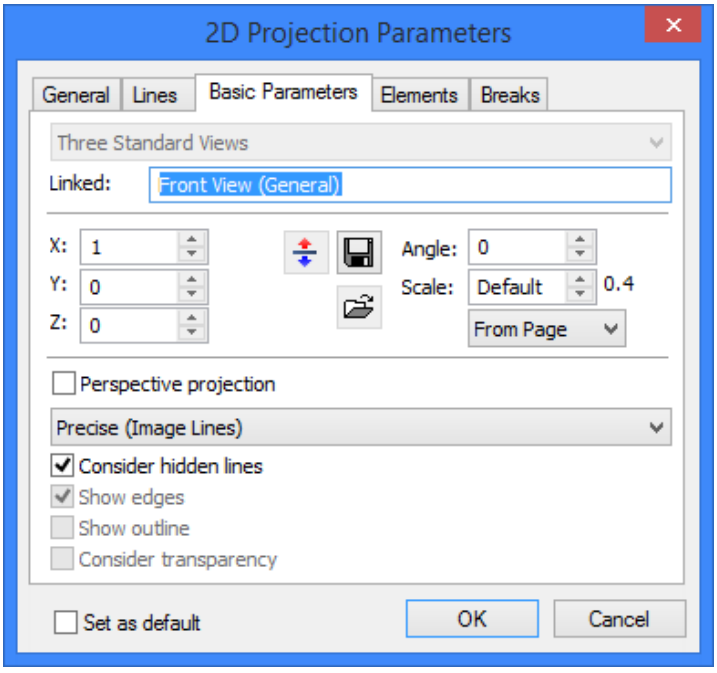


Recalculation time: 0,577 seconds.

Recalculation time: 0,088 seconds.

"Basic Parameters" tab

The drop down list found at the bottom of the window defines the type of the projection: **Precise (Image Lines)**, **Vector Picture**, **Rendering**, **Shading**, **Wireframe**.



The following projection parameters are defined here as well:

Perspective projection. Parameter defines type of created projection: parallel or perspective. When the flag is set, perspective projection is created. Dimensions, based on elements of such 2D projection, also drawn according to the perspective parameters.

Consider hidden lines. This parameter defines whether the hidden lines are shown on the projection. This parameter is not available for projections of the following types “Shading”, “Rendering”, and also for sections and cuts.

Show edges. This parameter controls the view of the edges on the projection (only for projection of the types “Shading”, “Rendering”).

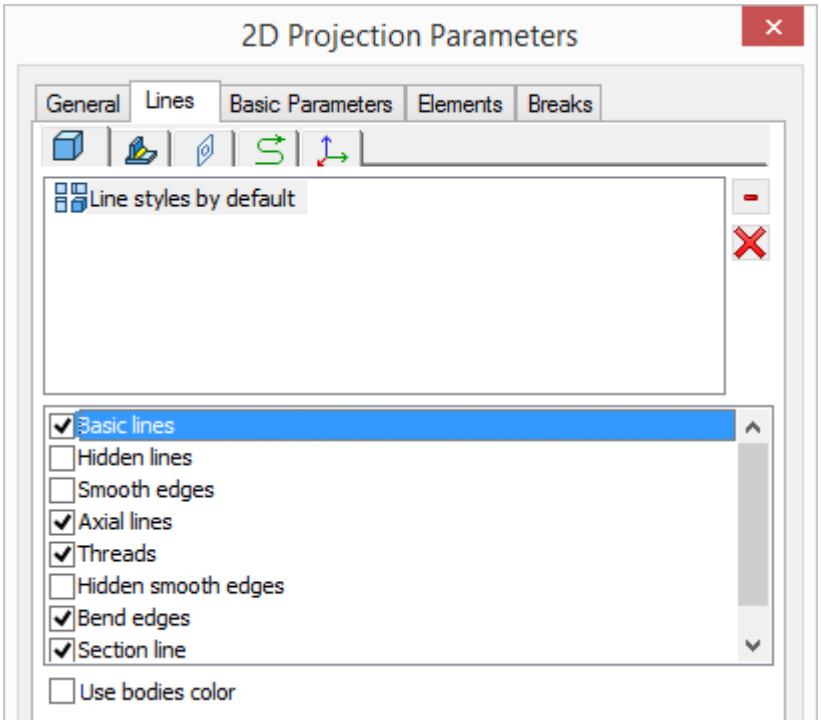
Show outline. With the help of this parameter, the view of the outline edges of the projected bodies is controlled on the projection (only for projections of the types “Shading”, “Rendering”, “Wireframe”).

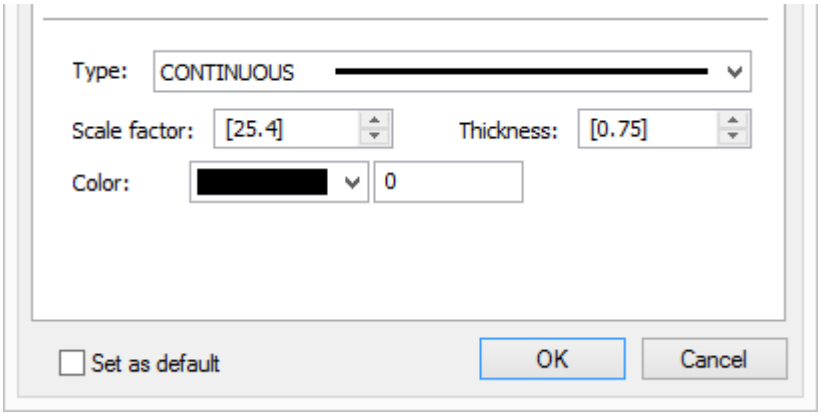
Consider transparency. When the flag is set, transparency of objects is considered on the projection. Elements, located behind transparent objects will be shown on projection as well.

The rest of the parameters on this tab reflect the information about the view direction, the scale and relations with other elements (see the detailed description above).

“Lines” tab

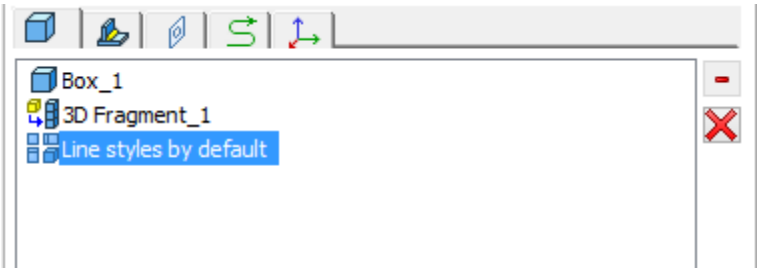
On the **Lines** tab you can change image line parameters for particular 3D construction elements, bodies and fragments.





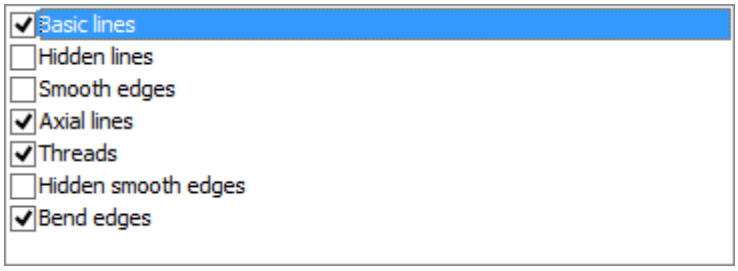
The upper pane displays the list of selected items. You can change lines parameters for the elements.

You can select elements in the **Properties** window by clicking "..." in the list.



You can select lines parameters for all elements on projection when you select *Line styles by default*, but if lines parameters are already set for any element in the list, they will not change.

In the center, there is a list of categories of projection lines. If you want to display lines of the category set flag near its name.



Lines Categories

Basic lines. Allows to set parameters of basic lines.

Hidden lines. This parameters of this group control the display of invisible lines and parameters of the view on the 2D projection.

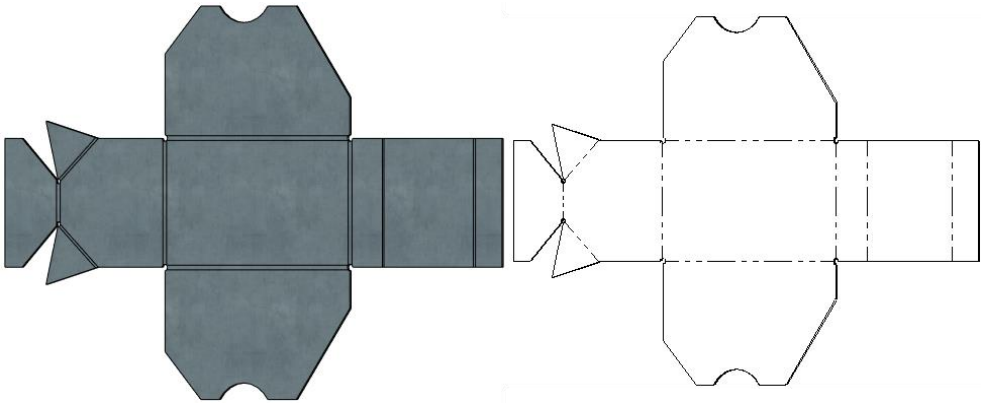
Smooth edges. The projection may or may not display the edges between the smoothly adjoining faces (connected without sharp corners). This group of parameters defines whether to display smooth transitions, and defines the way of displaying graphic lines (type, color, etc.) used for this purpose.

Axial lines. When projecting surfaces of revolution (cylinder, cone, torus), the axes, or centerlines, can be created automatically. In the case when the body has several surfaces of revolution, the option **Unite** can be conveniently used for avoiding overlapping of axes.

The flag "Automatic creation of axial lines" also controls the display of bend axes on the projections of bodies obtained by applying operation **SMU: Unbend Bend**.

Threads. When creating 2D projections, the threads existing in the model are recognized automatically. With the option turned on, the lines of the thread notation are created in the cases, when it is viewed on the given projection from the front or side, or when the section plane passes through the thread axis. When creating a dimension on a thread in one of the above representations, the system automatically creates the appropriate notation. The data for the notation is taken from the thread creating operation.

Bend edges are designed to display lines on the projections of unbent sheet metal parts.

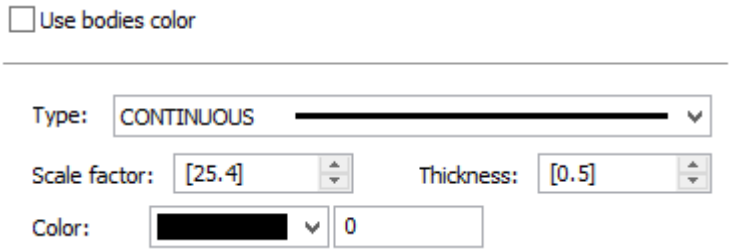


Section lines is used for drawing of visible segments that appear when the projected body intersects planes of stepped sections orthogonal to the direction of the projection.

Hidden smooth edges. Shows smooth lines that are invisible on the projection.

Lines Parameters

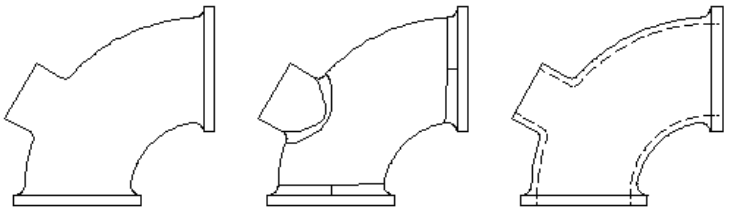
Parameters of lines available for changing are displayed in the lower part and depend on the selected category.



You can select several categories at the same time and set common parameters for them.

Any auxiliary type of the projection lines can be assigned **Line Type and Thickness**, **Scale factor**, **Color**.

Use bodies color. With this parameter turned on, the color of all projection lines and section hatches is determined by the color of the operations being projected. All bodies are colored only in the shading mode.



Type. You can select lines type for the selected category from the drop-down list.

Thickness. Specifies lines thickness.

Scale factor. Allows to set scale factor for non-continuous lines.

Color. You can select lines color from the drop-down list or enter number of the color.

Shorten Line parameter is available for smooth lines. If this option is checked, the lines of smooth edges are drawn shortened at both ends.

In the case of the thread, there is an additional parameter, **Minimum thread height**. It is used to make an exaggerated, to the specified degree, cosmetic thread profile, when the line thickness used in the drawing or the small scale of the projection do not allow to maintain the cosmetic thread legible in its actual size.

Depending on the type of a 2D projection, the dialog of the option can have additional tabs **Gaps**, **Sections**, **Coordinate systems**. The contents of these tabs duplicates the contents of the similar sections of the properties window when creating/editing 2D projections.

EDITING AND MOVING 2D PROJECTION

Editing of 2D projections is done by the command **3EJ: Edit 2D Projection**:

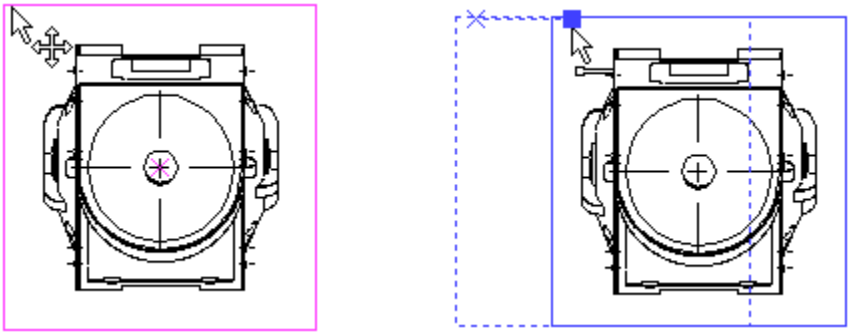
Keyboard	Textual Menu	Icon
<3EJ>	Edit > Draw > 2D Projection	

The editing command can also be called from the context menu by right clicking on a projection or its direct child.

Editing is little different from the projection creation. Upon calling the command, all options become available in the automenu that were provided at projection creation.

The already described options allow editing or redefining the list of elements to be projected, and selecting operations to include or exclude from the section, etc.

To move the projection, place the cursor on the projection perimeter, wait until symbol appears and move projection by holding the left mouse button.



Dynamic view of projection being moved, allows you to see the full image after moving the projection. When you move the projection of large drawings system offers option to disable dynamic view not to slow down operations.

To move a projection, you can also change the position of its attachment point. Activate one of the two options for attaching the projection and specify the new projection position (a free point or a 2D node).

> In the case when the projection is attached to a 2D node, its position can be changed by simply modifying the respective 2D node position and updating the projection. There is no need for launching the projection editing command.

USE OF 2D PROJECTIONS

Working with 2D Projections of Type "Precise (Image Lines)"

A 2D projection of type "Precise (Image Lines)" is a special element in the system. The drawing only displays the direct children of this element, that is, graphic lines and hatches. Together, these elements make the image of the projection. Most of the properties of each element are taken from the projection customization settings at the time of creation. These are, for instance, types of various projection lines, Color, Scale factor, priority, etc. however, if desired, each element properties can be modified separately. This can be done via the element parameters dialog box, conveniently accessible from the context menu. As usual, it appears in the right click over the element.

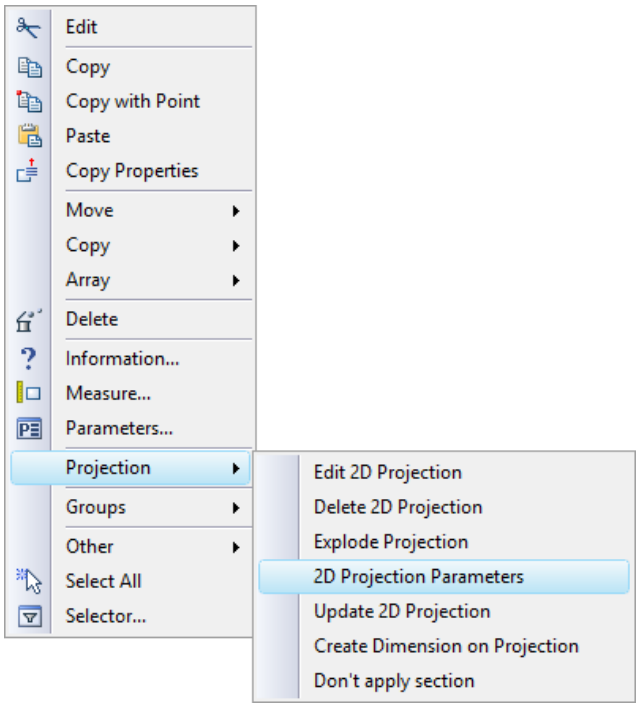
> Depending on which element was selected by right-clicking (whether the 2D projection itself or a child element), the context menu will have different contents.

If an immediate child of the projection is selected (such as a graphic line or a hatch), then its own context menu will have some additional specific commands to work with the 2D projection. In this case, there is also a difference in the action of the standard **Delete** command . In this case, it does not actually delete, rather merely hides the selected element. This can serve to hide some "unnecessary" elements of the projection.

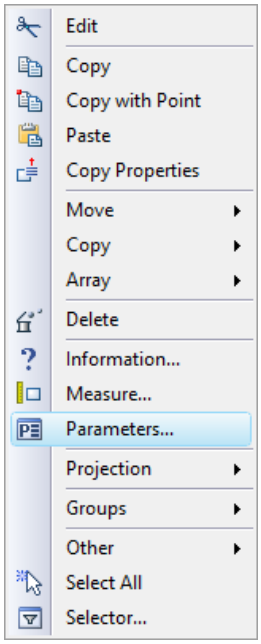
Projection editing is activated by the **Edit 2D Projection** command. To delete the whole projection, use the command **Delete 2D Projection**.

Next in the list is the command **2D Projection Parameters** that is used for calling the projection parameters dialog box. This dialog allows defining some general properties of the projection elements (line types, Color, Type and hatch display), as well as all the rest of the projection settings. Details on handling this dialog box are presented in the respective section of this chapter.

The projection is not regenerated as the 3D model updates. To have the model changes reflect on the drawing (projection), either start the full model regeneration or update the projection manually by the command **Update 2D Projection**. A special parameter on the tab "General" of the projection parameters dialog box controls the ways of updating the projection (see description above). The command **Update locally** provides greater time saving when handling complex models and drawings. Unlike the previous one, this command does not start regeneration of the children elements. This is convenient for working with projections that take long time for regeneration and are subject to projective relations.



The context menu upon selecting a projection line



The context menu upon selecting a 2D projection

The context menu of the 2D projection proper has several additional useful commands.

There is the command **Restore deleted elements** that serves to restore the "deleted" elements (lines or hatches hidden by the **Delete** command).

If necessary, you can break the relation between the projection image and the 3D model, and turn the 2D projection into a collection of nonparametric unconstrained graphic lines and hatches. You use the **Explode Projection** command for this purpose.

If the 2D projection is constructed based on a model from an external file, then you have the command **Open** which opens the document related with the projection.

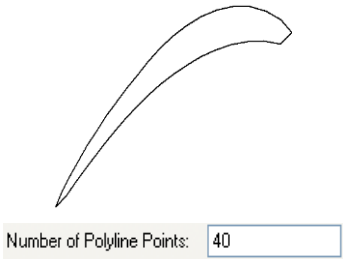
The command **Create Dimension on Projection** serves to carry the 3D annotation elements over to the 2D projection (see the next paragraph).

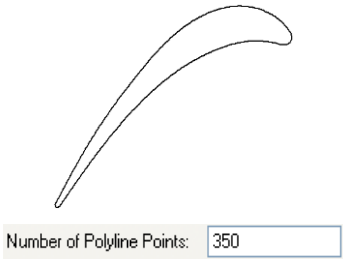
When projecting two or more bodies penetrating each other, there could be inaccuracies in defining the lines of bodies intersection and the lines visibility. Therefore, it is recommended to join such bodies in one by using the Boolean operation whenever possible or use the parameter "Allow intersecting bodies" before projecting (see the description of the parameters dialog box). Turning on this option increases the time of the projection regeneration.

Some 3D elements can be created based on a 2D projection, such as workplanes and 3D sections. The respective options are provided in the automenu sets of the commands creating the respective elements.

2D projections can be later used as references for construction lines, dimensions, detailing elements, etc. When creating construction lines, object snapping is engaged to the projection elements, distinguishing circular arcs, ellipses, splines, line segments, end points (nodes) of lines, arc centers, etc.

Some graphic lines created based on a 2D projection have additional parameters. For example, there are graphic lines of a special type "spline curve", which may sometimes be generated when projecting. You can additionally specify the number of spline points for a graphic line of this type, if you find the mapping accuracy not satisfactory (it is 40 points by default).





Working with 2D Projections of Types "Vector Picture", "Shading", "Rendering", "Wireframe"

On the drawing 2D projections of types "Vector Picture", "Shading", "Rendering", "Wireframe" represent themselves (in contrast to 2D projections of the type "Graphic Lines") inseparable objects. Parameters of separate elements of such projections cannot be modified.

In other respects, working with 2D projections of these types is similar to working with the projections of the type "Precise (Image Lines)". The context menu of similar 2D projections duplicates the context menu of the 2D projection of type "Precise (Image Lines)".

When creating 3D annotations on the 2D projections of the types "Vector Picture", "Shading", "Rendering", "Wireframe", the annotation elements being created are attached directly to the elements of the 3D model.

Creating Drawing Annotation Elements

To accomplish a drawing created based on a 2D projection, you can use standard 2D commands – **D: Create Dimension**, **IN: Create Leader Note**, **RO: Create Roughness Symbol**. Dimensions, leader notes and roughness symbols can be created on the projection lines in the same way as on a common drawing.

> If the 3D model was modified significantly, the composition of projection lines and some element IDs might change to some extent upon the 2D projection recalculation. In such a case, some of the annotation elements based on the lines produced by the 2D projections may lose their relation with their source elements . The system will notify the user about it with a message in the diagnostics.

A 3D model may include 3D annotation elements - dimensions, leader notes and roughness symbols, applied directly in the 3D window on the model faces. Those serve to make the 3D model appearance more intuitive and simplify modifying its geometrical parameters. Besides that, the information stored in 3D annotation elements (such as tolerance parameters in a dimension, its additional text lines, roughness symbol parameters, etc.) can be carried over to the counterpart 2D annotation elements in the projections of the given model. For details on creating 3D annotation elements refer to the chapter "3D Annotations".

If there are 3D dimensions or roughness symbols on the current 3D model, then, when creating 2D dimensions and roughness symbols on the respective projection lines, the system can automatically find the correspondence between the element being created and its counterpart element in 3D. If such a correspondence is found, then all parameters from the 3D dimension or roughness symbol (the nominal, tolerance, text lines, etc.) will be carried over to the 2D element being created. If any of such parameters are to be changed in the 2D element, then you should modify its counterpart element in 3D.

The data transfer from 3D dimensions onto a 2D projection can also be accomplished by a direct "mapping" of the dimensions from the 3D space onto the 2D projection. This is done using the **Create Dimension on Projection** command available in the context menu when called over a 2D projection.

Upon calling the command, the system determines, which of the 3D dimensions existing in the model can be mapped on the selected view (projection). The system-selected dimensions are drawn on the projection in red. The user needs to specify, which of the system-supplied dimensions shall be created on the given 2D projection.

You select the dimensions using the following options of the command's automenu:

	<M>	Select Detailing Elements

	<M>	Cancel Selection
	<*>	Select All Elements/Cancel Selection of All Elements
	<End>	Finish Element Selection

When the option is active, the dimensions are selected that shall be created on the given projection. To select a dimension, simply point the cursor to the desired dimension in the 2D window and click . The selected dimension color will change to blue. In this way, you can select any number of dimensions in a series.

Activating the option starts the mode of *canceling* the selection of dimensions. In this mode, picking a dimension by cancels its selection. As a result, the dimension color turns back to red.

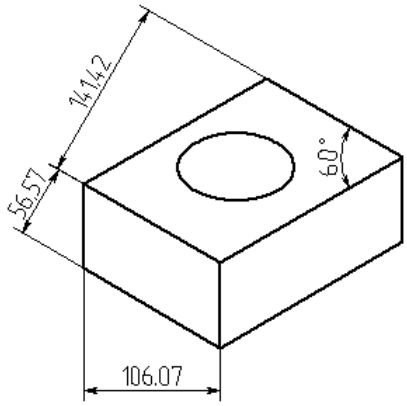
The effect of the option depends on which of the above-mentioned options is active. If the option is active, then using this one will cause all system-provided dimensions to become selected, whereas with the option active – it will do the opposite, canceling all dimension selection.

You can confirm the creation of the selected dimensions on the projection using the option .

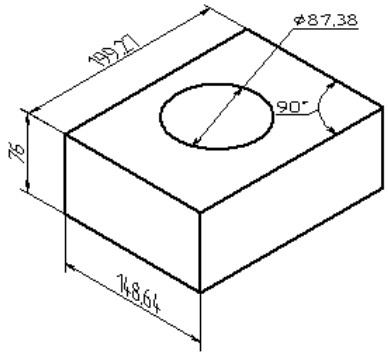
Modes of Displaying Dimensions on 2D Projections

Dimensions created on 2D projection lines can be displayed in two ways. The first method is the standard display of dimensions (in the plane perpendicular to the viewing direction). The second method is "spatial" display of annotation elements. The latter is usually used in nonstandard and "spatial" views (such as in the axonometric). In this mode, the dimensions created based on projection lines will be drawn in the plane of the selected elements. For example, if a dimension is applied to an elliptical projection image of a circular edge, then the dimension will be maintained in the plane of the circular edge. This capability helps creating dimensions with actual values on axonometric views.

This capability is controlled by the **Save information for 3D dimensions** option on the **General** tab in the 2D projection properties (see the section "2D Projection Parameters"). You need to update the projection after enabling this option.



Dimensions on a 2D projection in the regular mode



Dimensions on a 2D projection in the "3D" mode

The "spatial" method of displaying dimensions on a drawing can be used with all types of dimensions except construction ones.

Using Dimensions on a Projection to Manage the 3D Model

2D dimensions created on projection lines can sometimes be used to modify the 3D model. This is possible in the cases when the system managed to find a correspondence between the projection lines on which the dimension was created, and the control elements of the 3D model. Such elements may include faces of the 3D model dependent on the geometrical parameters of operations or on 2D constructions that provide the base for the 3D model operations. The system automatically tries to find such correspondence when dimensions are constructed on the 2D projection. If the system succeeds finding such correspondence, then the changes to the dimension values will affect the 3D model and its projections. You change the dimension value using the command **PE: Set Dimension Values**, described in detail in the chapter "Modifying the drawing using dimensions" of the "Fundamentals. Two-Dimensional Design" guide.

Sheet Metal Operations

SHEET METAL OPERATIONS

Sheet metal commands facilitate modeling of parts made of forming sheet metal. Those help create an initial flat part (workpiece) of the specified thickness, and then perform various shape manipulations, such as bending about a selected line, attaching flanges to the workpiece, making punches. Once modeling is finalized, the part can be unbent to the unfolded pattern, as for the purpose of drafting, and then re-bent to the actual part shape.

The special "Forming Feature" command in the sheet metal group of commands helps creating various commonly used forming features, such as beads, ribs, flanges, louvers, pockets.

When working with sheet metal, you can use not only the specialized commands described in this chapter, but also general-purpose commands (Extrusion, Boolean operations, Hole and so on).

In addition, there is a capability of creating sheet bodies and bended sheet objects on the basis of 3D models with the help of the Convert solid to sheet metal operation and create sheet bodies based on two sections using Lofted Bend operation.

Note that the sheet metal operations do not account for possible material deformations. Rather, those are based purely on geometrical calculations.

DEFAULT PARAMETERS FOR SHEET METAL

General Sheet Metal Parameters

You can define common settings for the whole group of the sheet metal commands. Those determine the values of main parameters of the group operations:

Metal thickness;

Main bending parameters – the bending radius, the position of the neutral layer of plastic deformations, the shape and dimensions of the relief slots and some other parameters.

The specified settings will be used across all sheet metal commands in the case of using Set as default value. As general parameters change, the operations that use those will adjust their parameters.

Defining Parameters of Sheet Metal Operations

To define default parameters for the whole group of sheet metal commands, call the command SMP: Sheet Metal Operation Defaults:

Icon	Ribbon
	3D Model → Special → Sheet Metal → Defaults
Keyboard	Textual Menu
<SMP>	Operation > Sheet Metal > Defaults

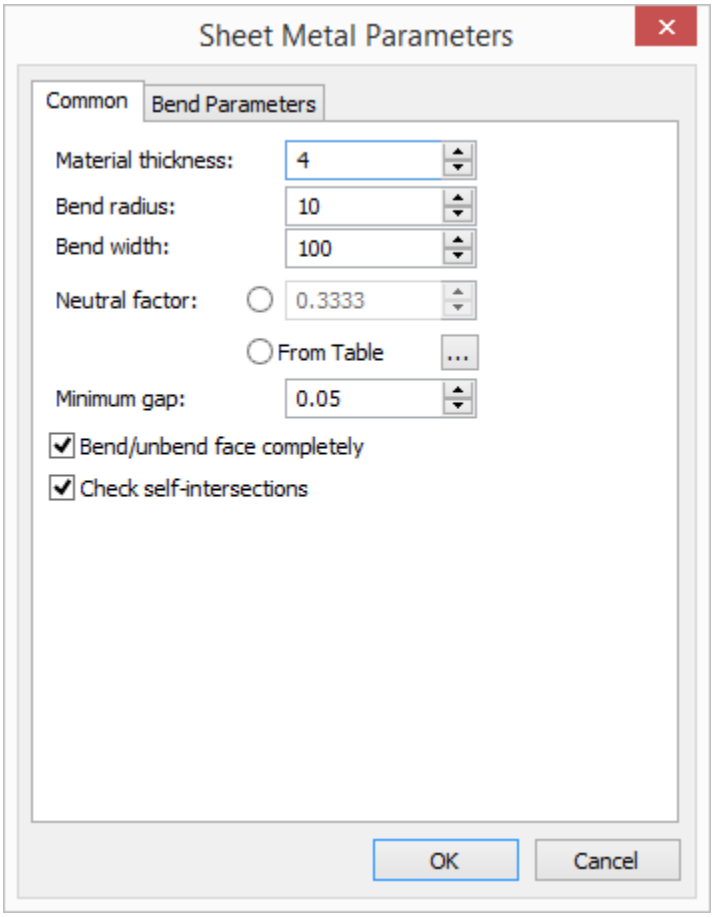
The command brings up the dialog box **Sheet Metal Defaults**. Default parameter values entered in its input boxes affect all sheet metal operations on the given 3D model:

Material Thickness. Defines the default thickness of the workpiece in the command Create Part.

Bend Radius. Defines the default bend radius used in Bend command.

Bend width. Sets default value for the bend width.

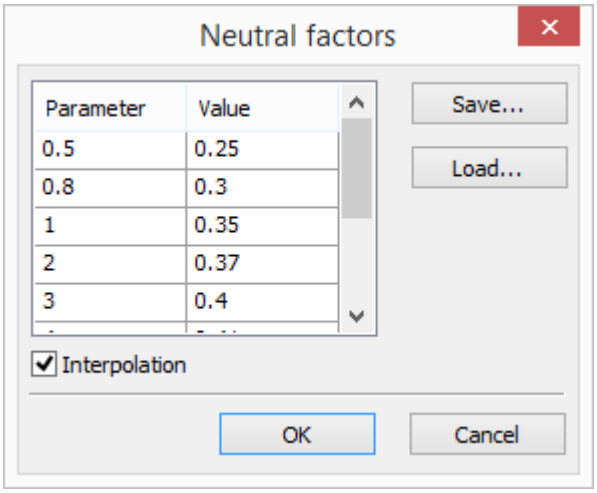
Neutral factor. Controls neutral factor definition for part bending and unbending.



The parameter value can either be entered by the user or defined by the system automatically, based on the table of Neutral factors versus "bend radius/workpiece thickness" ratio. The definition is controlled by a radial-button switch. To use automatic Neutral factor definition, set the button "From Table". Setting the other button allows entering an arbitrary number in the range from 0 to 1. The value "0" means the

neutral layer coincides with the workpiece face selected for bending, while "1" sets the neutral layer on the opposite face of the workpiece.

The Neutral factor dependency table is stored with the model document. To view or edit the table, use the button ⋯. Upon pressing the button, an additional window "Neutral factors" opens up.



This window displays the Neutral factor table of the current document. It represents a tabulated dependency of Neutral factor (the column "Value") on the ratio "bend radius/workpiece thickness" (the column "Parameter"). The flag "Interpolation" defines the type of interpolation of the table data: the set flag means linear interpolation – the Neutral factor linearly interpolated between the tabulated data; the cleared flag means piecewise constant interpolation – the Neutral factor is maintained constant until reaching the next entry.

The current table can be exported into a textual file by pressing the button [Save...].

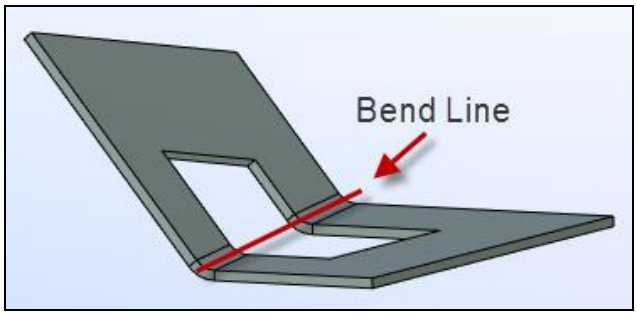
The button [Load...] is provided for importing table data from an existing user-created text file "**.tnf*" in the following format: the first line in the file defines the interpolation type: the keyword "Interpolation" means linear interpolation, "Table" – tabulated. Other lines contain the table data – the "bend radius/workpiece thickness" ratios and the respective Neutral factor values. The table data can be entered in an arbitrary way – in one line or in multiple lines, separated by spaces or tabs. For example, the default table file appears as follows:

```
Interpolation
0.5   0.25
0.8   0.3
  1   0.35
  2   0.37
  3   0.4
  4   0.41
  5   0.43
 10   0.47
 12   0.5
```

Once loaded from a file, the table is stored in the document of the current model and is further used in model manipulations.

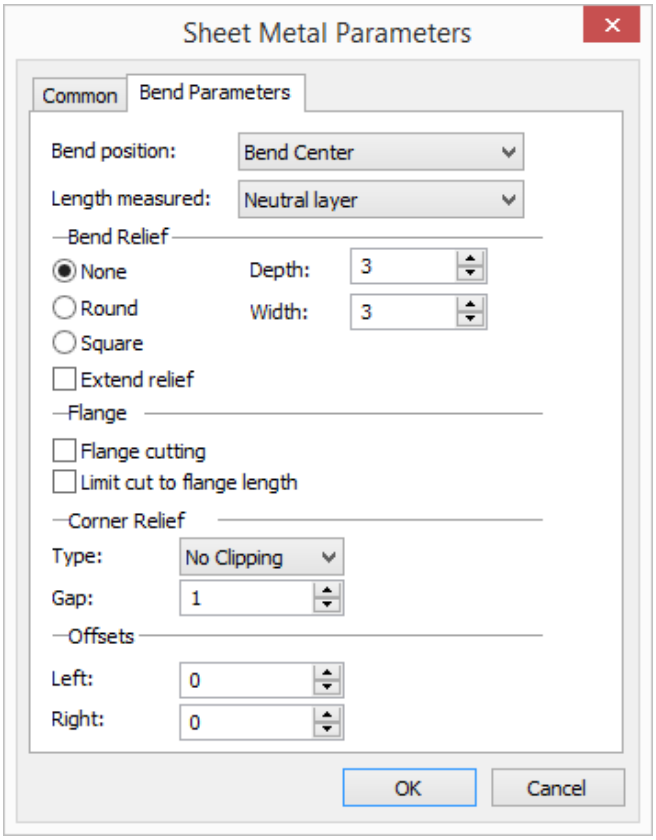
Minimum gap. Define the width of a cut when creating operations with sheet metal.

Bend/unbend face completely. This parameter is important when performing operations on the sheet metal of certain geometry (for example, when it is required to bend the sheet body with a cut along the specified line). When this parameter is enabled, the entire body along the selected line will be bent without violating the construction logic.



Check self-intersections. This parameter is important when creating the «Bend» parameter. It allows us to monitor self-intersection of bent sheet faces.

The «Bend parameters» tab



The parameter "Bend position" defines the position of the bended area with respect to the baseline of the bend: "Bend start", "Bend center" or "Bend end".

The way of defining the flange length is controlled by the parameter "Length measured". The four values in the combo box are: "Neutral layer", "External side", "Internal side", "Flat part". Selecting one of them sets the desired default way of length calculation.

The group "Bend Relief" defines the type and dimensions of the reliefs created by default when bending:

The type of reliefs is defined by the radio group: "None" (no reliefs), "Round", "Square".

The parameters "depth" and "width" allow defining the default dimensions of a relief: the incision depth (length) ahead of the bend line, and the width of the relief slot.

Setting the flag "Extend relief" sets the mode of cutting the slot up to the borders of the workpiece.

The group "Flange" provides various options for default material trimming when making flanges. Gaps are introduced by reducing the flange dimensions when bending or attaching a flange, if directed by the flag "Flange cutting". When the flag is set, gaps are introduced by default. The flag "Limit cut to flange length" controls the ways of material removal from the workpiece under the flange: when set, the material will be removed by the flange length (by default); when cleared – up to the borders of the workpiece.

The parameter "Corner relief" controls trimming of intersections between the attached flange and the workpiece faces. For trimming, select the option "On" in the combo box and enter the desired amount of Gap between the trimmed faces. If "Off" is set, the intersecting portions will not be trimmed.

The last group of parameters – "Offsets" – defines offsetting from the ends of the bend base line. The offset sides ("Left", "Right") are assigned by the system arbitrarily.

The parameters defined in the command **SMP: Sheet Metal Operation Defaults** will be used by default in all sheet metal handling commands. Those values will appear as **Default** in both text and numerical parameter entries. For numerical parameters, the actual value will be displayed next to the parameter input box.

Initial Workpiece for Sheet Metal Operations

Main concepts

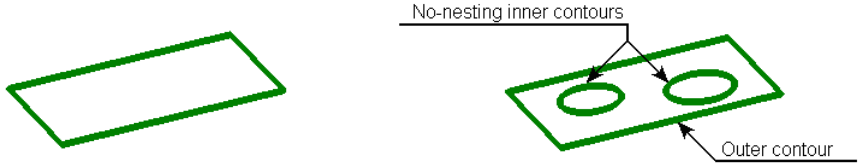
All sheet metal operations manipulate with the initial preformed workpiece. The workpiece can be a solid body of virtually any shape. A system requirement for defining various sheet metal operations is existence of two flat parallel faces in the area of deformation. One of such faces will be used as the base when defining the desired type of a sheet metal operation.

We recommend using a flat body as the sheet metal workpiece, with the side (end) faces perpendicular to main (top and bottom) faces. Such body can be created, for instance, by the extrusion operation or by the special command for creating a sheet metal workpiece.

The special sheet metal workpiece creation operation is essentially an extrusion in the direction normal to a flat contour, to the specified depth. Use of this specialized operation instead of extrusion or other general-purpose operation simplifies workpiece creation, minimizing the number of parameters and settings to define.

This command serves to create a sheet body workpiece using one of two methods: by the defined shape of the workpiece or by the defined shape of the workpiece cross-section. Both methods use a flat 3D profile as the source.

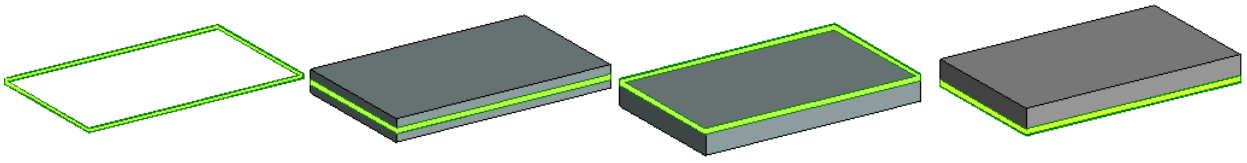
When creating a sheet workpiece *by the defined workpiece shape*, the source 3D profile defines the contour of the would-be workpiece. We recommend using a single-contour 3D profile. It is still possible to use a multi-contour profile, if it satisfies the following requirements: the 3D profile must have a single outer contour, and the inner contours with no nesting.



Single contour 3D profile

Multi-contour 3D profile suitable for creating a sheet metal workpiece

In this method of creating a sheet workpiece, besides the 3D profile, the thickness of the would-be workpiece and the direction of extruding its profile are defined at creation. The selected profile is extruded along the normal, to the specified depth equal to the width of the workpiece sheet metal. The extrusion is done in one of the three ways: upwards from the original profile, downwards from the original profile, or in both directions.



Original profile

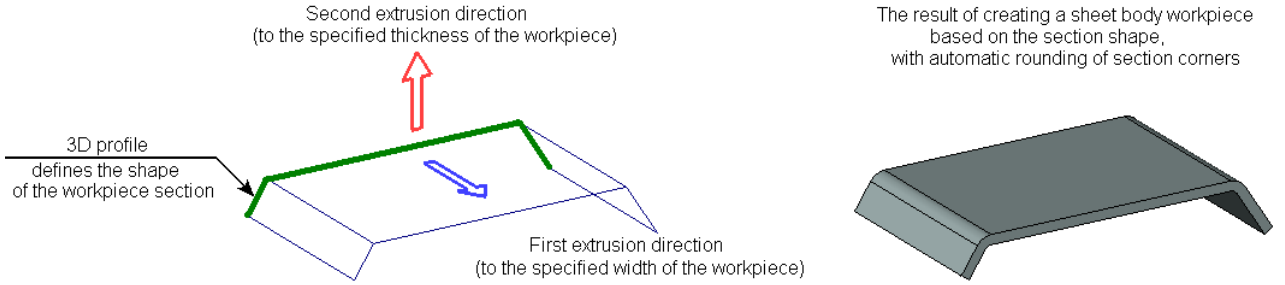
Workpiece symmetrical about original profile

Workpiece created downwards from original profile

Workpiece created upwards from original profile

When creating a sheet workpiece *by the workpiece cross-section shape*, the 3D profile determines the shape of the workpiece body. In this case, only a flat single-contour 3D profile can be used. It can be closed (which will result in a "pipe"-type workpiece), or open.

To create the workpiece, the selected 3D profile is extruded in two steps. First, the extrusion is made in the normal direction to the plane of the selected 3D profile to the specified width of the workpiece sheet. The edges of the resulting sheet body (the corners of the workpiece cross-section) are automatically rounded. This defines the shape of the would-be workpiece. Then, this sheet body is extruded normally upto the specified workpiece thickness. As in the previous case, extrusion by the sheet thickness can be done in three ways relative to the surface resulting from the first extrusion.



From now on throughout this chapter, any body subjected to a sheet metal operation will be referred to as **"**workpiece**"** regardless of the means of the body creation.

Creating a Base Part

To create a workpiece part, use the command **SMC: Create new Base Part**:

Icon	Ribbon
	3D Model → Special → Sheet Metal → Create Part
Keyboard	Textual Menu
<SMC>	Operation > Sheet Metal > Create Part

To create a workpiece, just select the source 3D profile in the model tree or in the 3D window. Profile selection activates automatically upon entering the command. This is indicated by the pushed automenu option:

	<R>	Select 3D Profile

To select a profile in the 3D window, point at one with the mouse. The pointed element will be pre-highlighted, and a tooltip next to the pointer will display its name. To select it, click.

The source profile will be selected automatically, if you call the command while drawing on the active workplane or from the context menu of an appropriate-type 3D profile.

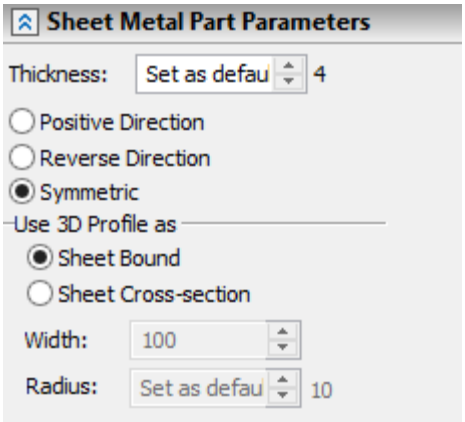
The workpiece creation method (by the workpiece shape or by the workpiece cross-section shape) is defined in the command's properties window. The default method depends on the 3D profile selection. If a closed profile is selected, then the method of creating by the workpiece shape is set. If an open 3D profile is selected, the system activates the mode of creating the workpiece by the workpiece cross-section shape.

The workpiece creation method can be altered by setting the desired value in the parameter "Use 3D Profile as" ("Sheet Bound" or "Sheet Cross-section").

The rest of the would-be workpiece parameters are also defined in the properties window. First of all, that is the workpiece thickness (the parameter "Thickness") and the direction of extruding to the specified thickness ("Positive Direction", "Reverse Direction" or "Symmetric").

When creating the workpiece by the cross-section shape ("Use 3D Profile as > Sheet Cross-section") the width of the workpiece sheet can be defined in the properties window (the parameter "Width"), as well as

the rounding radius of the section corners (the parameter "Radius"). By default, the rounding radius is the value of the "Bend Radius" parameter in the command **SMP: Sheet Metal Operation Defaults**.



BENDING. MAIN CONCEPTS AND CAPABILITIES

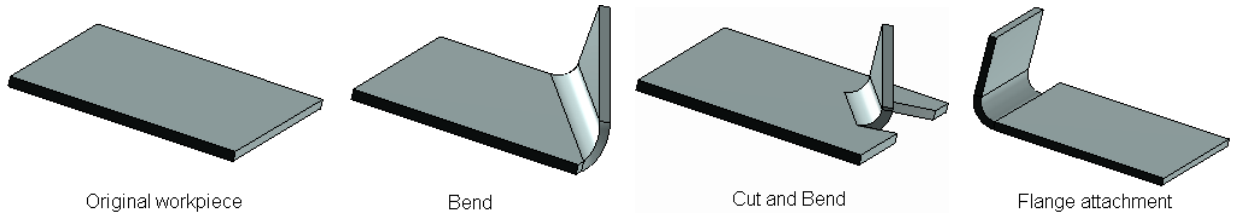
"Bend" operation imitates one of the main shape-defining sheet metal operations, bending. The operation is applied to a solid body satisfying the above conditions on the sheet metal workpiece.

Types of Bending

Bend operation supports various types of manipulation with a sheet metal workpiece, such as: bend the whole workpiece along a line; bend a protruding portion thereof; make an incision and bend an inner portion of the workpiece. If the flat pattern shape is not known in advance, the user can start modeling from a simple box-shape workpiece, attaching to it rectangular pieces as necessary and bending those right after.

All these actions are supported by the four bending modes. You can select a mode in the operation properties window.

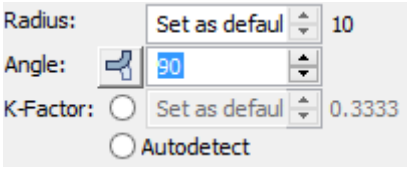
- ✓ Bend – bends the whole workpiece along a specified line.
- ✓ Cut and Bend – makes an incision, and then bends a portion of the original workpiece.
- ✓ Attach Flange – attaches a flange of the specified size to the original workpiece. The attached flange is bent about its attachment line.



- ✓ Hem – attaches a hem of the selected shape to an outer contour of worpiece.

General Bending Parameters

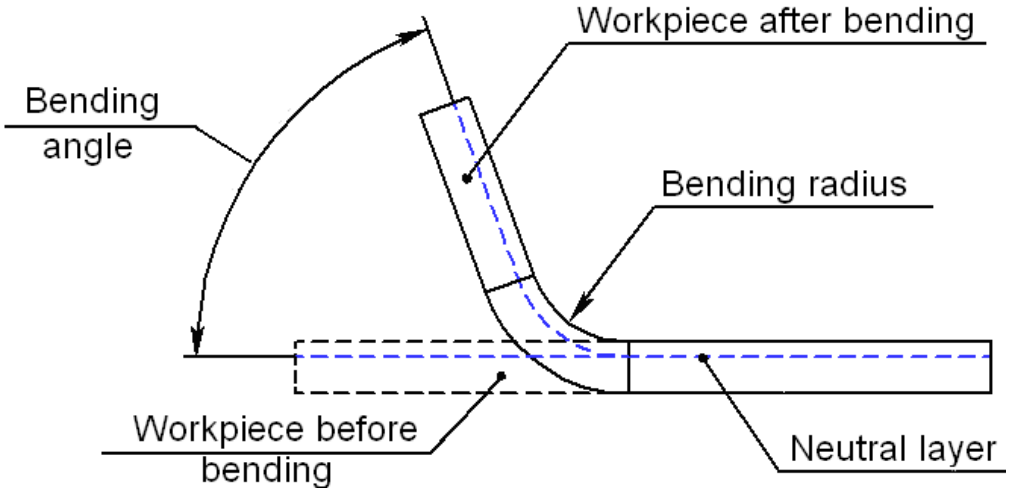
Bending of any type is affected by the following parameters of the operation:



Radius of bending – the radius of the base face in the bended area;

Angle of bending – the angle between the original and the bent position of the bent portion of the workpiece;

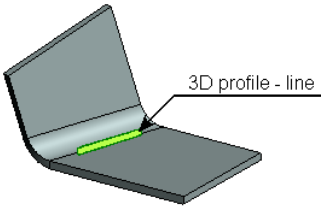
Neutral factor (the neutral layer factor) – a parameter defining the position of the neutral layer in the workpiece. The neutral layer of a workpiece is its section parallel to the main faces, whose geometry does not change in bending. All geometrical parameters of bending (the bending radius, the length of the attached piece, positioning with respect to a specified bending line) are calculated with respect to the neutral layer. This parameter value can be defined by the system automatically, based on the table of Neutral factor values versus the ratio of the bending radius and the workpiece thickness. The Neutral factor dependency table can be manually modified by the user as desired.



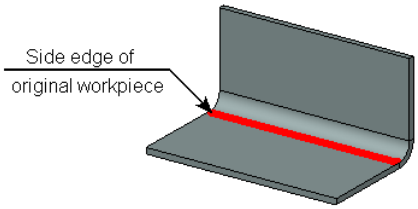
Main Concepts of Bending

Bend operation is defined by a *base line*, specified by two 3D points or a 3D object suitable for defining a straight line. The base line serves several purposes: it allows the system to define the body being deformed and the base face of bending, determines the position and size of the bended area, determines the possibility of creating one or another type of bending.

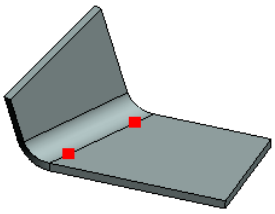
The base line must lie in the plane of one of the workpiece faces being bent (normally, the top or bottom face). The base line does not have to fully fit within the boundaries of a base face; however, at least some its portion must belong to the face.



Base line defined by an edge of a straight open profile



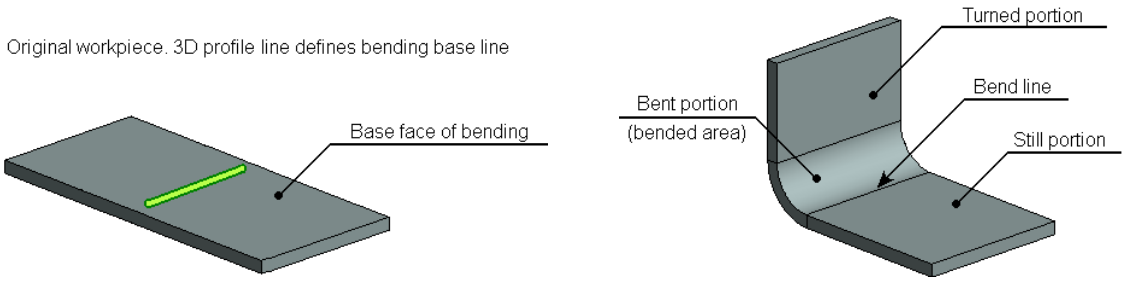
Base line defined by an edge of original workpiece



Base line defined by two 3D Nodes

The face that holds the base line is the *base face* of bending. The base face is used as the base reference when defining bending parameters. The body to which the selected face belongs is actually the workpiece being transformed.

Transformation of the workpiece is done in the following way. The base face and all original body are divided by the base line into three portions: still portion, bent portion (the bended area) and the turned portion. The still portion is left unchanged by the operation. The bent portion is replaced by a cylindrical element of the appropriate shape, with the radius equal to the bending radius. The turned portion is turned about the axis of the bent portion by the bend angle.

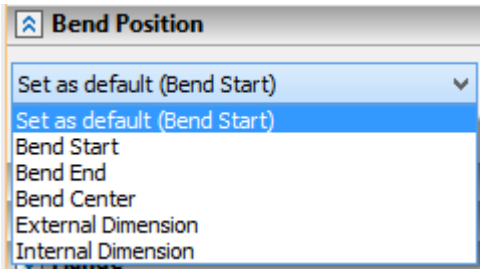


The border line between the still portion of the workpiece and the bended area will be referred to as *Bend line*.

Bend line position relative to base line

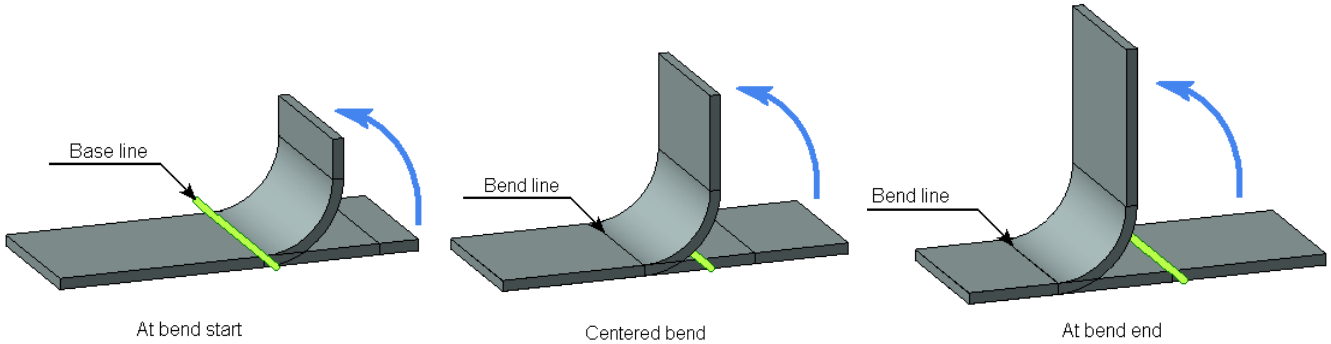
The kind of bend line (its length and position) depends on the specified base line, and type and parameters of bending. The bend line can have the same length as the base line, or be longer than the other one, passing through all base face, or, on the contrary, be shorter than the base line. The position of the bend line with respect to the base line is defined by the location of the bended area with respect to the base line of the bending.

A position is specified in Bend Position section of the properties window of the operation.

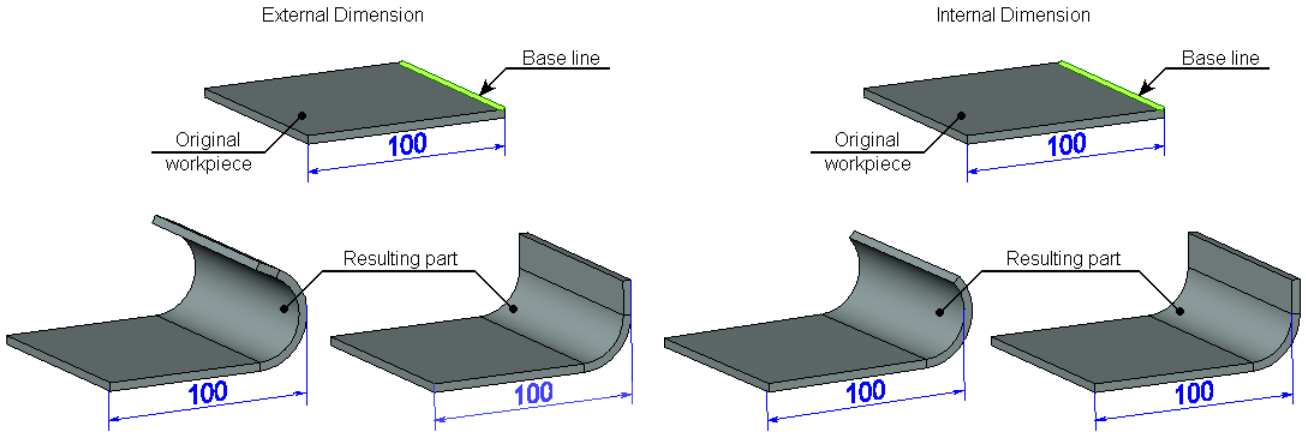


There are following possibilities:

- *Bend start* - The beginning of the bended area is aligned with the base line (used as default);
- *Bend center* - The bended area is centered about the base line;
- *Bend end* - The end of the bended area of the workpiece is aligned with the base line.



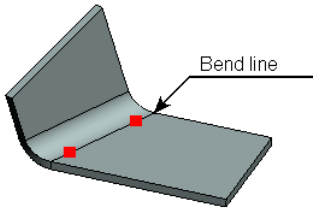
- *External Dimension/Internal Dimension* – the bend area is positioned in such a way that the overall size of the resulting part measured from the outer/inner wall of the bent/attached flange to the opposite trimmed face of the part remains the same as for the original part.



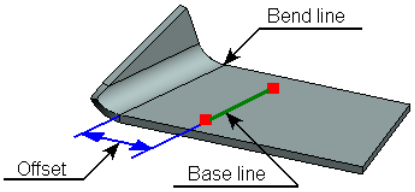
The base line location at the start of the bended area is most common. This configuration will be used throughout the further description of Bend functionality.

Shifting the bend line off the original position

An auxiliary parameter *Offset* allows shifting the bended area and, therefore, the bend line off the base line. The bended area shifts to either side of the original position, depending on the offset sign. Positive offsets are towards the bent (attached) portion, negative – away from it.



Original location of bended area (no offset).
Base line defined by two 3D Nodes



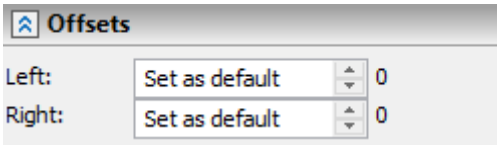
Bended area under positive offset.
Base line defined by two 3D Nodes.

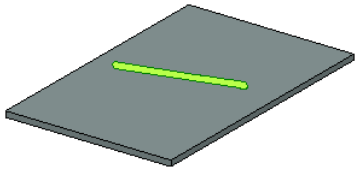
Shifting the bend line often helps in the cases, when the user is not satisfied with the location of the bended area, or when bending is outright impossible in the current location.

Offsets from base line ends

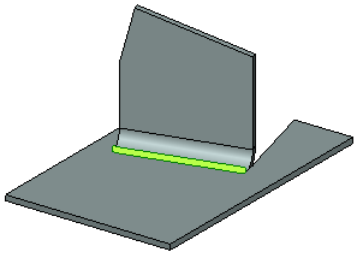
The length of the bend line can be adjusted by *Offsets* from the ends of the base line.

Offsets are specified in the corresponding section of properties window.

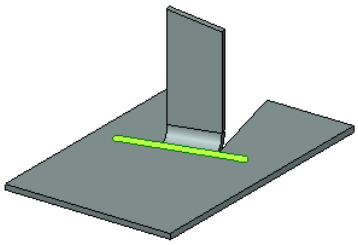




Original workpiece. 3D profile edge defines base line



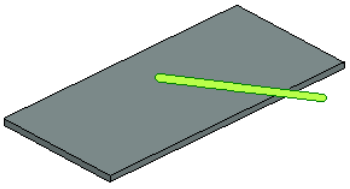
Bending of a portion of workpiece along base line without offsets



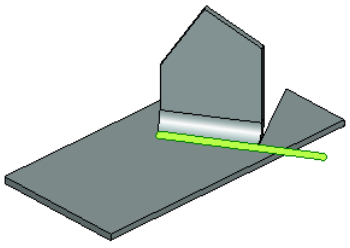
Bending of a portion of workpiece along base line with offsets from either end of the line

Offsets are introduced in the cases when the selected base line exactly defines the desired position of a bend, but is longer than the portion of the workpiece to be bent.

In some cases, use of offsets allows creating a bend even if the original base line was incorrect. This could be, for instance, the situation, when the base line for the bend does not fully fit in the face of the workpiece. Without using offsets, bending cannot be created in this case. By offsetting from the line end, one can reduce the length of the bend line in such away, that the bend area fully fits in the bending base face.



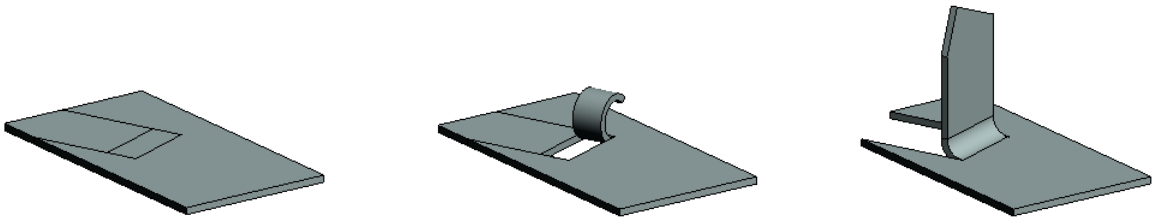
Originally, bend cannot be constructed, as the bend line is not fully in the face



Bend is created after defining offset from line end

Flange Reduction

In many instances of bending or attaching a flange, the side walls of the bent "ear" touch the faces of the workpiece base. During further manipulations with such a bend, the bent "ear" may "stick" to the workpiece elements it is touching. This causes errors in successive unbendings and rebendings of the model. Should such situation occur, we recommend using capability of automatic width reduction of the bent or attached portion in order to introduce gaps between it and faces of the workpiece base. The reduction amount is constant and is set to 0.05 millimeter on each side of the "ear".



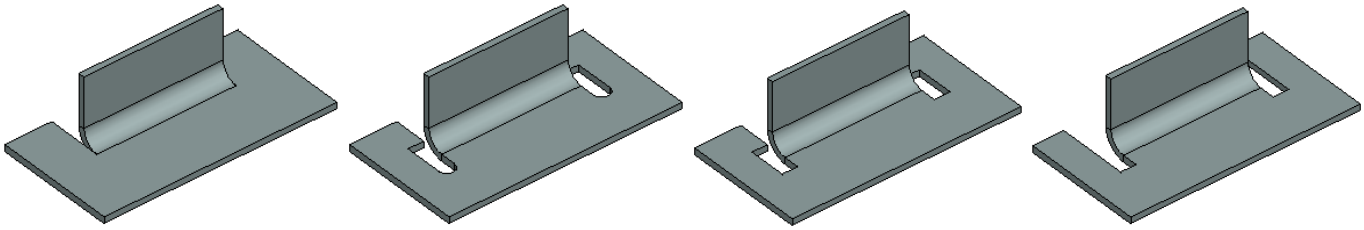
Unbent part | Re-bending error (without flange reduction) | Correct re-bending (with flange reduction)

When attaching a flange inside the workpiece with material removal by the length of the flange, the flange length will also be reduced by the same amount as the width.

Corner Relief in Sheet Metal

When making parts by bending sheet metal, stresses develop in the metal around bend corners. To treat stresses that may cause cracking, special slots can be introduced around the bend.

"Bend" operation supports creation of such slots – *reliefs*, introduced automatically when bending and attaching flanges. Introduced reliefs can have round or square shape.

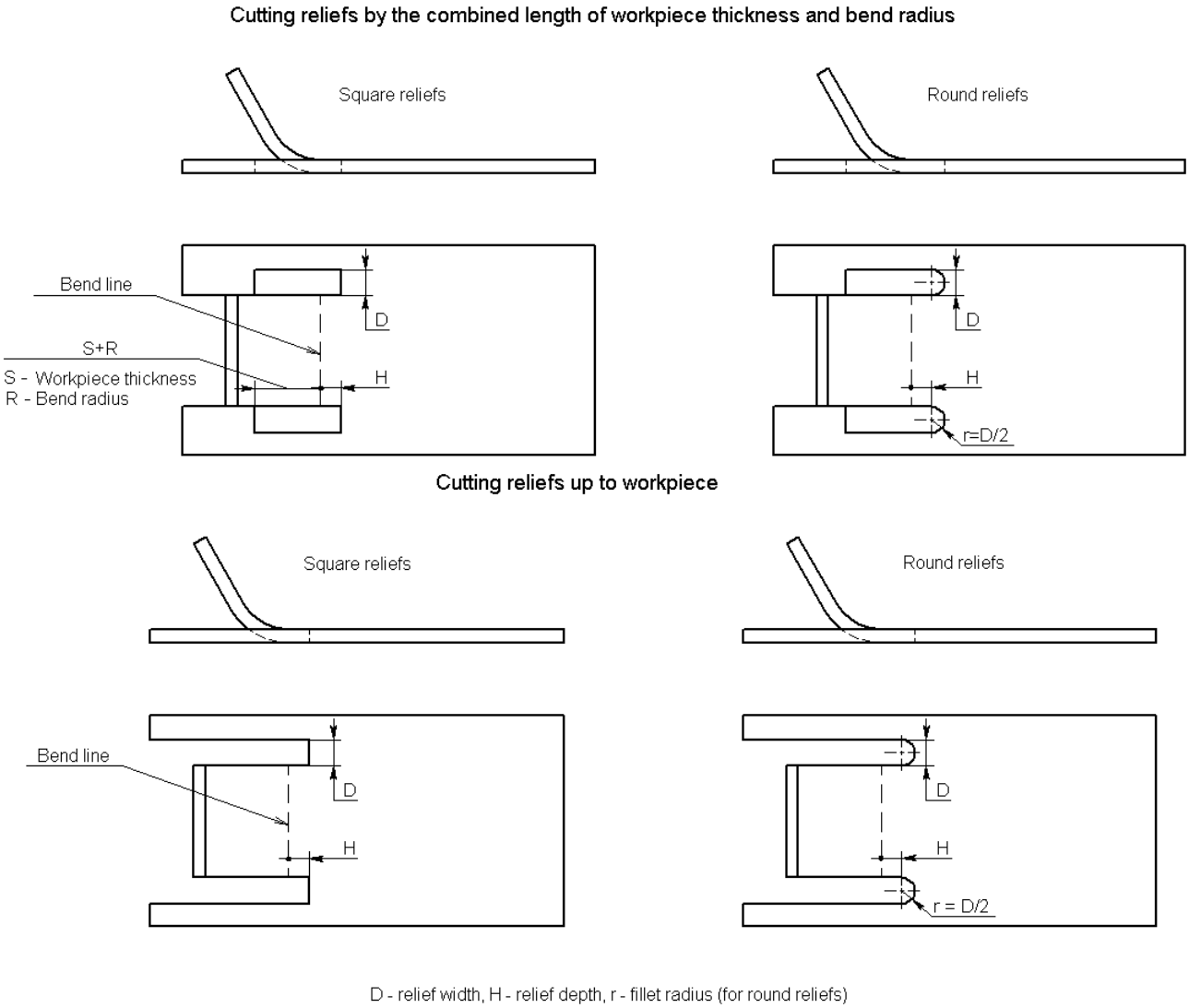


No reliefs | Round reliefs | Square reliefs | Square reliefs extended up to the borders of base face

Dimensions of a relief slot are controlled by two parameters: *"width"* and *"depth"*. For a square relief, those define the width of the slots and the length of the slot portion ahead of the bended area, respectively. In the case of the around relief, the length of the head portion of the slot is increased by the radius of the end fillet. The fillet radius is equal to half the slot width.

The rear portion of a slot of any type can be arranged in one of two ways:

- have the length equal to bend radius plus workpiece thickness;
- *extended* up to the borders of the workpiece (or to the end of the punch, if attaching a flange inside the workpiece with material removal by the flange length).



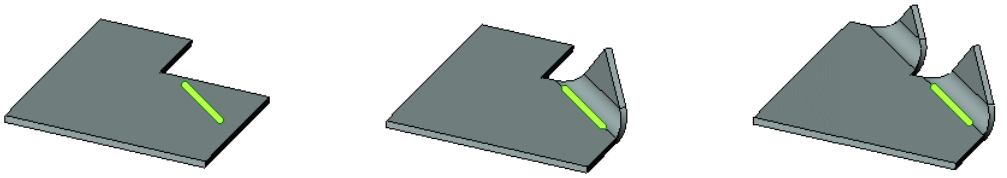
D - relief width, H - relief depth, r - fillet radius (for round reliefs)

Bend

In plain Bend, the original workpiece is just bent along the base line, without incisions or material removal. In this type of bending, the bend line is formed by extending the base line segment at either end until intersection with side edges of the base face. There are two ways of extending the base line:

Through the face – the bend line extends to the outermost edges of the base face.

To nearest edges – the bend line extends to the nearest edges of the base face (only a local protrusion of the workpiece is bent).

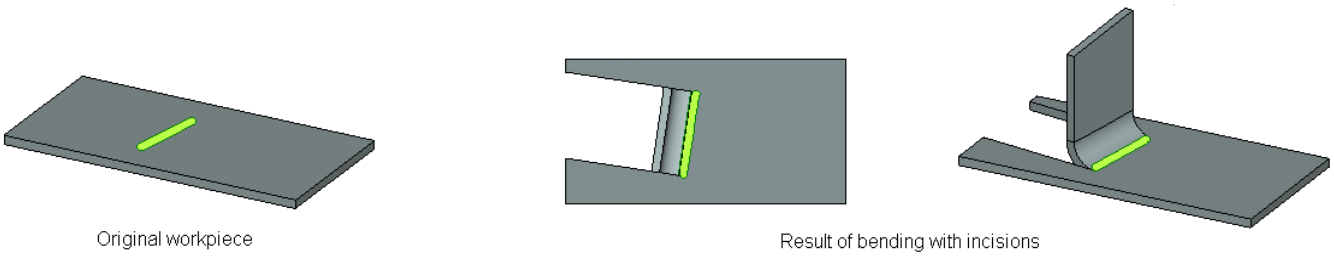


In plain Bend, offsetting from the ends of the base line is not supported.

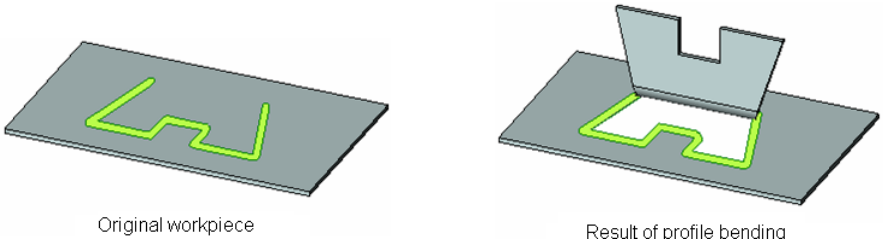
Cut and Bend

This is bending with incisions, when a portion of the workpiece between cuts is bent. The cutting lines of the bent portion originate from the ends of the base line (or from the points defined by offsets from the ends of the base line), perpendicular to the base line and extending to the boundaries of the base face.

This type of bending requires the bend line to be a segment lying fully within the base face. You can also define the base line by one of the side edges of the workpiece. However, in this case you must define an offset for the bend line.



A special type of cut-bending is *profile bending*. An open 3D profile created on a face of the workpiece defines the pattern for the flange being bent. The base line is determined by the profile end points.

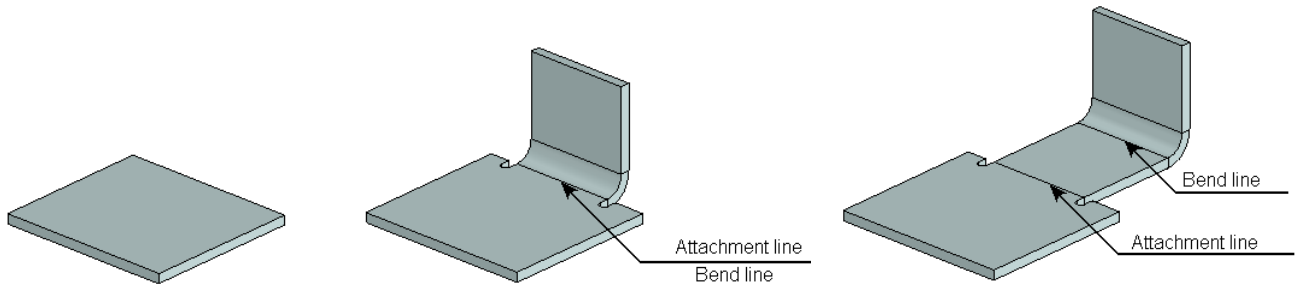


In profile bending, the base line always defines the start of the bent area. You cannot specify any offset for the bend line or offsets from the ends of the base line in this case.

Flange Attachment

Attach Flange mode is convenient in the cases when you do not know in advance even an approximate pattern of the part being designed. In this case, start with a simple-shape workpiece (commonly, a flat block). Then, work it out by adding rectangular "ears" of the specified length in the desired locations, and bending those off. The thickness of the attached "ears" is the same as the thickness of the workpiece, while the width is defined by the base line (including offsets).

To proceed with the description of attaching, let us introduce an additional term for this type of bending –*Attachment line*. By this term, we mean the line along which the flange is attached to the original workpiece. Initially, when creating a bend, the attachment line coincides with the bend line. In other words, the flange is bent along the same line as it is attached. In this way, the position and the length of both lines is defined by the base line of bending (including offsets) and by the location of the bended area with respect to the base line. The attachment line and the bend line can be separated by offsetting the bend line from the base line. In this way, the bended area of the flange will be shifted away from its attachment.



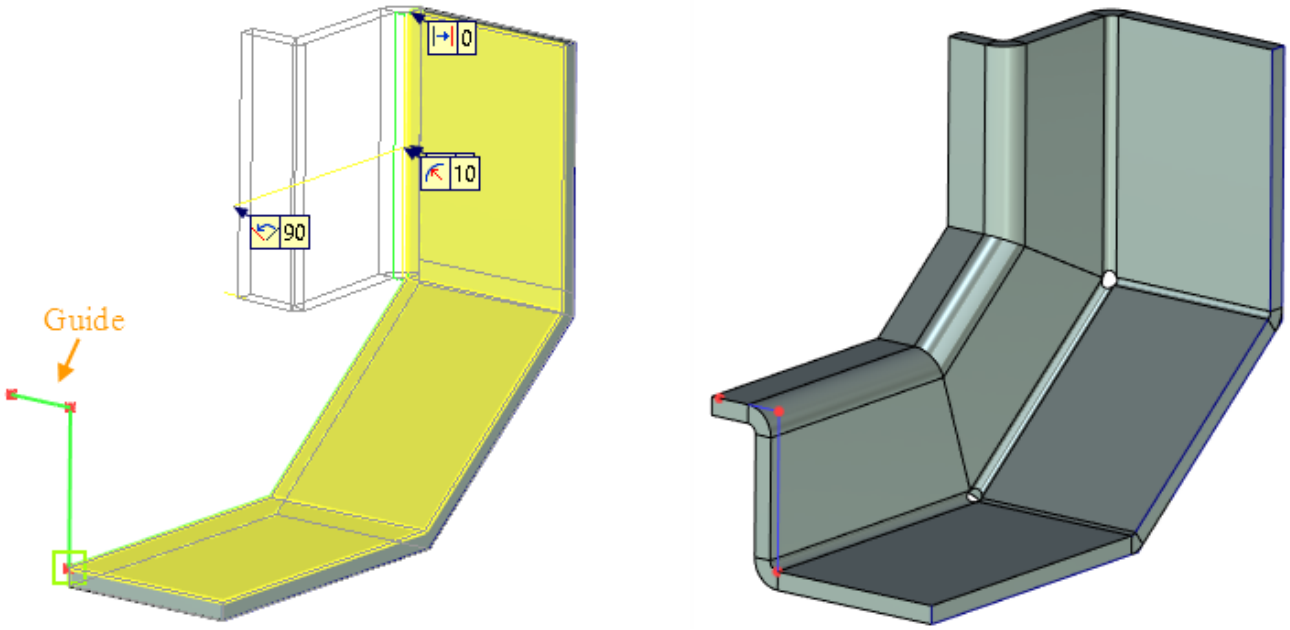
Original workpiece
Base line is the side edge

Flange attachment line is coincident with bend line

Distinct flange attachment line and bend line

Contour Flanges

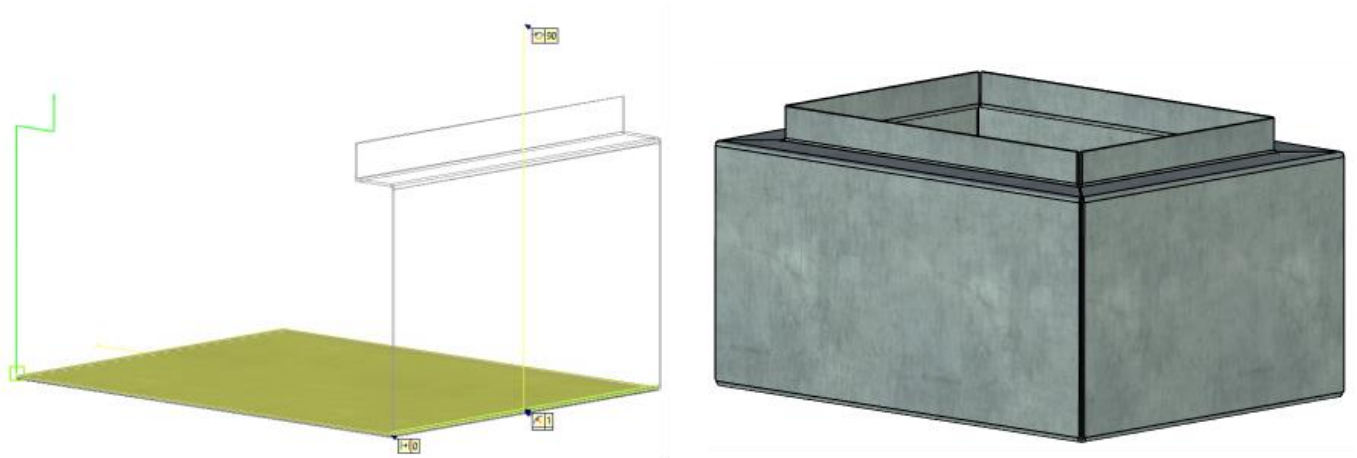
A contour flange can be attached for workpieces. A contour flange is the sheet metal operation, which shape is specified by a guide curve. The result of this operation is attached to the edges of the base sheet body, forming a single model.



You need to select a profile or a path as a flange guide to determine the flange geometry. Flange guide cannot be closed. All points should locate in one plane. The flange guide can be set by straight lines or circular arcs.

It is necessary to select a rib to which the contour flange will be attached and a flange guide. After that you need to select adjacent point and profile direction. If the guide is already adjacent to the rib, the contour flange is created immediately after guide curve selection.

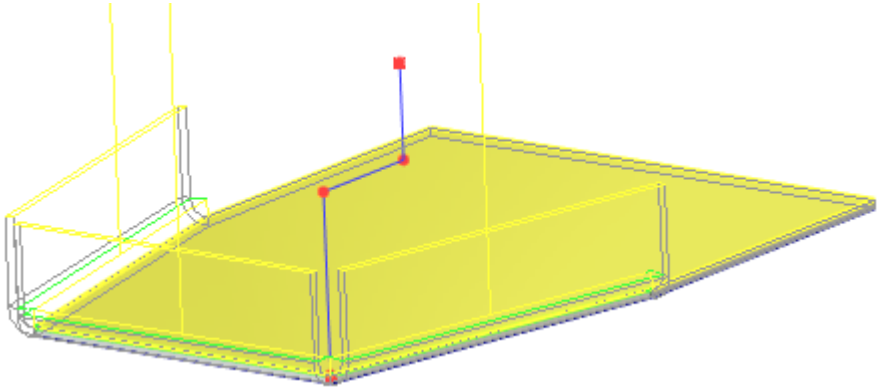
Flanges can form a closed contour. You can specify bend relief and gaps for the flange junctions.



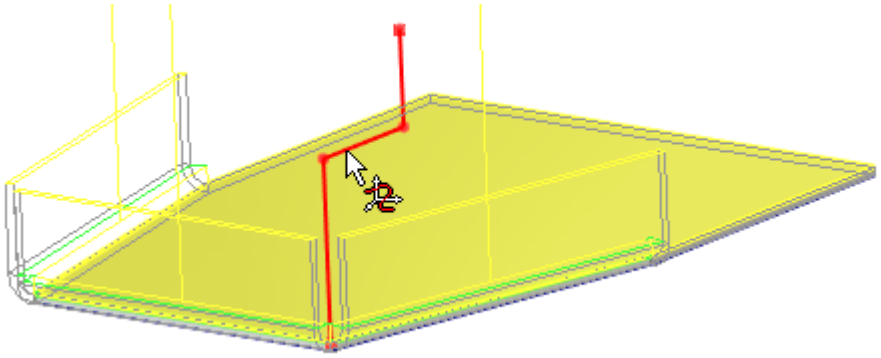
Contour Flange Creation

To create a contour flange you should:

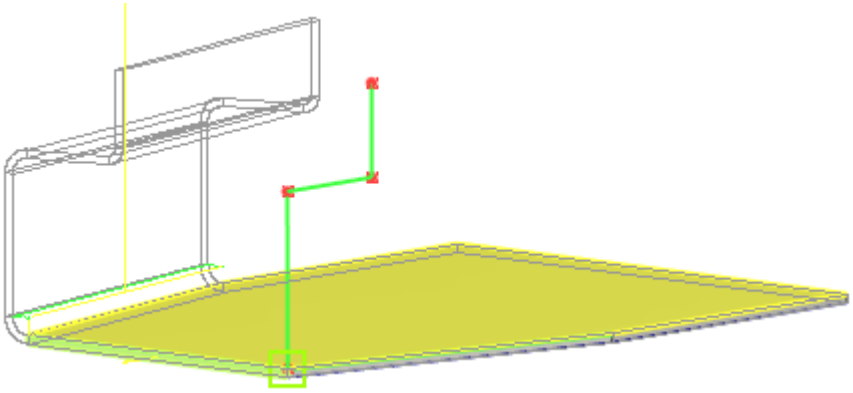
1. Select edges for the operation using **Add Edges** option.



2. Select guide – existing profile, route or path using **Select flange guide** option.



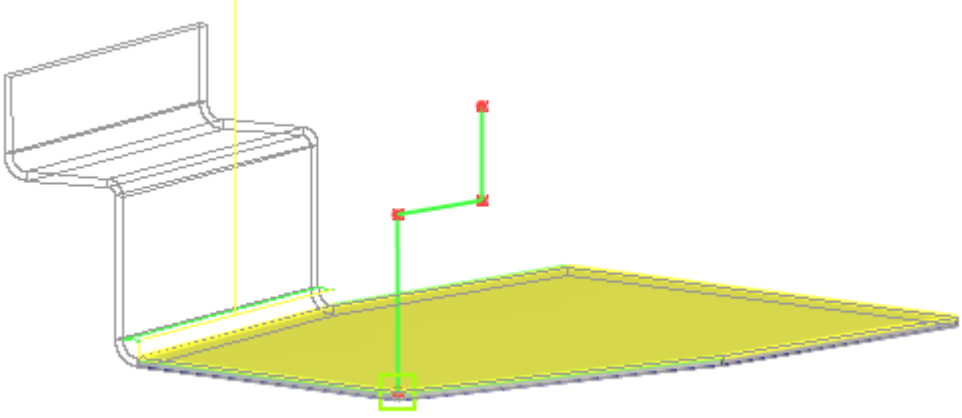
3. If the selected guide is already adjacent to the rib, the contour flange is created immediately after guide curve selection.



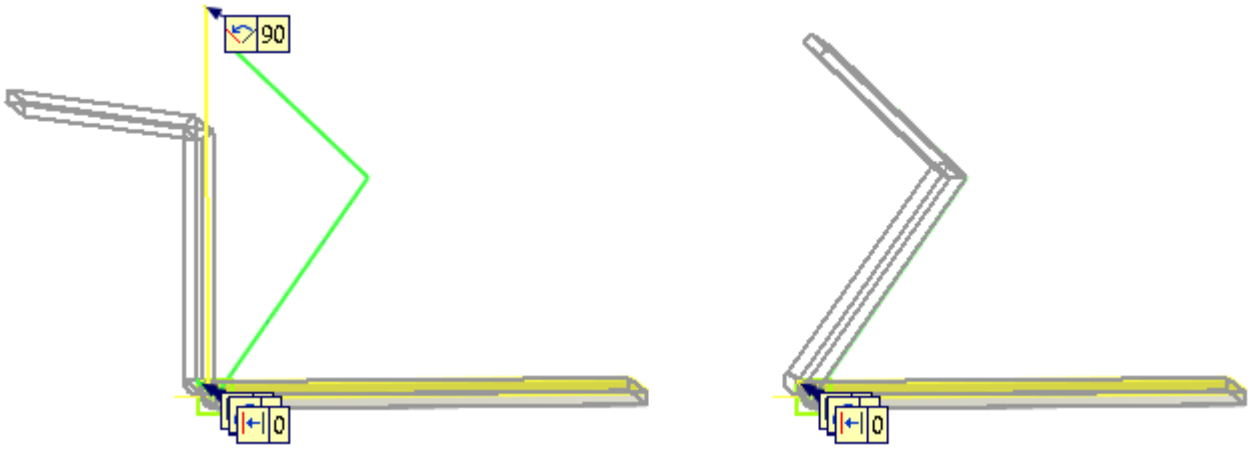
Otherwise, you may use Select adjacent point option and select a point on the guide.

Upon creating several flanges for a single workpiece, manipulators and decorations will be displayed only for the last created flange. This is done in order not to complicate the visual perception of the model.

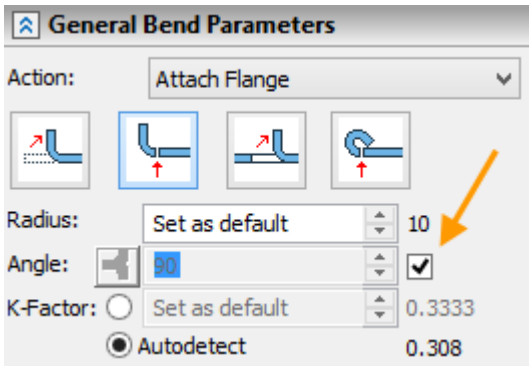
4. A direction of the flange can be changed using **Select profile direction** option. You need to specify a direction using existing geometry.



When you create a contour flange, you can specify angle for the flange manually or you can use a flange guide angle.



To specify the parameter use the **Use flange guide angle** flag near the **Angle** field.

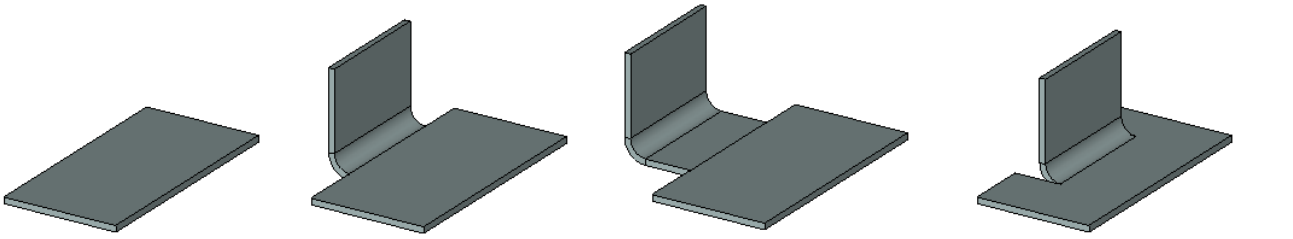


Shifting bend line off the base line

The result of attaching a flange with offsetting the bend line from the base line depends on the offset direction (sign).

When the offset is positive (that is, offsetting from the workpiece onto the flange), the attachment line is unchanged, that is, defined only by the position of the base line. At the same time, the flange bend line shifts by the amount of the offset. As a result, the flange being attached gains additional flat portion whose length is defined by the offset. This length of the flat portion is not included in the flange length defined in the operation parameters.

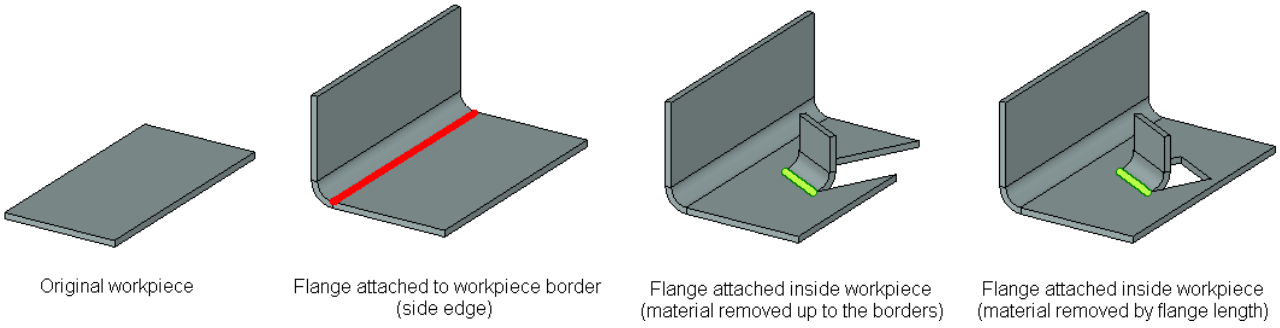
When the offset is negative (that this, offsetting onto the side of the workpiece), both lines shift – the attachment and the bend line.



Original workpiece | Flange attachment without offsetting bend line | Flange attachment with offsetting bend line (adding flat portion) | Flange attachment with offsetting bend line (the flange attachment line shifts together with the bend line)

Flange attachment inside workpiece

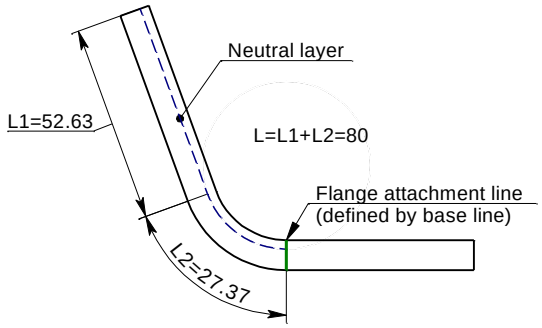
In most cases, flanges are attached to one of the side edges. This is done by selecting the desired edge of the workpiece as the base line, without offsetting. However, attachment can be done inside workpiece as well. This would happen if the base line were defined by an edge of a 3D profile lying within the face of the workpiece, or if attachment were made to a side edge with a negative offset. In this case, the material of the workpiece on the way of the attached flange is removed. The width of the removed portion is determined by the perpendiculars to the base line (including offsets), constructed on its ends, just like in the case of incision bending. In this way, two kinds of removal are possible: either all material is removed up to the extents of the workpiece, or just the portion thereof by the length of the attached flange.



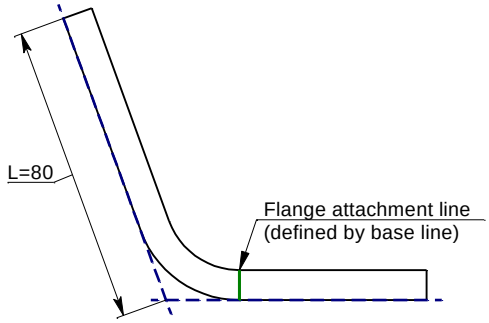
Original workpiece | Flange attached to workpiece border (side edge) | Flange attached inside workpiece (material removed up to the borders) | Flange attached inside workpiece (material removed by flange length)

Calculation of attached flange length

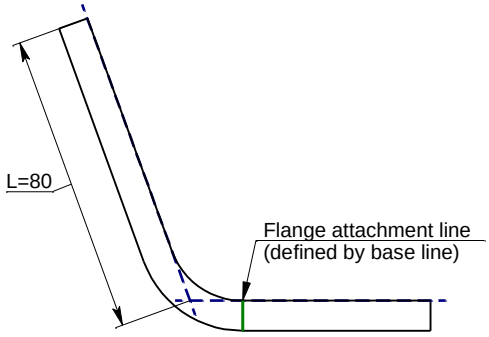
The length of the attached flange can be calculated in four ways: *by neutral layer*, *by external side*, *by internal side* or *by flat part* of the flange being attached. The following diagrams show the respective cases of length calculation when attaching a flange of length L=80 millimeters.



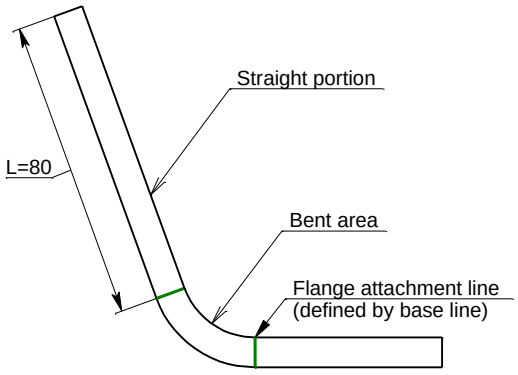
Flange length calculation by neutral layer



Flange length calculation by external (outer) side



Flange length calculation by internal (inner) side

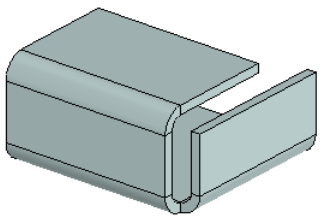


Flange length calculation by flat portion

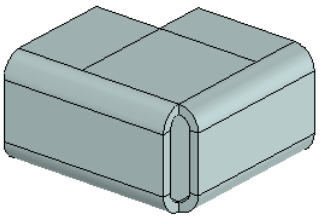
Intersecting ends trimming

When attaching a flange, intersections may occur between the flange and the workpiece. In such a case, automatic trimming can be performed. Trimming is done in such a way to leave a diagonal gap of the specified width in the intersecting area.

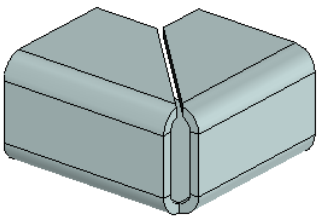
Trimming is possible if the flange and the intersected portion of the workpiece have the same thickness and lie in one plane. Certain exceptions from these rules are allowed, such as tolerance on flange thickness and bending angle when matching with the workpiece.



Original workpiece



Flange attached without trimming



Flange attached with trimming

Hem

Operation attaches a hem of the selected shape to an outer contour of a worpiece. You can select several workpiece ribs that belong to the same face. Flanges can form closed contours.

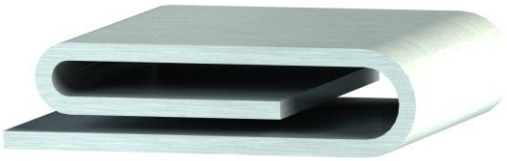
There are six types of hem available:

Rolled

Open

Teardrop

Closed



Double

S- shaped

Creating Various Types of Bend

The command **SMB: Create Bend** is used for creating various types of Bend:

Icon	Ribbon
	3D Model → Special → Sheet Metal → Bend
Keyboard	Textual Menu
<SMB>	Operation > Sheet Metal > Bend

Upon calling the command, do the following steps:

- ✓ Define the bend line (if not selected automatically upon entering the command) and the bend type (if other than the system default). You can select several bend lines for Attach Flange and Hem operations.
- ✓ Select hem type (only for Hem operation).
- ✓ If necessary, select a different face to bend and the side of bending. Not available for Hem operation.
- ✓ Define the shift of the bend line and offsets from its ends. In this way, you adjust (if necessary) the location and width of the bent (attached) flange. Not available for Hem operation.
- ✓ Modify (if necessary) the default Bend parameters (the bend radius and angle, Neutral factor, the flange length and the way of measuring it when attaching the flange).

- ✓ Modify (if necessary) the default type and dimensions of reliefs.
- ✓ Confirm operation creation.

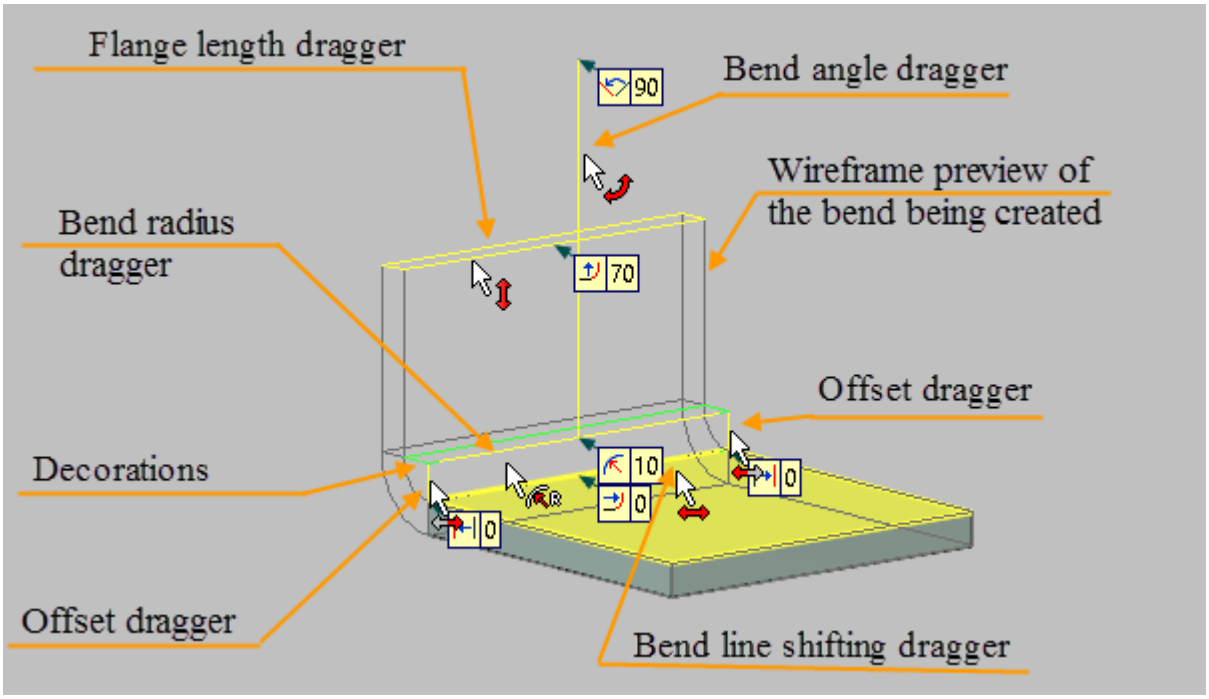
Available options and parameters set can differ for various types of bend.

Auxiliary Bend elements displayed in the 3D scene

During bending (after selecting the base line), auxiliary graphic elements - draggers and decorations - are displayed in the 3D scene. Those schematically outline the shape and size of the bend being created. Additionally, each of those objects has its specific purpose. The draggers serve for modifying the bend dimensions directly in the 3D scene. The decorations let the user know whether the bend is possible or impossible under the specified conditions.

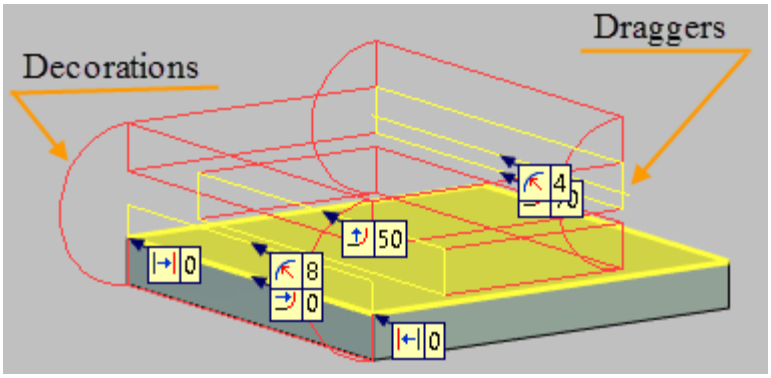
When the bend is possible, the wireframe preview image of the operation is displayed in the 3D scene over the *blue* decorations (by default).

Use of a dragger in bending is similar to using draggers in other 3D commands. As the pointer approaches to the dragger's image, it gains a respective glyph. If you start moving the mouse with the depressed at such moment, the wireframe image of the operation being created will rubberband, and the respective parameter update in the property window.



If the conditions are incompatible, no bend image is displayed, while the decorations change the color to the forbidding *red* (by default). In such a situation, either redefine the base line or fix the bend by shifting the bend line off the base line or offsetting from its ends. If, however, no decorations or draggers are displayed in the 3D scene upon specifying the base line, that means, the system could not find any face

suitable for creating a bend about the specified base line. In this case, the only option is specifying a new base line.



Altogether, six types of draggers can be found in the 3D scene. Those are the bend angle, bend radius, bend line shift from the base line, length of the attached flange and offsets from either end of the base line. Depending on a particular bend type, some draggers may not be there. For example, the flange length dragger is only used when attaching a flange. In the plain bend, there are no draggers for defining offsets.

Defining bend line and bend type

To define the base line by specifying a 3D element suitable for describing a line, activate the following option in the command automenu:

	<L>	Select Line

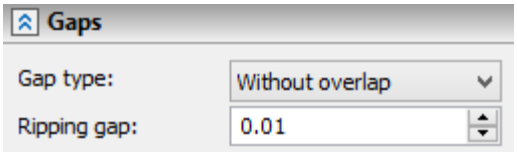
This option allows selecting a 3D element suitable for defining the base line of the bend. The drop-down list behind this option contains filters for selecting certain objects.

Add edges option is available for Attach Flange and Hem operations.

	<M>	Add edges

The option allows to select several consequent edges that belong to the same face of the workpiece.

Gaps section appears in the **Properties** window if you select more than one edge. There you can set **Gap Type** and **Ripping Gap**.



To define the base line by two 3D points, use other automenu options (activated successively):

	<S>	Select first Vertex or 3D Node
	<E>	Select second Vertex or 3D Node

The set of admissible elements for selection is defined by the state of the filters in the drop-down lists of the described options.

The base line of the bend or one of its defining 3D points will be selected automatically, if calling the command while drawing on active workplane or from the context menu of the appropriate 3D element.

The base line for bending by profile is defined by the option:

	<R>	Select profile to bend

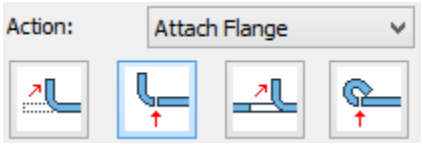
Upon turning on this option, you can select a 3D profile that will be used as the pattern for bending the flange and simultaneously define the base line.

Once the base line is defined, the system assesses possible bending configurations. The faces will be identified that host the specified line. One of those faces is determined as the subject to bending and is highlighted in the 3D scene (by default – *yellow*). If the system identifies more than one suitable face, the automenu will provide the option for selecting other faces:

	<F>	Change active face

After that, the system will tentatively define the type of bending, depending on the specified base line. The system choice depends on the type and way of defining the base line. For example, if a side edge was selected as the base line, the system will offer attaching a flange. If the option is used, the Cut and Bend mode will be set. In other cases, bend is assumed.

The choice is reflected in the command dialog in the property window.



The system-offered choice can be altered by selecting another bend type from the list. The four options in the list correspond to the four described bend types: Bend, Attach Flange, Cut and Bend, Hem.

If the face is identified and a bend can be created in the specified bending area without adjustments (such as shifting or offsetting), then the wireframe preview of the future bend will be displayed in the 3D scene. The bent side of the workpiece (with respect to the base line) is selected by the system automatically. The user can alter it by the automenu option:

	<D>	Change the part to be bent

When making a flange by profile, the bent part cannot be modified.

Defining bend parameters

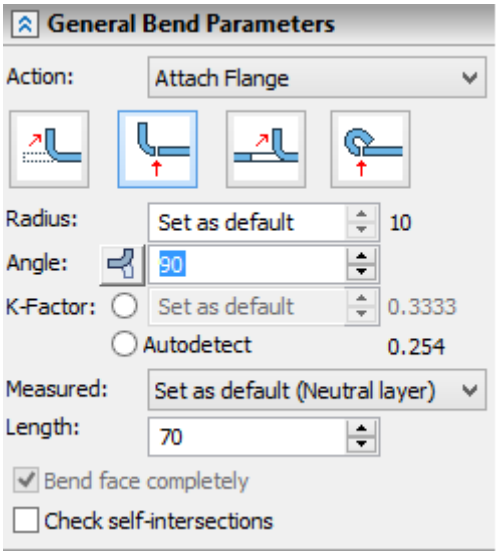
The values of main geometrical parameters of a bend can be easily and quickly modified directly in the 3D scene by draggers. Those include the radius, angle, flange length, shift from the base line and offsets

from its ends. Adjustments to the parameter values can be done in the property window or in the parameters dialog box invoked by the option .

All geometrical parameters of a bend can be defined in the property window. As was mentioned earlier, you can select there the bend type (the item "Action"), specify a numerical value of the bend radius and angle (the entries "Radius", "Angle"). An additional button next to the angle input box allows quickly flipping the bending direction by 180^0.

Neutral factor of the bend is controlled by the radio group "Neutral factor": "From Table" – directs the system to take a value from the table (the resulting value is displayed next to the item); the other choice allows entering an arbitrary value.

The entry "Length" is accessible only when attaching a flange. It allows you to specify the length of the flange being attached. The way of measuring it is controlled by the parameter "Measured".

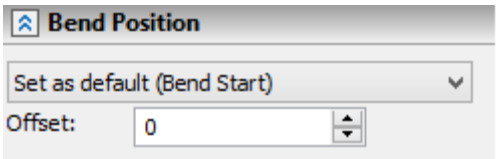


Bend face completely. This parameter is important when performing operations on the sheet metal of certain geometry (for example, when it is required to bend the sheet body with a cut along the specified line). When this parameter is enabled, the entire body along the selected line will be bent without violating the construction logic.

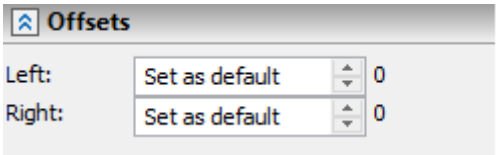
Check self-intersections. This parameter is important when creating the «Bend» parameter. It allows us to monitor self-intersection of bent sheet faces. When the flag is set system doesn't allow creating bend, which result leads to an intersection with source body.

The group "Bend Position" permits selecting the desired location of the bend with respect to the bend line ("Bend start", "Bend center", "Bend end", "Default", "External Dimension", "Internal Dimension") and specify the line shift.

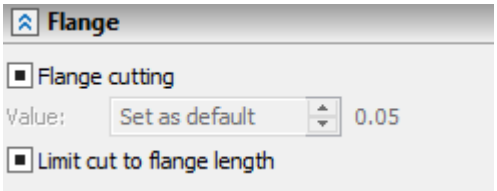
The rest of the parameters in the dialog are accessible only when bending with incision or attaching a flange (the settings "Cut and Bend" and "Attach Flange" in the "Action" combo box).



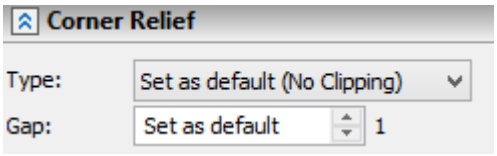
Offsetting from the ends of the bend line is defined in the group "Offsets". Offset amounts from the left and right ends of the bend line are specified separately.



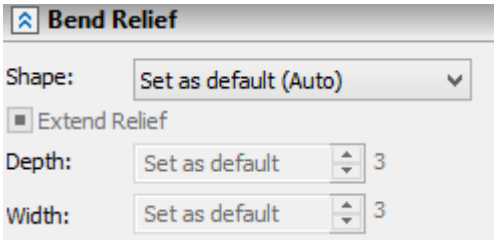
An additional flag Flange cutting is recommended in the cases when the side faces of the bend flange intersect with the faces of the workpiece base. This helps error-free unbending and repeated bending.



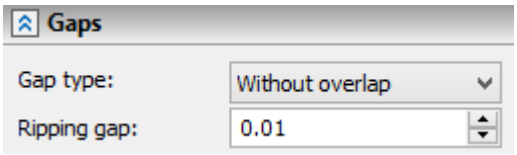
Trimming of the intersecting portions of the flange being attached and the workpiece base is defined in the group "Corner relief" (those parameters are available only when attaching a flange).



The group "Reliefs" in the dialog defines the shape and dimensions of relief slots created with the bend. First, select a relief type in the parameter "Type": "Round", "Square" or "None" (in the latter case, reliefs are not created). If one of the first to options was selected, the input boxes become accessible that define dimensions of the slots. Those are the depth (length) of the relief portion ahead of the bend line ("depth") and the slot width ("width"). Setting the flag "Extend relief" makes the relief slots cut up to the borders of the workpiece.

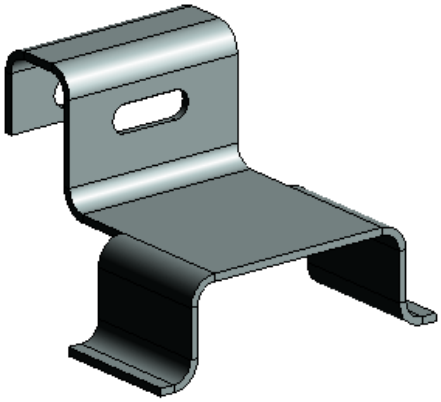


Gaps section appears if you select more than one edge using Add edges option. There you can set **Gap Type** and **Ripping Gap**.

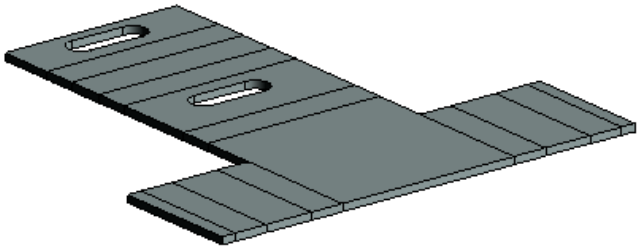


UNBENDING

Once the desired part model is created, you can use a special Unbend operation. It allows "unfolding" the part and getting the unfolded pattern model of the design. This may be needed, for example, for drafting the part unfolded pattern.



Original part

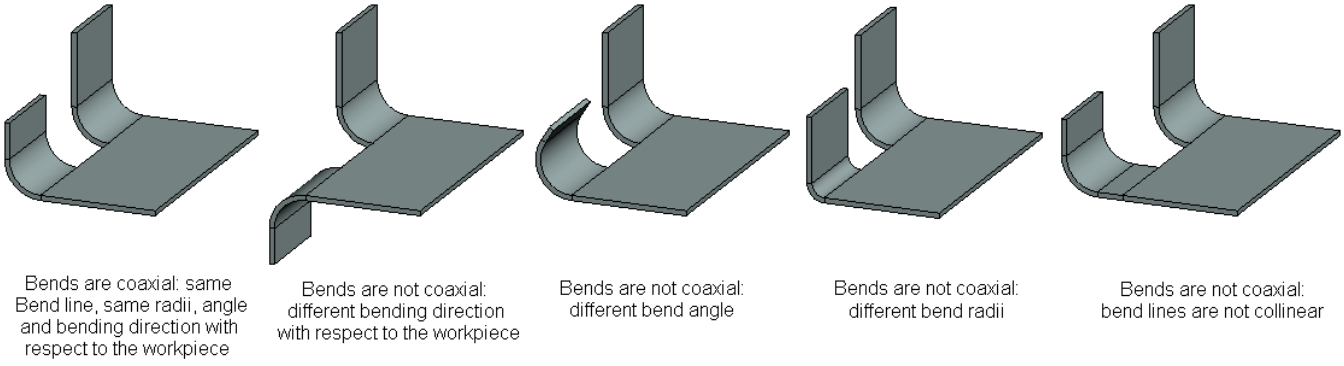


Unfolded pattern

This operation is more than just unbending genuine sheet metal parts. Imported bodies can also be "unbent", as well as cylindrical faces resulting from extrusion or rotation. Several bends can be unbent simultaneously.

Bends to be straightened are selected by picking cylindrical faces created by bending. You can unbend all coaxial bends by selecting just one of them.

Coaxial are bends satisfying the following conditions: they belong to one body, use collinear bend lines, have the same radius, angle and direction of bending with respect to the workpiece.



When unbending, the workpiece neutral layer factor is significant. If a part is being unbent that was not bent from sheet metal, the user shall define the neutral factor in the command parameters.

Unbend operation preserves all topology previously created by bending. That means, the number of faces and edges is the same before and after unfolding. This allows determining bend areas in the resulting flat pattern and use those, for example, in repeated bending.

Unbend Operation Creation

To flatten bends, use the command **SMU: Unbend Bend**:

Icon	Ribbon
	3D Model → Special → Sheet Metal → Unbend
Keyboard	Textual Menu
<SMU>	Operation > Sheet Metal > Unbend

Bends and other cylindrical body elements subject to flattening are selected by picking their defining cylindrical faces. Upon launching the command, the option is automatically activated in the automenu for selecting faces to flatten:

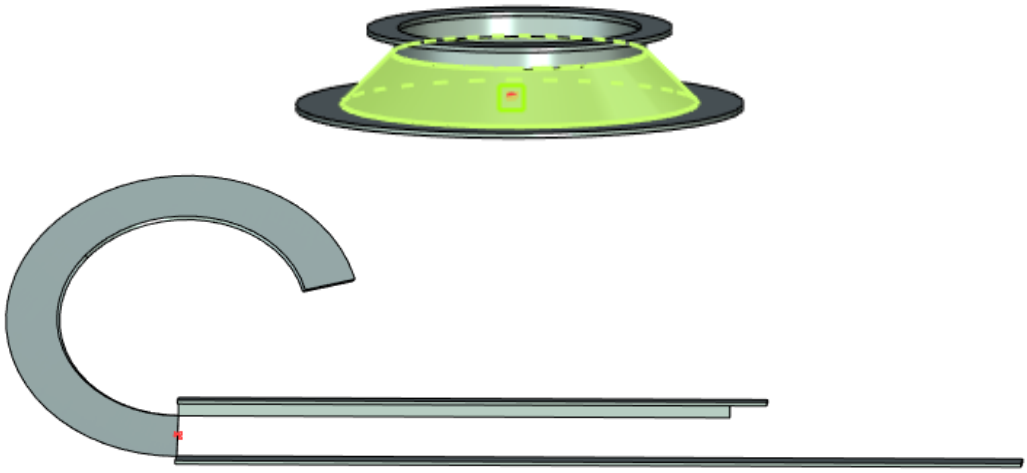
	<E>	Select cylindrical bend face

To select a face, simply click it with in the 3D scene or pick the respective operations in the model tree. The selected faces are highlighted in *green* (by default).

When you select closed face of the body the following option becomes available:

	<P>	Select node to define split surface

The option allows to select a 3D node to create a split surface.



You can determine unbending direction for closed contour using option:

	<D>	Select unbending direction

All bends in a body can be quickly selected by the option:

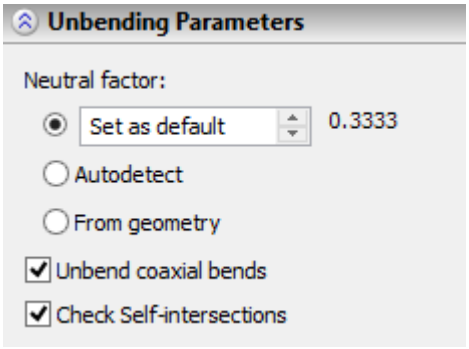
	<A>	Select operation

The portion of the part left still in unbending is determined by the system automatically. By default, such face is marked *yellow* in the 3D scene.

To alter the result, the still face can be manually selected by the automenu option:

	<K>	Select immovable unbending face

The Neutral factor for unbending selected faces is picked from the parameters of the respective bends. In the cases when the faces were created by other means than bending, the Neutral factor shall be defined in the command dialog in the property window.



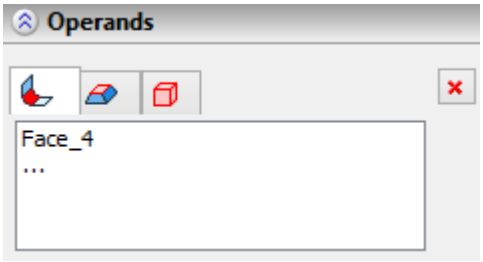
Autodetect. The factor is determined from the table according to the model thickness.

From geometry. The factor is determined according to the neutral factor for one of the bended faces, used in unbending.

All faces used in the unbending operation have the same neutral factor.

The flag Unbend coaxial bends (set by default) forces unbending of all bends coaxial with the selected one. When cleared, only the selected bends are flattened.

Check self-intersections. This parameter is important when creating the «Bend» parameter. It allows you to monitor self-intersection of bent sheet faces. When the flag is set, system will not create bend that intersects with the source body.



All deformed faces, fixed faces and operations are displayed in the **Operands** section.

RE-BEND

Re-bend operation restores surface bends previously flattened by Unbend operation. This functionality is convenient in conjunction with saving the unfolded pattern instance in the model structure. For example, Unbend and Re-bend operations can be conveniently used for drafting the unfolded pattern and the formed part in the same drawing.

Re-bend uses the same workpiece neutral factor as that used in unfolding.

Re-bend Operation Creation

For repeated bending of all surfaces flattened by Unbend operation, use the command **SMR: Re-bend**:

Icon	Ribbon
	3D Model → Special → Sheet Metal → Re-bend
Keyboard	Textual Menu
<SMR>	Operation > Sheet Metal > Re-bend

Upon launching the command, the option is automatically activated in the automenu for selecting faces to be bent:

	<E>	Select face for bending

To select a face, simply click it with in the 3D scene. Selected faces are highlighted (by default – *green*).

To select all faces of unbending of specific operation used option:

	<A>	Select operation

The piece of the part, which should remain immovable when bending, is selected by the system automatically. Such a face is highlighted *yellow* (by default) in the 3D scene.

To alter the result, you can manually select the still face by the automenu option:

	<K>	Select immovable face

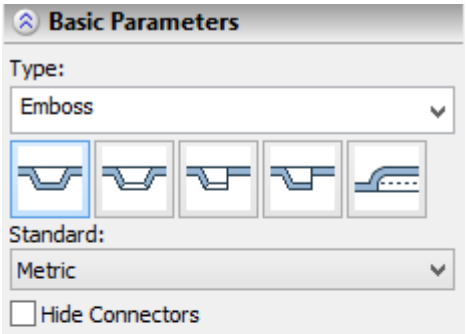
The option is not available if the **Select operation** option is used.

Library of Frequently Used Forming Features

You can create forming features on bodies using Forming features operation. Forming feature from standard or custom libraries can be used.

To create one of frequently used sheet metal forming features, use the command **SMF: Create Sheet Forming Metal Feature**:

Icon	Ribbon
	3D Model → Special → Sheet Metal → Forming Feature
Keyboard	Textual Menu
<SMF>	Operation > Sheet Metal > Forming Feature



Type of a sheet metal forming feature can be selected either from the quick access drop-down list or by clicking the corresponding icon. If you select type from the list and it is not presented among icons below, the new icon will replace the last right icon.

Item "Type" in the drop-down list allows you to select element from custom libraries added to selection list.

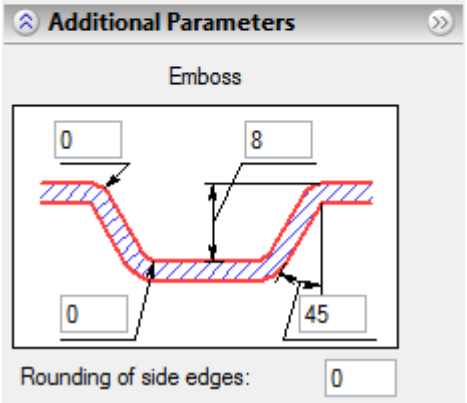
The content of the selection list is linked with the value of the Standard field and will update accordingly.

You can select sheet metal forming features library using option:

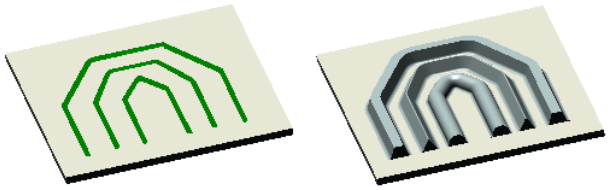
	<B>	Library configuration

More information about configurations can be found in "Holes" chapter.

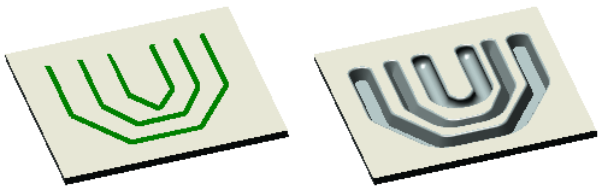
A special library of typical forming elements is used for forming feature creation, "*Sheet Metal Features*", shipped together with the system. If necessary, the library can be appended with custom elements.



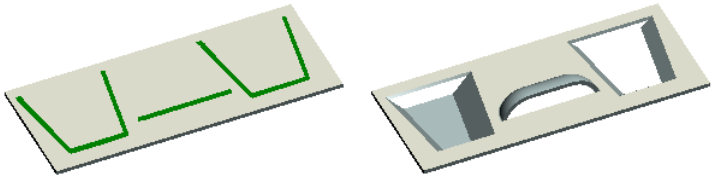
If linear units are set to inches in the command **ST: Set Document Parameters**, on the tab "3D", then the forming features will be used from the library *Sheet Metal Features Inch*.



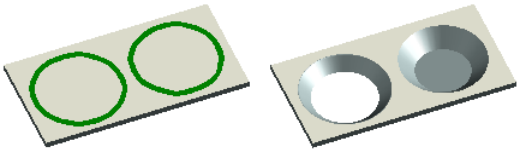
Beads: U-shape, V-shape and circular



Ribs: U-shape, V-shape and circular



Embossing non-closed, Round Louver, Flange non-closed



Flange and Embossing

The operation is based on a flat 3D profile lying fully on one of the workpiece faces. Depending on a forming feature, the profile should satisfy different requirements. Thus, for example, majority of the standard forming features (except "Embossing" and "Flange") strictly require a single-contour profile. The elements "Flange non-closed", "Embossing non-closed" and "Louver" require an open profile, "Embossing" and "Flange" – strictly a closed one, while the rest of the forming features can use either a closed or an open profile.

Creating Typical Sheet Metal Forming Features

To create a forming feature, do the following steps:

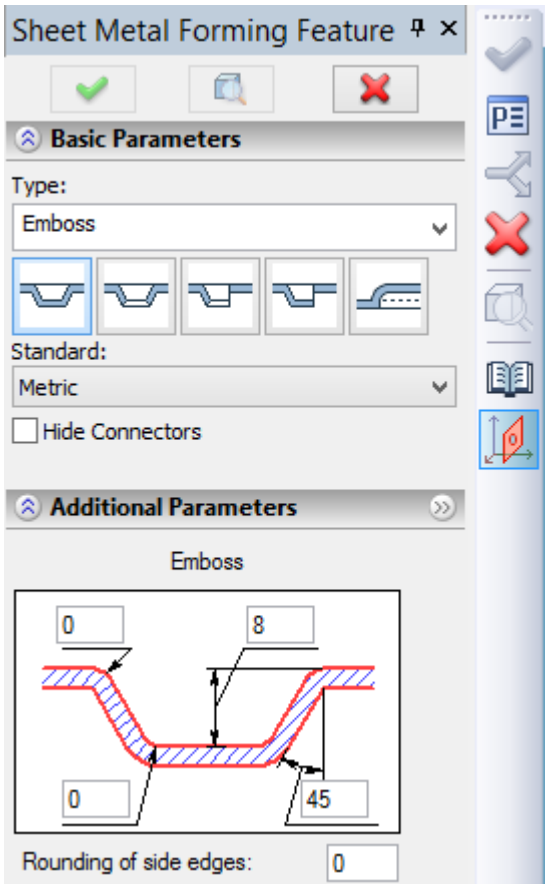
1. Select a source 3D profile for the operation. The profile selection mode is activated automatically upon entering the command. This is indicated by the pushed automenu option:

	<A>	Select 3D Profile

To select a profile, click one in the 3D scene by or pick it in the model tree. The selected profile will be highlighted.

The source profile can be selected automatically, when calling the command while drawing on the active workplane or from the context menu of an appropriate-type 3D profile.

2. In the command property window, select the desired forming feature from available in the library. The dialog of the selected forming feature parameters will be displayed in the lower portion of the window.



3. In the dialog of the selected forming feature parameters (displayed in the operation property window), define the required parameter values.

   The parameters dialog for each forming feature has a schematic chart of the feature being created with input boxes for all geometrical dimensions. To define a dimension, placed the pointer in the respective input box in the chart and enter the desired value.

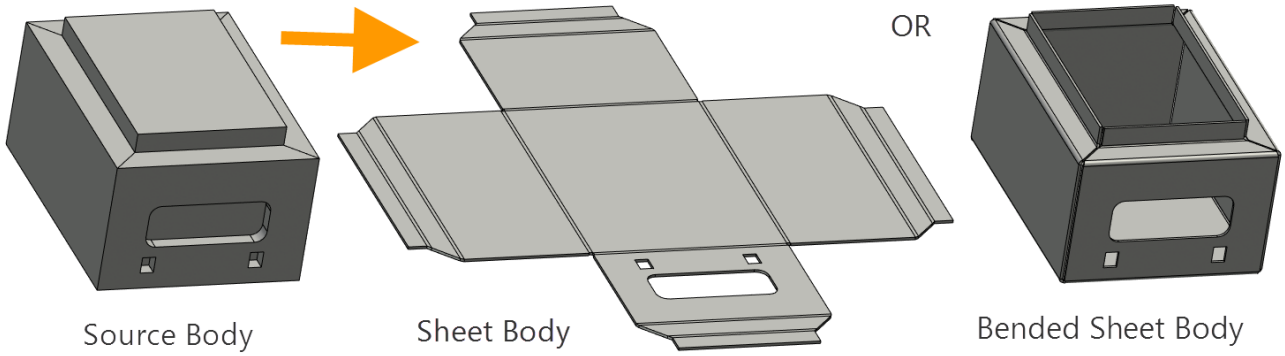
Confirm operation creation.

> More information about forming features creation can be found in "Create custom elements for sheet metal and holes libraries" chapter.

Convert Solid to Sheet Metal

The Convert Solid to sheet metal command is used for quick creation of bodies from the sheet metal on the basis of solid bodies.

The result of execution of the Convert Solid to sheet metal operation is the sheet body or bended sheet body that retains the shape of the original object.



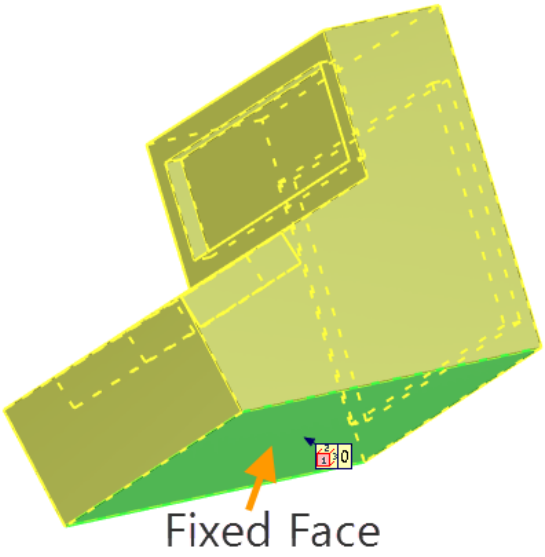
Main Concepts. Operation's Capabilities

This operation significantly simplifies the creation process of flat and 3D bodies from the sheet metal. Solid bodies with flat faces can be used as the source bodies for creation of this operation.

Fixed face

Creating of the body is carried out relative to the first selected face which is considered fixed and defines location of the plane. This face is called a fixed face.

Any flat face of the body can be selected as a fixed face. The edges for bending which will participate in the operation are specified relative to the fixed face. After being selected the fixed face is highlighted with a green color.

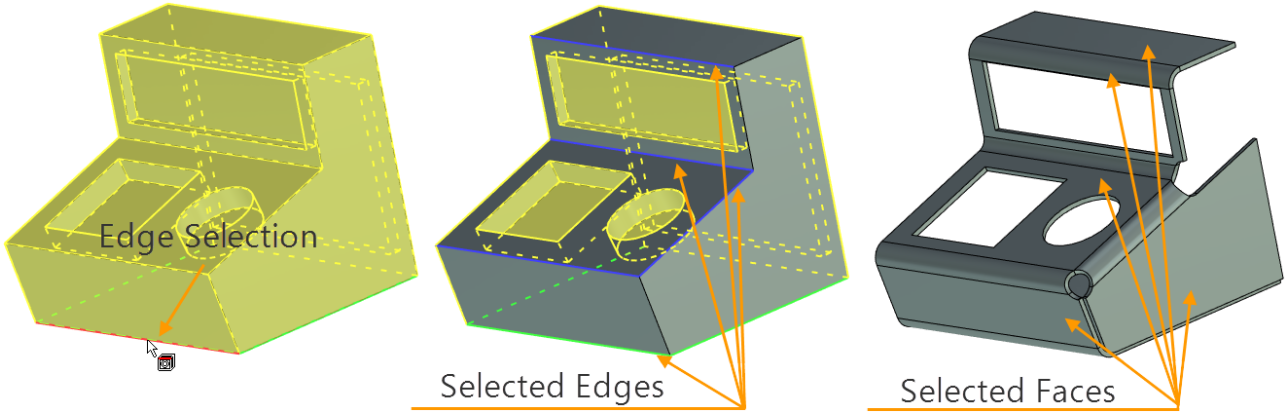


Edges for bending

After selection of the fixed face it is required to select edges for bending. Either the operation of fillet with a specified radius value or the operation of sheet body creation by the arc length of the resulting fillet is applied to the edges selected for the operation.

One of the edges of the fixed face should be the first among the selected edges. After selection of the edge its second forming face is included into the operation. Other edges of this face immediately become available for selection as the edges for bending. Thus, in order to select the face it is required to indicate at least one edge adjoining the face.

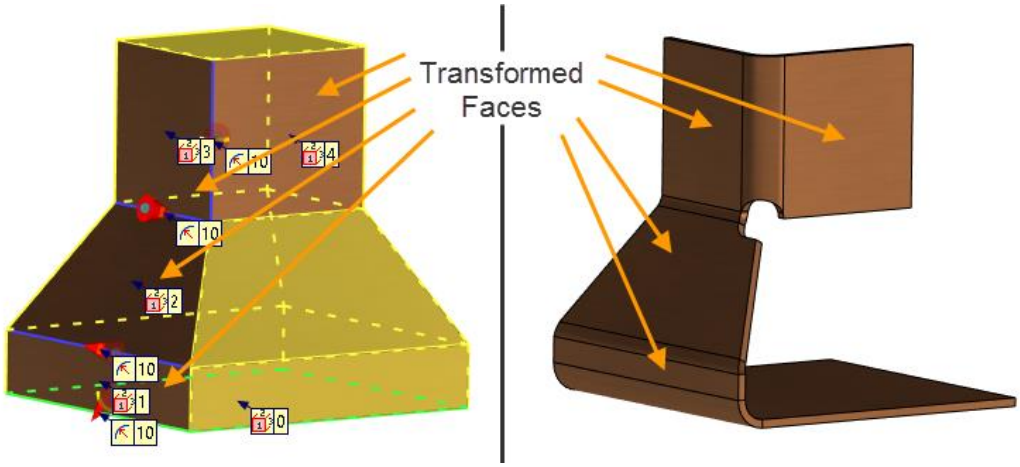
All faces to which the selected edges belong are highlighted with grey color. The faces that do not participate in the operation are highlighted with yellow color.



When selecting edges it is important to ensure that the resulting sheet elements do not penetrate into each other. This is related to the fact that with the help of this operation it is impossible to obtain self-intersecting flat template.

Creating thickness

For each selected face an equidistant surface is created at the given distance equal to the specified thickness. The new surface is constructed from inside of the face. After completion of the operation all selected faces are transformed into the sheet material, and the faces which were not selected are removed.

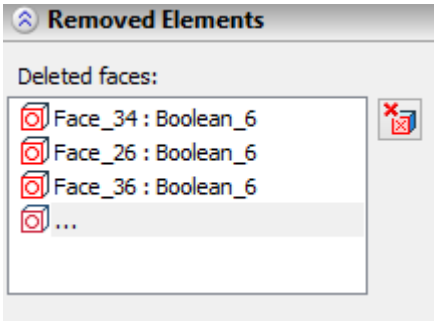


Removing faces

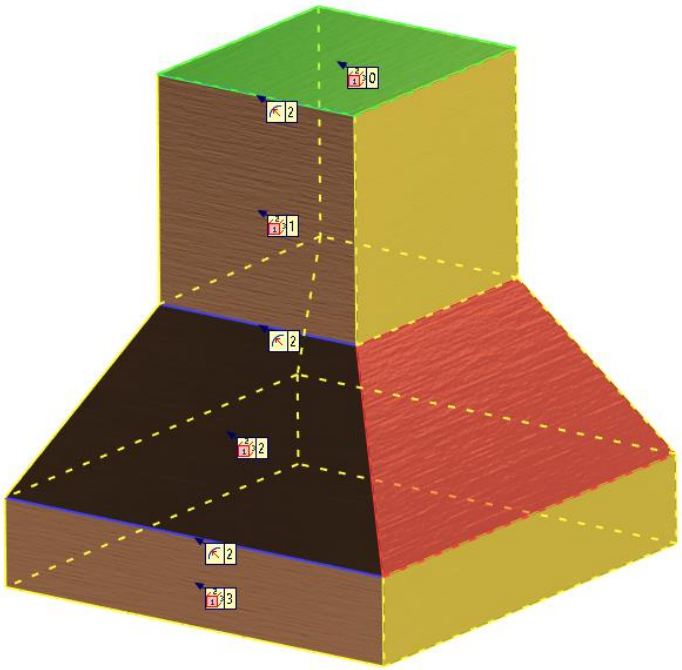
By default all faces highlighted with yellow color will be removed upon creation of the operation. If it is required to remove already selected face, the following option is used

	<D>	Select removable faces

The selected face is added to the «Removed Elements» section in the properties window and is marked with red color once the face is selected in the list.



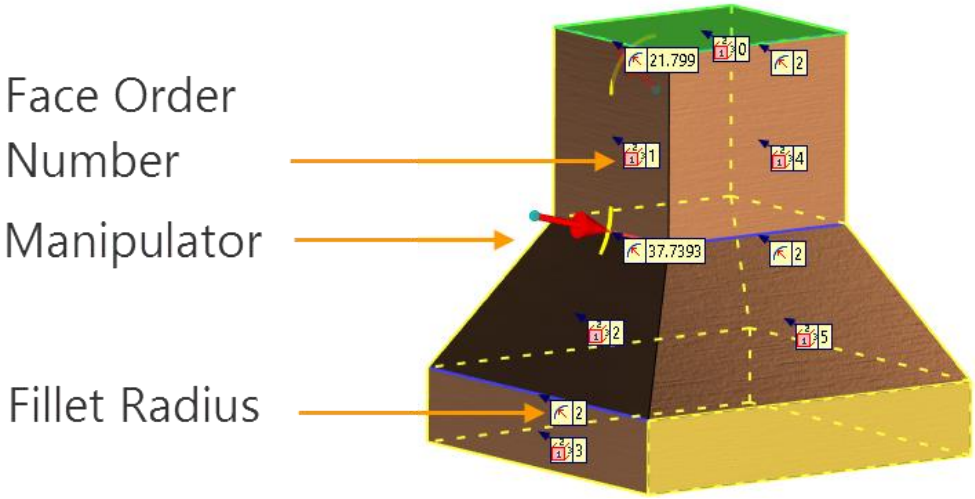
In order to cancel selection of the face for removal it is required to go to the properties window «Unfold Parameters» to the «Removed Elements» section, select the face and press the «Delete face from list» button . This button is located to the right of the list of faces.



Draggers and markers

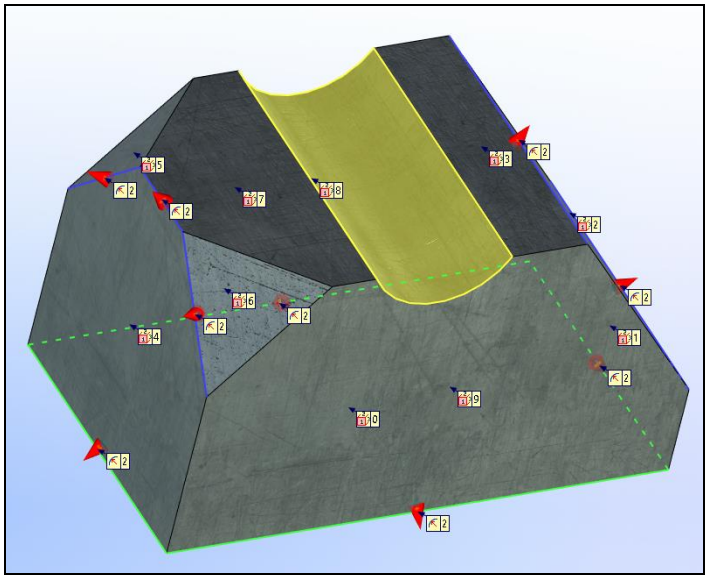
For each edge selected for the operation, the dragger and marker appear in the scene. The fillet radius is shown on the fillets markers. It can be changed if we click on the marker with the left button of the mouse. If we press on the dragger's arrow and move it forward or backwards, then we can also change the radius value. At the same time the arc by which the fillet is created will be shown.

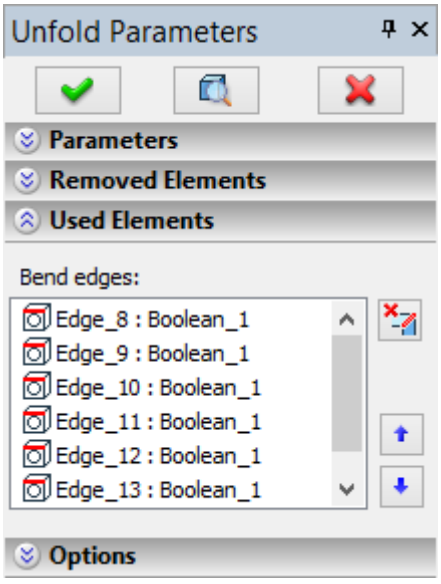
All faces are numbered in the order of their selection for participation in the operation. The number is displayed on the marker («0» – this is a fixed face, «1» – face forming the first edge from the selected edges etc.).



Elements being used

All edges and faces of the body that are used in the operation are selected manually. Their list is displayed in the «Used Elements» section of the properties window.





The edges selected for the operation are assigned the sequence numbers («Edge_2», «Edge_3» etc.). The number is assigned to each newly selected edge sequentially, i.e., if the edge selection is cancelled, and afterwards the same edge is selected again, it will have a different sequence number.

The «Delete bend edge» button located to the right on the «Used Elements» section allows us to exclude the bend edge selected in the list from the operation. It is also possible to delete the bend edge from the list by selecting it with a mouse click and by pressing the <Delete> button on keyboard. In the work window the edge selection can be cancelled by clicking on it again with the left button of the mouse.

The buttons up and down located to the right below the « Delete bend edge» button are used for changing the order of bending. The use of this option is described below.

Creating «Convert Solid to Sheet Metal» Operation

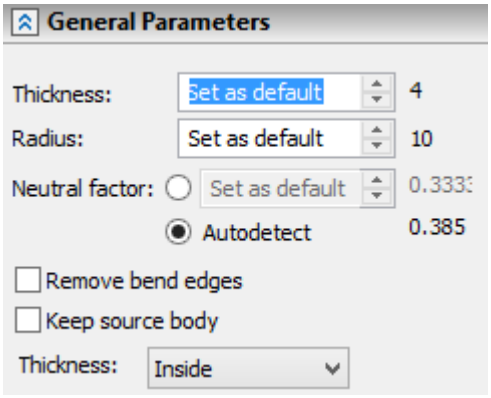
For transforming the body into a sheet metal the **Operations > Sheet Metal > Convert Solid to Sheet** Metal command is used:

Icon	Ribbon
	3D Model → Special → Sheet Metal → Convert Solid to Sheet Metal
Keyboard	Textual Menu
<SMS>	Operation > Sheet Metal > Convert Solid to Sheet Metal

To create this operation it is first required to carry out the following sequence of actions:

1. Select the fixed face of the body
2. Specify thickness of sheet material and fillet radius of the faces, otherwise these values will be selected automatically from the status
3. Select edges which will participate in the operation
4. Enable or disable the options: bend result, remove bend edges, keep source body (if necessary)
5. Specify additional capabilities of the operation: relief type, gap type, ripping gap (if necessary)
6. Confirm creation of the operation.

Parameters of the Operation

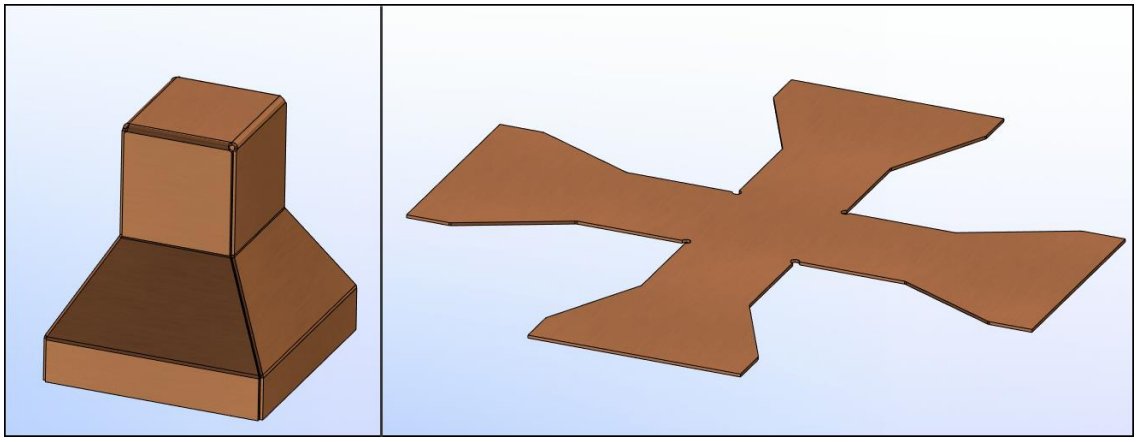


Thickness – allows us to specify the thickness of the resulting sheet metal. The thickness can be specified automatically or manually.

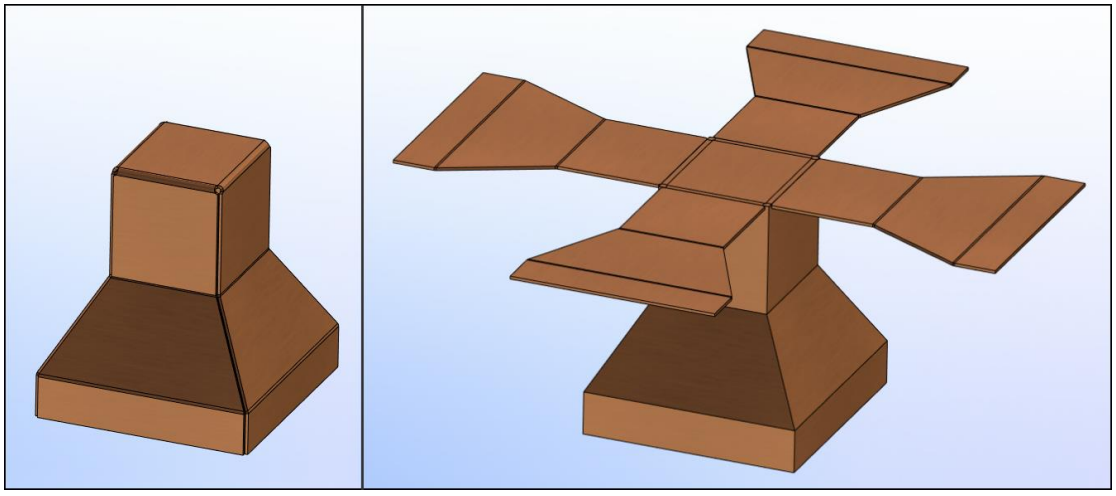
Radius – allows us to specify the fillet radius of sheet metal faces. The radius can be specified automatically or manually. If the «Bend result» option is disabled and the flat body is created, the radius is taken into account when calculating the value of the fillet's arc that is unfolded on the plane.

Neutral factor. Controls neutral factor definition

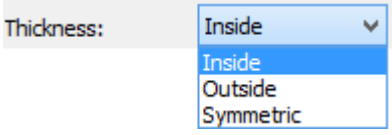
Remove bend edges. To activate this option, it is required to enable the flag in the corresponding field on the «Parameters» section. When this option is enabled, an absolutely smooth sheet body without lines at the places of bending is created. When this option is enabled, the «Bend result» operation is not executed.

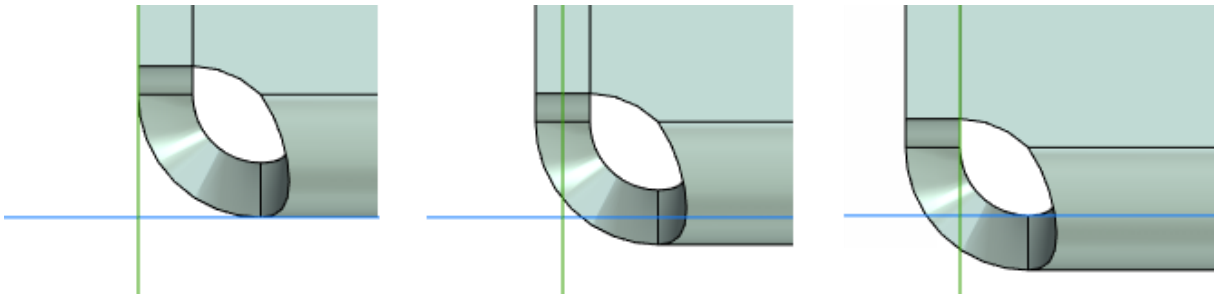


Keep source body. To enable this option, it is required to enable the flag in the corresponding field on the «Parameters» section. When the option is active, the source body is added to the result of the current operation. This option is convenient in estimating how the bent workpiece fit in with the overall dimensions of the source body.



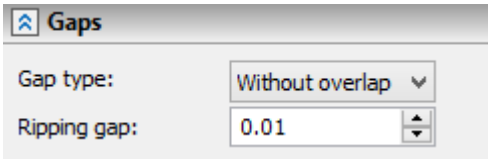
Thickness direction. Option allows to specify thickness direction for the sheet body. Inside, outside and symmetric modes are available.



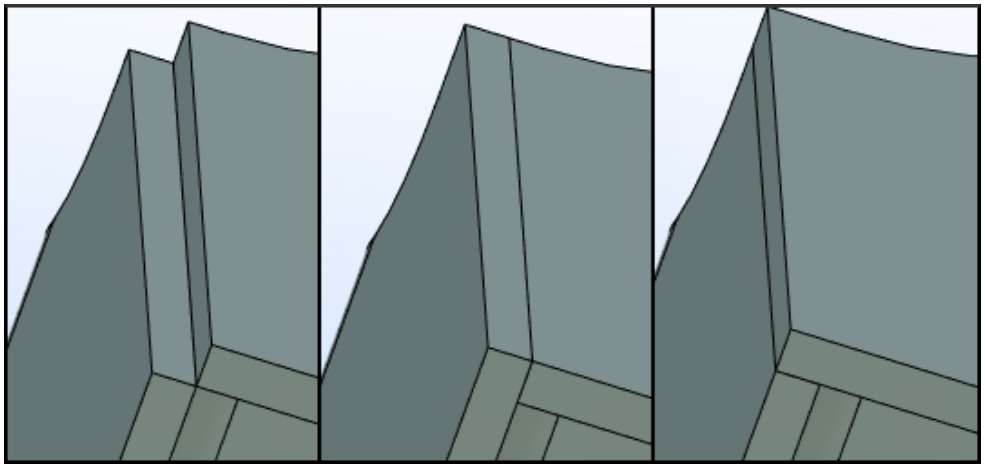


Inside Symmetric Outside

Gaps group:

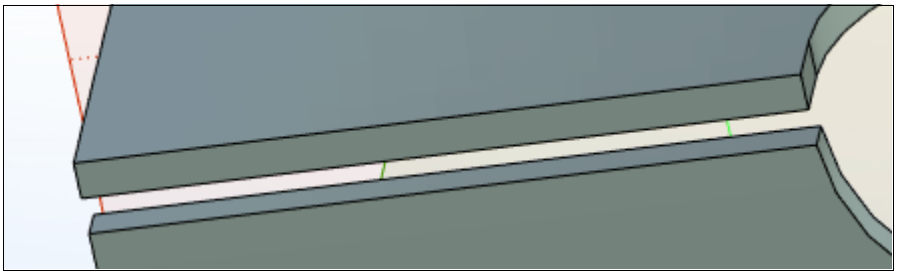


Gap type – this parameter defines location of adjoining sheet faces upon bending relative to each other. From the drop down list it is possible to select one of the three options: without overlap, overlap, reverse overlap. This parameter is used when the «Bend result» option of the automenu is used.



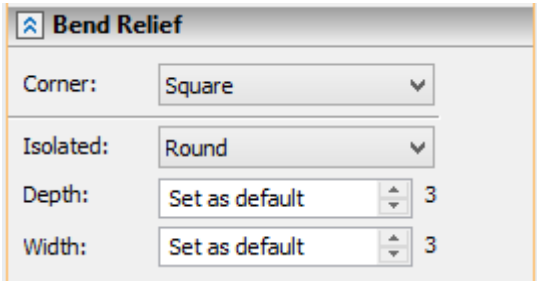
Without overlap Overlap Reverse overlap

Ripping gap –allows manually specifying the distance between sheet faces adjacent to each other.

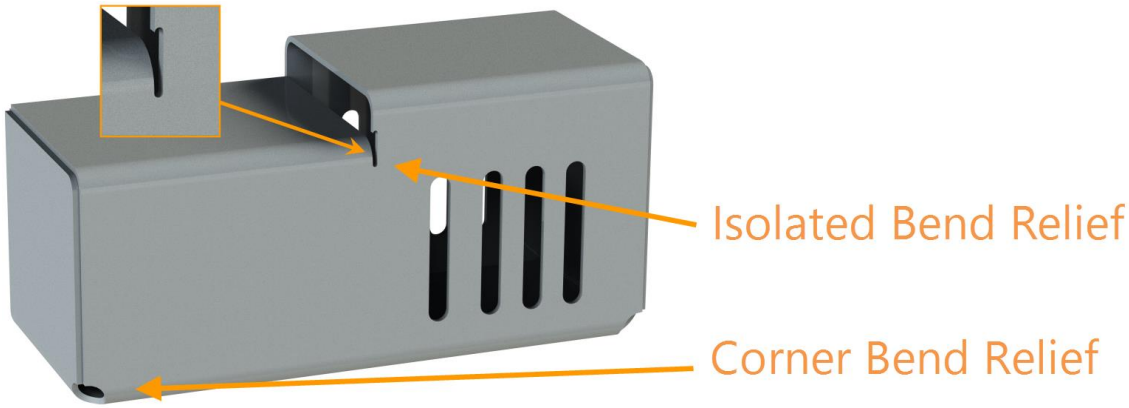


Bend relief group:

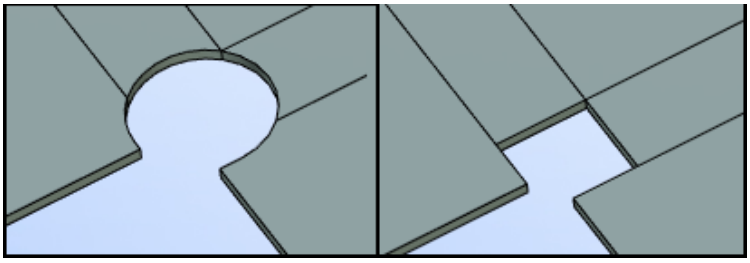
Corner, Isolated. Options allow to specify the shape of holes for relieving the stresses in the sheet metal for corner and isolated bend reliefs correspondingly. Three bend relief types are available: circular, square and welding. Welding type is unavailable for isolated bend relief.



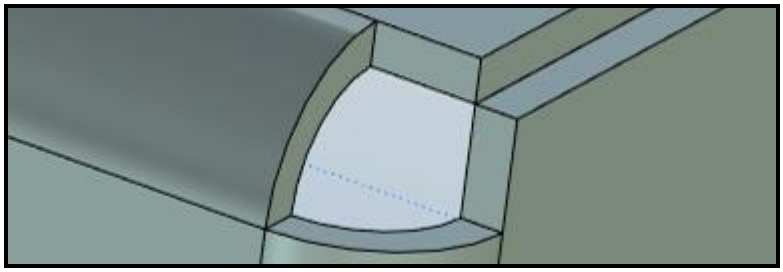
Width and **Depth**. Allows to set width and depth of an isolated relief.



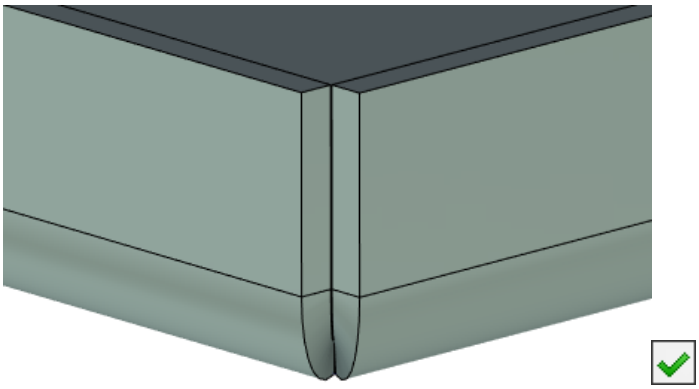
Radius of the circular hole is equal to the radius specified in the parameters. The square hole is constructed by three or more points of intersection of the given radius circle with the planes selected for the operation.



For 3D sheet bodies, the square holes are created only on the non-rounded faces.



When welding type is selected the same distance is specified on straight and curved sections at the junctions.



Automenu Options

The following automenu options are available:

	<F5>	Preview operation result
	<K>	Selected fixed face
	<R>	Cancel selection
	<B>	Select bend edges
	<D>	Select removable faces
	<E>	Edit rip edges
	<2>	Change relief type at selected vertex
	<C>	Bend result

To select fixed face, relative to which the operation will be carried out, the option is used:

	<K>	Selected fixed face

To cancel selection of all faces and edges participating in the operation, we use the option:

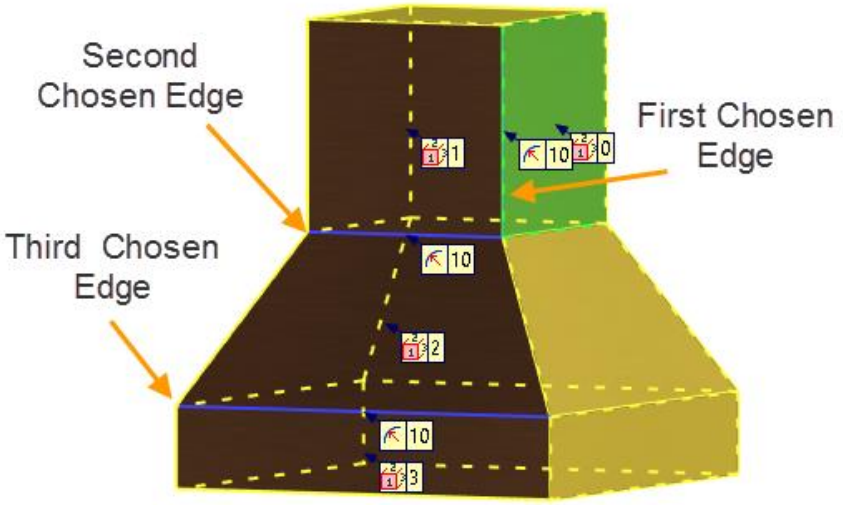
	<R>	Cancel selection

This option returns to the start of the operation. A fixed face and edges must be selected again. The values already specified in the «Parameters» section (thickness, radius, etc.) will not be changed.

For selection of edges for bending which are participating in the operation, the following option is used:

	<B>	Select bend edges

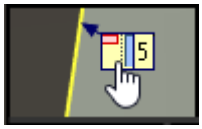
After this option is enabled it is required to select the edges adjacent to the faces participating in the operation, starting with the edges of the fixed face. The indicated element will be highlighted and the tooltip with name of the element will appear next to the cursor. For selection it is required to press .



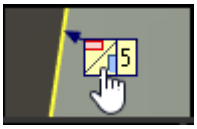
To change the value of the gap and the type of the overlap for each face individually, the option is used:

	<E>	Edit rip edges

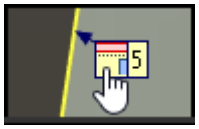
After this option is enabled it is required to select the edge on the common boundary of two faces participating in the operation and, in the window that appears, indicate the value of the gap and, if necessary, select one of the three types of the overlap.



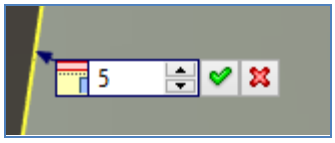
Overlap



Without overlap



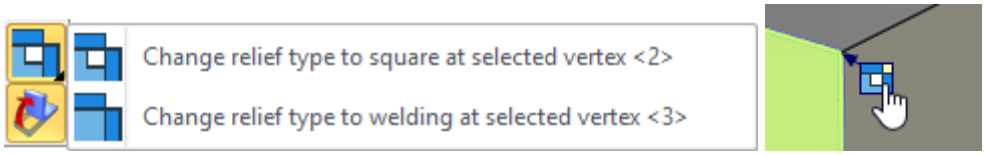
Reverse overlap



To specify a relief type in the selected vertex use option:

	<2>	Change relief type at selected vertex

For the selected vertex you can select a relief that differs from the default relief type of the operation. A relief type can be chosen from the drop down list of the option before the vertex selection. The specified relief type may be changed by clicking on the special marker.



To exclude faces from the operation, the following option is used:

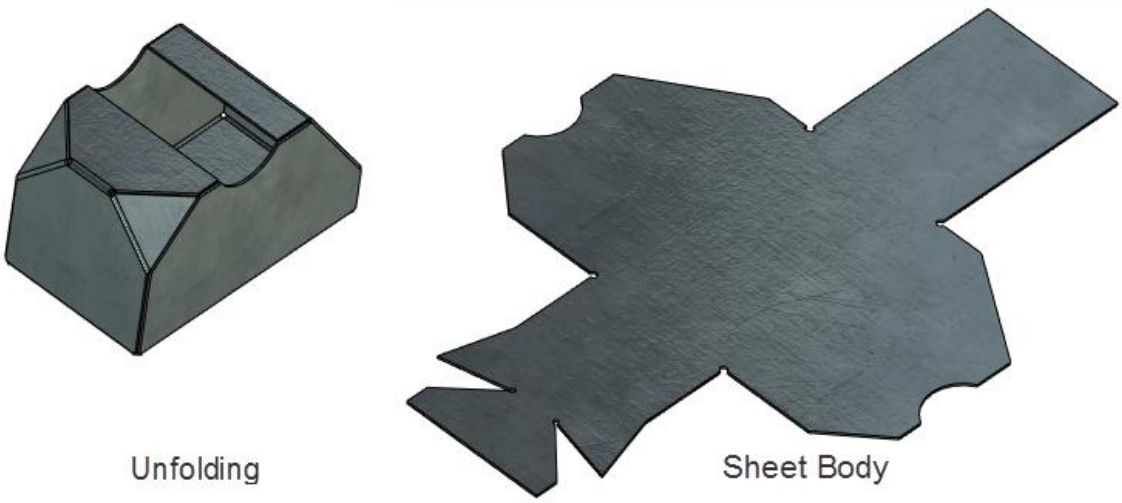
	<D>	Select removable faces

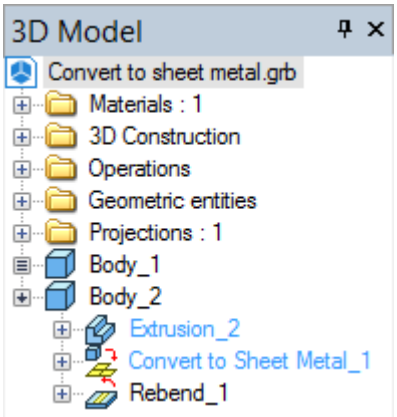
After selection of the faces participating in the operation, it is possible to enable this option of the automenu and indicate the selected faces that will be removed as a result of the operation. The edges adjacent to the faces being removed become unavailable for selection.

To switch between creation of flat and 3D body, the «Bend result» option is used. For activation of the option is used:

	<C>	Bend result

If this option is enabled, the sheet body will retain original shape of the object. Otherwise, the body will be located on a plane which will be determined by the fixed face.





When the «Bend result» option is enabled, after creation of the operation, in addition to conversion to sheet metal, one more operation – «Rebend» is added to the constructions tree.

If we change the properties of already created operation, the «Bend result» button of the automenu become inaccessible. This is done in order not to destroy logic of operations execution when other operations are present in the «3D model» window after the «Convert to Sheet Metal» operation.

For previewing the result of the Boolean operation, and also the changes introduced by the Boolean operation, the automenu option of the preview must be used:

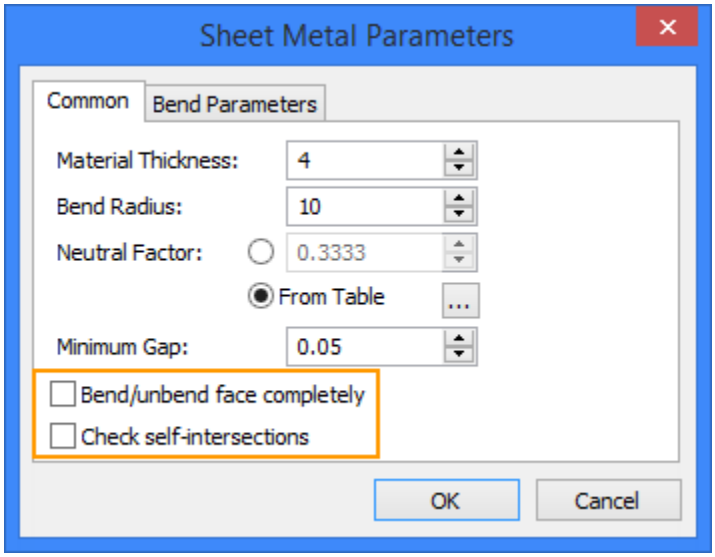
	<F5>	Preview operation result

Operation Execution Example

The use of the «Convert solid to sheet metal» operation is demonstrated in this example. Moreover, we consider the case when it is required to change the order of bending.

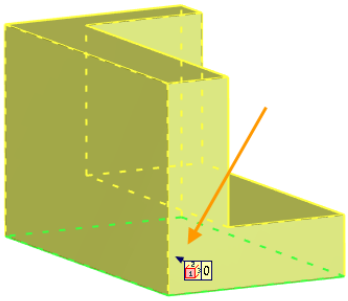
After creation of the cube in 3D window, we go to the parameters of the sheet metal using **SMP: Defaults** command.

We remove the flags from the «Bend/unbend face completely» and «Check self-intersections» options. This is done for the system to be able to display the resulting body on the screen when self-intersections of the metal sheets occur. If the flag are retained, the system will not be able to create a self-intersecting body and issue the error message.

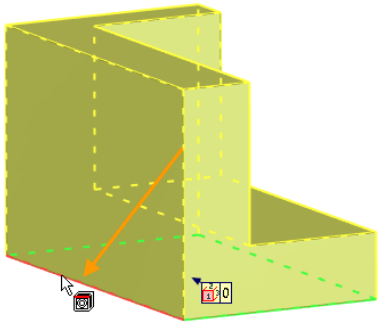


For creation of the «Convert Solid to sheet metal» operation, the **SMS: Convert Solid to sheet metal** command must be invoked:

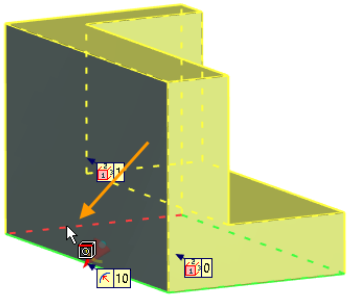
The order in which the construction elements are selected for creation of the operation is shown in the figure below. In the first step the fixed face was chosen (picture 1). Next, three edges of the body were chosen in order (pictures 2 – 4).



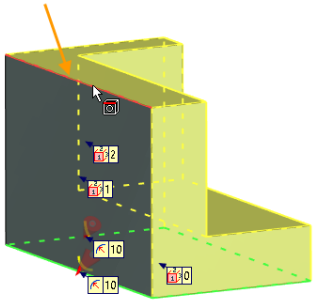
Fixed face selection



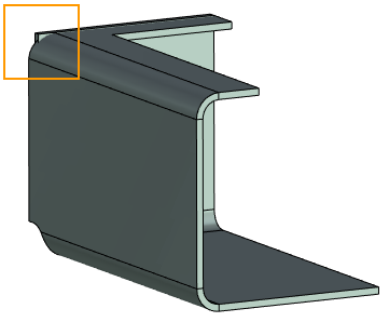
First edge selection



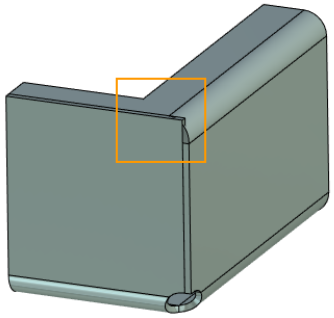
Second edge selection



Third edge selection

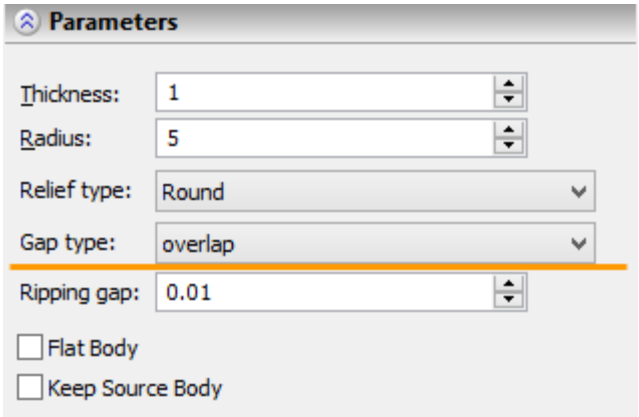


Result



Result

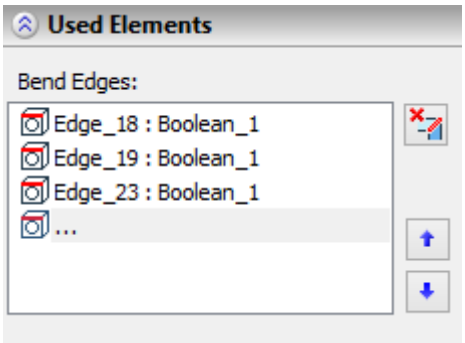
Before execution of the operation in the properties window on the «Parameters» section it was indicated that the edges of the sheets must lie on each other with the overlap.



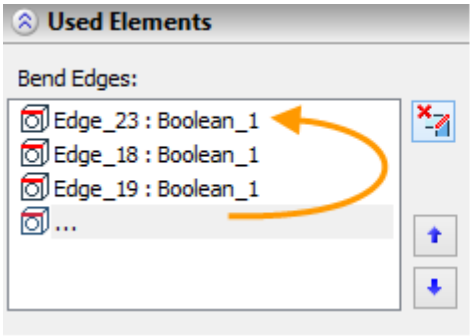
After execution of the operation, we obtained the solid body in which two sheets with the angles that are not deformed intersect the rounded face (Result). This is caused by the incorrectly specified sequence of body bending.

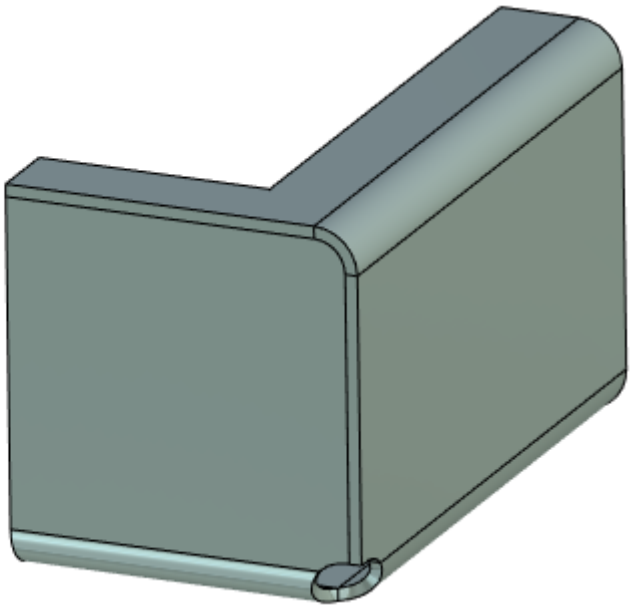
For these very cases it is convenient to use the change of order in bending of edges. This allows us to avoid self-intersections of bodies. At the same time there is no need in redefining of the edges manually.

In the «Used Elements» section of «Parameters» window, the edge selection order for the operation is shown. Using the up and down arrows we change the order in which bending is executed.



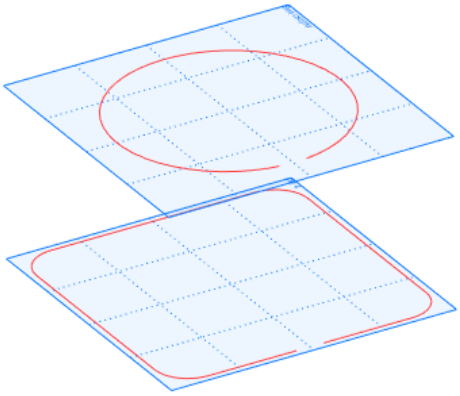
Now we move the last edge selected for the operation up by two lines by selecting the desired edge and pressing the up arrow, and afterwards we repeat the creation of the operation. Now the sheets are smoothly adjoining each other and do not intersect.

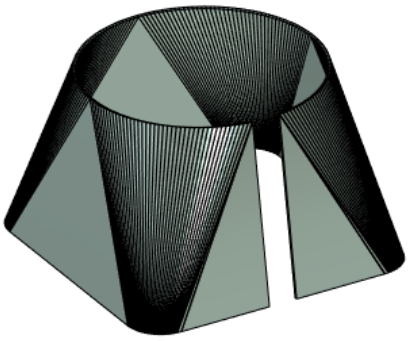




LOFTED BEND

Lofted bend operation allows to create complex sheet bodies based on two sections.

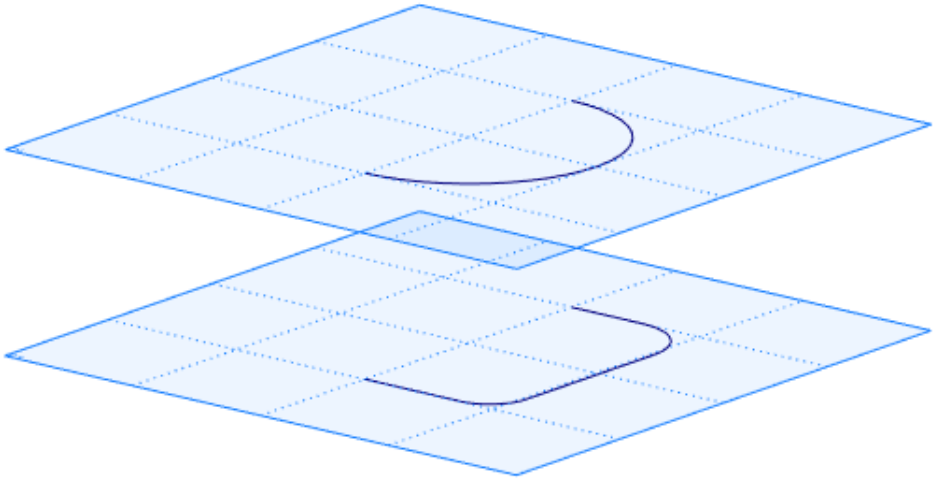




You may call the operation via command:

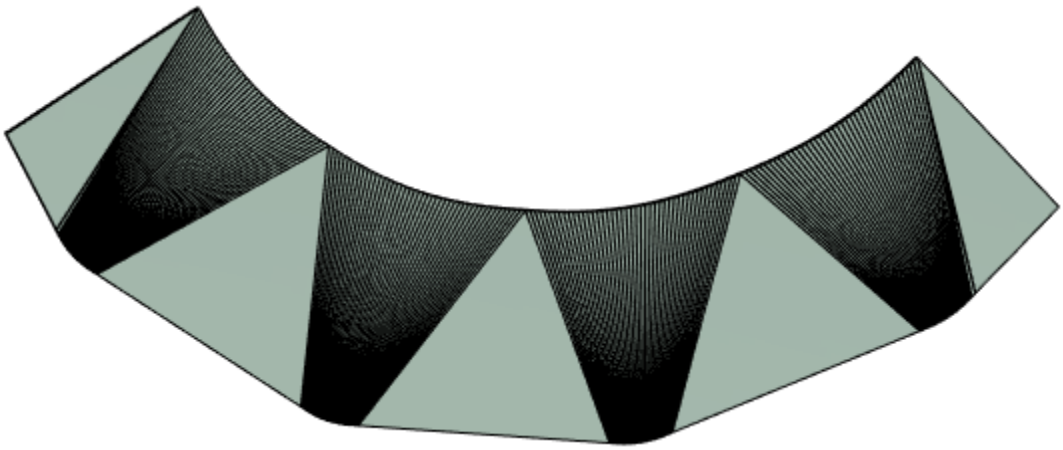
Icon	Ribbon
	3D Model → Special → Sheet Metal → Lofted bend
Keyboard	Textual Menu
<SML>	Operation > Sheet Metal > Lofted bend

To create a new sheet body using this operation you need to select two non-closed sections located on two parallel planes. Sections can consist of smoothly docked arcs and straight lines.



There can be different number of arcs and straight lines in the sections. Arcs are connected with arcs, straight lines are connected with straight lines.After that, cylindrical bend segments are created.

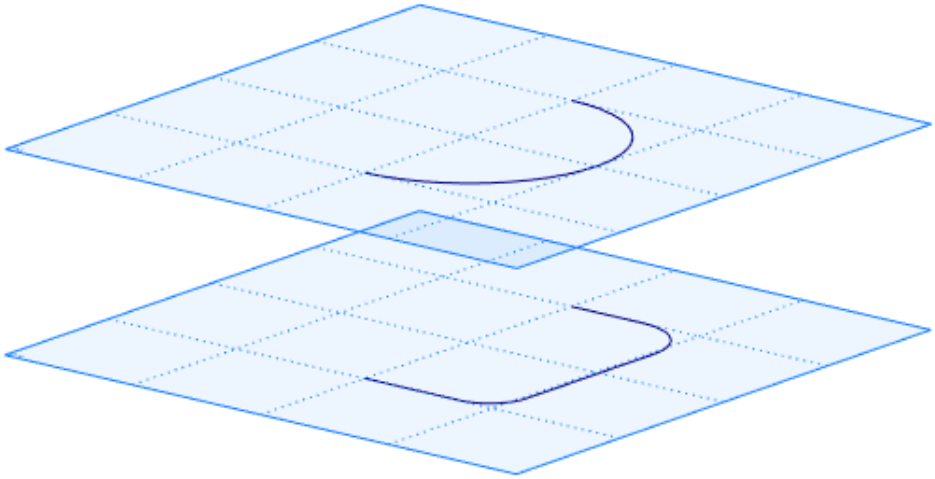
A body produced as a result of this operation can be unbent with **SMU: Unbend operation**.



Operation Creation

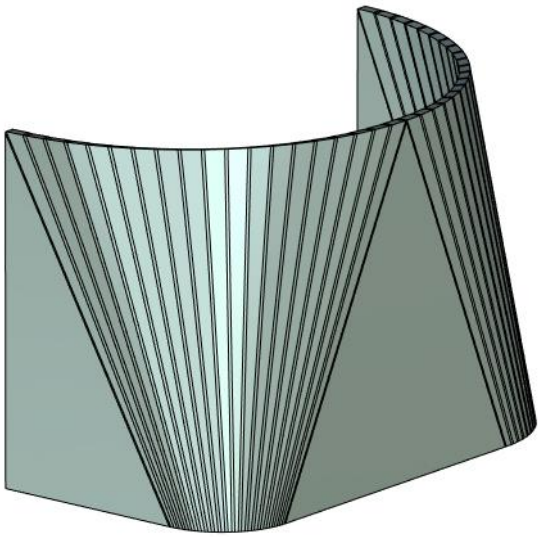
Non-closed Sections

You may create a sheet body using non-closed sections.



You can select sections using:

	<U>	Select first section
	<V>	Select second section



A sheet body may be created using part of the contour. For this purpose you should select starting point and direction for the both of the contours. The directions should coincide.

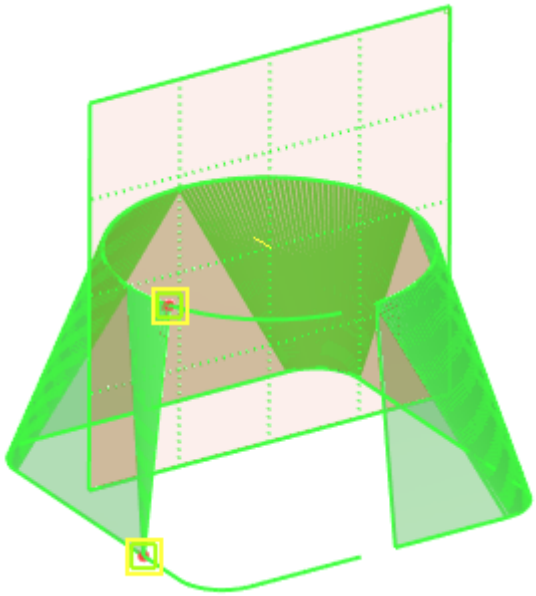
To select starting points for sections use options:

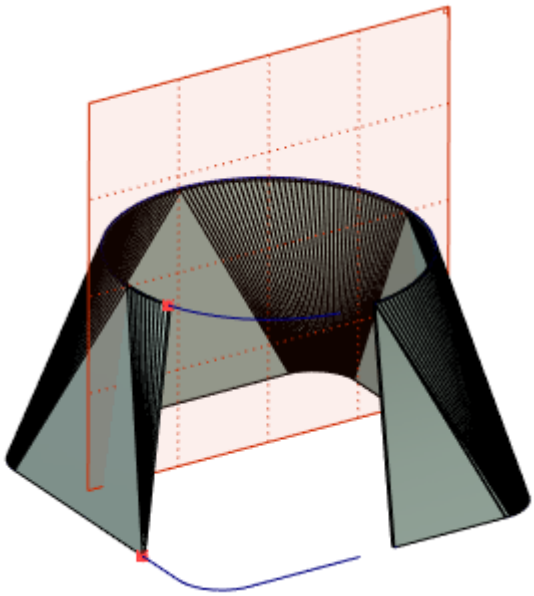
	<1>	Select starting point for the 1st section
	<3>	Select starting point for the 2nd section

To select the direction use options:

	<2>	Select direction for the 1st section
	<4>	Select direction for the 2nd section

In the example a workplane is selected to define the direction.





Closed Profiles

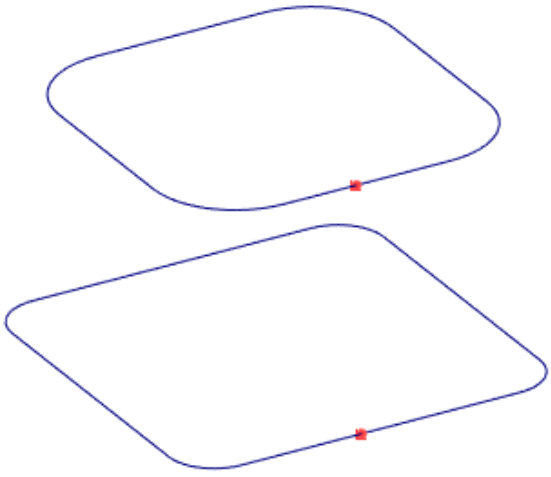
You can create sheet metal body from two closed contours.

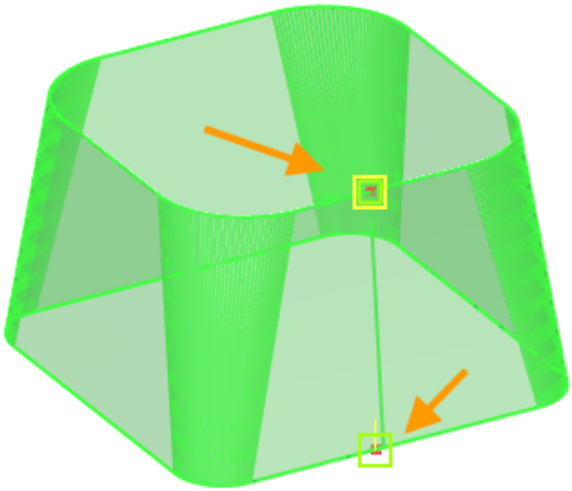
For each of them you need to select a starting point and a direction for bends creation.

For closed profiles, you need to define a parting line using starting points.

To select starting points for sections use options:

	<1>	Select starting point for the 1st section
	<3>	Select starting point for the 2nd section





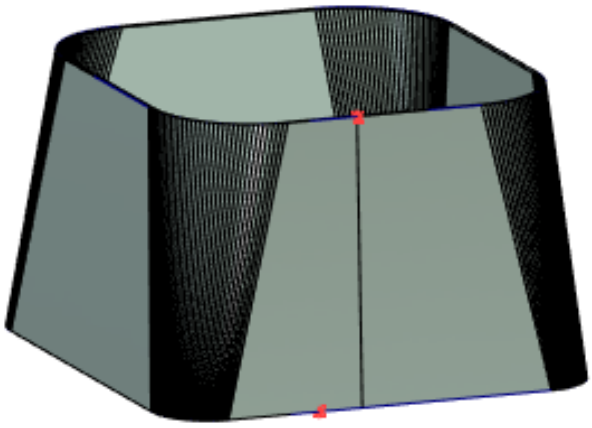
To select the direction use options:

	<2>	Select direction for the 1st section
	<4>	Select direction for the 2nd section

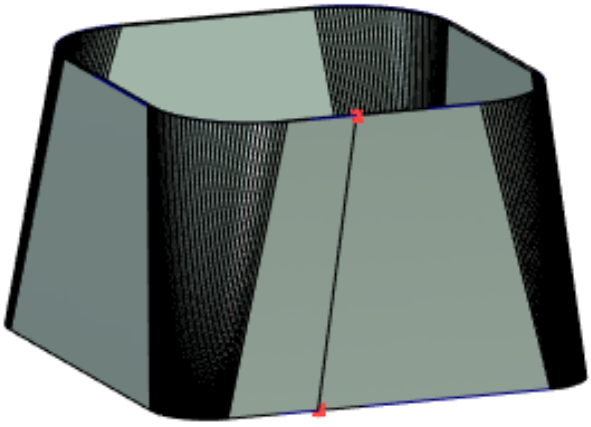
There are several ways to define parting line. One way is to select a starting point on the section and a direction for the section to define a parting line.

Other way is to select two starting points on the sections and specify a direction.

The result will differ according to the selected way,



Starting point and direction are selected on the upper section



Two starting points on the sections and direction are selected

Command Parameters

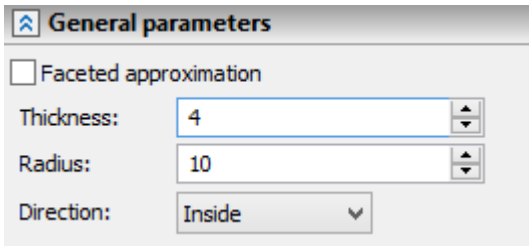
General Parameters section:

When you use **Faceted approximation** option, a surface body is created. The body can be converted into a sheet metal body.

Thickness defines thickness of the created sheet body.

Radius defines radius of all bends of the created body.

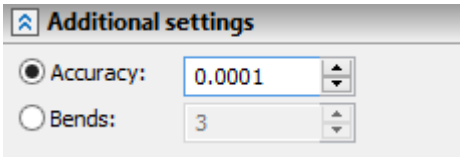
The values by default are set in **SMP: Parameters**.



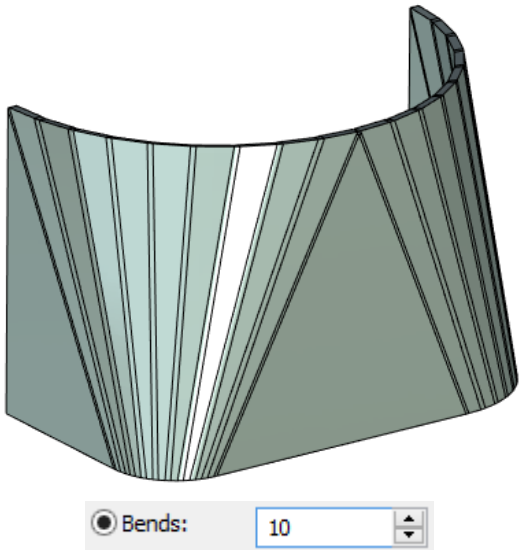
Direction. Allows to determine direction of body creation corresponding to the plane created between two selected sections: Inside, Outside, Symmetric.

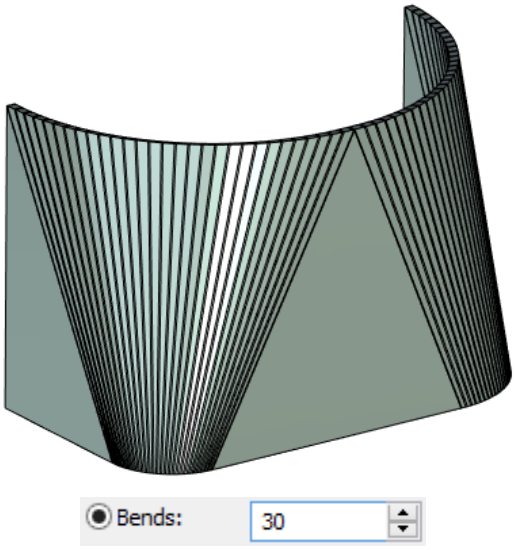
Additional settings section:

Accuracy. The smaller is the value, the higher is the accuracy. High accuracy increases the number of bends, but requires more time to create the body.



Bends. Allows to configure number of bends, which are created in the segments formed by two arcs or straight lines.

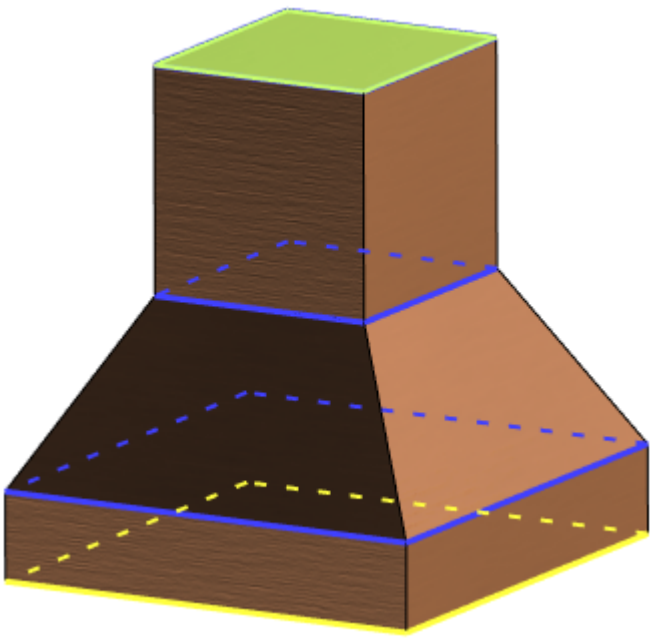




BEND TABLE

Bend Table, **Bend Note** and **Bend Table Data** commands are used for bend tables creation.

You can add a bend table to the drawing using **Bend Table** command. A table with information about all bends of the bodies in the scene will be created.



Number	Radius	Angle	Direction
1	2	27	down
2	2	27	up
3	2	87	down
4	2	87	down
5	2	87	down
6	2	27	up
7	2	27	down
8	2	27	down
9	2	27	up
10	2	87	down
11	2	27	up
12	2	27	down

You can create bend table via command:

Icon	Ribbon
	3D Model → Special → Sheet Metal → Bend Table
Keyboard	Textual Menu
<SMO>	Operation > Sheet Metal > Bend Table

Bend Table Data command creates a new product structure of the “Bend table” type:

Icon	Ribbon
	3D Model → Special → Sheet Metal → Bend Table Data
Keyboard	Textual Menu
<SMT>	Operation > Sheet Metal > Bend Table Data

Information about bends is added to the corresponding product structure columns.

Bend Number	Operation	Radius	Angle	Bend Direction	Position
Body: Body_3					
1	Rebend_1	2	27	down	1
2	Rebend_1	2	27	up	2
3	Rebend_1	2	87	down	3
4	Rebend_1	2	87	down	4
5	Rebend_1	2	87	down	5
6	Rebend_1	2	27	up	6

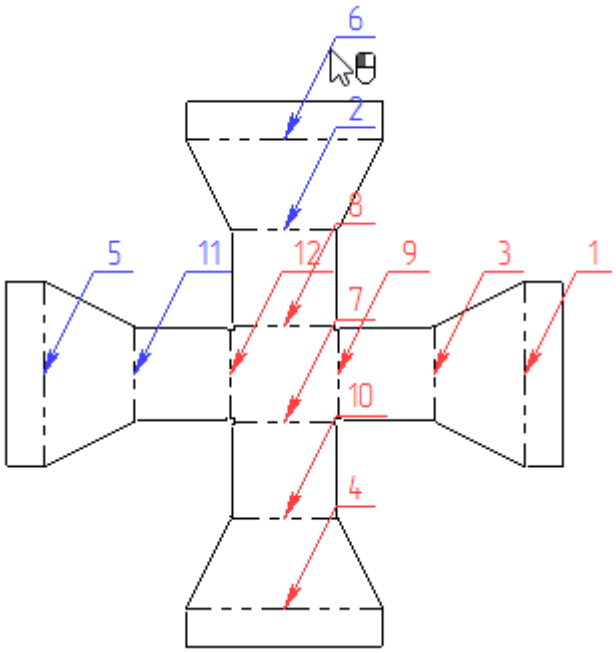
You can also select a "Bend table" type in the **Product Structure** window manually and create a report.

You may add bend notes for the projection of an unfolded model via command:

Icon	Ribbon
	3D Model → Special → Sheet Metal → Bend Note
Keyboard	Textual Menu
<SMZ>	Operation > Sheet Metal > Bend Note

If there is no product structure for a bend table in the document, it will be created automatically upon the command call.

After the command call you need to select all bend notes that should be added to the drawing and press .



Face Operations

FACE PROCESSING OPERATIONS

The set of commands under the "Face" group provides means for direct manipulation with a body's faces. This becomes necessary in special modeling situations, when handling 3D models imported from other systems, or when preparing 3D models for FEM analysis.

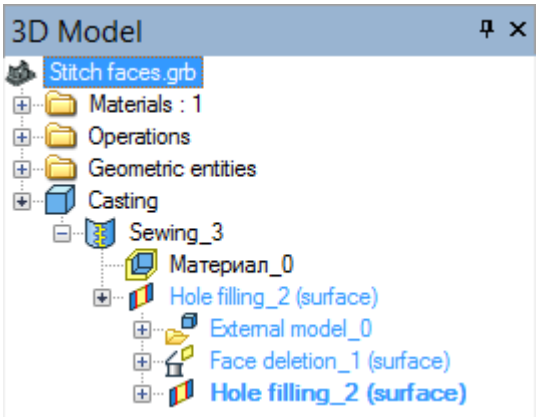
Use of those commands results in operations that are added to the model structure. This insures correct regeneration and proper behavior through possible model modifications.

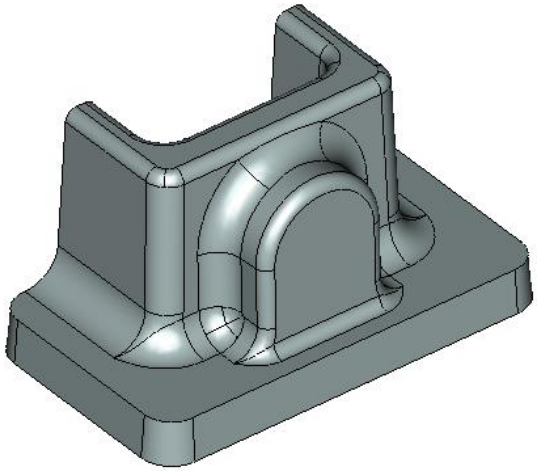
The face group of commands includes the following commands:

- ✓ Sew
- ✓ Imprint Elements
- ✓ Delete Faces
- ✓ Separate Faces
- ✓ Replace Faces
- ✓ Change Faces
- ✓ Transform Faces
- ✓ Extend Faces
- ✓ Fill Hole

The commands are accessed from the textual menu item **Operation**, the group **Face**. Besides, some commands can be called from the context menu accessed by right clicking over a selection of one or several faces or a closed loop of edges.

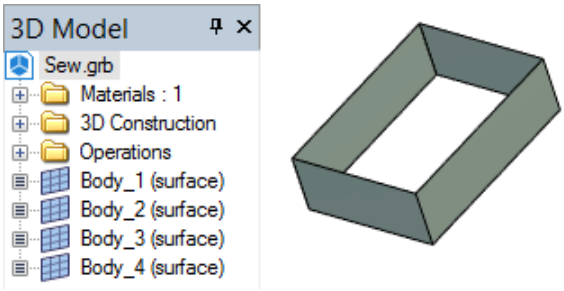
Similar to other operations, the face processing commands have a corresponding representation in the **3D Model** window.

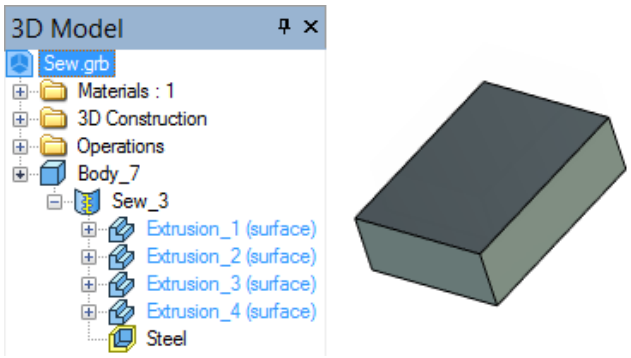




SEW

This command unites a set of disjoint surfaces into one sheet body and, if necessary, creates a closed volume, yielding a solid body.

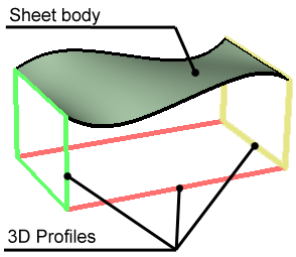




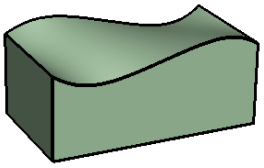
To create the operation, you need to define the set of surfaces to sew. You can select 3D profiles, faces or operations as the original surfaces. In the latter case, the operation must be a sheet body, for example, the result of extruding or rotating a wire contour, or an import surface from other applications.

The surfaces being sewn must adjoin without overlapping.

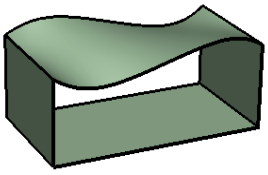
The result of sewing can be either a solid body or a new sheet body. By default, a solid body is created. In this case, the system attempts to create missing faces to make a closed volume.



Original model

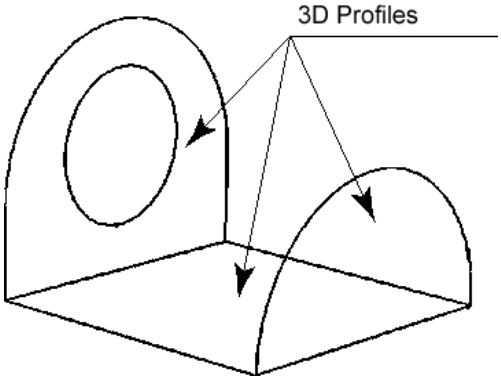


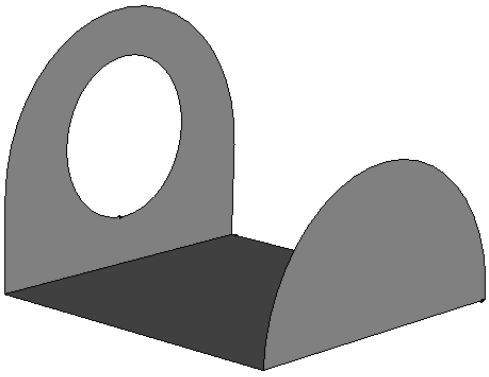
Solid body created



Sheet body created

In some cases (for example, when the missing surface requires more than one face), the solid body cannot be created.





Call the command **3SS: Sew Surfaces**:

Icon	Ribbon
	3D Model → Special → Faces → Sew
Keyboard	Textual Menu
<3SS>	Operation > Face > Sew

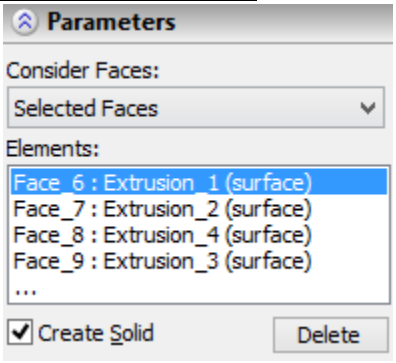
To execute the command, do the following:

- ✓ Select surfaces to sew.
- ✓ Specify advanced parameters (optional).
- ✓ Confirm operation creation.

To make a sewing, you need to specify the 3D elements defining the set of surfaces to sew. This can be done with the help of the following automenu options:

	<R>	Select 3D Profile
	<F>	Select Face
	<O>	Select surface operation

The selected 3D elements are entered in the **Elements** list in the command's properties window.



When selecting faces of some body (or several bodies), you can additionally tell the system which set of faces of this body (bodies) to sew: **Selected Faces** or **All Except Selected Faces**. This is done with the help of the parameter **Consider Faces**. In other cases (when selecting 3D profiles or entire operations), this parameter does not affect the sewing result.

There are two ways to cancel a 3D element selection:

- Pick this element again in the 3D scene using ;
- Pick it in the "Elements" list in the command's properties window and click **[Delete]**.

To cancel the selection of all elements at once, use the option:

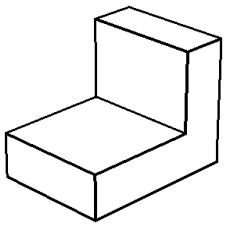
	<K>	Cancel selection

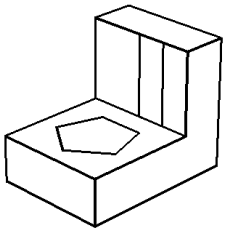
The **Create Solid** flag in the command's property window specifies whether to create a solid body from the sewn faces. The option is turned on by default.

Sewing can be used not only to join isolated surfaces into one body. This operation can also be used to create a surface from one or multiple body faces.To do this, simply select the face or faces to sew. The flag "Create solid" must be cleared. As a result, a new sheet body will be created. In the future, it can be used to construct an equidistant (offset) surface using the "Shell" operation.

IMPRINT ELEMENTS

This command is used for defining areas of specified shape on already existing faces. This might be necessary, in particular, for imprinting separator lines and surfaces into a 3D model as part of preparation for FEM analysis. Those elements could be used, for example, for defining forces applied to the model.





An element being split (the imprint target) can be one or several faces, or a whole operation. Those elements will serve as a basis for creating the new faces. Depending on the splitting method, the shape of the new area is defined based on either the shape of the dividing element (the imprint tool), as Face, Edge, 3D Profile, Operation, or based on the geometry of the imprint target elements.

The command for splitting faces is **3ZS: Imprint Elements**:

Icon	Ribbon
	3D Model → Special → Faces → Imprint Elements
Keyboard	Textual Menu
<3ZS>	Operation > Face > Imprint Elements

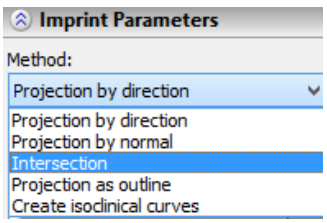
To create the operation, do the following steps:

- ✓ Select the splitting method.
- ✓ Select the objects to split (the imprint targets).
- ✓ Select the imprint tool objects (optional).
- ✓ Specify the direction (optional).
- ✓ Confirm operation creation.

Selecting Splitting Method

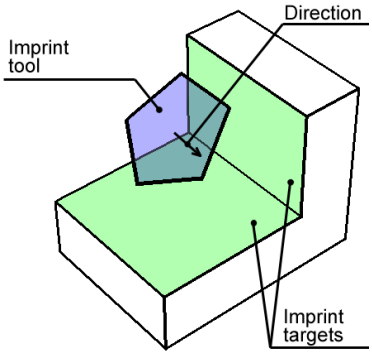
A splitting method defines how the boundaries of the new faces will be determined, and, therefore, what data you need to specify.

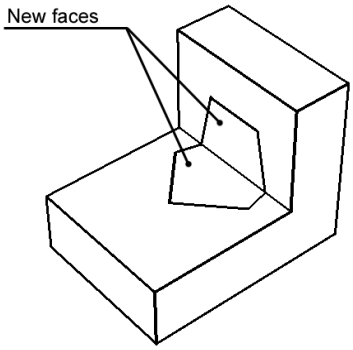
Select the splitting method from the pulldown list of a combo box located at the top of the property window. Upon entering the command, the method "Projection by direction" is set by default.



Projection by direction

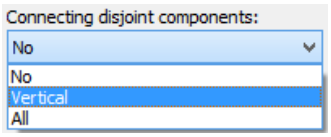
In this method, the imprint tools are mapped on the imprint target objects along the specified direction.



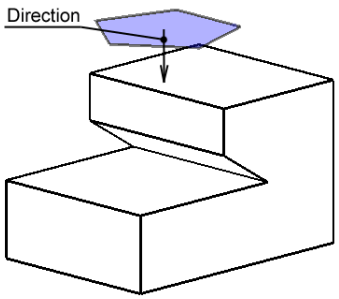


In this method, you need to specify the imprint tool and the imprint targets. Additionally, you need to specify the mapping direction. If the imprint tool is a flat 3D profile, then the assumed direction is along the 3D profile normal. Make sure that the profile mapping direction is towards the imprint target objects. To use a different direction, specify it manually.

It is possible that the imprint tool mapping occurs along the surface, or even on the opposite side of the surface. In the "Projection by direction" method, upon selecting an imprint tool element, the option "Connecting disjoint components" appears for selecting the mode of joining edges in this case.

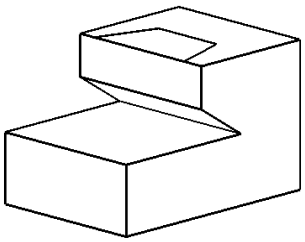


"No" means the system will attempt to create new edges only on the visible faces as viewed in the specified direction.

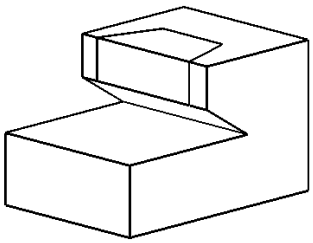


"Vertical" means the system will attempt to create new edges on the faces visible in the specified direction or parallel to this direction. "All" means the system will attempt to create the edges on all faces, including the obstructed faces, as viewed in the specified direction. In this case, the new edges on the obstructed faces must allow a connection with new edges on other faces.

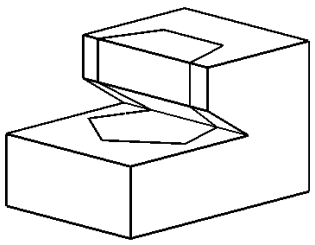
The following diagrams show the results produced by each option.



No

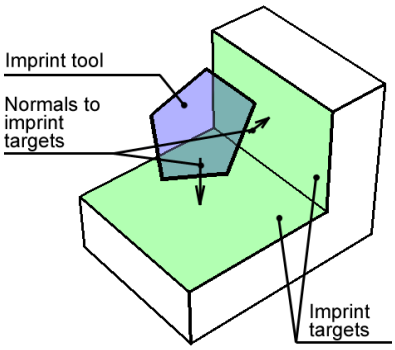


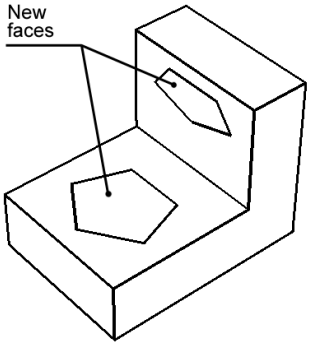
Vertical



All

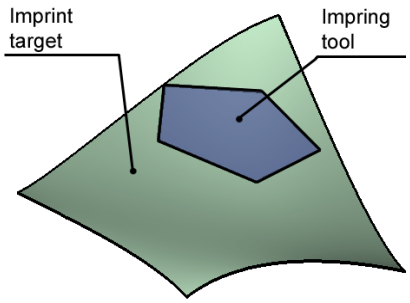
Projection by normal

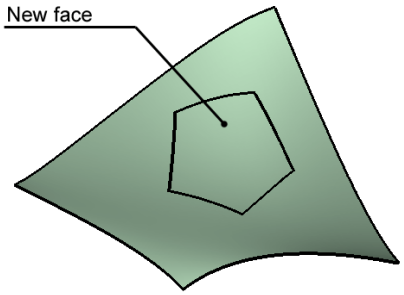




In this method, the boundaries of the new faces are determined by mapping the imprint tool objects on each imprint target object along the normal to the imprint target. All you need to specify is the imprint tool and target objects.

If the imprint target is not flat, the normal changes from a point to a point of the surface. In such a case, the imprint tool object is mapped on the target as follows: each point along the edges of the imprint tool is mapped onto the point of the target that is the origin of the normal passing through the point on the tool.

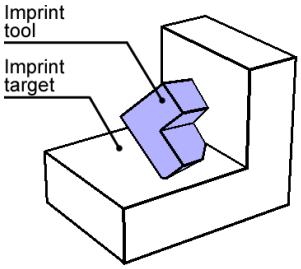


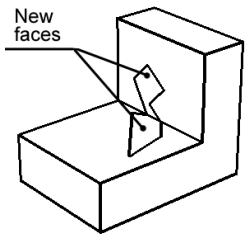


It is important that all points of the imprint tool had a unique mapping on the surface of the imprint target. The command will fail otherwise.

Intersection

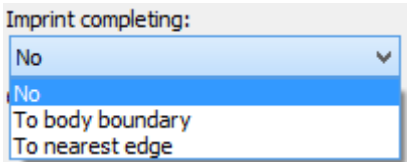
In this method, the boundaries of the new faces are the intersection lines between the tool and the target objects.

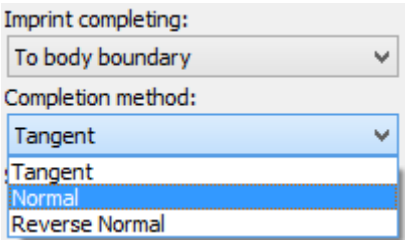




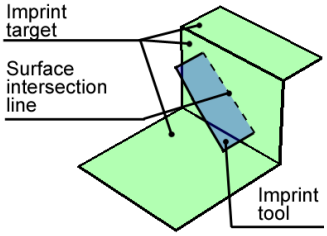
You need to specify the imprint tool and target objects.

If sheet bodies are selected as the imprint target and tool objects, the comboboxes “Imprint completion” and “Completion method” appear in the property window.

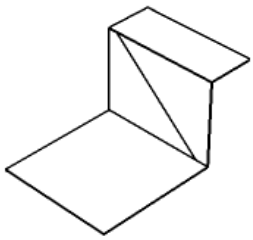




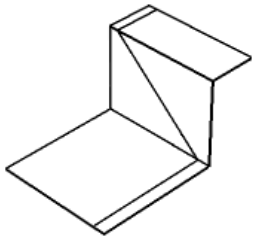
Those can be used in the case when the face of the tool sheet body does not fully intersect the imprint target. In this case, setting “Imprint completion” to “No” results in no new edges created upon executing the command.



Alternatively, with the setting **To nearest edge** or **To body boundary**, the intersection line between the bodies will be extended (in the first case – up to the nearest edge, in the second – through all body), and that will be the edge.

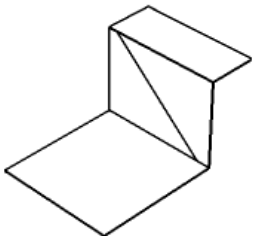


Edge extended up to the nearest edge

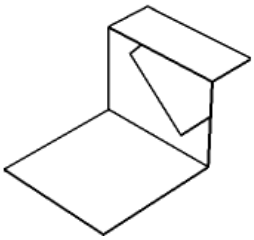


Edge extended up to face bound

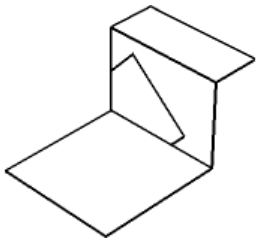
The option “Completion method” lets you specify the direction of extending the edge: either tangent to the edge, or normal (reverse normal). The effect of each parameter is shown on the diagram below.



Tangent direction



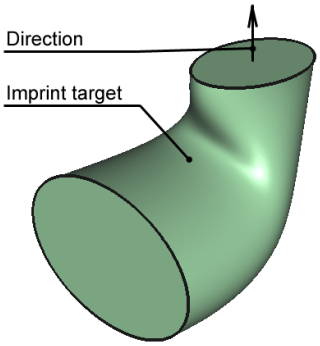
Normal

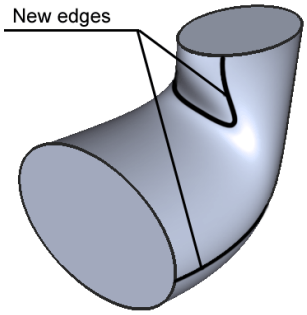


Reverse Normal

Projection as outline

In this method, the boundaries of the new faces are determined by the outline edges of the imprint target object.

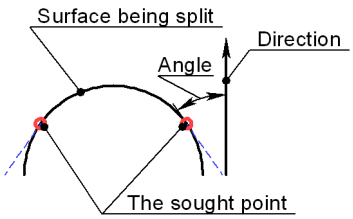


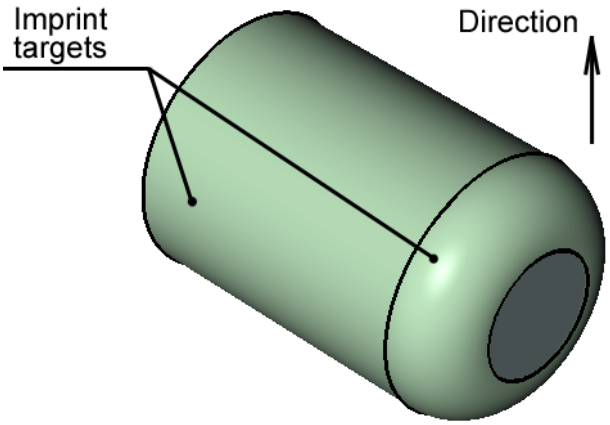


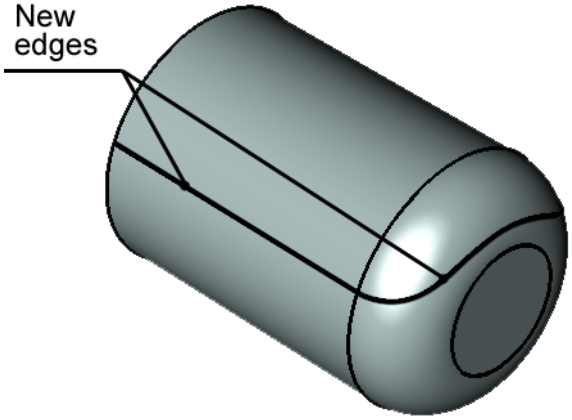
You need to specify the imprint target object and direction.

Create isoclinical curves

In this method, the boundaries of the new faces are determined by an isocline. An isocline is a line connecting points on the surface being split, in which the tangency line to the surface makes the same constant angle with the specified direction. In 2D, this can be easily sketched (see the right-hand diagram). In 3D, the system builds a set of points – a line.







Isoclines are convenient in mold design. In some cases, an isocline is required for making "Taper"operation.

Selecting Imprint Target Objects

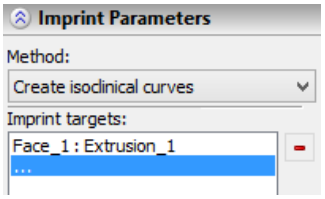
The imprint target are the objects that will be split as a result of the operation.

It is possible to use multi-solid operations and multi-contour profiles as the imprinted elements.

To launch the imprint target selection mode, use the automenu options:

	<B>	Select target operation
	<F>	Select target faces

Another way to select is by clicking the mouse on an empty line in the "Imprint targets" pane. After that, you can proceed with selecting the desired faces or operations in the 3D window.



Selecting an operation as the imprint target means that all faces of the operation will be selected for imprinting. The system will correctly handle the situations when the number of the original operation's faces changes after regeneration.

To cancel selection use option:

	<R>	Cancel target selection objects

Selecting Imprint Tool Objects

Selection of imprint tools takes place only in the following imprinting methods: "Projection by direction", "Projection by normal" and "Intersection". In the other cases, you do not need to select the imprint tools.

It is possible to use multi-solid operations and multi-contour profiles as the imprinting elements.

In the first two cases, you can use the option:

	<C>	Select tool contours

The option allows selecting elements (3D Paths, Edges, Profiles, Faces and Loops) that define boundaries of new faces.

With the method "Intersection", use either of the options:

	<O>	Select tool operation

(In this case, the system constructs an intersection with all faces of the operation), or the option:

	<A>	Select tool faces

(the system considers the selected faces only).

If an operation was selected as the imprint target, then you can select a plane as the tool. In this case, use the option:

	<P>	Select tool plane

To cancel selection use option:

	<K>	Cancel selection of tool objects

Defining the Direction

You must specify the direction if using one of the following split methods, "Projection by direction", "Projection as outline" or "Create isoclinicalcurves". In all those cases, the direction can be specified by selecting two 3D nodes. For this purpose, use one of the options:

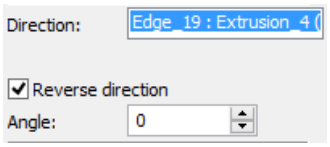
	<1>	Select first point
	<2>	Select second point

Alternatively, use the option:

	<D>	Select direction

That allows selecting elements for explicitly specifying the direction.

Make sure that the profile mapping direction is towards the imprint target objects, otherwise the application will display a message about an error regenerating the model, upon confirming the operation.



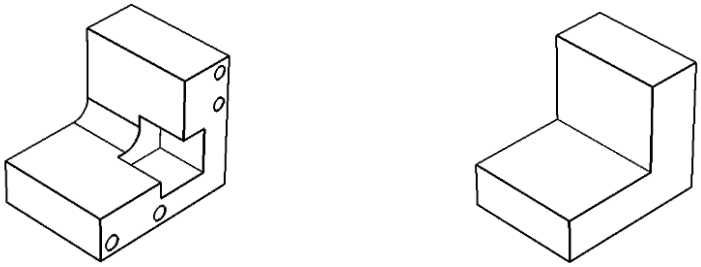
If the mapping direction is away from the imprint targets, use the option **Reverse direction** to flip the direction. The option is located at the bottom of the property window.

To cancel selection use option:

	<U>	Cancel selection of projecting direction

DELETE FACES

This command allows deleting one or several selected faces. Face deletion breaks solid body topology. The introduced voids invalidate the closed volume. If required, the system can attempt healing those by various means. Otherwise, the result will be a sheet body.



The operation is used in surface and solid modeling, when local changes to the model topology are desired, without affecting the original relations. It can also be helpful when preparing a model to finite element analysis, for such tasks as removal of small model features that are insignificant to the calculations.

You can directly select the faces to delete by explicitly picking the faces to delete. Alternatively, you can indirectly specify such faces. In this way, you either select a chain of edges surrounding the faces to delete, or a face on which the faces being deleted are dependent.

Call the command **3ZD: Delete Faces** via:

Icon	Ribbon
	3D Model → Special → Faces → Delete Faces
Keyboard	Textual Menu
<3ZD>	Operation > Face > Delete Faces

To execute the command, do the following:

1. Select faces to delete.
2. Select a healing method.
3. Confirm operation creation.

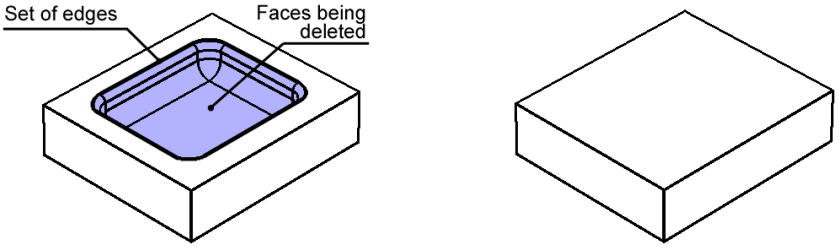
Selecting Faces to Delete

The option “Select Elements” serves for selecting objects defining the faces to delete:

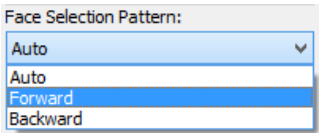
	<F>	Select Elements

By default, you can select faces, edges and loops.

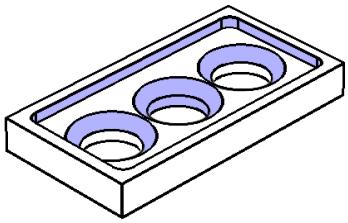
In the simplest case, you can select the exact faces to be deleted. However, besides that, there is a convenient option of selecting a closed loop of edges that will define the faces to delete (the selected elements are added to the **Faces** list). This option is not just for speeding up the selection of faces to delete. In a parametric model, if a new face is created as a result of model modifications in the area of faces deletion, this face will be correctly handled by the system.



A closed loop of edges divides all body faces into two sets: one to be deleted, and the other to be left out. To determine what particular set of faces will be deleted and what left out, use the option “Face Selection Pattern”.



The default option value is "Auto", for the system to delete the set of faces with the least area. The options "Forward" and "Backward" allow manual selection of the set of faces to delete.



The set of faces to be deleted is automatically highlighted in the 3D window.

Note that you can select several closed loops of edges simultaneously (intersecting and nested loops are not allowed). This makes possible defining more intricate sets of faces to delete.

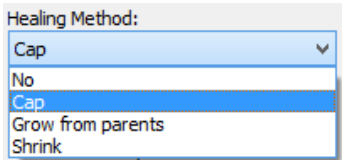
When selecting a set of edges, the following option is turned on by default:

	<G>	Smooth edge chain selection mode

If you need to select specific edges, turn off this option.

Selecting Healing Method

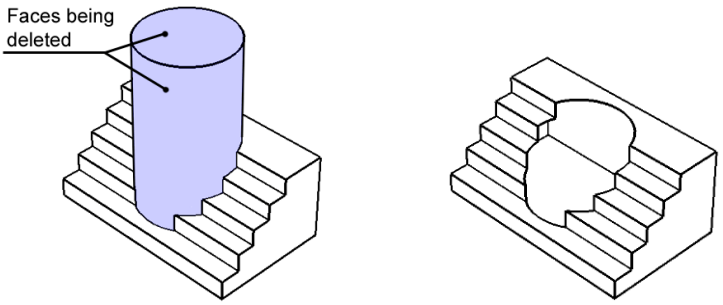
"Healing method" determines the system actions after deleting the selected elements. It instructs whether to close the introduced voids, and, if so, how.



Select a method from the pulldown list of the combo box in the property window. The command default method is "Cap".

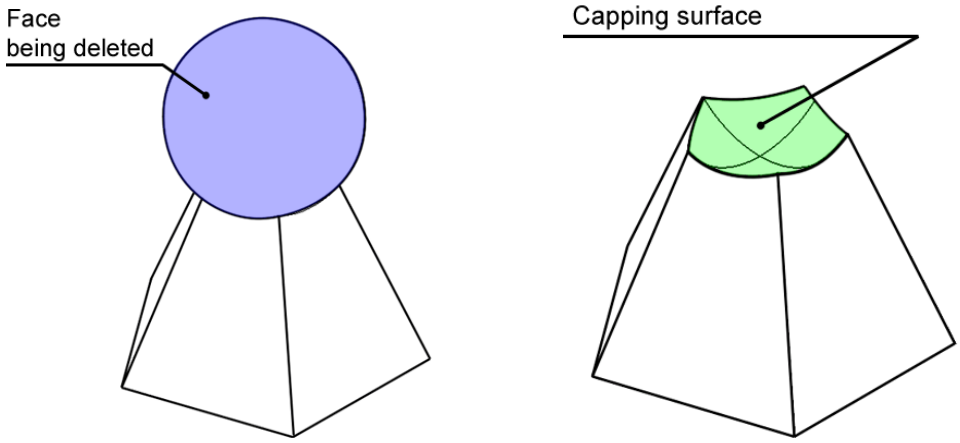
No healing

In this case, the system just deletes the select faces, leaving the resulting void in the model as is. If the original operation was a solid, then the model will lack the closed volume after the deletion, and will become a sheet body.



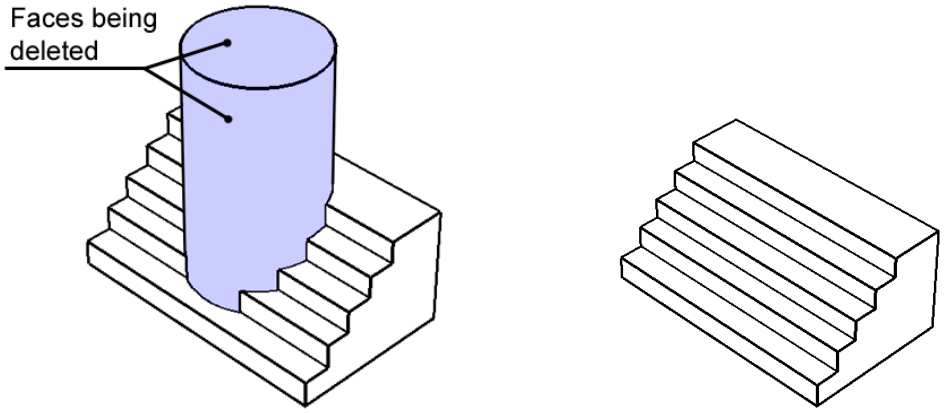
Cap

In this way, upon deleting the faces the system tries to form a surface for closing the resulting void. The capping surface must pass through all edges surrounding the void. As a result, this method works in the cases, when all edges surrounding the voids are obtained by an intersection with a single surface. This actually becomes the capping surface.



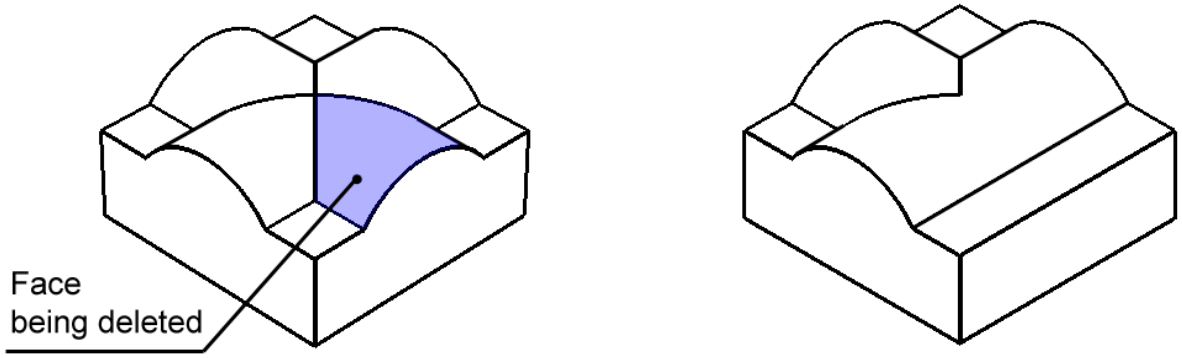
Grow from parents

In this method, the system tries to close the voids caused by surface deletion by growing the faces adjacent to the deleted faces.



Shrink

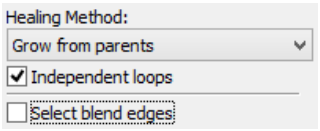
In this method, the system tries to close the voids caused by surface deletion not only by growing the faces adjacent to the deleted (as in the previous method), but also by trimming them.



Advanced Parameters

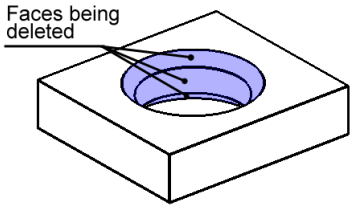
Upon selecting the deletion method and the faces to delete, you can specify some advanced parameters.

When surfaces are deleted, one or several sets of new loops (of edges) can be created, surrounding the model voids.

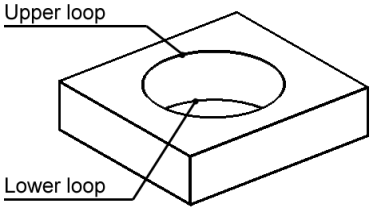


By default, the system will try to process all those loops simultaneously, as part of processing the model. In the process, the capping surface is being formed that is passing through all loops (that's a ruled surface). However, in some case,s it is necessary to make separate capping surfaces for each loop. The option **Independent loops** accounts for this.

The effect of this option can be demonstrated on the example of a hole with two chamfers. Suppose, the surfaces of the hole and both chamfers are being deleted. In the course of the command, the faces of the hole are deleted, and two loops come into play (the upper and the lower loops), bounding the void introduced in the body.

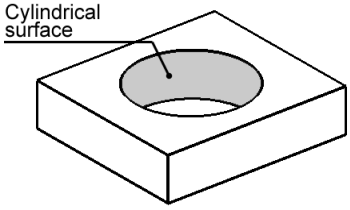


Original model

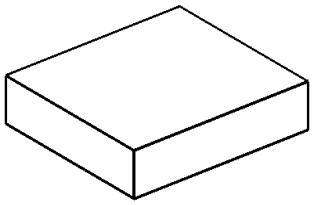


Intermediate stage

If the option **"Independent loops"** is turned off, the system unites the two loops in order to close voids, which results in another hole of a larger diameter than the original one. Instead, with the option "Independent loops" turned on, the system processes each loop separately, which in this case results in capping each with a flat surface. The hole completely disappears.

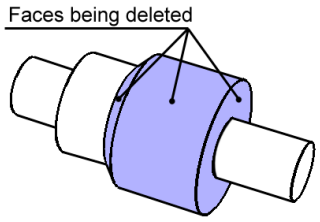


Independent loops Off

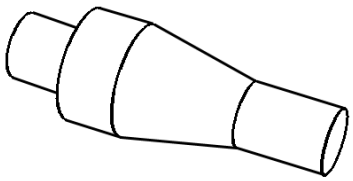


Independent loops On

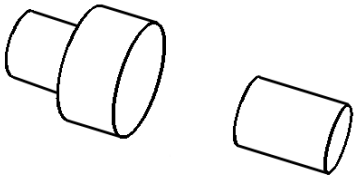
When using the **"Independent loops"** option, a situation can occur when two or more bodies are produced by the operation, instead of one.



Original model

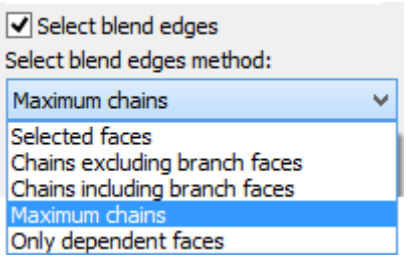


Independent loops Off



Independent loops On

The option "Select blend edges" is provided specifically for the cases of deleting faces created by blending.



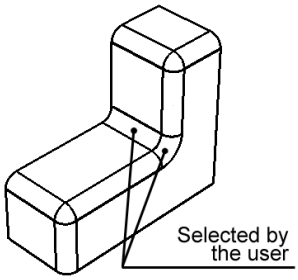
When deleting blends, one often needs to delete, besides the particular blend face, also some other faces in order to correctly cap the voids introduced in the model.

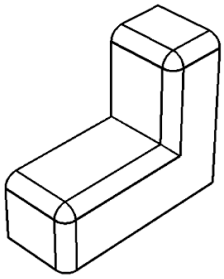
Moreover, sometimes it is necessary to delete all blended faces adjacent to some flat face. Of course, you could select such faces manually; however, more convenient is to select just the flat face. This lets the system automatically select the blend faces to delete. Such automatic selection of faces insures the model parametric behavior through changes in the number of blend faces, maintained by the system.

The option **Select blend edges method** tells the system which faces to select and delete automatically. There are several ways of selecting blend faces.

Selected faces

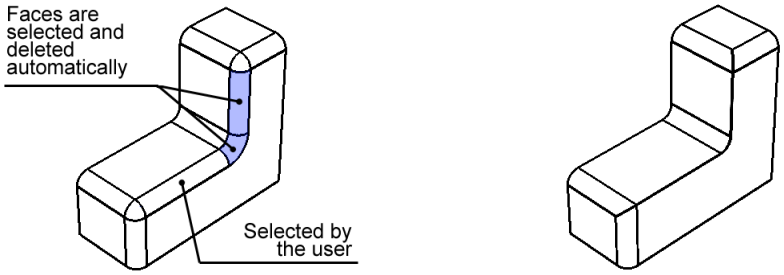
The system will attempt to delete only the faces that were explicitly selected by the user.





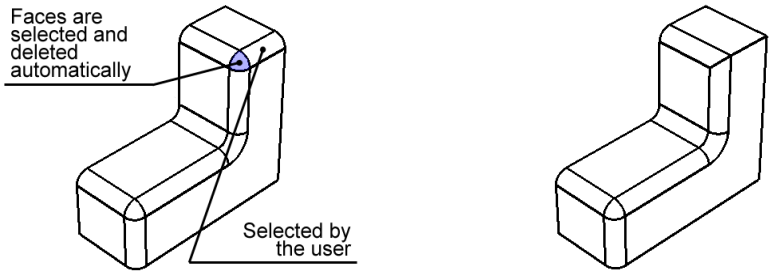
Chains excluding branch faces

A smooth sequence of blend faces will be automatically selected for deletion. The sequence extends up to the faces at which the chain branches into several chains of blend edges (up to branching faces).



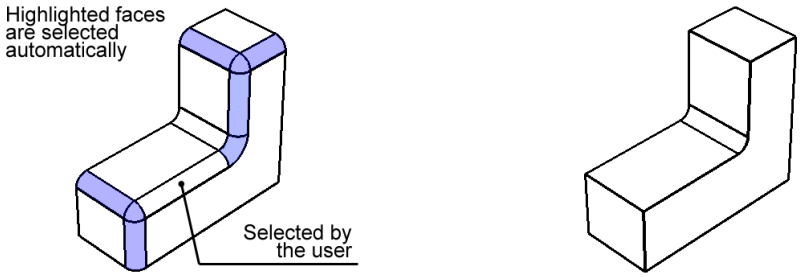
Chains including branch faces

A smooth sequence of blend faces is automatically selected, extending up to and including the branching faces.



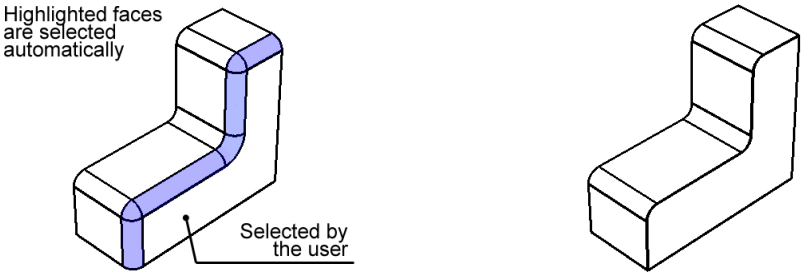
Maximum chains

A smooth chain of as many blend faces as possible is selected automatically.



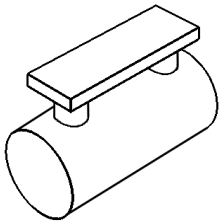
Only dependent faces

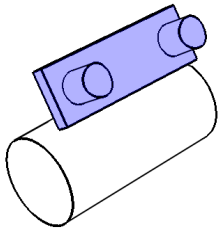
Only those blend faces are selected automatically that are dependent on the faces selected by the user. The user-selected faces themselves are not deleted.



SEPARATE FACES

This command allows excluding selected faces from an already existing body, and creating a new body based on them. The resulting voids in the bodies can be closed by one of several methods.





When selecting the separation faces, that is, the faces that will be excluded from the existing body, the same mechanism is used as in the command **"3ZD: Create Delete Faces".** The separation faces can be specified explicitly or indirectly.

Call the command **3ZX: Separate Face** via:

Icon	Ribbon
	3D Model → Special → Faces → Separate Faces
Keyboard	**Textual Menu**
<3ZX>	Operation > Face > Separate Faces

To create the operation, do the following steps:

- ✓ Select separation faces.
- ✓ Specify the methods for processing the original and separated faces (optional).
- ✓ Confirm operation creation.

Selecting Separation Faces

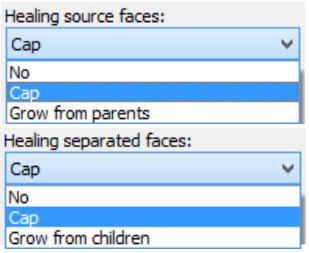
To select separation faces, use the option:

	<O>	Select Elements

When using this option, you can specify not only the particular faces to be separated, but also edges and loops defining the faces to separate. All selected elements are automatically added to the list of separation faces.

Specifying Healing Methods for Original and Separated Faces

As a result of separating faces from the original body, voids may occur in the original and the resulting bodies, that invalidate the closed volume. The options "Healing source faces" and "Healing separated faces" allow selecting a healing method for such voids. The sets of methods for the original and separated faces are the same.



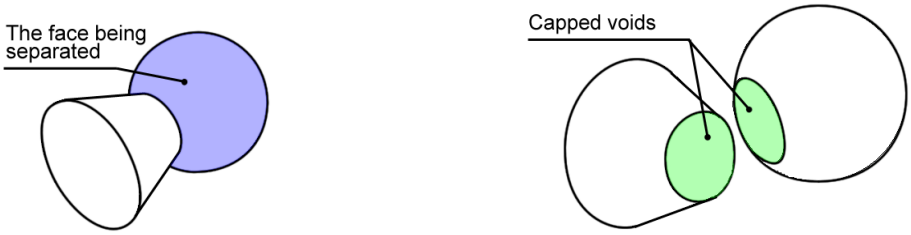
Note that the original faces and the separated faces are either healed ("Cap" or "Grow from parents/children"), or left as is.

No healing

In this case, the system does not process the introduced voids. The resulting bodies are sheet bodies.

Cap

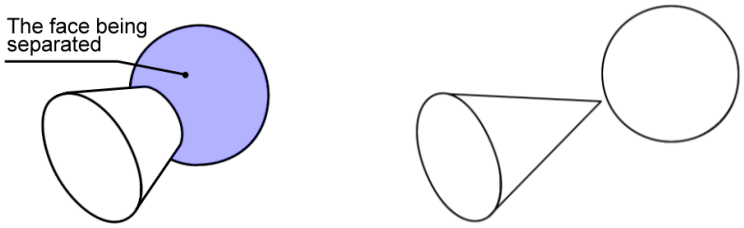
The system attempts to process the introduced voids by creating a single capping surface. The capping surface must contain all edges surrounding the body's voids. Therefore, this method works in the case when those edges result from intersecting the body with a single surface.



Depending on the original model geometry and the selected separation faces, more than two bodies could result from the operation. In the model tree, the resulting bodies will have references to the parent operation (the "Split Face") named: in the first two bodies - "First part" and "Second part", in other bodies – "Part", "Part_0", "Part_1", etc.

Grow from parents/children

In this case, the system tries to heal the introduced voids by growing the faces adjacent to the voids (the child faces or parent faces).

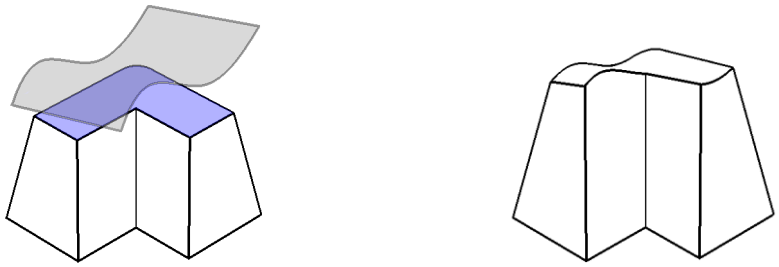


Depending on the geometry of the original model interfaces selected for separation, the operation could yield more than two bodies.

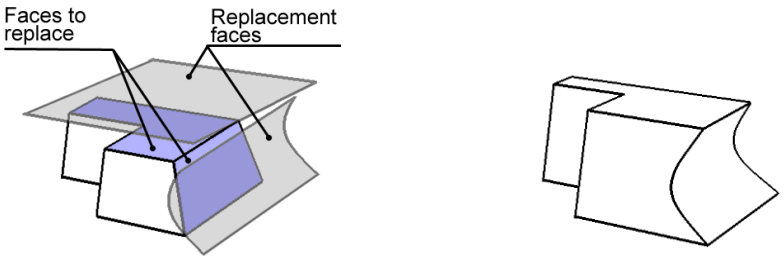
To get the desired result on the original (source) and resulting faces, you can use various methods of healing the voids (except for the "No healing" case).

REPLACE FACES

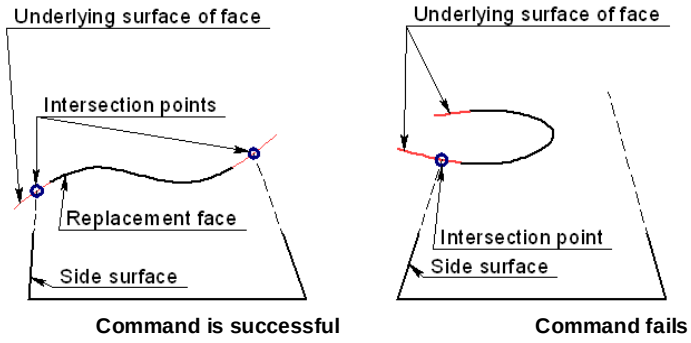
This command allows replacing a geometrical surface underlying the selected face, by another geometrical surface.



In the usual command flow, the faces to be replaced are selected in pairs with the replacement surfaces (by selecting elements with these underlying surfaces). The surface underlying the face being replaced is substituted by the replacement surface. As a result, the face being replaced changes its shape according to the shape of the replacement surface. The side faces that bound the face being replaced are extended or trimmed to the intersection with the replacement surface. If several pairs of faces to replace - replacement surfaces were selected at once, this procedure is repeated for each pair.



It is possible that no intersection between the side faces and the replacement surfaces will be found despite extending the side faces. Then the command will end up in an error message.



The command **3ZR: Replace Face** can be called as follows:

Icon	Ribbon
	3D Model → Special → Faces → Replace Faces
Keyboard	Textual Menu
<3ZR>	Operation > Face > Replace Faces

To create the operation, do the following steps:

1. Select sets (pairs) of faces to replace and the replacements.
2. Define advanced parameters (optional).
3. Confirm operation creation.

Selecting Faces for Replacement

You can select multiple pairs of faces to replace and replacement surfaces.

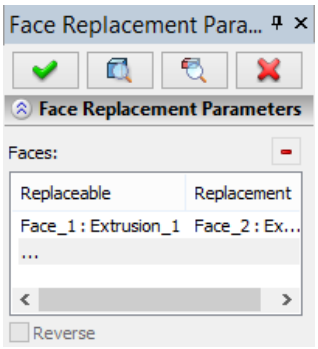
To select faces to replace, use the option:

	<F>	Select replaceable faces

A face to replace could belong to a sheet body, but in that case it should be "surrounded" by adjoining faces on all sides.

To select replacement faces, use the option:

	<R>	Select replacement faces



When specifying replacement faces, you can select not only faces, but also edges, workplanes and 3D profiles. Selecting a workplane or a flat 3D profile is similar to selecting an appropriately positioned flat face. If a non-flat 3D profile was selected, the system will use its underlying surface as the replacement. An edge can be selected if it fully lies in a plane, but is not straight.

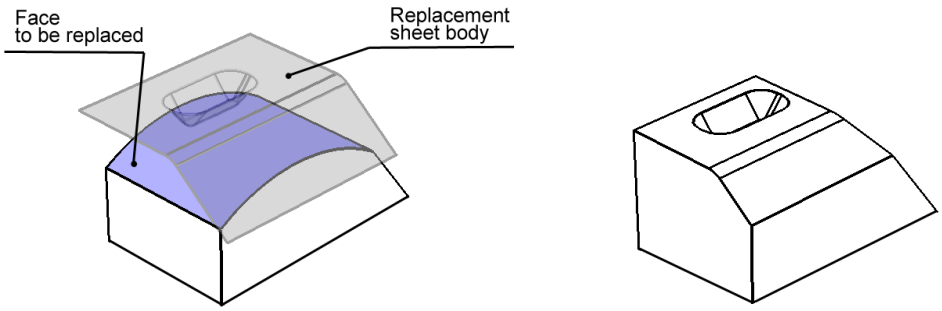
You can also start selecting a face to replace and a replacement surface via the property window. Working with the "**Faces**" list is similar to working with other such lists.

Besides simply selecting the replacement surfaces, it is also possible to select a sheet body. The following option is used for this purpose:

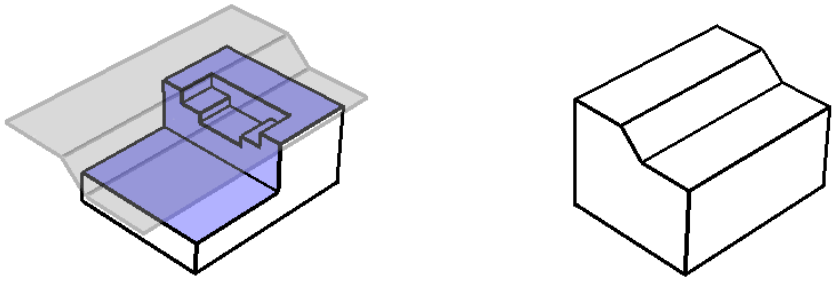
	<S>	Select sheet body

When using this option, you can select sheet bodies, faces and 3D profiles.

The faces being replaced will be replaced by one or multiple new faces copying the shape of the replacement sheet body.



While only a single sheet body is allowed, you can select multiple faces to replace at once.



When using a sheet body, the faces to be replaced may not have holes, otherwise the command will fail. Besides that, the replacement sheet must be large enough, so that the intersection could be constructed between the replacement sheet and the extensions of the adjacency faces.

To cancel selection of a particular pair of the face to be replaced - the replacement surface, you can use the properties window and the button . As for canceling selection of all faces, you can use the option:

	<X>	Cancel selection of all faces and surfaces

Selection of the replacement sheet body can be canceled by the option:

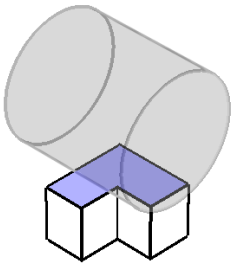
	<K>	Cancel sheet body selection

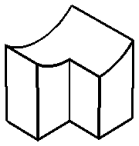
Defining Advanced Parameters

As was mentioned above, the adjoining faces are grown or trimmed to the intersection with the replacement surfaces. If two intersections are found, which one will be picked by the system? By default, the intersection is accepted with the part of the surface where the normal points towards the faces to be replaced.

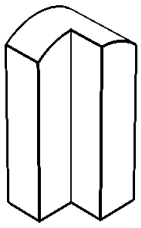
However, in some cases, you may want the system to pick the second intersection choice. For this purpose, use the “Reverse” flag, located at the lower portion of the property window.

If the flag is set, the system will pick the intersection with the portion of the replacement surface, whose normal points away from the face being replaced.





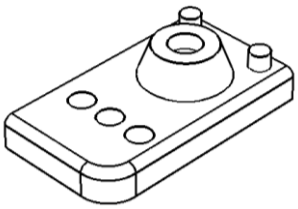
"Reverse" is Off

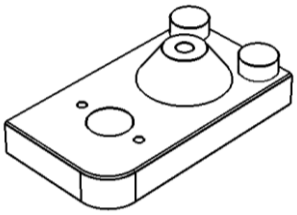


"Reverse" is On

CHANGE FACES

This command allows modifying parameters of the faces whose underlying surfaces are analytical (cylinder, cone, sphere and torus), as well as the parameters of faces created by a blend operation. Most commonly, this command is used for handling imported models.





To use the command, select one or several faces to modify (faces being modified), and specify their parameters. Parameters vary depending on the face type, and can be shared across all faces or be specific to a face. The required condition for executing the command is, the face being modified must be "surrounded" by adjacency faces.

The command **3ZC: Change Face** can be called as follows:

Icon	Ribbon
	3D Model → Special → Faces → Change Faces
Keyboard	Textual Menu
<3ZC>	Operation > Face > Change Faces

To use the command, do the following sequence of steps:

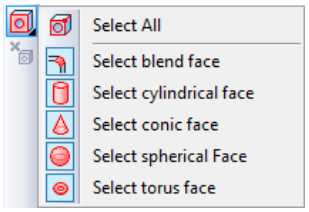
- ✓ Select faces to modify.
- ✓ Define face parameters.
- ✓ Define advanced parameters (optional).
- ✓ Confirm operation creation.

Selecting Faces to Modify

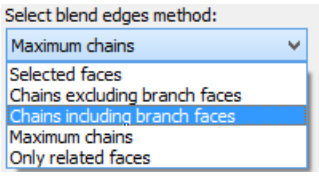
To select the faces to modify (the faces, whose parameters will be changed), use the option:

	<F>	Select faces

When selecting faces, the system allows selecting only the faces that can be modified, that is, cylinder, cone, sphere and torus type surfaces and blend faces.

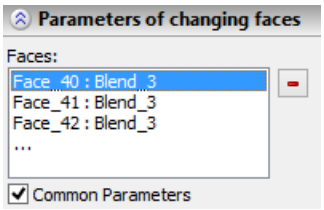


When selecting blend faces, you can use the **Select blend edges method** combo box. Exactly the same combo box is used among the properties of the command **Delete Faces**. Each item of this combo box was described in details in the respective section of this chapter. The command **Change Faces** utilizes this combo box in a similar way, except that the automatically selected faces are subjected to parameter changes, rather than deleted.



You cannot modify blend faces and faces of other types simultaneously.

You can select several faces of different types at once.



Once a face is selected, it is added to the "Faces" list. Working with this list is similar to working with other such lists in the application.

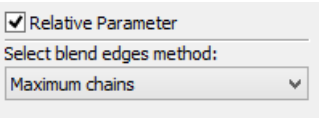
To cancel selection of all faces at once, use the option:

	<X>	Cancel selection of all faces and surfaces

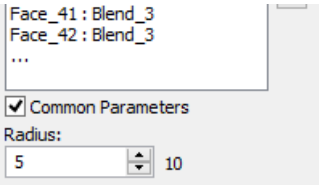
Defining Face Parameters

New parameters of surfaces are always defined in the property window. However, different parameters can be defined depending on a face type. For the faces that are blends, cylinders and spheres, you can define the radius. For cone faces, define the cone angle. For the faces – tori, define the major and minor radii of the torus. Faces of different types may be selected when defining the command. The appearance of the property window changes accordingly, depending on the type of the face selected in the "Faces" list.

If you check the "Relative Parameter" option, the respective face parameter will be changing relative to the original parameter. For example, if the original cylindrical face had the radius equal to 20, then, with the option "Relative Parameter" set, entering the radius value of 5 yields a cylindrical face with the radius 25 resulting from the operation.

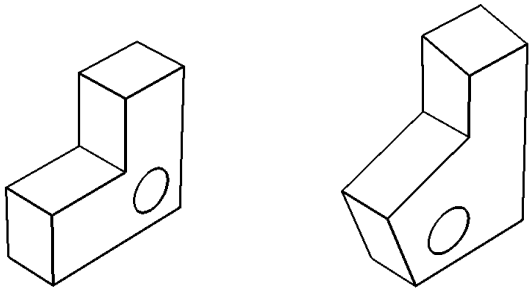


By default, each face in the list of faces being modified has parameters shared across other faces, while having the "Common Parameters"option set in the property window. To define specific properties of a face, turn off this option. If necessary, you can turn off the "Common Parameters" option for all selected faces.

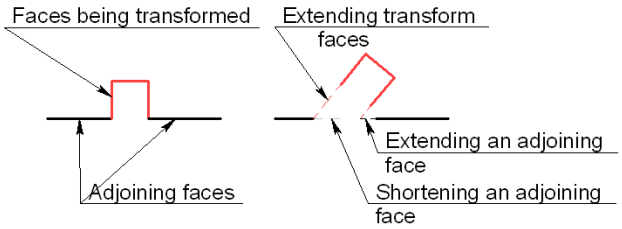


TRANSFORM FACES

This command applies a transformation to one or several selected faces. The command is mostly used for handling imported models, however it can be helpful in other cases as well.



By executing the command, the selected faces are moved to a new position and, if requested, extended (or trimmed) to achieve an intersection with the surfaces of the adjoining faces. The adjoining faces, in turn, are extended (or trimmed) to the intersection with the faces being transformed.



The correctness of the command behavior can be guaranteed only if the topology of the resulting model is unchanged. If the model topology changes, the result of the operation may be incorrect.

When selecting the faces to transform, the same mechanism is used as in the command **3ZD: Delete Faces**: the faces to transform can be specified directly or implicitly. Defining transformation parameters of the selected elements is done similar to that in the command **3EG: Transform Element**.

The command **3ZT: Transform Faces** can be called as follows:

Icon	Ribbon
	3D Model → Special → Faces → Transform Faces
Keyboard	**Textual Menu**
<3ZT>	Operation > Face > Transform Faces

To use the command, do the following sequence of steps:

- ✓ Specify faces to transform.
- ✓ Define transformation parameters.
- ✓ Confirm operation creation.

Specifying Faces to Transform

To specify the faces to transform, use the option:

	<F>	Select Elements

When using this option, you can select not only faces, but also edges with loops. The selected elements are put in the **Faces** list.

To cancel selection of all elements, use the option:

	<X>	Cancel selection of all faces and surfaces

If edges or loops are selected, the **Face Selection Pattern** parameter becomes available. The effect of this option is described in the command “Delete Faces”.

When selecting a set of edges, the default option is:

	<G>	Smooth edge chain selection mode

To select individual edges, turn off this option.

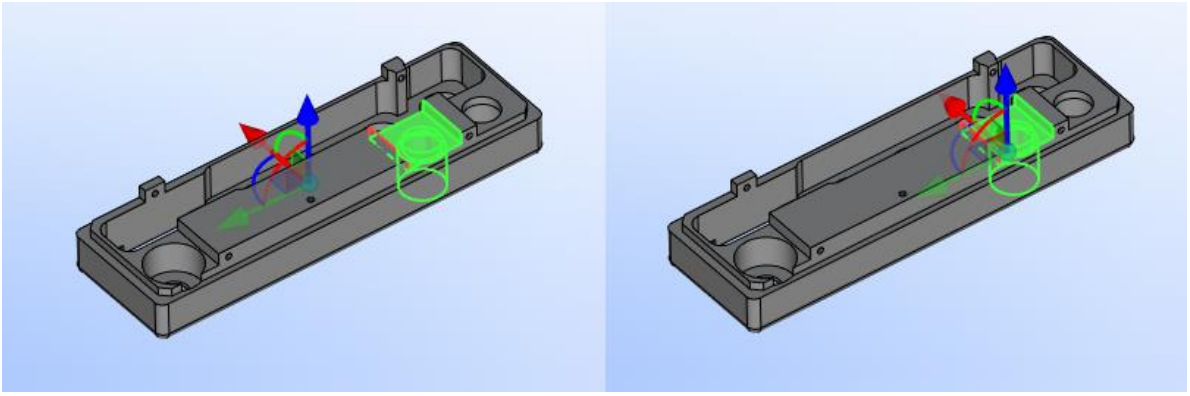
SPECIFYING A TRANSFORMATION

Special manipulator appears in the scene after selection of the first element. It helps you to change the position of the selected faces. Procedure of specifying transformations is described in detail in the "3EG: Transform Element" command description. The exact values of translations and rotations can be set in the properties window.

Use the following options to select the coordinate system relative to which the transformation will be performed:

	<1>	Source CS on source operation
	<2>	Select LCS for current transformation

Option is active by default. The manipulator is in the center of original operation bounding box. The drop-down list of the option has the second variant of coordinate system positioning - option . When you select this option, the manipulator will be in the center of the bounding box that bounds all movable face.



Source CS on source operation | LCS for current transformation

Option allows you to select any LSC for specifying the transformations.

The operation can be confirmed either at the stage of selecting the elements to transform or at the stage of defining the transformation parameters.

EXTENDING SURFACES

This command allows extending a selected face (or several faces belonging to a sheet body) in the specified direction, for the specified length amount. The direction of extending the face is defined by selecting the side edges of the faces being extended. 3D profiles can be extended as well. In this way, an enlarged face will be created as a result of executing the command, identical in shape with the underlying surface of the 3D profile.

Sometimes, self-intersections occur when extending a face. In such cases, the command execution ends up in an error message.

The command **3SX: Surface Extend** can be called from the textual menu:

Icon	Ribbon
	3D Model → Special → Faces → Extend Faces
Keyboard	Textual Menu
<3SX>	Operation > Face > Extend Faces

To use the command, do the following sequence of steps:

✓ Select the object to extend.

- ✓ Select edges.
- ✓ Specify extension amount.
- ✓ Define advanced parameters (optional).
- ✓ Confirm operation creation.

Selecting Object to Extend

To select an object to extend, use the option:

	<R>	Select surface operation

When specifying an object to extend, you can select faces, 3D profiles and sheet bodies.

Selecting Edges

Once the object to extend is selected, you can select one or several edges bounding this object. Those edges will indicate the direction of extending the given object. To select edges, use the option:

	<H>	Select Edge

The default option for selecting edges is:

	<S>	Smooth edge chain selection mode

To select individual edges, turn off this option.

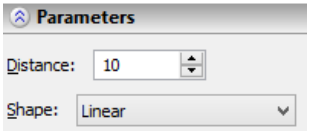
To select all edges surrounding the selected object, at once, you can use the option:

	<A>	Select all edges

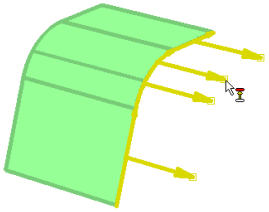
To cancel selection of any of the edges, click it again.

Defining Amount of Face Extension

The entry **Distance** means the distance from the selected edge to the new edge bounding the extended surface, as measured along the surface.



Enter the amount of face extension in the **Distance** input box of the property window.

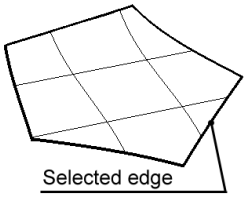


Besides the property window, this value can be modified in the 3D window, using arrow-shaped draggers. The draggers appear after selecting at least one edge. To "grab" a dragger, move the pointer to the tip of the arrow (the pointer will then change its appearance) and depress the left mouse button.

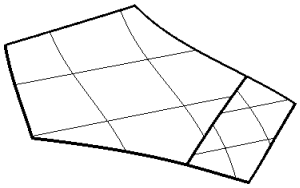
Note that the same extension amount is applied in all specified directions.

Defining the Advanced Parameters

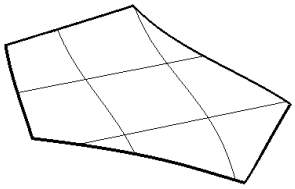
The parameter "**Modify source surface**" tells the system, whether to modify the original face topology, or leave it as is. The effect of this parameter on the result is shown below.



Original model

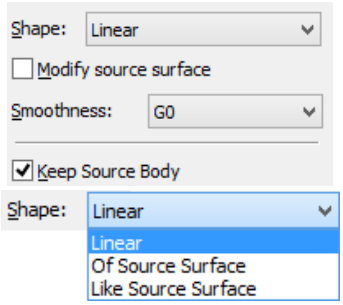


"Modify source surface" parameter is off

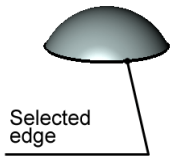


"Modify source surface" is on

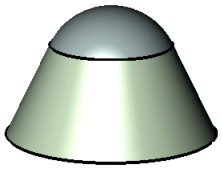
The parameter **Keep source solid** specifies, whether to leave the source body in the 3D scene (the whole body altogether, rather than just the selected face). If this parameter is off, the source body disappears upon the operation completion.



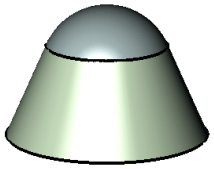
The parameter **Shape** defines the shape of the new face created as a result of executing the command. This parameter can take several values. **Linear** means, the face extends straight, staying tangent to the original face at each point of the selected edge. In this way, the surface curvature has a jerk near the selected edge. With the parameter value set to **Of Source Surface**, most of the newly created surface is linear (ruled), as in the previous case. However, the surface curvature around the selected edge changes smoothly.



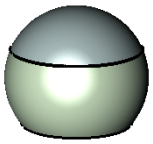
Original model



"Linear" Shape



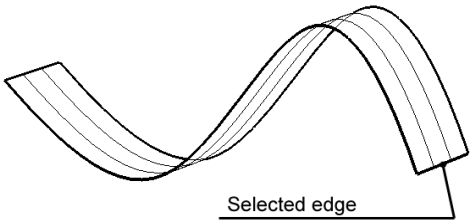
"Of Source Surface" Shape



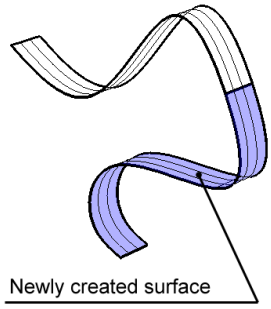
"Like Source Surface" Shape

In real life, the angular difference between the new surface with the parameter **Linear** and that with the parameter **Of Source Surface** is about 3 degrees. Therefore, the difference between the two results is practically vanishing.

The parameter **Like Source Surface** means the new surface will rely on the definition of the source surface. The example below shows the effect of this parameter on a spline surface.

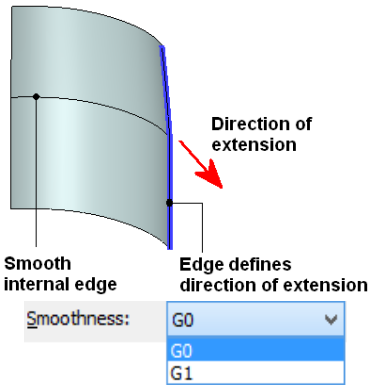


Original model



Resulting model

Original faces may contain internal smooth edges. Internal edge is considered to be "smooth" when converging faces that form this edge are tangent to each other at the point of intersection of the edge with another edge that defines direction of surface extension.



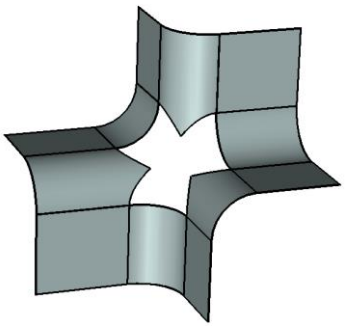
The "smooth" state of internal edges may be affected in the process of surface extension. To avoid this set **G1** mode for **Smoothness** option (G1 – is geometric continuity by first derivative). This option will recommend system to retain tangent continuity, keeping "smooth" state of the internal edge on surface extension.

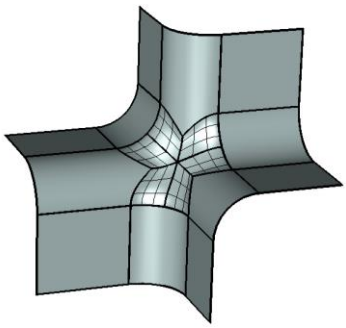
By default **Smoothness** option is set as **GO** (recommendation to ensure only surface continuity).

FILL HOLE

This command serves to patch "hole" contours (regions) in a sheet body. Filling is done by creating one or several faces closing the selected regions. It is also possible to patch regions in a solid body, however with numerous restrictions.

The command is used in surface and solid modeling. It can also be useful when working with imported models whose bodies contain voids resulting from loss of faces due to export/import deficiencies.





When executing the command, the system may fill the gap using an analytical or ruled surface, depending on the original geometry. It is also possible to make patches based on a specified sheet body or geometrical surface.

Selecting Regions to Fill

You can close several regions of one body at once by performing the operation.

Each region is defined by its contour - the set of edges making up the given contour. To define a contour, you need to specify all body edges that belong to it. Contours must be closed.

When defining contours, you can use the mode of automatic selection of all edges. In this mode, you just need to select one of the edges in each contour. The system will select the remaining edges automatically. The composition of the selected contours can then be manually edited.

An also provided mode to select all contours serves to automatically select all holes to patch in the specified body.

Hole-Filling Methods

The specified regions of a sheet body can be patched by various methods:

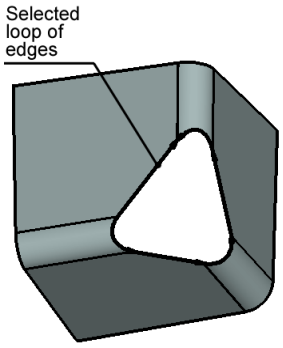
- Automatic generation of the filling surface;
- Fill by the sheet body;
- Filling by a geometrical surface.

The surface resulting from patching regions is by default merged into the source body. If necessary, it can be saved as a stand-alone operation.

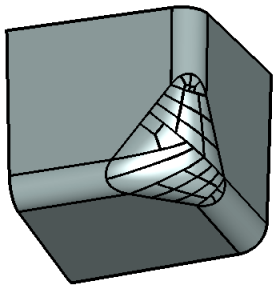
Filling with Automatic Patch Surface Generation

In this way of patching, the system automatically creates the faces that can close the selected body regions. The user has a way to influence the type of the created faces. This is done by defining a preferred form of the closing surface:

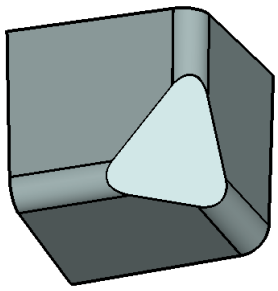
- **Smooth** – the system will attempt to create an analytical or ruled surface closing the selected region;
- **Preferably plane** – the application attempts to close the selected region with a plane first, and then, in the case of a failure, with a spline surface;
- **Plane only** – the system attempts to fill the region using only a plane. If this attempt fails, the command execution ends up with an error message.



Original model



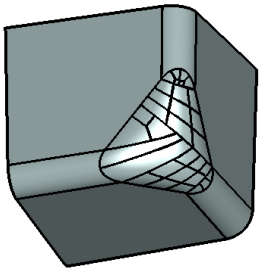
"Smooth" Filling



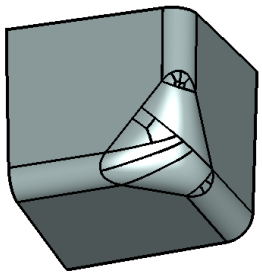
"Preferably plane" Filling

The preferred form of the filling surface is defined separately for each selected contour (that is, separate for each region to be patched). By default, the faces filling the area approach the original model smoothly (tangentially) at each edge bounding the area to fill. The smooth edge condition can be disabled for each individual edge or for all edges in a contour.

When closing a region with a ruled surface, there is an option to minimize the topology. In this case, the system will attempt, whenever possible, to close the gap with a minimum number of faces.



"Minimize topology" flag is off



"Minimize topology" flag is on

Fill by sheet body

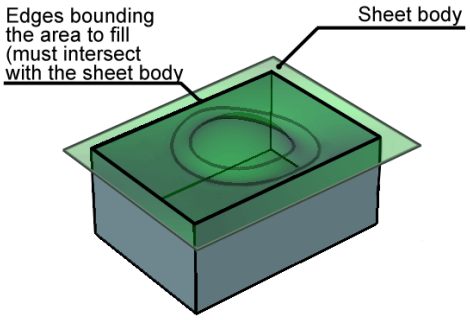
In this filling method, an additional sheet body shall be defined. The selected regions of the source body will be closed based on it. Please note that in this case the same filling method is applied to all selected contours of the source body.

The sheet body defined as the filling surface must satisfy the following conditions:

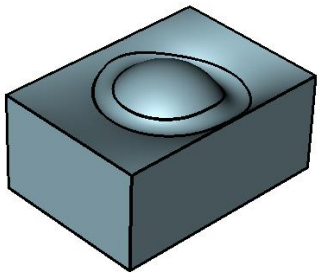
- ✓ Fully close all holes being patched;
- ✓ Have only one boundary (for example, a sheet body like a side surface of a cylinder does not satisfy this condition);
- ✓ Not have holes.

Filling regions by a sheet body can be done in two ways.

The first method is filling with trimming by the borders of the patched regions. In this case, the system trims the shape-defining sheet body along the boundaries of the patched regions. The resulting surface is the sought filling surface. This filling method can be used only under the condition that the filling sheet body goes through all edges that surround the regions to be patched.

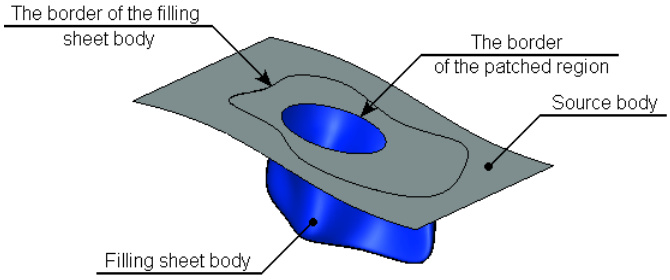


Original body

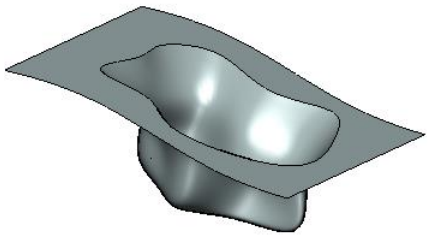


Result

The second method is filling with trimming by the sheet body boundaries. In this case, a region is cut from the source body along the boundary of the filling sheet body. The filling sheet body then "patches" the cut region. That means, the entire sheet body is used as the filling patch. The necessary condition to use this method is that all border edges of the sheet body lie on the source body.

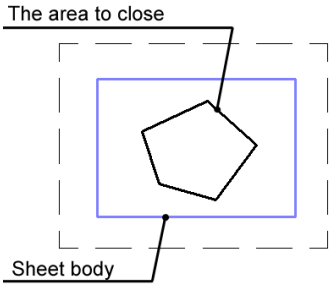


Original body

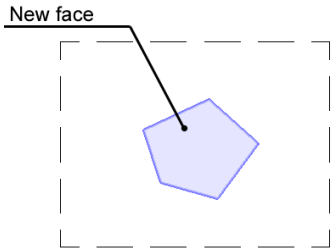


Result

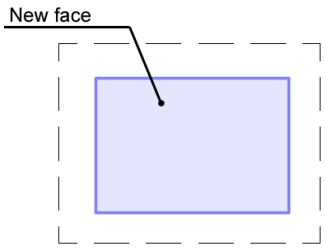
Shown below are schematic diagrams that demonstrate the difference between these two methods.



Original model



Filling with trimming by the patched region borders

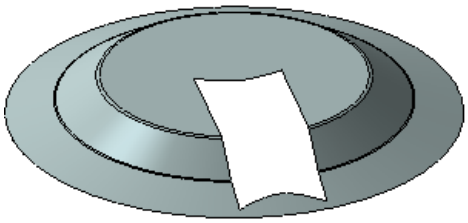


Filling with trimming by the sheet body borders

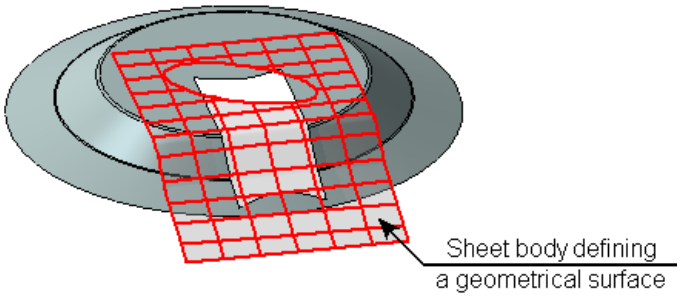
Filling by a geometrical surface

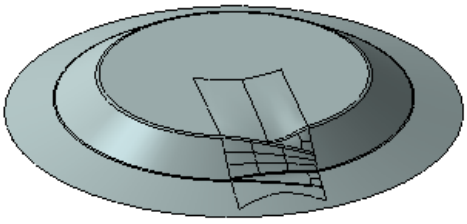
In this filling method, a geometrical surface is additionally defined, based on which the selected regions of the source body will be patched. The surface is defined by selecting a 3D object of an appropriate type (a worksurface or workplane, a face-underlying surface, a planar edge, etc.).

In this case, as in the previous one, the same filling method is applied to all selected contours of the source body.

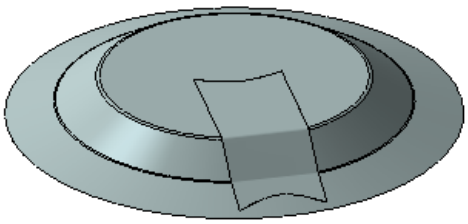


Source body with a hole





The result of patching
with automatic creation of the filling surface



The result of patching
by a geometrical surface

The surface defined as the filling specimen must close all regions to patch in the source body. At the same time, you need to pay attention to the type of the selected surface. Some types of simple surfaces can be extended by the system beyond its defining element. For example, if a plane underlying a flat face was selected as the filling surface, then the system can extend this surface up to the infinity, and it will be always closing all selected regions. On the other hand, a surface beneath a spline face is strictly limited to the face borders.

Besides that, to successfully fill the gaps, the borders of all regions being patched must lie on the selected surface.

When patching, the system constructs a sheet body by the specified surface and trims it by the borders of the regions to be patched. The resulting body will be the filling one.

Specifics of working with solid bodies

The region-filling command allows processing holes in solid bodies as well. However, in this case the command capabilities are limited.

When patching the surface of holes in a solid body, only the "Filling by a geometrical surface" method can be used. You cannot use filling by a sheet body. In the cases when the surface is not specified, but the system managed to perform filling on its base, the system will automatically find the methods of closing this surface (you cannot specify it, unlike the case of the sheet body). Besides that, the result of filling a region in a solid body is always created as a stand-alone body (a separate operation). It cannot be merged with the source body.

> Please note that the face - filling operation cannot be used to remove holes in solid bodies. The "Delete Faces" operation is instead used for this purpose.

Defining the Operation

The command **3ZF: Fill Hole** can be called as follows:

Icon	Ribbon
	3D Model → Special → Faces → Fill Hole
Keyboard	Textual Menu
<3ZF>	Operation > Face > Fill Hole

To use the command, do the following sequence of steps:

1. Select the areas to fill.
2. Select the filling method by specifying the preferred form of the filling surface or a filling surface itself (a sheet body or a geometrical surface).
3. Define additional parameters for the selected filling method (optional).
4. Confirm operation creation.

Selecting regions to patch

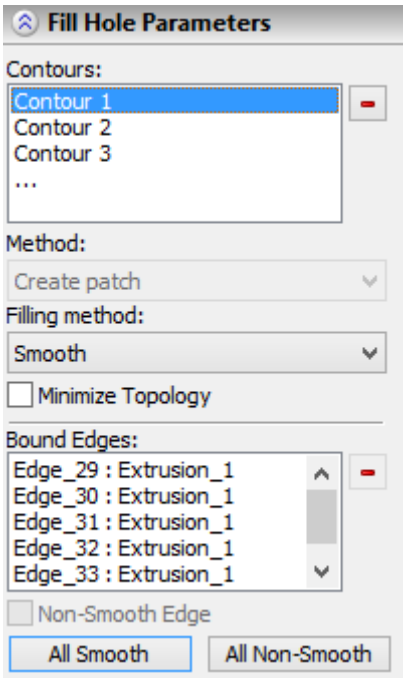
Upon launching the command, the option to select contours of the regions to be patched is active by default in the automenu:

	<C>	Select Contour

The additional option is turned on by default:

	<G>	Contour selection mode

This option serves to create a new contour and select all edges belonging to it by picking just one edge.



If the option is enabled, then you just need to select one of its member edges in the 3D window in order to have the contour selected. The system will automatically find the remaining edges of this contour and make them selected. After that, you can right away proceed with selecting an edge of another contour, and so on.

The selected contours are entered in the **Contours** list in the properties window. If you select one of the contours in the list (by picking it with), then the **Edges** field will display the list of edges that make up this contour.

To manually select contour edges (this may be required, for example, when working with a solid body), disable the option . In this mode, selecting an edge with the option will create a new contour in the list of contours (with a single selected edge). You can then select more edges for the contour using the option:

	<E>	Select Edge

To define a second contour, you need to use the option again, create a new contour with a single edge in it, and then select more edges using the option , and so on.

To delete one of the defined contours, select it in the "Contours" list in the properties window, and then click the button beside this list.

To delete an individual edge in a contour, select the intended contour in the list of contours. The "Edges" list will display the edges of that contour. Select the desired edge and click the button beside the list of edges. The selected edge will be removed.

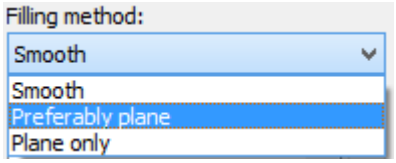
To automatically select all holes in a sheet body as the contours to patch, use the option:

	<A>	Select all Contours

After calling this option, you just need to select the sheet body, in which you want to close the holes. All contours will be defined automatically.

Defining parameters when filling with automatic generation of the patch surface

The preferred form of the patch surface is defined in the **"Filling"** field of the properties window. Please note that defining a preferred filling form of the patch surface is done individually for each selected contour. That means, in the case of selecting multiple contours, you need to pick each contour in the list in a sequence and define the preferred surface form for each of them.



When creating a continuous patch surface for all edges of all contours, the smoothness condition is set by default. To disable this condition for an individual edge, select the contour in the list of contours, which includes the given edge, and then select the desired edge in the list of edges of that contour. Set the **Non-smooth edge** flag for the selected edge. You can similarly reenable the smoothness condition for an individual edge of a contour, by selecting this edge and disabling the flag **Non-smooth edge**.

To disable the smoothness condition for all edges of all contours, you just need to click the button **[All Non-Smooth]**. The button **[All Smooth]** allows to reenable the smoothness condition for all edges of all contours.

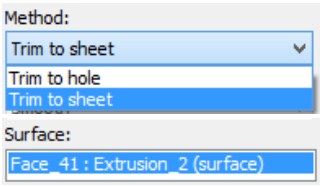
The minimal topology mode is enabled by setting the flag **Minimize Topology** – separately for each patched contour.

Defining parameters when filling by a sheet body

You can define the sheet body that will be used as the specimen to create the patch surface by using the option:

	<S>	Select surface operation

The selected sheet body is entered in the **Surface** field of the command's properties window.



The trimming method is defined in the field **Method**. To trim by the borders of the regions being patched, use **Trim to hole**, to trim by the sheet body borders – **Trim to sheet**.

A selection of a replacement sheet body can be canceled by the option:

	<D>	Cancel sheet body selection

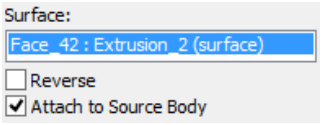
Defining parameters when filling by any geometrical surface

You can specify a 3D object defining the geometrical surface to use for patching body regions, using the option:

	<U>	Select surface

The selected 3D object is entered in the "**Surface**" field of the properties window.

In the complicated cases, the system may be mistaken with the direction of closing the selected contours. That means, the new faces could be applied not on the selected holes in the body, but rather around them. In such a case, use the additional flag "**Reverse**" that alters the direction of closing the selected regions.



Selection of a replacement sheet body can be canceled by the option:

	<D>	Cancel sheet body selection

Adding the filling surface to the source body

By default, the created surfaces become part of the source body. This is indicated by the enabled flag **Attach to Source Body**.

> This flag is unavailable when working with solid bodies.

Service 3D Tools and Entities

EDITING

This chapter gives basic ideas and recommendations on editing a 3D model and its related elements of a 2D drawing in T-FLEX CAD. By editing, we mean certain user actions aimed at making changes to an existing 3D model. There could be various reasons for editing:

- ✓ correcting model flaws in the event of getting critical errors in the diagnostics window;
- ✓ introducing improvements to existing model geometry, rearranging or rectifying a model, inserting sequences of operations in the middle of the model tree;
- ✓ rearranging graphic elements in a drawing or in 2D constructions. Replacing elements. Resolving recursive references;
- ✓ redefining geometrical or other properties of individual model elements;
- ✓ modifying variable values;
- ✓ modifying the model structure, reordering operations within the model tree;
- ✓ modifying the methods of snapping and relations between 3D assembly elements, optimizing mates and setting up the function of a mechanism model;
- ✓ deleting elements, excluding model objects from a sequence of related elements. Deleting sequences of related elements, deleting "extra" elements. Other.

A T-FLEX CAD model is usually a parametric one. Therefore, depending on a case it may become necessary to undertake a group of measures aimed at editing the model, since certain changes, once introduced, often cause a need to correct other relations. Therefore, before going ahead with profound editing, it is advisable to do some research and prepare the best plan of actions. When you do such preparations, you may face the following tasks:

1) identifying (researching) the model structure and the steps and methods of its creation;
2) identifying the sequence of related parents and children for a given model element;
3) finding elements:
   - operations;
   - 3D construction elements;
   - 2D construction or graphic elements;
   - variables that control the specific parameters;
4) tracing parametric dependencies: geometrical relations and element dependencies on variables;
5) geometry measurements and checks;
6) identifying invalid elements and the reasons behind model regeneration failures.

In this chapter we review the system tools that may be useful in various scenarios of the editing process. The specifics of editing certain elements is given in dedicated chapters of the documentation, which describe the workflow specifically for those elements.

> We recommend to set aside a backup copy of the source file before proceeding with significant modifications to the model structure.

BEFORE YOU EDIT

Researching the Model Structure and Selecting Elements

The user may often face the task of understanding the sequence of the model creation, the order in which elements and operations are calculated, and identifying relations between model elements. In particular, you may face this task if you need to modify a model created by another author. The following model research approaches are possible:

- reviewing the model structure in the model tree window;
- using the rollback function to determine the sequence of operations creation and analyzing the methods used for creating operations at various steps of model creation;
- "Destructive method". This implies sequential deletion of operations and detailed analysis of the model at certain creation steps, identification of individual operation creation methods, experimenting with the model in various ways at its early stage of creation. We recommend doing this kind of experiments with a copy of the source file;
- going into an operation editing mode, but then canceling the editing, in order to see what elements this operation was based on and how it was created. When you enter the editing mode, the time is spent solely to calculate the rollback state, and when it is canceled, the original state is quickly restored.

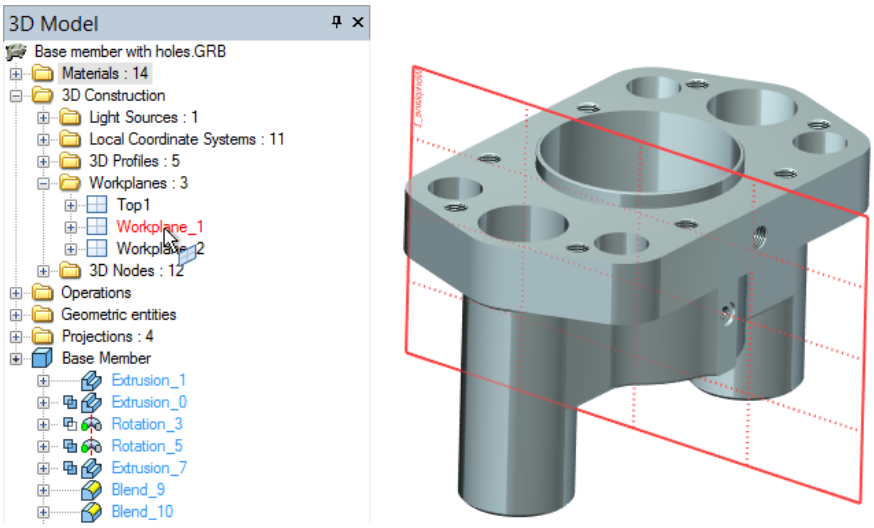
"3D Model" Window

The "3D Model" tool window serves to display the information about the 3D model structure and its creation history, as well as for selecting and managing the model elements. The 3D model tree displays all 3D construction entities and operations of the model. This information is displayed as a tree. One can select 3D elements either on the tree branches which illustrate the history of creating Bodies of the given a model, or on the auxiliary "Constructions" and "Operations" branches, where the elements are sorted by types.

The model structure can be traced by expanding the model tree branches and watching the order of operations creation. Each model tree branch represents the history of creating a certain Body of the model. Any element in the structure that possesses a creation history has an associated or glyph to expand or collapse the tree branch. Other elements have an associated or glyph for viewing the list of source parent elements. Some elements do not have any associated glyph since they have neither parents nor a creation history within the model structure.

By expanding the model tree branches one can watch the elements corresponding to a given operation, which were used as the base for the operation's creation. To view or edit the respective parameters, you can select the object by . The context menu containing the element's commands is displayed upon .

You can select 3D construction elements and operations in the 3D model tree within the folders containing the elements of the relevant type. Upon pointing the cursor to a 3D element name in the model tree, the respective element is highlighted in the 3D scene - this helps to keep the track of the location when there are many elements of the same type in the model. If you select an operation which is buried deep in the model structure, the system will attempt to highlight in the 3D window the model topology elements (edges, faces, vertices), which were created by this operation. This is possible only if such elements are still identifiable within the final Body. Such highlighting helps to understand what part of the model is represented by a given operation. By subsequently selecting operations in the model tree, one can understand the order and the purpose of the operations. This is how you proceed with the preliminary general analysis of the model structure.



To enable the selector and filter settings at the time of selecting elements in the 3D model tree, enable the “Use active View Filter” flag.

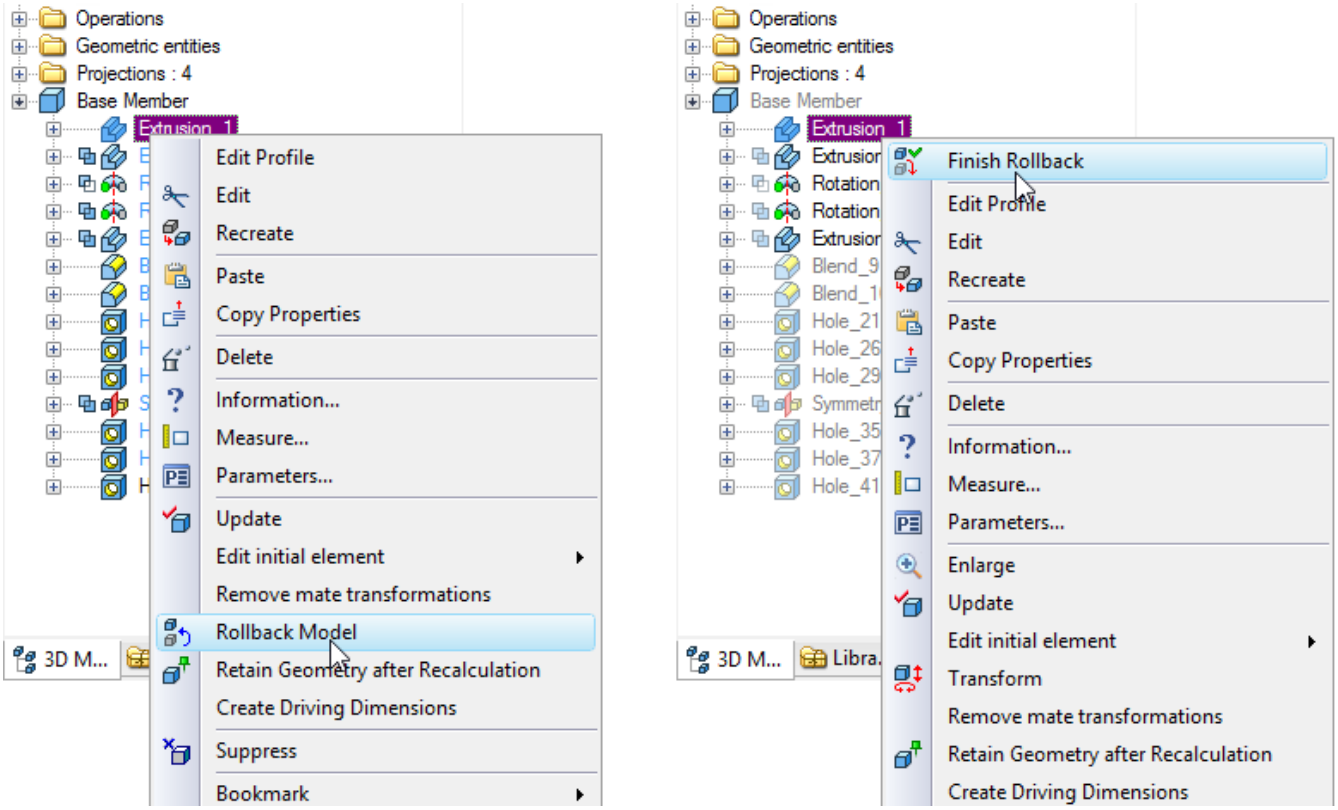
When analyzing the structure, it is helpful to know that the last operations in the creation history of the model's Body, that is the ones that are displayed in the 3D window, are by default entered in *black* in the model tree. Other operations are entered in *blue*. The names of the visible 3D construction elements are also entered in *black* in the 3D model tree. The invisible elements (that is, those hidden through the use of layers, the hide command or otherwise) are displayed in *gray* in the 3D model tree. When selected in the 3D model tree or simply pointed to with the cursor, the hidden 3D elements are highlighted in the 3D scene.

For detailed information on working with the “3D Model” window see the chapter “Basic 3D Terms and Concepts of Modeling with T-FLEX CAD”.

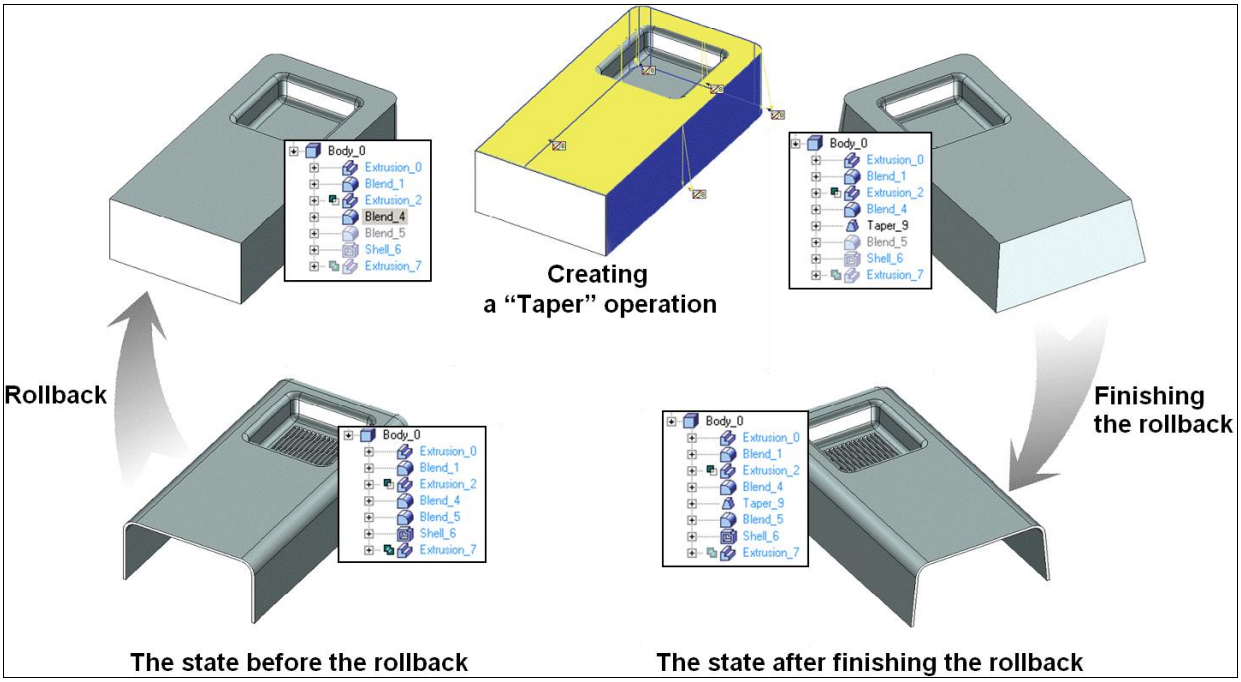
Rollback Model

The system provides the function of rolling back a three-dimensional model to the level of a specified operation. This functionality is helpful in the cases when it is necessary to work more on the model somewhere in the middle of the model tree, that is, revert it to an earlier creation stage.

To perform a rollback, use the "3D Model" window. Select the operation, up to which you need to roll back. Click . Select the "Rollback Model" button in the context menu. To return from the rollback, select the "Finish Rollback" command in the context menu.



Elements and operations that follow after the rollback point in the model's history become hidden in this mode and are inaccessible for you to work on. In this state, those are marked by semitransparent icons in the model tree. All elements and operations newly created in the rollback mode are automatically inserted in the model's history. At each such instant, the system automatically rebuilds the relevant dependencies as necessary to be able to re-create the following operations. If for some reason the introduced changes result in errors in the model, then the system will offer restoring the model state before the rollback. This protects the user from introducing invalid modifications to the model.



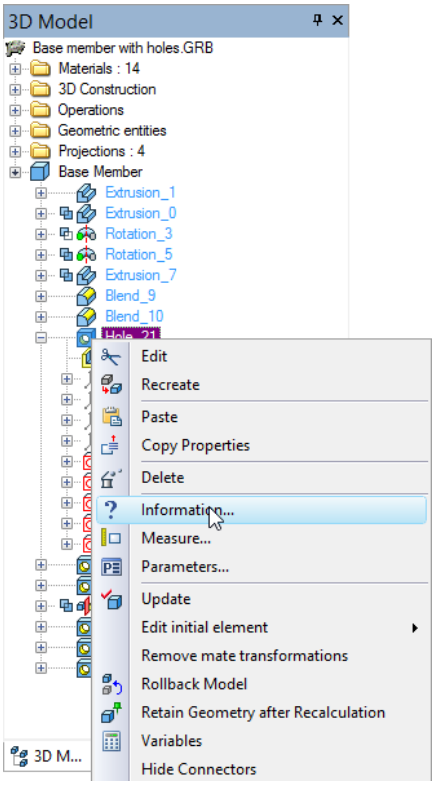
Sometimes, it is helpful to subsequently delete elements and operations in order to get a detailed understanding of the model structure as part of other experiments. In this case, the system will not be spending time for saving and rebuilding relations, unlike in a rollback. But the model itself may be damaged by this, therefore this method should be used on a copy of the source file.

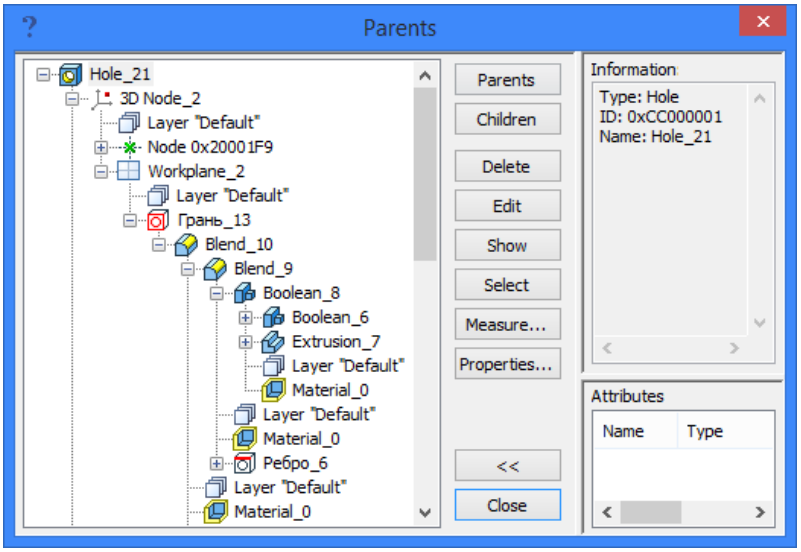
Tracing parameter dependences and relations between variables

One may often need to find out what elements directly depend on a given variable, or which geometrical or parametric relation makes the selected element interact with other objects in the model. This information is not reflected in the "3D Model" window.

The **Help** > **Info** command brings up a special dialog window in which the full model structure is displayed that pertains to all model elements. Sequences of related elements are displayed as a tree. Unlike the "3D Model" window, the "Info" window's structure provides more details not only on the 3D elements, but also on the drawing elements and all auxiliary objects.

This command can be conveniently called from the context menu of the selected element of interest. In this case, the model structure in the command's window is given relative to the selected element or group of elements. The model structure can be displayed in two relational directions: Parents→Children or Children→Parents.

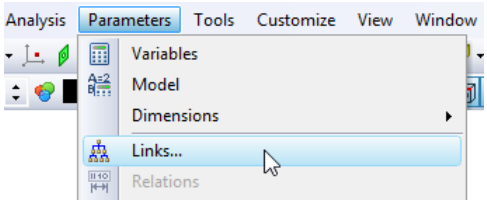


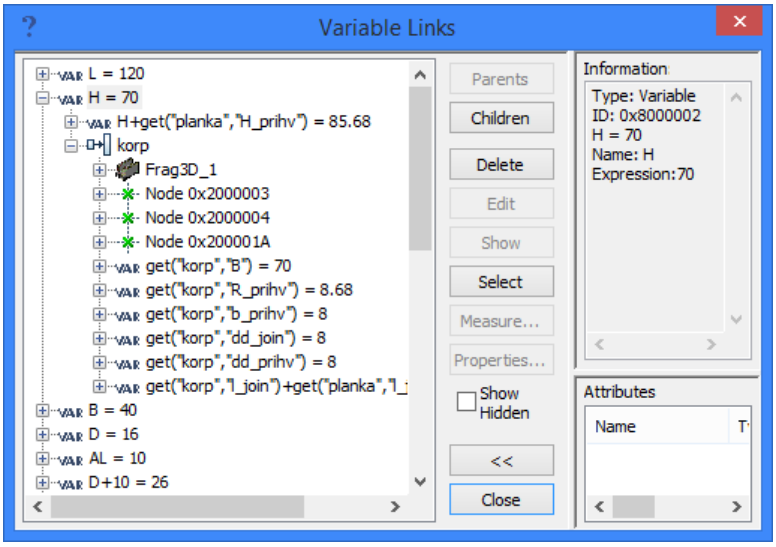


The command's dialog window provides the buttons for launching the general commands for editing, sorting elements and running measurements. Such commands are applied to the element selected in the model structure pane in the left-hand side of the dialog.

The "Info" pane in the right-hand side displays the data pertaining to the currently selected element. This includes its creation method, some general parameters, the ID, etc.

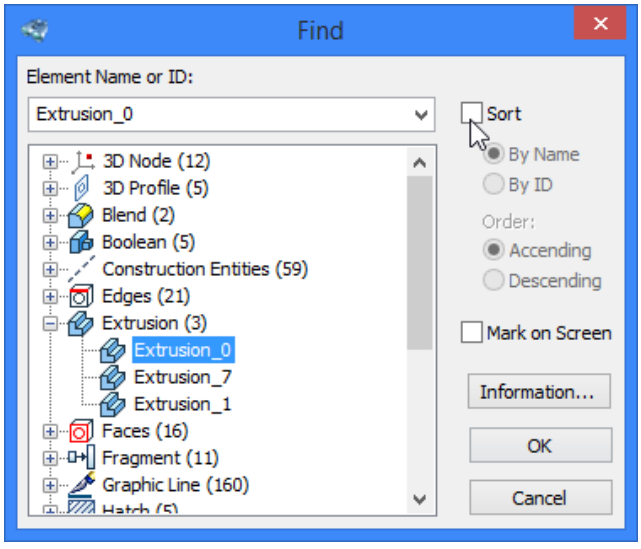
The special command **Parameters** > **Links...** serves to trace the elements which are directly controlled by each system variable or expression that involves that variable. This command's dialog window is similar to the window of the **Help** > **Information...** command. It displays all system variables and expressions, and the structural sequences of dependent elements relying on those variables and expressions.





Element Search

You can use the search to select an element, if you know its individual name or ID. To search an element, there is the dedicated command **Edit** > **Find...** You then need to enter the element ID or name in the command's dialog. To run the search, close the dialog by clicking the **[OK]** button. If the system finds the element, it will become selected and highlighted in the 3D window, and its name will be marked in the model tree. After that, you will be able to proceed with editing, as by calling a desired command or by accessing the context menu via .



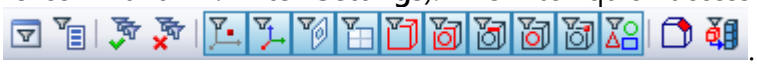
In the search window, all existing system elements are also arranged by types in the relevant folders. Selection can be done directly in the dialog window.

Selecting elements in the 3D scene

By default, you can only select the *visible* 3D elements in the 3D window. The elements hidden by whatever means - whether through the layers mechanism, or through suppressing or hiding 3D elements, etc. - are not selectable in the 3D scene.

> If necessary, the user can enable selection of invisible elements in the 3D window (the **View** tab of the **ST: Set Document Parameters** command).

To select an object, point the cursor to it. The system will offer the current selection choice by highlighting the respective element and display a tooltip with the element name. To confirm the selection, click or . In the case of the context menu will appear with the set of the element's commands.

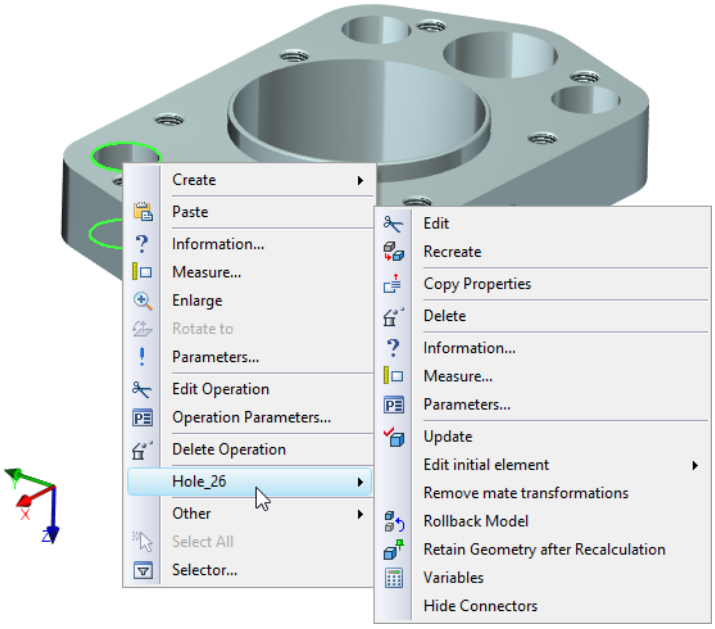
When selecting elements, pay attention to the selector's filter settings. To control 3D elements selection in a densely populated 3D scene, configure the selector (the command **FT: Set Selector Configuration**) and the filter (the command **FL: Filter Settings**). The filter quick access buttons are available on the system toolbar:  . When all filters are enabled, then, upon pointing the cursor to a Body, the topology elements (faces, edges, vertices) are offered first.

The cursor may often point to several objects at once. In this case, if the system-offered selection choice is not suitable, then, after a 2-second delay, the multichoice cursor appears. It lets you make another choice by spinning the mouse wheel.

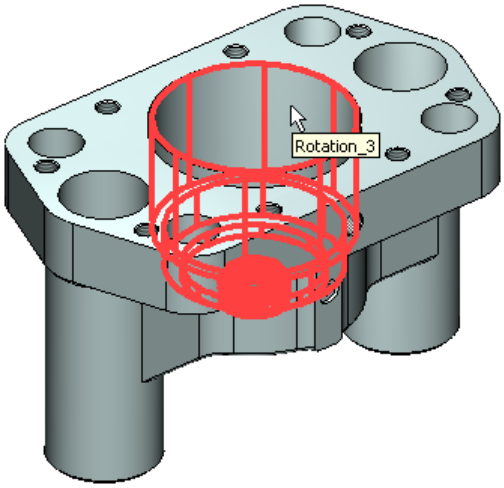
Selecting Operation by Created Topology

The system allows to select a topology elements (face, edge, vertex), and then switch the selection to its parent operation. This is done by using the context menu. The context menu of the topology element has a drop-down submenu for the operation. A line of the context menu matches the name of the operation, for which the submenu is provided.

One option among the set of selector filters on the system toolbar is "Select Operation by Created Faces", which serves to select earlier operations in the model history via the topology elements. When this option is enabled, the context menu contains the names of the operations that directly contributed to the creation of the selected topology element. Otherwise, the context menu offers the operation which is the last in the Body's creation history.



The option "Select Operation by Created Faces" used together with the "Body Selection" filter allows to directly select "faraway" operations. When the cursor is pointed to a body's face in the 3D scene, the system will offer to select the operations that were involved in creating or modifying this face. The offered selection is explained with the pop-up tooltip displaying the name of the operation. If the system finds multiple operations for a given face, then the multichoice mark will appear by the cursor.



Geometry Measurements and Checks

The system has several different tools to perform various checks and measurements of geometrical data. This paragraph lists them all without providing a detailed account of the respective working techniques. For details please refer to the dedicated sections of the documentation.

- The **Parameters** > **Measure** command serves to get geometrical data about an object or to measure relations between two selected objects. This command operates mostly in the 3D window or in the drawing area where objects to be measured are selected. The achieved results can be bound to a variable. This function is used when you need to save the results

of measurements and include the achieved data in other expressions that drive some model parameters. This command is also accessible via the context menu on right-clicking a geometrical object or from the context menu called from an input box in any of the dialogs.

- The sets of commands in the **Tools** > **Geometry Analysis** > ... group serves to perform various studies of the model, as getting mass-inertia properties of a selected body, the geometry validity check on various bodies, body intersection or contact analysis of the model, etc.

Sometimes, to perform a series of distance measurements between various elements of a part, it may be helpful to construct a temporary 2D projection somewhere on the 2D page and create the required dimensions. Once done with measurements, you can delete that projection unless useful for other purposes.

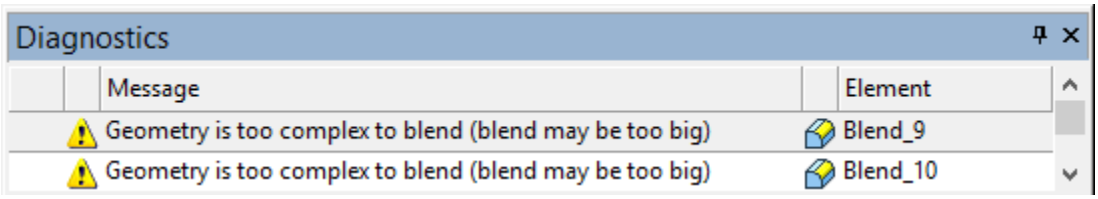
Identifying Failing Elements and the Reasons of Model Regeneration Errors

Upon encountering errors during the model recalculation, the system outputs relevant signals. The user shall be able to correctly interpret those in order to proceed with a sound and quick fix to the source of errors and restore the model validity.

Diagnostics Window

The "Diagnostics" tool window serves to output various system messages regarding errors in the model encountered during a recalculation, as well as warnings and information messages.

Messages are output in the diagnostics window line by line. Each line includes the message type symbol (error-type, info-type, etc.), a brief description of the message cause and the name/ID of the model element.

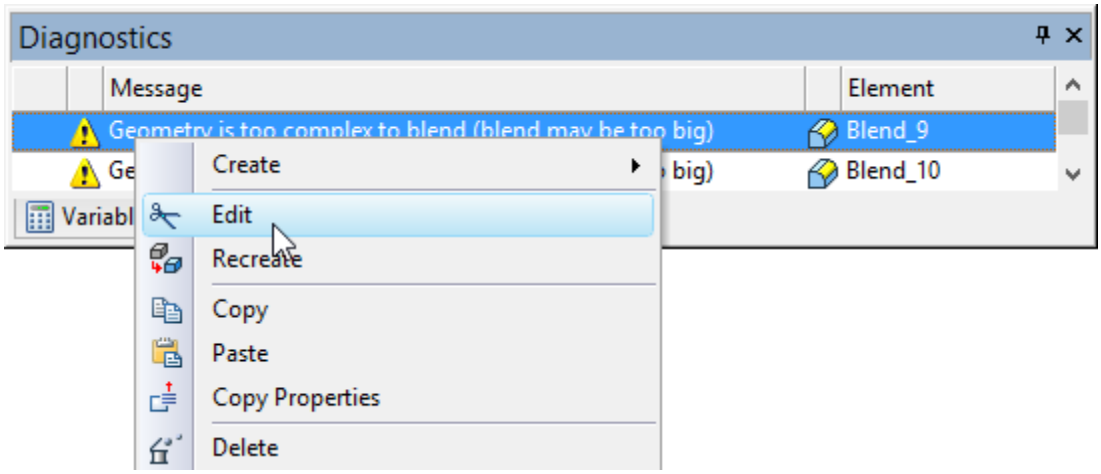


Below we review the most typical errors occurring in the course of model recalculation:

Message text	Remarks
Geometry is too complex...	A geometrical parameter specified in the operation's properties (for example, a blend radius) does not support a valid result under the current conditions. To correct the error, check the geometry near the location of the operation or modify the geometrical parameter value.
Error regenerating *a source/parent element*	One or more elements used as the base for the concerned operation failed the regeneration for some reason. The operation cannot be recalculated because of a missing source element. To correct the error, try to restore the source element or redefine the operation.

Degenerate *element*	The source data used for constructing the element (usually, a 3D profile or a 3D path) for some reason does not support a correct construction of the element. To correct this error, check the source 2D construction elements (for such issues as self-intersecting hatches, open contours etc.).
Error opening Fragment file Link to Nonexistent File	The system couldn't find the required file neither by the specified path nor in the assembly folder.
Error regenerating source *Element*	The system encountered additional errors when recalculating the element, which lead to an invalid result.
Input data is incorrect	The operation's input parameters do not support a valid result. Modify the source elements or the operation's geometrical parameters.
Missing operand in Boolean operation	A Boolean operand could be missing due to an error in its regeneration or due to parametric modifications of the model that caused the body to disappear. In this case, only one operand is left, and the Boolean operation no longer makes sense within the sequence of the model's operations. It could be the case, however, that such "blinking" elements are intentional instances within a family of parametric versions of the part. The Boolean operation provides an option among its parameters to allow a missing operand.
Error recalculating Fragment model	This is a general message conveying the fact that errors were encountered when regenerating a model of a Part or a 3D fragment. You need to investigate the fragment's model to correct the error. Use the command **Open Fragment**, **Detail**.
Recursion in Element links	In most cases, the system warns the user by this message in advance about an attempt to create recursive relations (that is, the element's dependencies on itself). This message being displayed during model regeneration means the model already has recursive relations. To resume the normal function, make sure you find and break the recursion by redefining the elements with invalid relations.

Upon error messages being displayed in the diagnostics, make sure to investigate and fix their causes. By selecting a message line in the diagnostics window you automatically select the respective element. You can call the context menu containing the commands for the selected element by , which includes the editing commands (see *"Editing Techniques"* below).



Diagnostics may include messages about the regeneration errors which occurred during recalculation of 3D fragments. In this case, the context menu will provide the command **Show Fragment Structure**. Once this command is called, the system will show in what fragment the error occurred. Next, depending on the situation, you can use the command **Open Fragment** or **Detail** to investigate the fragment's model.

Since model elements are related with each other, a failure in one element may simultaneously cause a sequence of errors in other elements. In a case like that, you need to start fixing problems from the very first failing element. Therefore, if you see several messages, first pay your attention to the beginning of the list in the upper part of the diagnostics window.

Sometimes, it is not the element triggering a recalculation error that actually needs to be fixed, but rather one of its "parents". For example, a 3D profile doesn't cause regeneration errors by itself, but its geometry may not support valid calculation of a further operation. In such a case, you need to look for the reason why the 3D profile changed. Maybe, that reason originates on the workplane page that contains the source construction for the profile. After that, the error in the child 3D element may disappear automatically.

"3D Model" Window

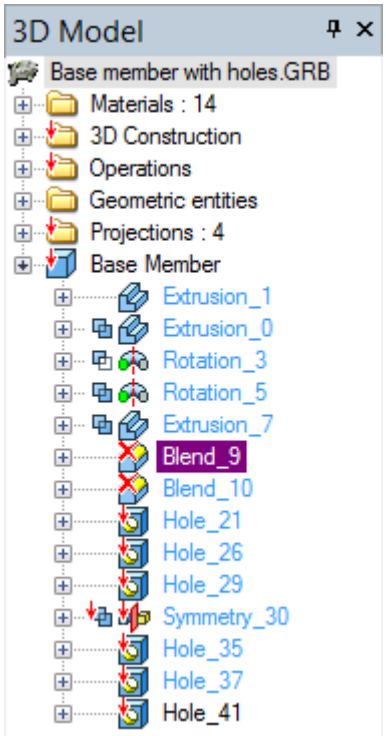
Working with the diagnostics may be enhanced by using the 3D model structure in the "3D Model" window for finding the desired element. The model elements within which errors are detected are marked by special signs:

- A red cross on an element (, ,) means the failure occurred in that particular element.
- A red arrow attached to the icon of a 3D element (, , , ,) means an error was detected among its parents. By tracing the history of the element's creation along the model tree branch, you can reach the first failing element. If there are errors, then the red

arrow will also show on the special folders of the model tree containing lists of elements (,).

- A blue cross on an element ( ) means the operation was suppressed by the user. This is just an info message, rather than an error.

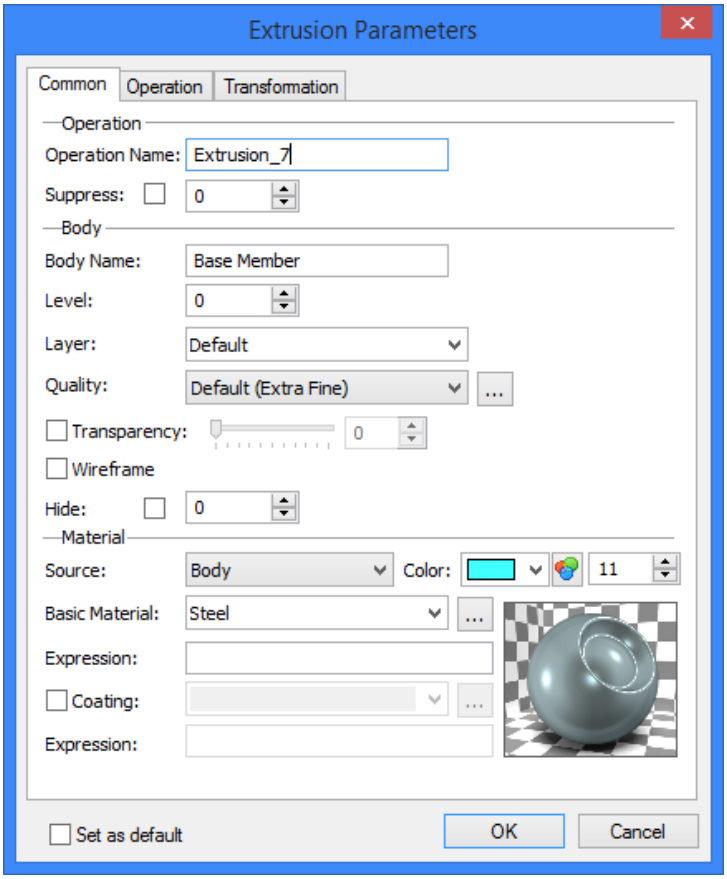
To reach the failing element, you need to expand the branches of the model tree and follow the tree structure, finding the so marked elements.



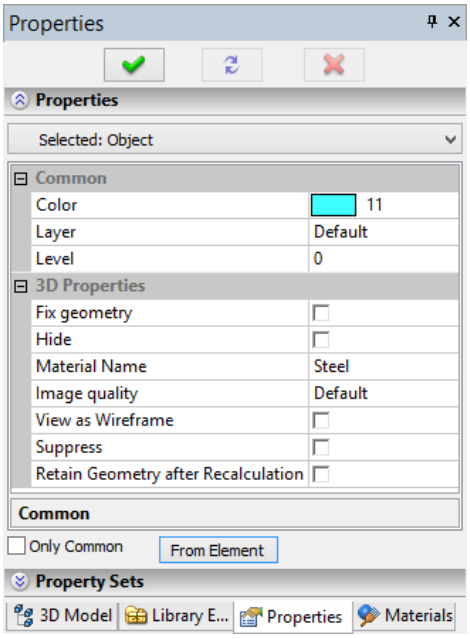
EDITING TECHNIQUES

Modifying Element Properties Without a Rollback

Each system element possesses a certain combination of settings and geometrical parameters. One can view and modify those parameters in the element parameters dialog window. One can call this dialog and modify the values at any time. This is convenient when the full-function editing command is not required, in avoidance of forcing the system to run a potentially time-consuming recalculation up to the edited element.



Select the desired element to call its properties dialog. This can be done directly in the "3D Model" window or with the help of the search function. You can select the element by to access the context menu, and then call the **Properties** command. When you work in the "3D Model" window, you can bring up the element's properties by .



When selecting the element by clicking , the properties window displays certain common properties which can be modified. You can handle common parameters of multiple simultaneously selected elements in the same way. You can conveniently select multiple elements in the "3D Model" window with the help of the <Shift> or <Ctrl> key. Once the objects are selected, a table with all parameters common across the group of the selected objects will be loaded in the properties window. Set the checks against the parameters which are to be edited, enter the new respective values, and then click the button in the properties window.

"Edit" Command Workflow

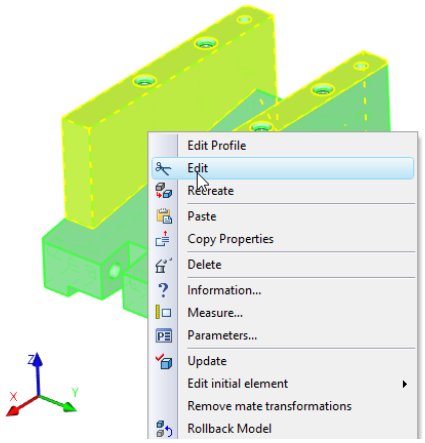
Each system element has its own editing command that serves to redefine the setup, modify the source data, etc. Several methods are available to launch the editing commands.

The commands to edit all types of 2D and 3D elements are located in the "Edit" textual menu:

Keyboard	Textual Menu	Icon
<3E..>	"Edit > Construction > ..."	...
<3E..>	"Edit > Operation > ..."	...

In most cases, the editing command name for a 3D element differs from the respective creation command by just one letter "E" (**Edit**). Examples: **3N** – **3EN** – these are the commands to construct and edit a 3D node, **3X**– **3EX** – the commands to create and edit an extrusion operation. The icons of the editing commands differ from that of the respective creation commands in the added scissors symbol .

The edit a particular element, you first need to select it, and then lunch the editing command. You can select the element in the 3D window or in the 3D model tree by , and the **Edit** command will be available in the context menu. You can also use the "Info" window (see above), which also has the **[Edit]** button, or launch the editing command on a preselected element, as by using the **Search** command.



When the editing command is launched for a particular element, the system appears to be as if in its creation command, with the parameters already defined and geometrical elements selected. The user can proceed using the same methods and techniques as one would use in the element's creation command. Please refer to the dedicated chapters of the documentation for the description of relevant techniques.

If no element was selected at the time of launching the editing command, then you will see the options for selecting an element and general edit-specific actions in the command's automenu:

	<Esc>	Cancel selection

This option clears the selection of the selected objects.

	<A>	Select All Elements

This option serves to select all objects of a particular type present in the model. Invisible objects will be selected if this was allowed by the setting in the command **ST: Set Document Parameters** on the **View** tab.

	<Z>	Select element by name

The list of elements available for selection will appear.

	<I>	Select Other Element

If the first 3D element was selected by the parent 2D element, then using this option will cause subsequent selection of all 3D construction elements of an editable type created based on 2D elements. If an element was selected in the 3D window, then the selection will loop through all similar 3D elements lying on the selection ray. The selection ray is directed perpendicular to the screen plane and goes through the point of the cursor position at the selection instant. Only those elements will be selectable which are located close enough to the point of the cursor. Using this option over causes the sequential selection of all such 3D elements.

	<P>	Set entity Parameters

This option allows to modify parameters of the selected 3D element. When this option is called, the element parameters dialog window appears on the screen. This option is fully analogous to the 3D element parameters editing command accessible from the context menu (by selecting the 3D element in the command-waiting mode).

	<F4>	Execute Edit Command

This option launches the element creation command and loads in it all source data attributable to the edited element. After that, the user can make changes to the set of source data using the same techniques as in the element creation command.

	<R>	Change Definition

This option allows to modify the creation method of the selected 3D element. This mode is fully analogous to the 3D element redefinition (recreation) command accessible from the context menu (by selecting the 3D element in the command-waiting mode).

	<Del>	Delete entity

This option's function is fully analogous to the **Delete** command accessible from the context menu (by selecting the 3D element in the command-waiting mode).

	<M>	Transformation

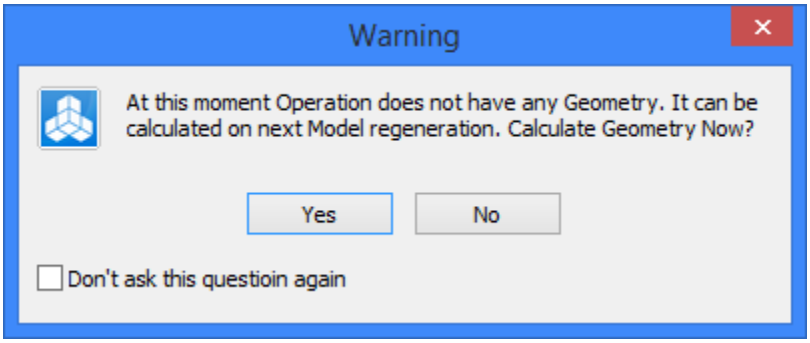
This option calls the **3EG: Transform Element** command for the given 3D element or group of elements. Working with this command is described in the “Assigning "Transformations" to 3D Entities” chapter of this manual.

If several elements of the same type are selected in the editing command using the option and the <Shift> or <Ctrl> key, then the only options available in the automenu will be the transformation and deletion.

Retain Operation Geometry after Recalculation

Upon the activation of the editing command, the system rolls back the model to the state in which it was at the time of creating the operation. The elements remaining in the scene are those independent from the edited object. When executing a rollback, a part of the model requires regeneration, therefore the command may take some time to activate.

To eliminate this regeneration phase and thus speed up the command activation, you can enable saving the intermediate geometry for a certain operation. This will increase use of computer memory and the model file size to some extent, but will help speed up editing. In fact, saving geometry can be used as a temporary measure while experimenting with model editing. To turn on geometry saving, right-select the operation (for instance, in the model tree) and choose the "Retain Geometry after Recalculation" item in the context menu. The system will query the user as to when it shall regenerate the model in order to calculate the geometry to be saved - immediately or upon the next recalculation:



When the "Retain Geometry after Recalculation" option is enabled, the operation icon in the model tree is marked with the special symbol .

Element Redefinition

The context menu of any 3D element accessed by has the **Recreate** command which serves to modify the element's definition method. The system rolls back to the state as before the element creation and launches the element creation command. After that, the user shall create that element anew. This command is typically used instead of simply deleting an element and creating a new one, in case it may be used somewhere else. Therefore, it is important to know that the old element's ID, name and subsequent relations are maintained.

Element Deletion

There are several ways to delete elements from a model. Depending on the case, it may be possible to delete groups of dependent objects or eliminate isolated objects from the model's structure, if possible.

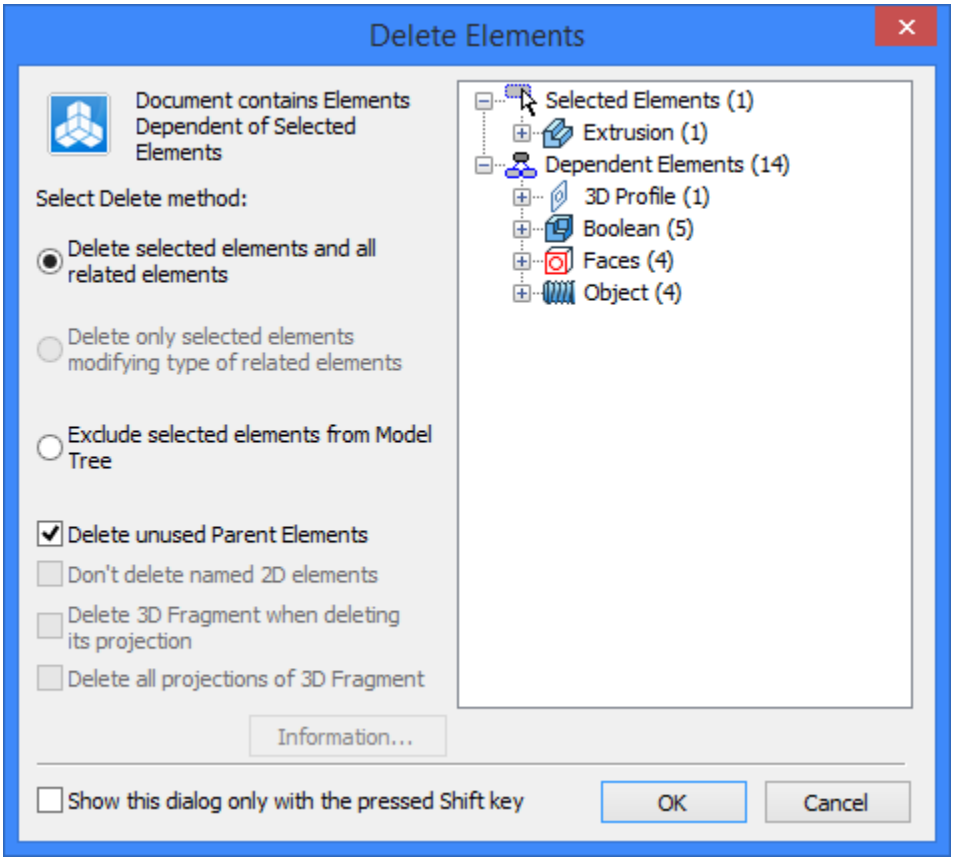
The system also allows to automatically trace the objects unused downstream in the model, and delete them.

"Delete" Command (the 3D element deletion command)

The **Delete** command is called from the context menu when an element or a group of elements is selected. The command serves to delete one or several elements (whether 3D or 2D elements). When the selected objects are deleted, their parent elements can also be deleted automatically, upon the condition that the latter are not referenced anywhere else in the 3D model.

When the command is called for the selected 3D element(s), there are two possible scenarios.

If the selected element(s) has neither dependents nor parent elements not referenced anywhere else otherwise, then the deletion will be instant and without any prompts. If the above condition doesn't hold, then, once the command is called, the **Element Deletion** dialog window will appear.



In the left-hand side of the dialog window, you will be able to select one of the possible options of the command's function:

Delete Selected Elements and all related Elements. If this option is chosen, then the selected elements will be deleted together with all their dependents.

To have the unreferenced parent elements (if any) deleted too, you shall set the flag **Delete unused Parent Elements**. If there are workplanes among such elements, those will be deleted together with their respective pages.

If some of the selected or dependent 2D objects are named, then an additional flag **Don't Delete Named 2D Elements** will be available. With this flag set, the found named 2D objects will be excluded from the deletion list.

When you delete layout elements (schematic images of 3D fragments on the assembly workplanes) the flag **Delete 3D fragment when deleting its projection** becomes available. You can delete 3D fragment that is linked with the deleted layout element.

When you delete 3D fragment, which has a layout, the flag **Delete all projections of 3D fragment** becomes available. It allows to delete all its schematic images from all workplanes.

Delete only selected elements modifying type of related elements. This option is available only when selecting 2D elements (for example, graphic lines on a workplane), which have dependent 2D objects. As a result of choosing this option, only the selected 3D elements will be deleted. As for the children of the deleted elements, the system finds an alternative definition for them.

For example, when LCS linked with a 3D fragment is deleted, the fragment will stay in 3D scene and its position will be fixed. All other links between child and other elements will be broken.

In some cases, geometry or elements of deleted fragments can be left in the scene. For example, if extrusion was created using external profile from the fragment document, the profile and the extrusion will be left in the assembly after fragment deletion.

Another example: when two elements are connected using LCS-connector, the connector will be left in assembly after one of the elements deletion.

Workplanes, nodes and LCS-s remain in a document if their parent element was deleted. Their location is kept according to all transformations. If such object is created using several other objects, and one of the objects is deleted, the link with all other parent objects will break too.

> The only exclusion are 2D nodes that define workplane borders. The workplane can't break links with them.

Exclude selected elements from Model Tree (by reassigning the references of dependent elements to other elements). When this method is used, the system attempts to switch the references of the dependent elements to other model objects. When an operation is deleted, the references of its dependent 3D elements are reassigned to the operation that stands before the deleted one in the 3D model tree. This option is available only in the case when the deleted 3D element has both a parent operation and a dependent 3D element.

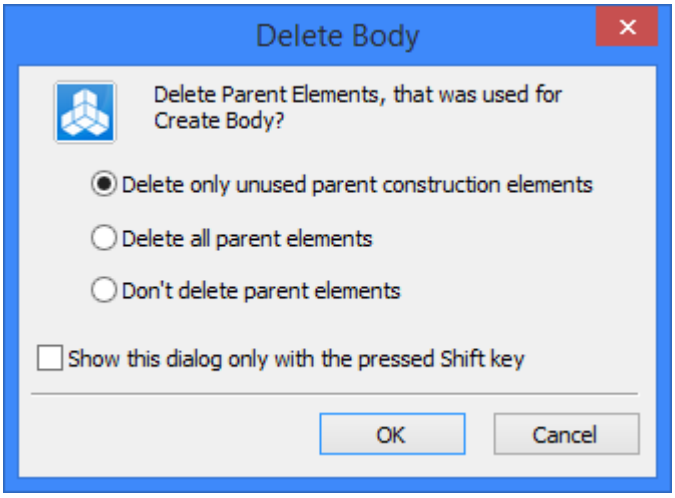
The list of the elements to be deleted is displayed in the right-hand side of the dialog window. The elements are sorted by types and are placed into the appropriate folders. To view the information on the elements selected for deletion, you can click the **[Info...]** button.

If the option **Only show this dialog when the Shift key is down** is flagged, then the "Element Deletion" dialog will not be displayed upon subsequent calls to the command. To have it display when the command is called, you will have to additionally hold <Shift>. The command will be working in this way until the user changes this setting or until the end of the current T-FLEX CAD work session.

"Delete Body" Command

When a Body is selected (which can be done in the 3D model tree) then instead of the **Delete** command the context menu will provide the command **Delete Body**. This command will delete from the 3D model all the operations which are part of this Body's creation history. Besides that, the command allows to delete the parent elements used in this Body's operations.

After calling the **Delete Body** command, the following dialog window is displayed.



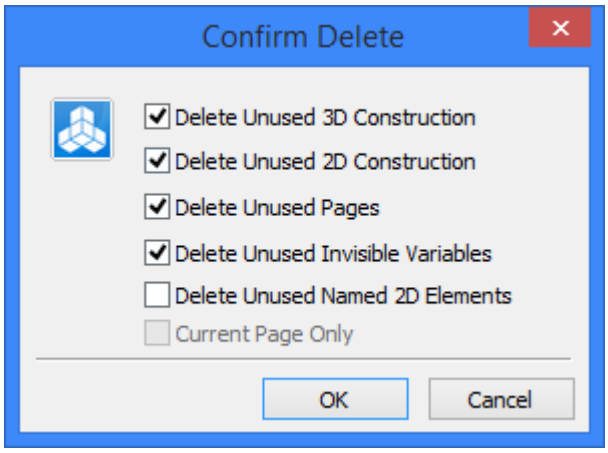
You need to select one of the following actions regarding the deletion of auxiliary elements:

- **Delete only unused parent construction elements**. As the Body is deleted, so are the construction entities and workplanes, together with their pages, if those are not referenced by other elements.
- **Delete All Parent Elements**. The selected Body is deleted together with all elements used in the body's operations (construction entities, variables, etc.), which are not referenced by the model's other Bodies.
- **Don't delete parent elements**. Only the operations that form the selected Body will be deleted.

The flag **Only show this dialog when the Shift key is down** has the function similar to the same-name flag of the 3D element deletion command.

"Purge" Command

The command **Edit** > **Purge** serves to delete all 2D or 3D construction elements that were used neither for creating graphic lines of the 2D drawing nor for operations.



After calling the command, the “Purge” dialog window is displayed. In it, you can set the following flags:

Delete Unused 3D Construction. All unused 3D construction elements will be deleted. 'Unused' are considered the 3D construction elements that are not referenced by a single operation.

Delete Unused 2D Construction. All unused 2D construction elements will be deleted, which are those not involved in creating 2D graphic lines, hatches, dimensions and other drawing elements. The named 2D constructions, that is the 2D construction elements assigned individual names are not deleted by default. To have them deleted, you would need to turn on an additional flag **Delete Unused Named 2D Elements**.

Delete Unused Invisible Variables. All variables marked in the variables editor as "Hidden" and not used anywhere in the model will be deleted.

Delete Unused Pages. All empty pages in the current document will be deleted.

The unused 2D pages also include the pages of workplanes created automatically by the **Draw On Face** command if the user didn't use them to create anything. Such pages are deleted together with the projections of source faces which were automatically created on those pages.

Delete Unused 3D IDentifiers. As the unused 3D IDs are deleted, the 3D model size gets reduced. As a result, the undo stack of the **UN: Undo Changes** command also gets cleared.

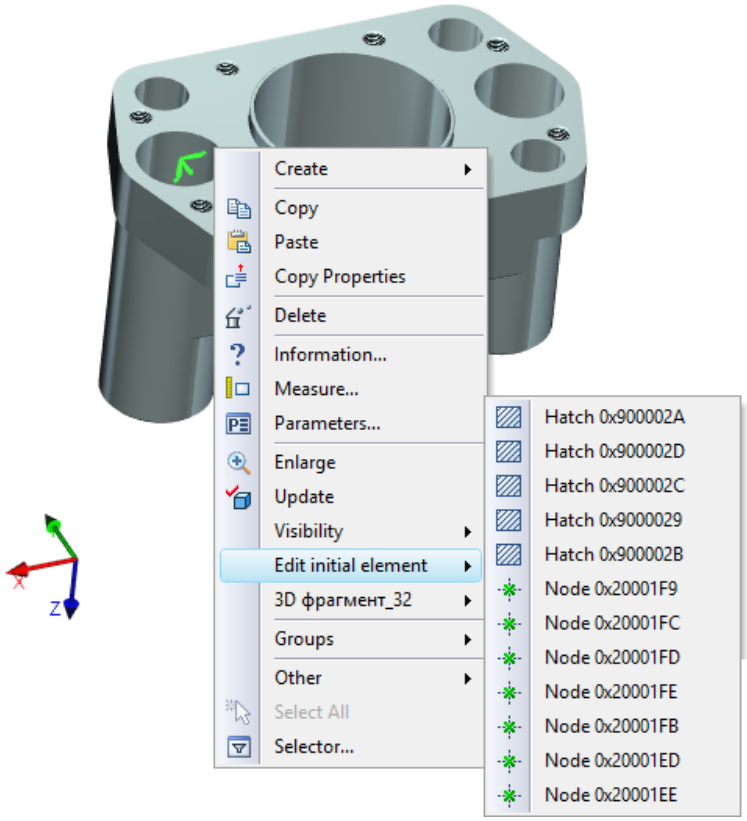
When 2D elements are deleted on a page of a 2D drawing or on the active workplane, the following additional flags are available in the command's dialog:

Delete Unused Named 2D Elements. When this flag is set and the unused 2D constructions are deleted, the named 2D construction elements will be deleted too. In addition, this flag allows deleting the workplanes created on faces, if the projections on those workplanes include named graphic lines.

Current Page Only. When this flag is set, the unused 2D constructions will be deleted only on the current page of the 2D drawing or on the active workplane.

Source Geometry Editing Command

This command allows to quickly select for editing the geometrical elements used for creating the selected 3D element. The list includes the 2D objects (hatches, 2D paths, 2D nodes, etc.), which were used for creating the selected 3D element and its parent elements. If a 3D element, for example, a 3D profile, was created based on a set of graphic lines on a workplane, then the "geometry" list also includes the workplane itself.



When you select a 2D element in the list of the source geometry, the appropriate drawing page opens and the mode of drawing on the workplane activates; at the same time, the editing command for the selected element is launched. Therefore, this command can be conveniently used to quickly go to the page of the source workplane of a given 3D object.

Command **Edit Profile** is available in context menu of operations, created from 3D profiles (for example extrusion and rotation), and workplanes, on which 3D profiles were created. The command activates the workplane on which the source 3D profile is located.

Handling the Drawing Elements while Editing

When modifying a 3D model, one may often have to deal with drawing elements, since those may be used as references for constructing base 3D model elements (3D profiles, 3D paths, 3D nodes, etc.). The user may proceed with the same editing techniques for 2D elements as one uses when working with a

conventional drawing. To modify the shape of a 3D element constructed on the basis of 2D elements, you need to appropriately redefine the source 2D elements. Some of them, for example, hatches directly related with 3D profiles, cannot be deleted and built anew, unless the entire model tree branch is to be deleted together with the children of that hatch. In order to redefine such hatches, one should use the function of editing hatch contours. Otherwise, the model's structural integrity can be broken. If a 3D profile was created based on a set of graphic lines on a workplane, then those lines can be freely modified, including deletion and subsequent new creation. When the session of drawing on the active workplane is over or when the system undergoes the next calculation, the 3D profile will be rebuilt according to the modifications to the graphic lines.

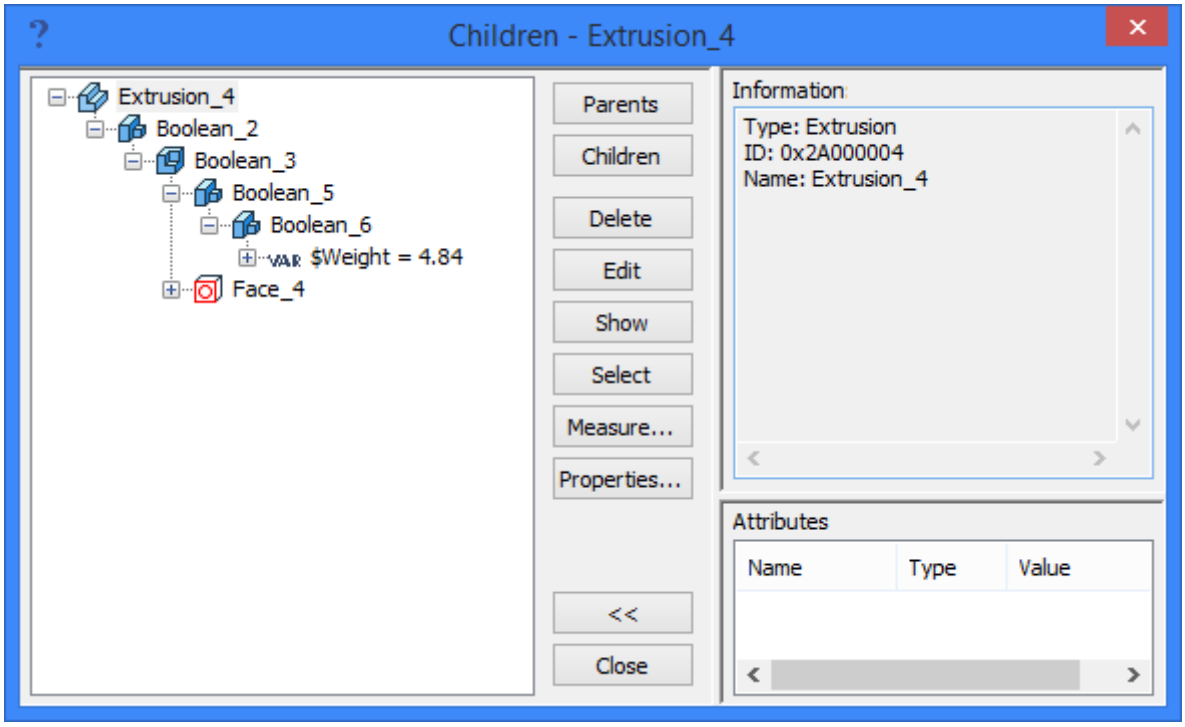
If a certain 2D construction element needs to be modified but cannot be deleted, then you can rebuild it by using another creation method for it in the editing command. The option to rebuild 2D construction elements is available in the automenu of the construction lines editing command.

Sometimes it is helpful to use the command **RL: Replace Element**. It allows to transfer all relations with parents and children from one 2D construction element to another one, and then delete the original element. In this case, the other construction element may use a different construction method and maintain its own relations or dependent elements.

Avoiding Recursion

When creating long chains of element dependencies, the user may run into an attempt of creating a recursive dependency, that is, a dependency of a certain element on itself. In most cases, the system automatically checks the created relations and does not allow saving the changes that cause a recursion. Yet in isolated cases a recursion may occur. Such dependences are seen most of the times in complex chains of relations like "3D elements → 2D elements → 3D → 2D and so on". One danger of the situation for the user is that the one may save the file with a recursion and not be able to undo the erroneous actions when opening this file later. In such a case, it won't be possible to normally work with the file in which the system detects a recursion, up until it is rectified.

In such a situation, the elements shall be edited in a special way. As was already mentioned, upon an attempt to make any changes, the system will run a check for invalid relations in the modified chain of dependent elements. If such invalid relations are found, the system will disallow the changes. This creates certain complications for the user in one's attempts to delete or edit elements.



The user's goal shall be detecting a recursive relation and breaking the chain of dependences at any suitable point, all in one editing step. It is often enough to change the definition method of a certain element in the recursive chain. This could be either a 2D or a 3D element. You can redefine the element by changing its definition method - for example, define its position in absolute coordinates or relate it with some other independent elements.

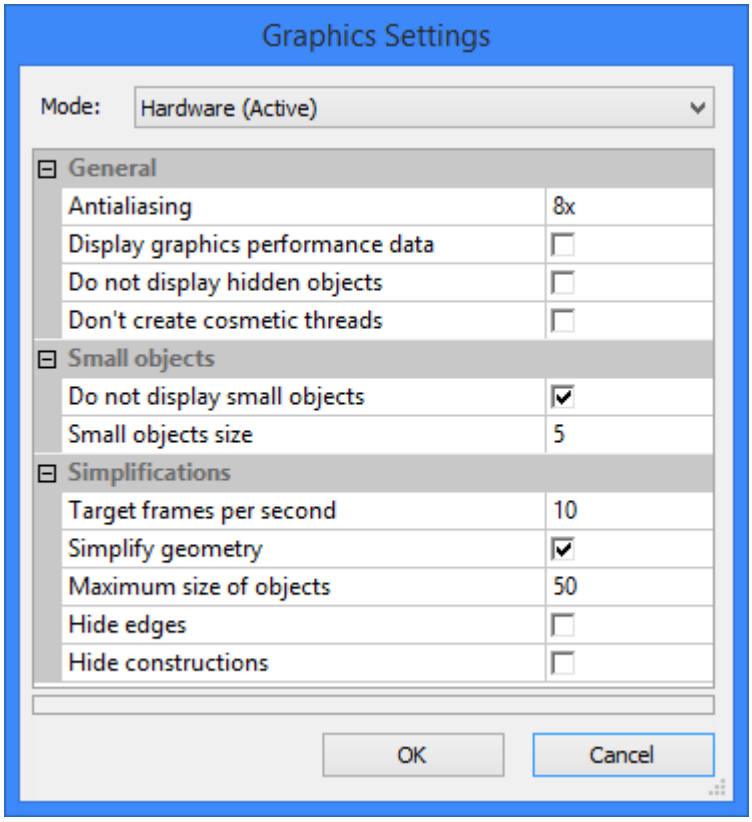
Such elements can be identified with the help of the diagnostics window - the names of the elements are indicated in the recursion warning line. Next, you need to investigate the dependencies between the elements - you can call the “Info” window for that purpose. After identifying the causes of the recursion, one can proceed to editing. Once corrections are made, run the model recalculation to check and make sure there are no new warnings.

Techniques of Simplifying the Model Structure and its Display in the 3D Window

T-FLEX CAD system provides means for speeding up primary work processes - this is achieved by freeing additional computer resources when some secondary processes and model details are temporarily dropped.

1. Display simplification:
   - the use of the wireframe display mode in most cases helps the system faster recalculate the objects in the scene and speeds up the scene spinning and object selection;

- The image quality deterioration affects the display of 3D objects in the scene. The objects appear coarser but are drawn faster. The image quality can be controlled in the **ST: Specify parameters of document command**, on the "3D" tab;
- The 3D model display simplification settings help when spinning the 3D scene densely populated with model elements. Those are system settings that are defined in the command **SO: Set System Options**, on the tab "3D". To call the setup dialog, click the button **[Graphics Settings]**. For details, please refer to the "Customizing System" chapter of the 2D manual;



- Any element in the 3D scene can be made invisible. Reducing the number of elements in the scene helps speed up the display. An element can be hidden by means of changing the level settings, the layer, or by using the **Hide** command in the drop-down "Visibility" submenu of the context menu of the element right-selected with .

2. Element Suppression. By suppressing individual operations, the user not just hides them from the 3D scene, but also excludes them from the regeneration process. This also helps reduce the total model regeneration time. An operation can be suppressed by calling the "Suppress" command from the context menu or by setting the element suppression in the parameters dialog window on the "General" tab.

3. Set of options to assist in handling a large assembly model. These options are provided in the command **ST: Set Document Parameters**, on the tab **Large Assembly Management**. For details, please refer to the "Auxiliary Tools for 3D Assemblies Modeling" chapter of this manual.

ASSIGNING "TRANSFORMATIONS" TO 3D ENTITIES

The command serves for translating, rotating, scaling and mirroring operations and 3D constructions. Transformation parameters are recorded in the properties of 3D elements. Moreover, properties of operation (or the element of 3D construction) can contain simultaneously several transformations that are executed successively one after another. It is also possible to form a group of objects and create common transformations for them.

To define transformation parameters of 3D elements, use the command **"3EG: Transform Element"**. However, a transformation can be specified directly in the parameters of specific elements as well.

Most common examples that involve transformations are assemblies of 3D fragments, the exploding mode and mates.

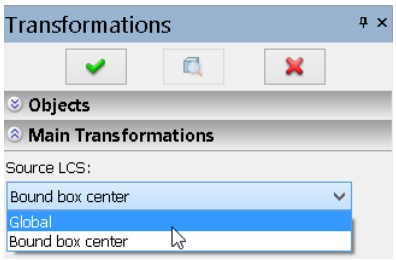
In T-FLEX CAD there are several types of transformations:

Translation. This type of transformation allows us to specify translation with respect to the coordinate axes X,Y,Z or in one of three coordinate planes along two axes simultaneously.

Rotation around axis. This type of transformation allows us to specify rotation with respect to any coordinate axis. Transformation is specified with a value of the rotation angle.

Scaling. This type of transformation allows us to change the size of the object with respect to the coordinate system or the point of scaling. The transformation is specified by the value of the scale.

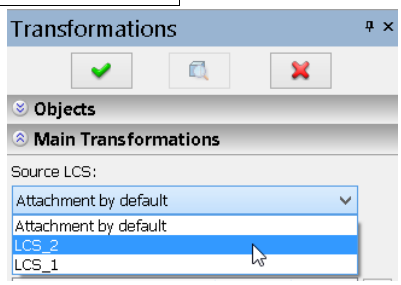
Symmetry. This type of transformation allows us to create the object symmetrically with respect to the selected plane. One of the planes of the coordinate system is used by default.



For transformations of operations and construction elements the Global coordinate system and the coordinate system at the center of parallelepiped encompassing the selected object can serve as an initial coordinate system. Direction of the axes of the coordinate system associated with the parallelepiped coincides with that of the Global coordinate system.

For operations and construction elements the local coordinate system can be selected with respect to which the transformation will be performed.

	<L>	Select LCS for current transformation



For transformations of fragments as an initial coordinate system we can use the default snapping (Global coordinate system), one of the external LCS of the fragment or the fragment's LCS created during the fragment insertion (snapping by fragment).

To select the target coordinate system of the transformation we use the option:

	<T>	Select or create target LCS

In case when it was not possible right away to orient LCS in a desired way, the functions of additional coordinate system specification can be employed. These functions allow us to quickly rotate the constructed LCS around its axes by 90°( ), perform cyclic turning of the axes of the coordinate system around the origin (when pressing the axes change their places).

The target coordinate system can be defined in the process of specification of transformation. To do so, it is possible to use the option of selection of geometric snapping points for the center of the coordinate system of the element:

	<N>	Select point

Directions of the axes in this case will be coincident with the global LCS.

In order to cancel all data that define the coordinate system at the present moment, the option of the automenu can be used:

	<C>	Cancel coordinate system for the current transformation

For selection of the source coordinate system of the transformation we can use the option:

	<S>	Select or create source LCS

After opening the new window at the beginning in addition to the standard options the following options are present in the automenu:

	<1>	Select 3D node or vertex as LCS origin
	<2>	Select point that sets direction of the X-axis
	<3>	Select point that sets direction of the Y-axis
	<C>	Reset target LCS
	<A>	Rotate LCS around the X-axis by 90°
	<O>	Rotate LCS around the Y-axis by 90°
	<Z>	Rotate LCS around the Z-axis by 90°
	<9>	Change LCS axes orientation

Selection of the option returns the user to the assembly and activates the mode of the target LCS specification.

WORKING WITH COMMAND

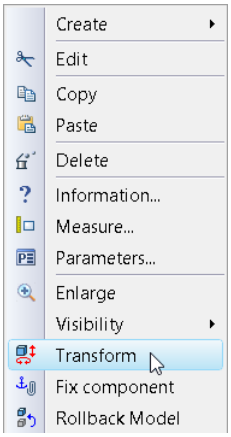
There are several ways to launch the **Transformation** command.

Firstly, the command can be called through the text menu of the program.

To invoke the **3EG: Transformation** command we use:

Icon	Ribbon
	Assembly → Additional → Move/Rotate
Keyboard	Textual Menu
<3EG>	Tools > Move/Rotate

The **Transformation** command can also be invoked through the context menu of object.



After entering into the command, it is required to select 3D element for which the transformation will be specified. To do so, we use the option:

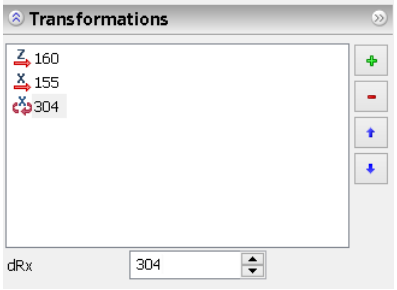
	<1>	Select objects for transformation
	<2>	Select object for individual transformation

Upon selection of it is possible to simultaneously select several objects for transformation. When is selected it is possible to select only one object for transformation.

After selection of operation, construction element or fragment the program enters into the mode of specification of transformation for the selected object.

The properties window changes its view.

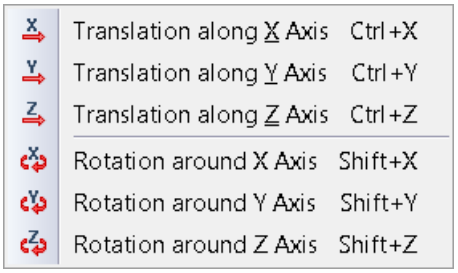
The **Transformations** section of the properties window can be used for control, correction of numeric values, creation and removal of transformations. In the special field the list of all transformations specified for the object at the present moment is displayed. Each transformation is marked with a special symbol that signifies its type (translation or rotation). If the value of transformation is specified with a constant, then the value of the constant is displayed next to the symbol.



Below the main list of transformations there is a field for entry and editing of values.

To input the values of transformation, we can also use the special field on the system toolbar.

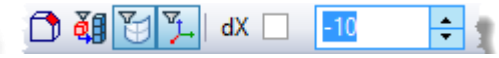
Graphics buttons are located to the right of the list of transformations. The button is used for addition of new transformation. When it is pressed the list of possible transformations appears from which the desired type is offered for selection. The list of possible transformations can also be invoked with the help of the context menu when pressing in the «**Transformations**» window.



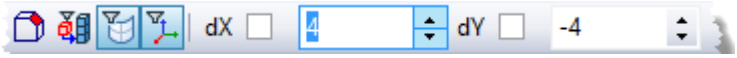
After selection of the type in the list of transformations a new line will appear. In the values entry field the value of transformation can be entered. The transformations' values can be specified both as numbers and with the help of variables and expressions.

To save a transformation, press **[Enter]**. In addition, for specification of the value we can go to 3D window and use the dragger which moves according to the selected transformation type. At the moment of transformation creation its value will be displayed in the special field of the system toolbar:

translation along one axis

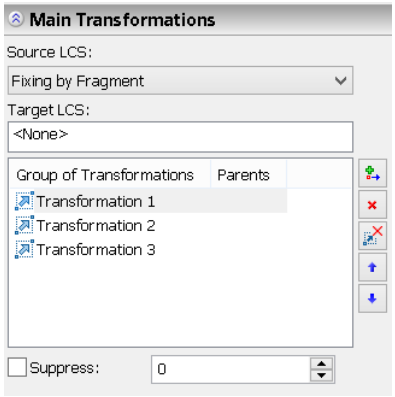


translation along several axes



rotation

In the **Main transformations** section it is possible to create groups of transformations. Each group has its name. Below the list of transformation groups there is the **Suppress** flag. With the help of this parameter the transformation group can be temporarily excluded from the calculation. Next to the flag there is a field for entry of values. It is possible to use numeric values, real variables or expressions as a value for the given parameter. If the value different from 0 is entered into this field, the transformation group becomes suppressed.



Graphics buttons are located to the right of the list of transformation groups. The buttons are accessible only if one object is selected.

With the help of the button it is possible to add a new group of transformations.

When pressing the button the last transformation group in the list is removed.

The button allows us to remove all transformations groups.

When pressing the buttons the order of execution of transformation groups changes.

> Note that when changing the order of execution of transformations (including groups of transformations) the final result changes.

The **3D Assemblies. Creation of 3D models-assemblies** chapter describes the use of transformation groups in more detail.

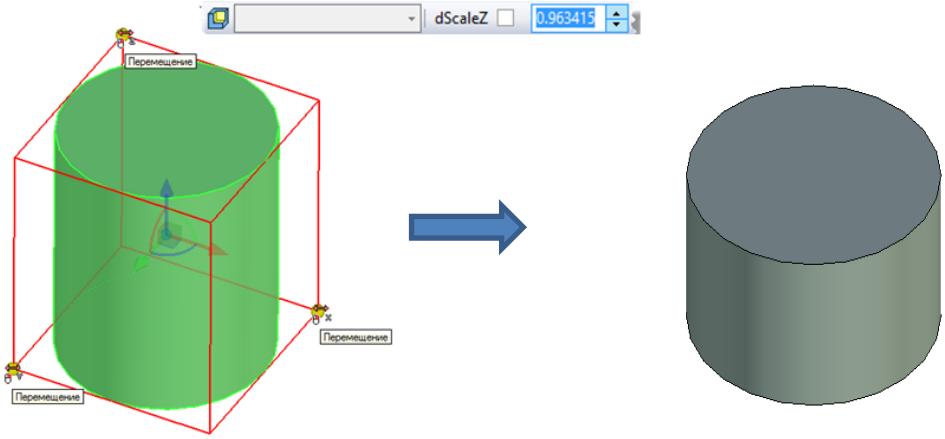
The option of scaling is invoked with the help of the icon on the automenu toolbar:

	<M>	Scaling

Scaling selected objects can be carried out in several ways.

The first method implies the use of the encompassing parallelepiped that bounds the domain around the selected object in 3D window.

When pressing on one of the vertices of encompassing parallelepiped the axis (direction) along which the scaling of the object will be carried out is highlighted. After that the value of the scale can be specified in the special field of the system toolbar.



Several successive transformations of the scale are automatically summed up.

The second method implies the use of the **Scaling** tab in the properties window.

Scaling can be common (all values by all axes will be identical) and with different numeric values along the axes X, Y, Z.

The value of the common scale can be specified with the help of the **Uniform scale** flag

To specify different numeric values for each direction there is the entry field next to the axes.

To cancel the result of scaling we can use the option on the automenu toolbar:

	<J>	Remove scaling

Scaling cannot be suppressed.

To create the symmetry of the selected object it is required to select the symmetry plane. To do so, we can use the option:

	<P>	Select mirror plane

Planar faces, workplanes, 3D profiles and edges can serve as element with respect to which the symmetry will be applied.

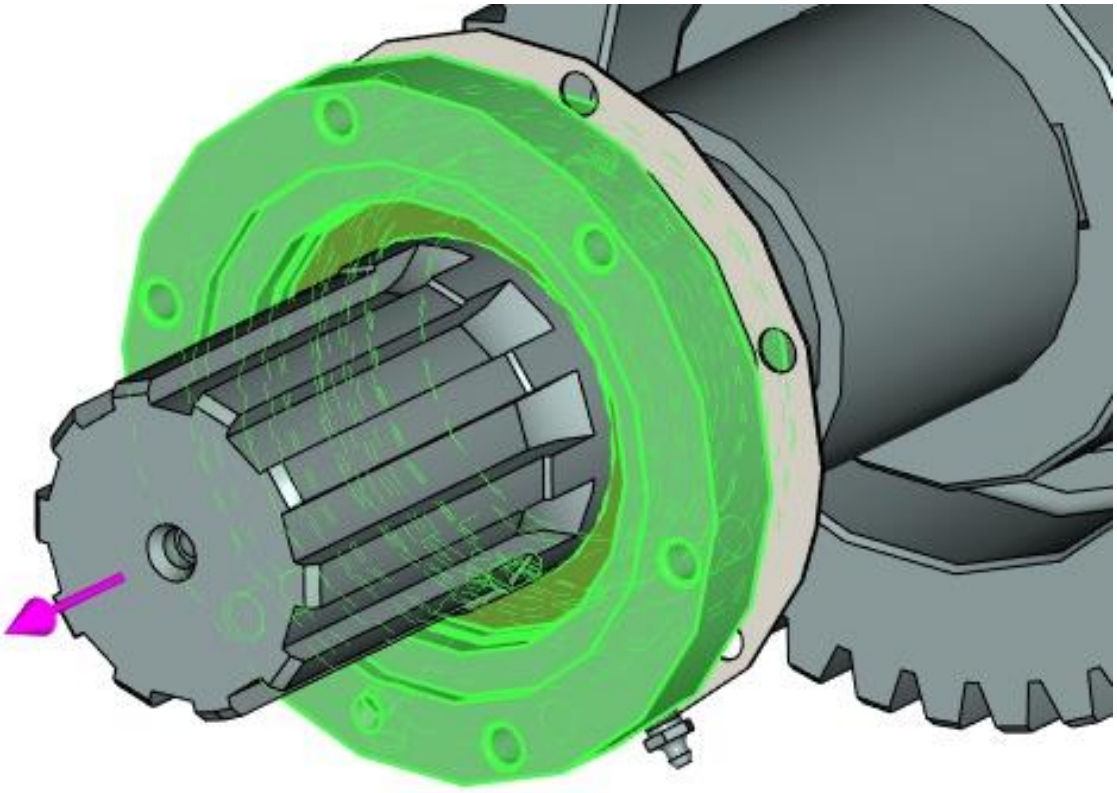
To move element along or around the defined axis, use options:

	<O>	Rotate around arbitrary axis

	<B>	Move along arbitrary axis

Children of the transformed objects can't be selected. This prevents recursion in the model.

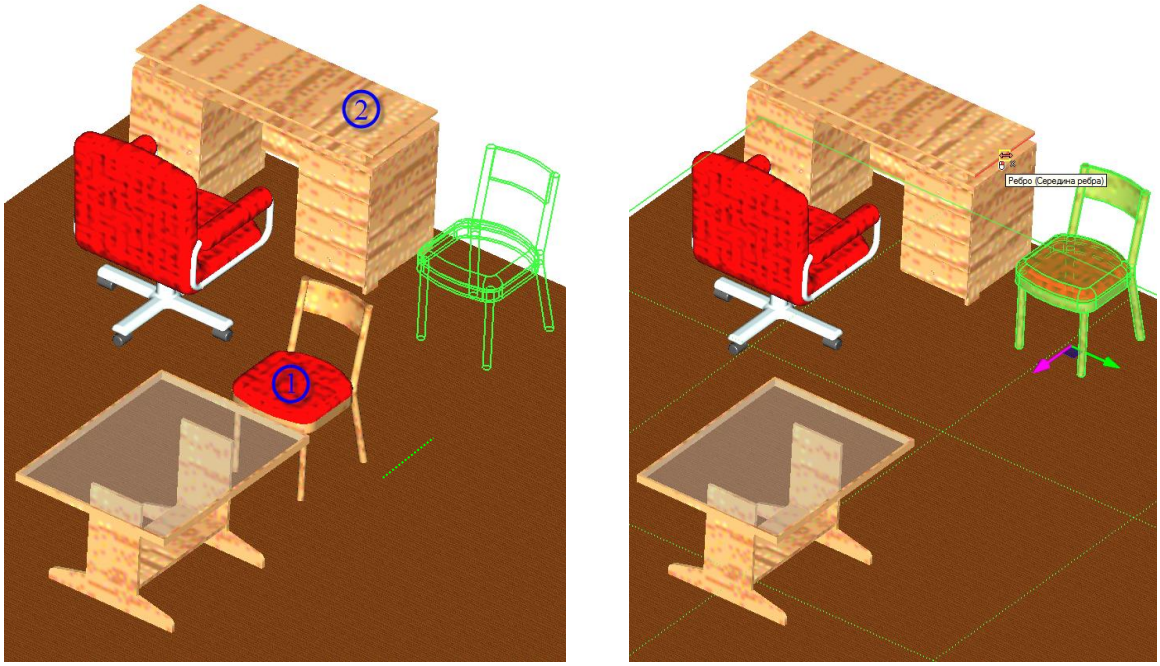
Manipulator that specifies transformation appears after axis selection. You can specify transformation value in special field on the system toolbar.



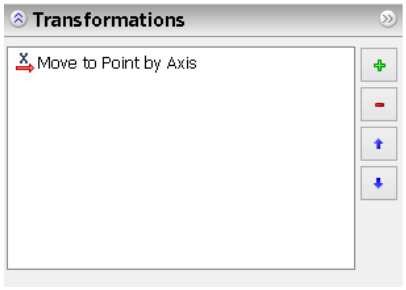
To create transformations it is possible to use geometric attachments. The **Transformations** command works with the following nodes: vertices, edges, faces, 3D paths, profiles, 3D nodes and LCS.

For example, let us suppose that in the existing 3D layout it is required to change location of the **object (1)** by aligning it with **object (2)**.

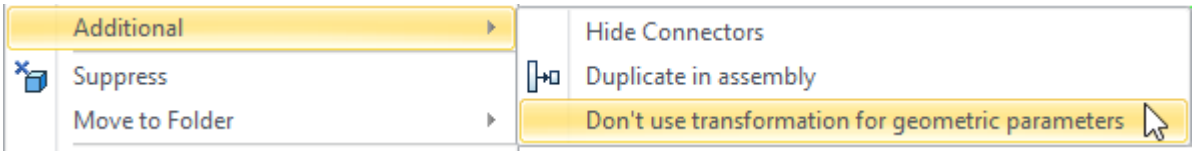
Let us invoke the «**Transformations**» command via the context or text menu. We select the chair, **object (1)** for transformation. After that on the dragger we select the X-axis along which the translation will be carried out. Next, with the help of the cursor on the **object (2)** we select the midpoint of the edge to which the X-coordinate of the chair being moved will be «snapped». When pressing **object (1)** will move to a new location.



In the «**Transformations**» tab a new recording will appear.



If you need to transform an adaptive fragment you should enable the option **Don't use transformation for geometric parameters**.



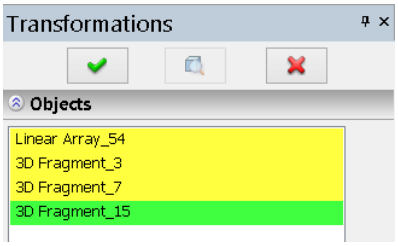
It is recommended to use the option if the geometry of the adaptive fragment does not depend on the adaptive parameters, but changes on their bases.

Group transformations

In the **Transformation** command it is possible to create transformations for several objects simultaneously.

Only the objects independent from each other can participate in the group transformation. Objects are considered independent if they are not connected with each other geometrically (i.e., they are not descendants of each other).

The selected objects are displayed in the «Objects» tab in the properties window.



To make selection of objects for transformation more convenient the command includes the following hot-key combinations:

– selection of object, ungrouping previous groups;

+<Shift> – addition of object to the group;

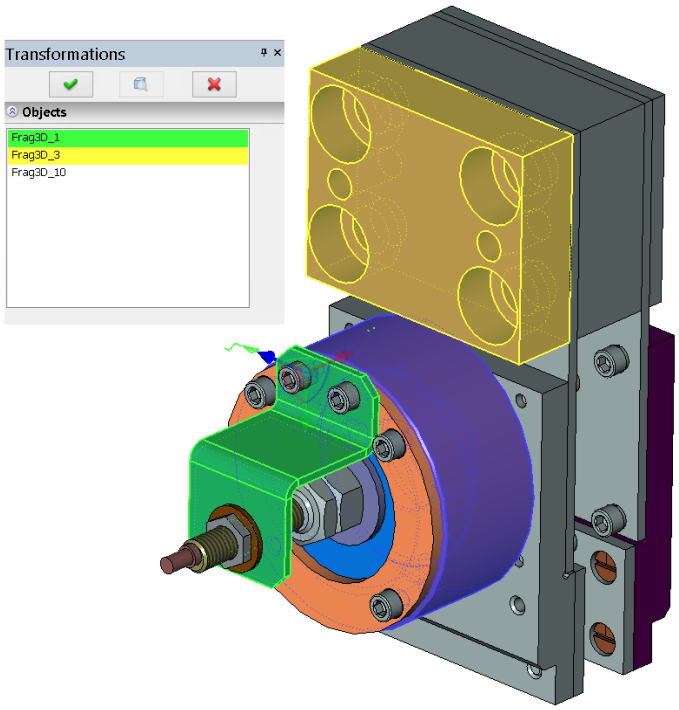
+<Ctrl> – if the object is absent in the group add it to the group; if the object is already in the group delete it from the group.

Group transformation can be performed either in «common» coordinate system or in individual coordinate system of each member of the group.

After selection of objects for transformation the mode of specification of transformations in «common» coordinate system is enabled by default.

	<G>	Mode of setting transformations for selected objects in general coordinate system

For a group transformation on the **Objects** tab and in the 3D window domain the object in the coordinate system of which the transformation is performed is highlighted with green color. All other members of the group are highlighted with yellow color.



If the mode of specification of transformations for selected objects in the «common» coordinate system is disabled the group transformation is transferred to the individual coordinate systems of each member of the group. The Global coordinate system or LCS of the fragment will serve as individual coordinate system for fragments. The Global coordinate system or the coordinate system at the center of the parallelepiped encompassing the selected object will be the individual coordinate system for operations and construction elements.

When selecting one object or a group of objects the command is launched in the mode of not allowing addition of new objects. When destroying the group, each object of the group is assigned the local transformation from the group transformation.

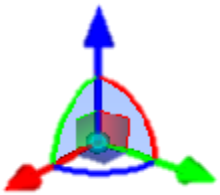
Group transformations can be used in the most efficient way in the mode of disassembly. More detailed information on using group transformations in the disassembly mode can be found in the **3D Assemblies Creation – Additional tools for working with 3D models-assemblies** chapter.

Using draggers

To specify parameters of transformation in 3D window the draggers can be used. A dragger constitutes an image of the coordinate system.

To change parameters of transformation with the help of a dragger, it is required to place the cursor over the dragger and move it to a new place. 3D element will also change its position together with a dragger.

Various parts of dragger are the active elements used for specifying transformations:



- the dragger axes are intended for defining translations;
- the dragger arcs are used for defining rotations;
- images of the planes are designed for translation in the planes **XY, XZ, YZ**;
- the dragger center is used for manual entry into the mode of dynamic snapping.

Upon bringing the cursor to different active parts of the dragger the cursor changes its appearance prompting the user to specify the corresponding types of additional transformations. Next to cursor a tooltip will appear providing information about the type of offered transformation. The selected element of the dragger is highlighted with a different color. When a user points at the axis he can start to specify translation along the selected axis.

Upon pointing at the arc, the system prompts the user to specify rotation in the corresponding plane. In addition, the axis of the dragger closest to the cursor is also selected so that a user could define a rotation by directing the selected axis towards a specific element of the model.

When pointing at the image of the coordinate plane the user can start defining the shift in the selected plane.

When selecting one of the dragger's elements mentioned above the following commands become available in the context menu:

- ✓ for translation along the axis;
- ✓ for translation in the plane;
- ✓ for rotation about three axes;
- ✓ for translation along three axes.

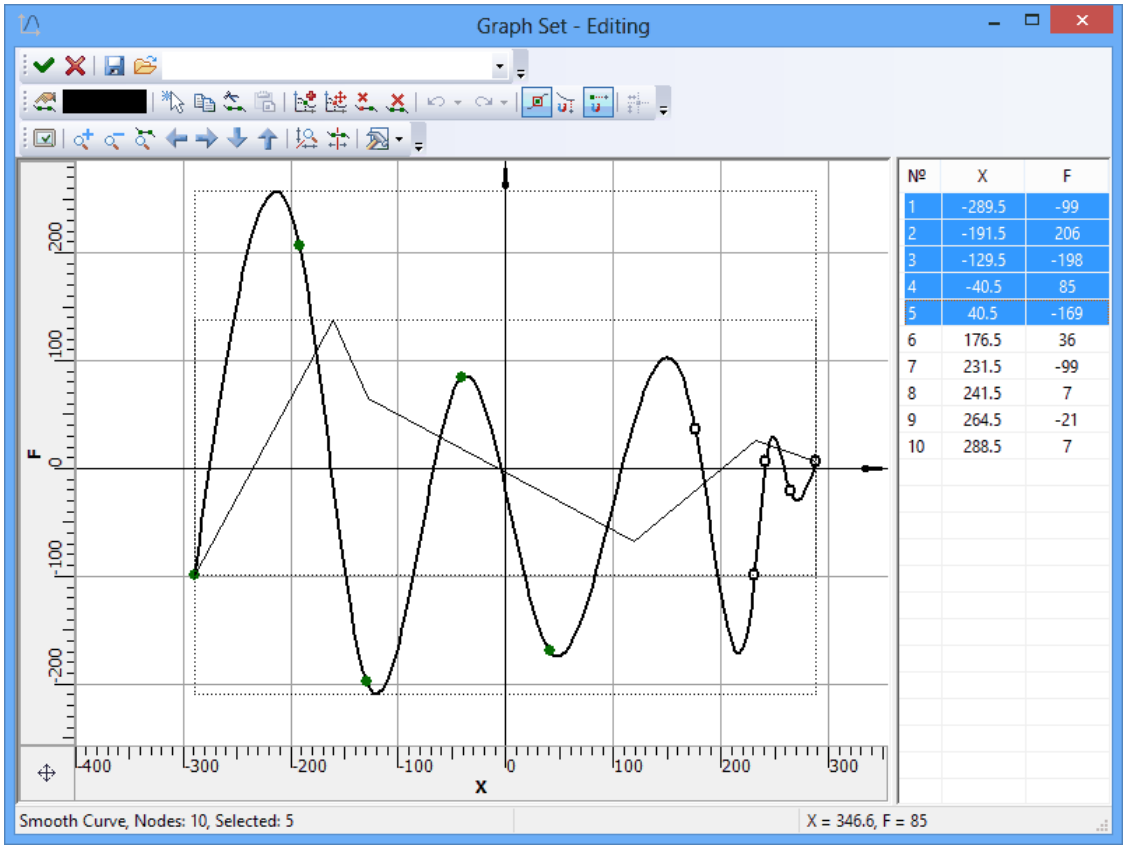
To specify transformation with the help of the dragger the user has to choose an active element of the dragger by pressing the mouse.

> Draggers are described in more detail in the "*3D Assemblies. Creation of 3D models-assemblies*" chapter.

GRAPHS

There is a special object type in the T-FLEX CAD system - the “Graph”. A graph is a function based on a set of points in its own two-dimensional coordinate system, connected by a polyline or by a smooth curve. This versatile instrument serves to define dependencies of different nature, for example, to specify a variable load in Analysis modules, or to create a parametric dependency based on an array of numbers from a database. Graphs are also used to store and display results of a dynamic motion analysis. Graphs can be used in the variables editor to read the values that they define. It is also possible to graphically define the laws of scaling or twisting in the “Sweep” operation, properties of an offset 3D path to define the offset distance, the variable radius value in the “Blend” operation.

Graphs are created and stored in the document together with the drawing and the 3D model. The system provides a specialized editor to create and edit graphs. It serves to manage arrays of points and their coordinates, supports multiple point selection as well as clipboard handling (copy/paste), undo/redo actions, axial zoom control, points dragging etc.



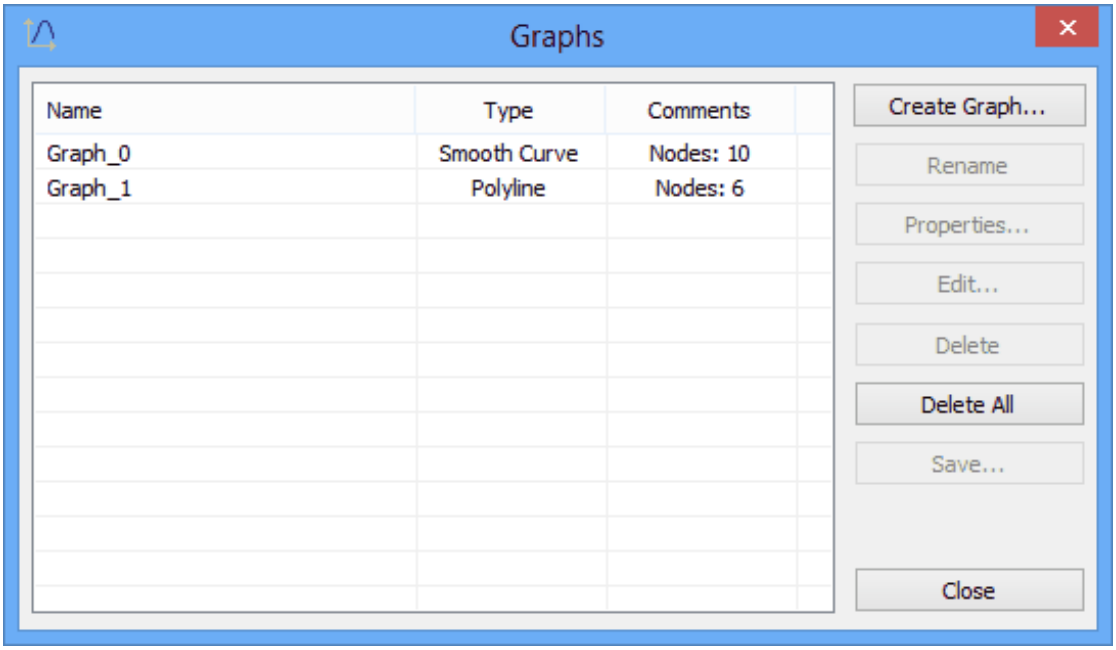
CREATING AND EDITING GRAPHS

Graphs Manager

Working with graphs is done via a special dialog window - the "Graphs" manager, which displays the list of all graphs in the current document, and the buttons to start all required commands. The list of graphs is accessed via the command:

Icon	Ribbon
	Parameters → Tools → Graphs
Keyboard	Textual Menu
<PL>	Parameters > Graphs

After calling the command, the **Graphs** window appears

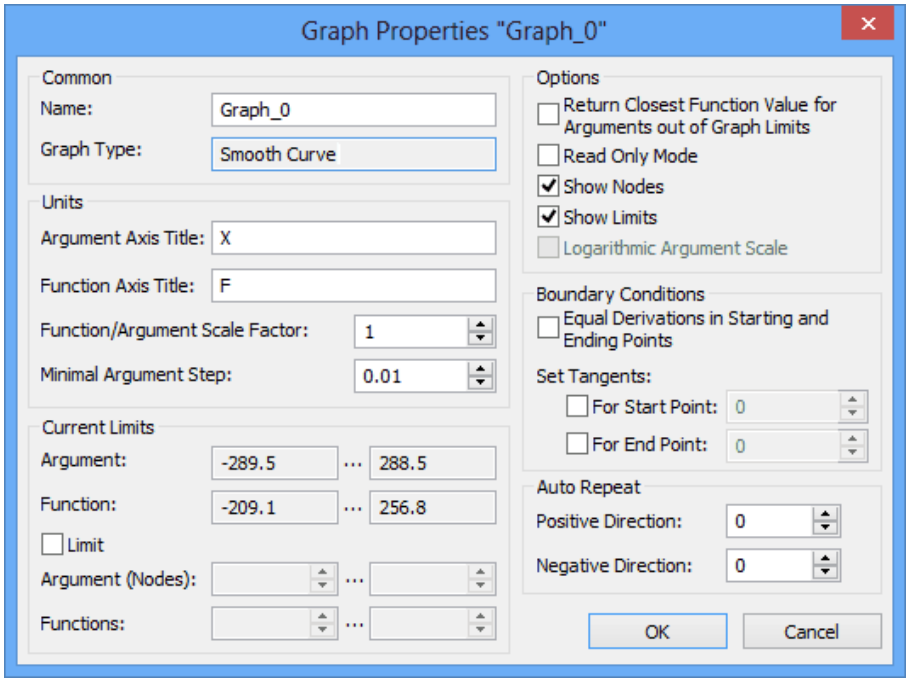


To create a new graph, use the button **[Create Graph...]**. When creating a new graph, you are prompted to select the connection type of the graph nodes: by a spline (a smooth curve) or by a polyline. A newly created graph is empty. It is assigned its own name – "Graph_1", "Graph_2" etc. If desired, the user can modify the graph name using the button **[Rename]** or in its properties (see below).

The buttons **[Delete]** and **[Delete All]** serve to delete existing graphs from the current document.

Graph Properties

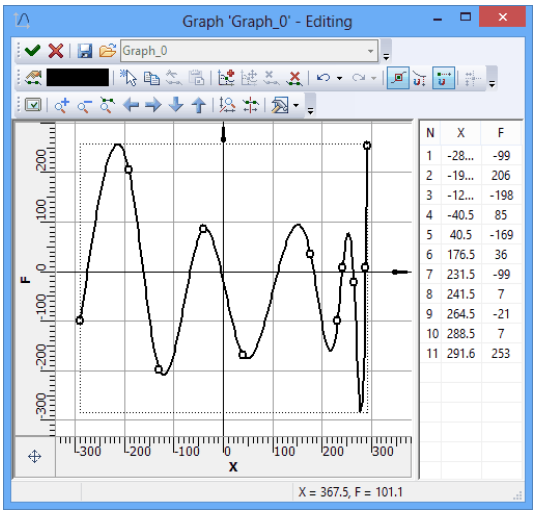
A dialog with descriptive and defining graph properties can be accessed by clicking the **[Properties...]** button in the graphs manager window.

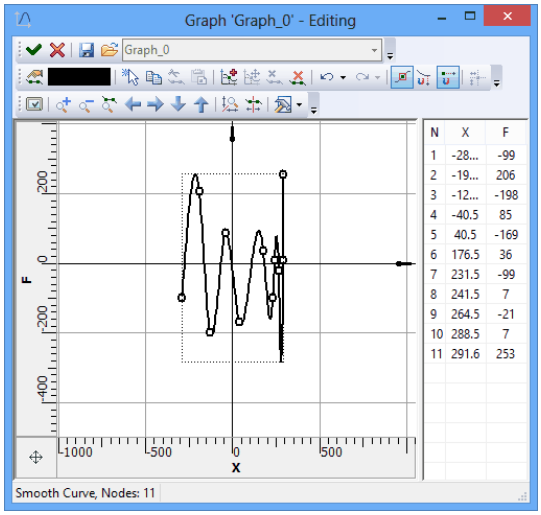


The graph name and type are displayed in the upper part. You can edit the graph name. The graph type is defined at creation and is not changed later.

You can define the following graph properties in the **Units** parameters group:

- The labels for the graph's coordinate axes – in the fields **Argument Axis Title** and **Function Axis Title**;
- The **Function/Argument Scale Factor** parameter defines the units scale along one axis relative to another one when plotting the graph. It is used to setup the image when individual axis scales need to be used. The graph editor can set this value automatically (see below);





- The **Minimal Argument Step** parameter determines the minimum allowed distance between two neighboring nodes of the graph in the X-direction.

The **Current Limits** group of parameters displays the information regarding the current graph bounds. You can also deliberately impose graph limits by the argument and by the function. When limits are enabled, the system will not allow creating points out of the limit bounds. By default, limits are not imposed.

The **Boundary Conditions** group of parameters defines the conditions for the ends of the graph line. Such parameters can be used only for the "smooth curve" graph type. You can define the tangent of the curve direction slope at the graph start and end. You can also set equalities of first and second derivatives at the graph start and end. Such boundary condition allows obtaining a smooth transition between copies when having a master graph segment repeated cyclically.

The **Read Only Mode** flag prohibits modifying properties and coordinates of graph points.

The **Show Nodes** flag enables the display of function nodes on the graph.

The **Show Limits** flag turns on graphic rendering of the range-of-definition bounds and the values range (actual) for the given graph. Graph bounds are displayed as a dotted frame.

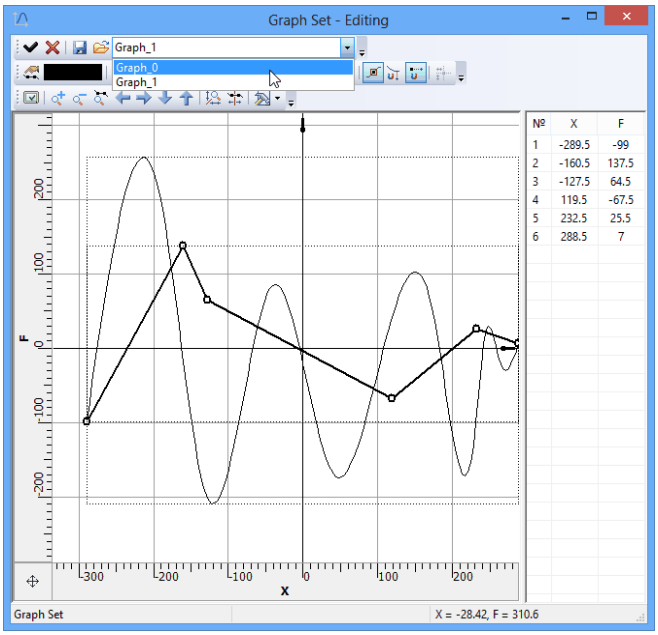
The **Return Closest Fuctions Value for Arguments out of Graph Limits** flag serves to use the graph for any argument values, including those outside the existing defining range. This parameter is used to work with the function `graph("GRAPH NAME", ARGUMENT VALUE)` of the variables editor.

The flag **Logarithmic Argument Scale** sets the logarithmic scale along the argument axis of the graph.

The **Auto Repeat** group of parameters defines a cyclic repeating of a master graph segment in both directions. When repeating in the positive direction, then each copy of the master segment is attached by its start point at the end point of the graph, and vice versa when in the negative direction.

Graph Editor

Graphs are edited in a special graph editor. The editor is called by selecting the desired graph in the "Graphs" window and clicking the **[Edit...]** button, or by double-clicking on the graph's row. To simultaneously edit several graphs, select them into graphs manager using <Shift> and use the **[Edit...]** button as well. All graphs will be simultaneously displayed in the editor. At the same time, one of them will be active and editable. To switch between graphs, use the interface control (the drop-down list) with graph names.



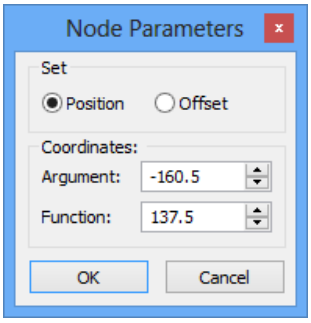
The main portion of the editor window is occupied by the **workspace**, in which graphs are displayed. The workspace is ruled for convenience with an automatically scalable coordinate **grid**. Image moving (panning) and scaling (zooming) is done by the mouse wheel – in the same way as when drawing in T-FLEX CAD.

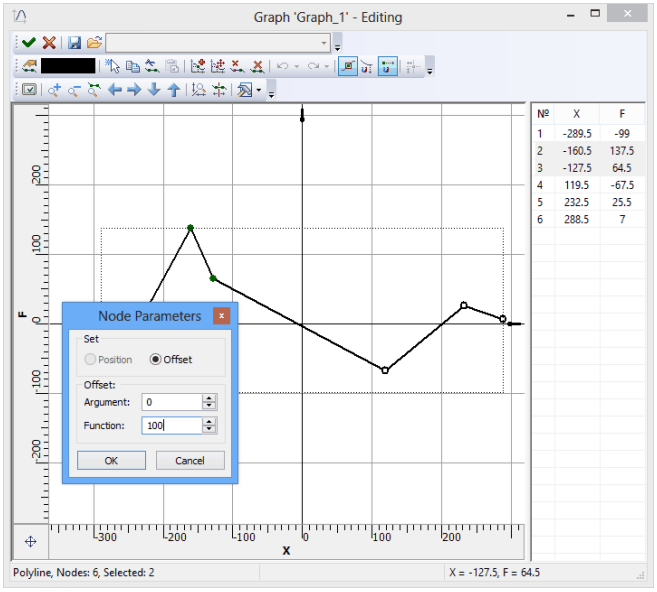
The coordinate **rulers** are displayed along the borders of the workspace. You can use these rulers in the same way as in the T-FLEX CAD drawing window – to pan and zoom the image. The graph editor allows separate zooming along the coordinate axes. Separate zooming is done using a ruler, when the appropriate mode is enabled in the graph properties, or by the icon on the editor's "View" toolbar. A specified ratio of coordinate axes units is automatically stored in the graph properties.

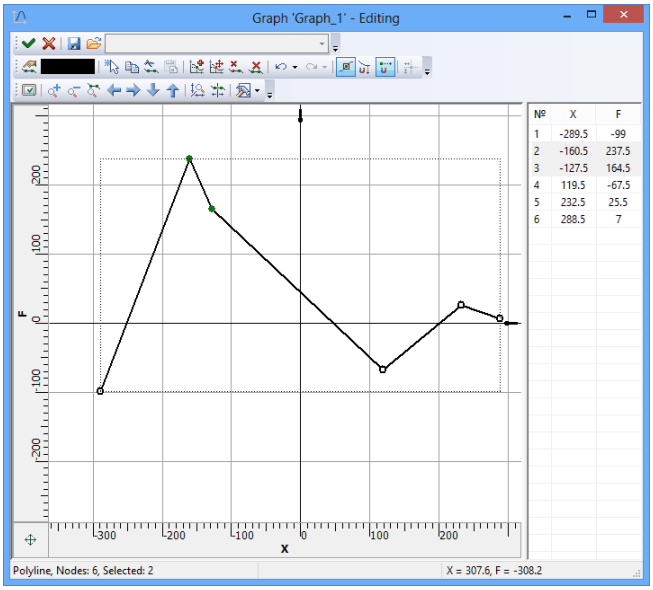
Clicking in the workspace creates a new graph point. All points are connected by a line (curve) of the specified type (a polyline or a spline). For user convenience, the current cursor position coordinates are displayed in the status bar in the lower-right corner of the graph editor.

At the right of the workspace there is the table of graph point coordinates. Point coordinates can be edited directly in this table. You can start editing coordinates after double-clicking in the desired table cell. To input changes, press <**Enter**> or switch the input focus to another window. The edited (selected) graph point is highlighted. To select several points, use the keys <**Shift**> or <**Ctrl**>.

A graph point position can be modified with the cursor, by "grabbing" the point with and dragging it to the new position. When double-clicking on a graph point, a special dialog appears to edit point coordinates. Here you can enter the absolute point coordinates (when switched to **position**) or set offset coordinates with respect to the current point position (when switched to **offset**). If, when double-clicking , several points were preselected (for example, in the table of coordinates), then only the offsets can be used. This is convenient, if you need to "move" a segment of the graph by a certain distance (see an example below).







Graph Editing Tools

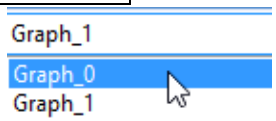
The graph editor has three bars with various tools and icons to launch utility commands.

The "**Standard**" toolbar contains:

- Commands to close the graph editor with or without saving changes:

	–	Save changes and close
	–	Cancel changes and close

- A drop-down list to select the name of the graph being edited is used when several graphs are edited simultaneously.



- The command to save the edited graph in the external file:

	<Ctrl><S>	Save to File

- The **"Download from file"** command that allows us to replace the graph being edited with another graph from the external file "*.tflaw". As a result of execution of this command all source points are removed, and the basic properties are replaced with other properties:

	<Ctrl><L>	Download from file

The "**Edit**" toolbar contains:

- An icon to call the edited graph properties dialog:

	-	Graph Properties

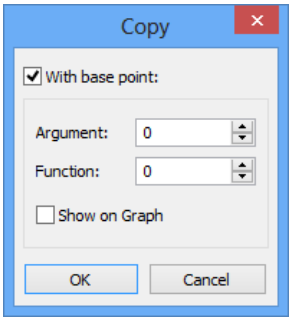
- A drop-down list (combo box) to define the colors of the graph line:

	-	Graph Color

- Commands to work with the clipboard:

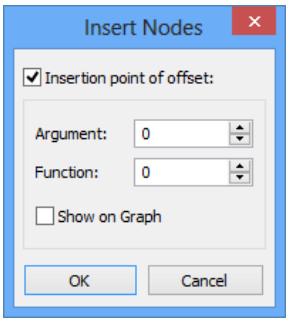
	<Ctrl><A>	Select all nodes
	<Ctrl><C>	Copy
	<Ctrl><X>	Cut
	<Ctrl><V>	Paste

To copy graph points to the clipboard, you first need to select points in the table of coordinates or in the workspace. Next, you can use the command "Copy" or "Cut". When copying graph points, the system must remember the reference point for future snapping when pasting from the clipboard.



To define the point, the **Copy...** dialog window appears. The user can define the coordinates of the reference point or select it on the graph. To select a point in the workspace, you need to enable the option **Show on Graph** and click **[OK]**. Next, the system goes into the mode of waiting for point selection. The respective help message appears at the bottom of the graph editor in the status bar line. The user can point at any location in the workspace or select any of the existing graph points. If the user did not specify a reference point, the system will use the origin instead.

When adding the copied points from the clipboard to the graph, the system will prompt for defining the target point to snap the insertion to. The **Insert Nodes** dialog window will appear. In it you can also define the coordinates of the insertion target point or enable the mode of selecting such point in the workspace. If the user did not define the reference point at the time of copying, then, when defining the coordinates for the insertion, the user will thus define the offsets of the new graph points from the original position. When specifying the target point, the preview image of new graph points rubberbands with the cursor.



- Commands to work with graph nodes:

	<Ctrl><N>	Add new Node by Coordinates
	<Ctrl><E>	Edit Node Coordinates
	<Del>	Delete Selected Nodes
	<Ctrl><Shift><Del>	Delete All Nodes

- Commands to undo unwanted actions and redo undone actions:

	<Alt><BackSpace>	Undo
	<Ctrl><BackSpace>	Redo

- Options to enable the modes of selecting nodes and object snapping:

	-	Mark / Select nodes
	-	Snap to Nearest Point
	-	Snap to Nearest Node
	<Ctrl>	Move Nodes by Coordinate Axes

The "**View**" toolbar contains:

- The command to refresh the image:

	<F7>	Update Window

- Commands to zoom in and out the image:

	<Ctrl><Shift><PgUp>	Zoom In
	<Ctrl><Shift><PgDown>	Zoom Out

- The command to automatically fit the entire image within the workspace window bounds:

	<Ctrl><Shift><End>	Zoom All

- Commands to pan (move) the image:

	<Ctrl><Shift><Left>	Left
	<Ctrl><Shift><Right>	Right
	<Ctrl><Shift><Down>	Down
	<Ctrl><Shift><Up>	Up

- The button to enable the mode of separate zooming by axes:

	-	Zoom separately by axes

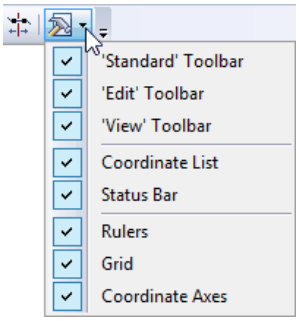
- The command to enable/disable the mode when a special vertical marker is displayed that reads the argument position and returns the respective function value. This command is duplicated with a special switch flag in the graph editor options (see below):

	-	Argument Marker

- An icon to call the graph editor options dialog (see options description below):

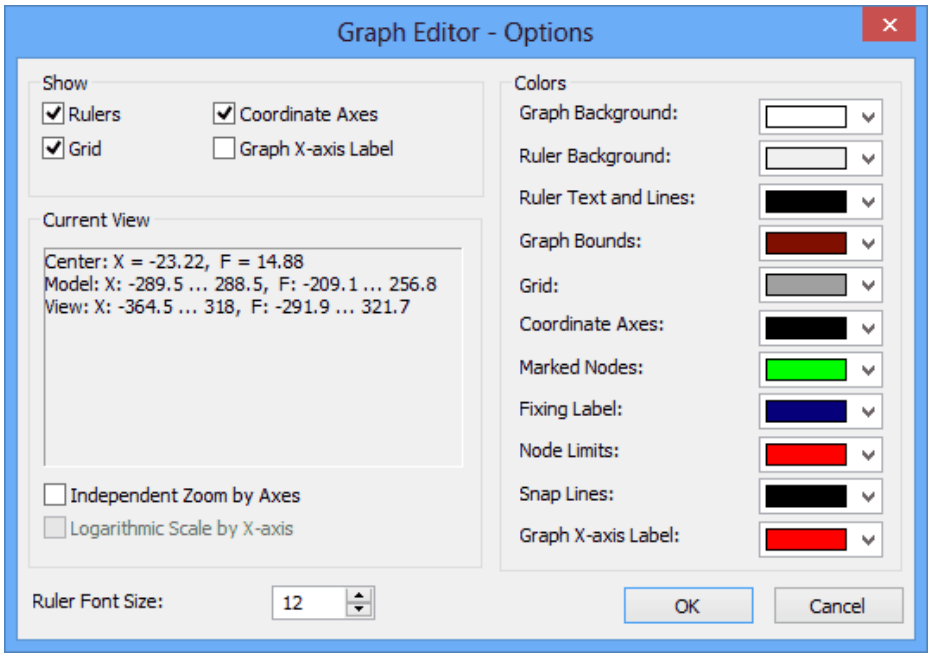
	-	Options

The drop down list, which pops up upon pressing the symbol ▾ to the right of the button, allows the user to customize the view of the graph editor window (display of the toolbars, rulers, grid, etc.).



Setting Up the Graph Editor

The dialog contains interface controls to set up the graph editor appearance.



The **Show** group of options contains the switches to enable/disable auxiliary utility objects – the ruler, the automatically scalable coordinate grid, coordinate axes, the argument marker.

The pane in the **Current View** group of options displays the coordinates of the displayed area for the user reference.

The flag **Independent Zoom by Axes** controls the mode of independent scaling along the axes. If it is turned on, then upon scaling with the help of the ruler the visual scale factor along the X-axis changes independently of the scale along the Y-axis.

The flag **Logarithmic Scale by Argument Axes** is a reference flag and is always unavailable for editing. This flag shows what argument scale is set in the properties of the graph being edited.

In the right part of the dialog there is a group of controls to set up the colors of various objects in the graph editor.

With the help of the parameter **Ruler font size** you can also define the font size for the graph editor ruler.

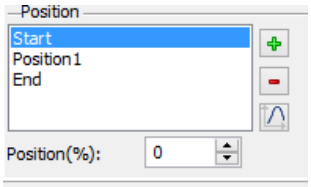
USING GRAPHS

As was mentioned in the beginning of the chapter, graphs can be used in various areas of the system. In this topic we will review specific examples of using graphs.

Using Graphs in 3D Operations

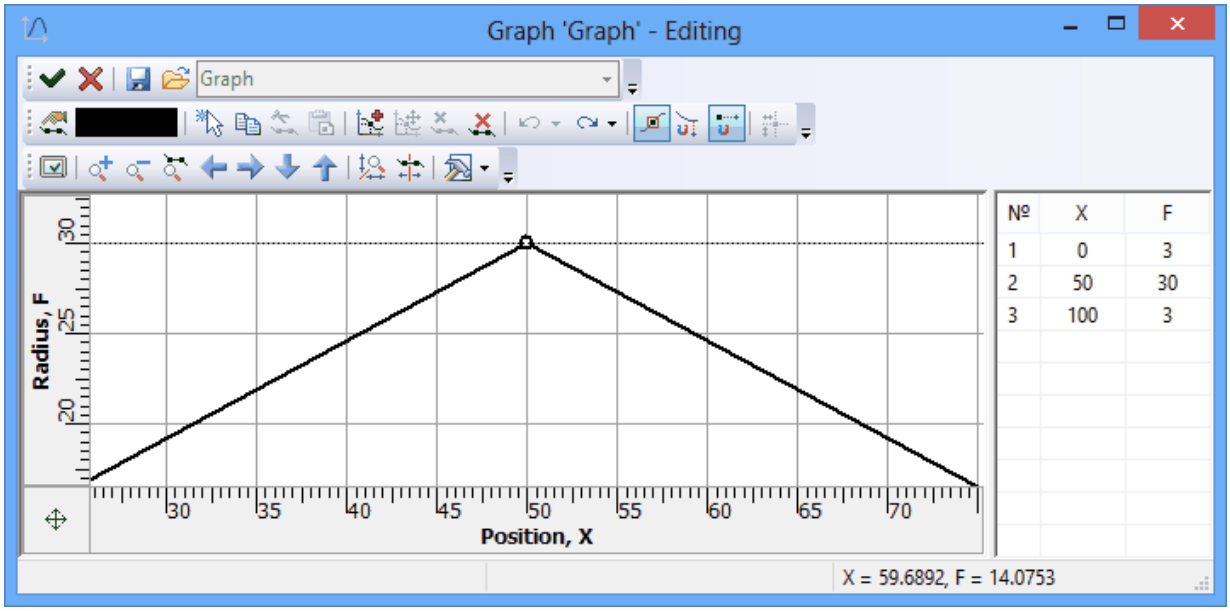
Graphs can be used for a more intuitive way of defining tabular dependencies in some 3D modeling commands. For example, in the “Blend Edge” command – to define the law of changing a variable radius, or in the “Sweep” operation – to define the law of changing the twist angle or scaling of the contour. It

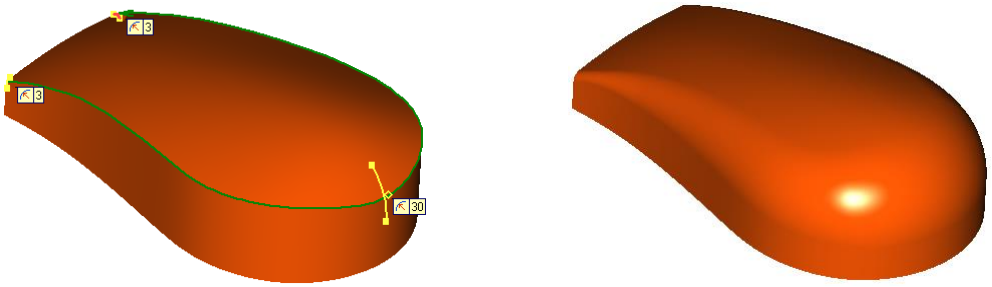
can be used wherever you need to define the law of changing some operation's parameter over a certain range, whose length is defined in percent (0%-100%).



To define a dependency by a graph, use the special "Graph" button in the operation properties window, next to the field for defining the entries of tabular dependencies. Upon clicking the button, the graph editor opens, in which there is already a graph constructed based on the input data. (By default, those are the point positions 0% and 100% with the respective parameters.) The graph type in this case is always a smooth curve.

Depending on the operation type, we can distribute various properties using a graph. The definition range and the graph values range are also limited depending on the operation type. The range of values allowed for input in the X-axis is always from 0 to 100. This defies the position of each point. In the Y-axis we reckon the operation parameter values for each point. For example, for a variable round blend, one can define the law of the radius distribution along the span of the selected edges. In this case, the system will not allow setting the function value less than zero, since this is restricted in the blend operation.





To save the results of editing a graph, exit the editor by clicking the button . After closing the editor, the system will transfer all graph points to the properties window to display the dependencies in the tabular form.

Using Graphs in the Motion Analysis, in the Variables Editor

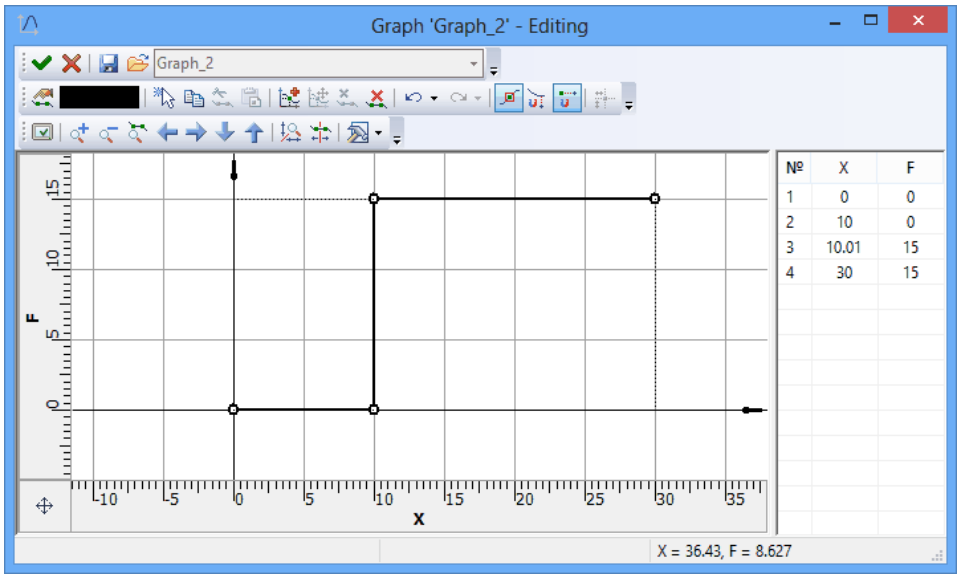
Graphs are used on several occasions in the «Dynamic motion" module:

1. To display the results of a dynamic calculation – to record the changing in time coordinates, forces, moments etc.
2. To define variables for loads, springs.

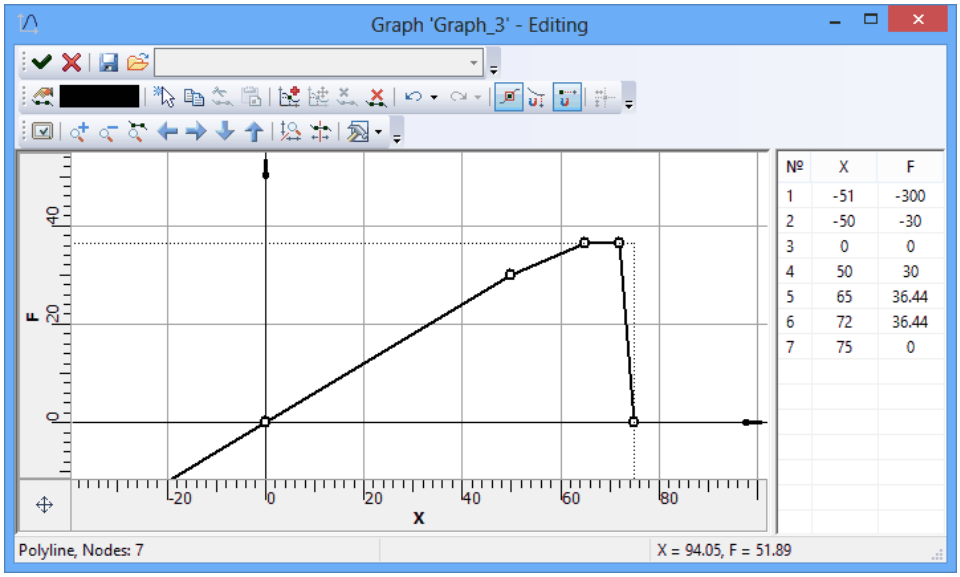
A prepared graph can be used in the variables editor for further calculations, by reading its function value in the variables editor using the expression `graph("graph name", argument value)`.

> Note that this chapter brings only the examples of using a graph in the dynamic motion calculations. For detailed information on the dynamic motion analysis refer to the respective chapter of the documentation.

As an example of a varying load, which can be defined using a graph, consider a motor (the "Torque" load), which turns on and off at a certain time instant. The graph for such load may look as follows:



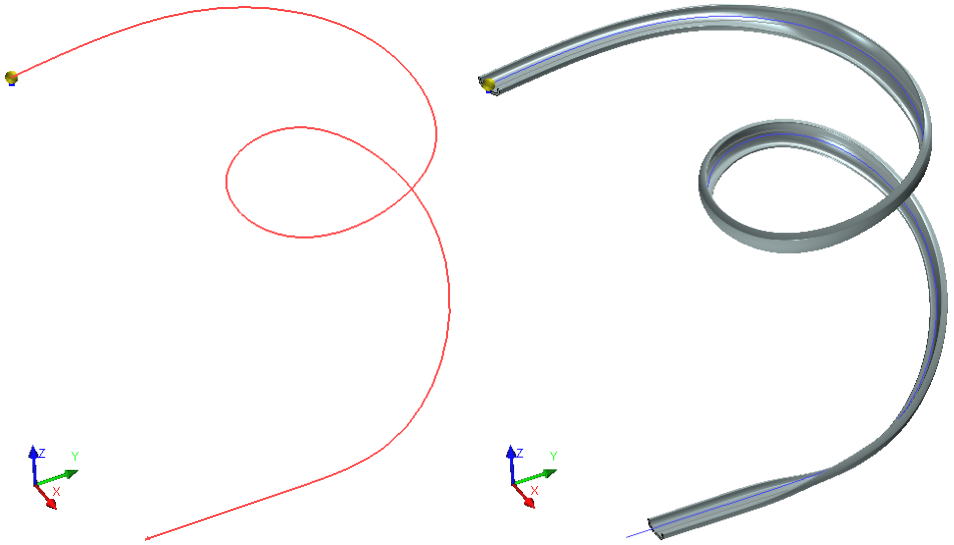
Sometimes, one may need to define nonlinear properties in spring, for example, to account for an imperfect spring. At the extreme compression, when its coils contact each other, the rigidity increases manifold. One can also account for the spring destruction instant at a certain deformation level value (the spring elongation). At that instant, the spring resistance disappears. We shall clarify here, that the "destruction" of such spring in the model will be reversible, that is, when the model returns to a state with the spring elongation within the normal range, then the spring will automatically restore. The graph for such spring will define the dependency of the bipolar force on elongation and may look as follows:



Let's review the use of graphs to record results of a motion analysis and using the function `graph("GRAPH NAME", ARGUMENT VALUE)` on an example of solving a typical problem of finding and maintaining an equilibrium of a moving body. For the example, we will create a simplified model of the "Waterslide" attraction.

Problem conditions. There is a spatial descending and trajectory (a 3D path). A body shall move strictly along this path under the gravity force. The problem is formulated as to design such a surface, along which the body could slide and follow the prescribed path. To simplify the example, we will use a ball as the moving body and neglect the friction force. Let's also skip some details of the design in the example description, which are not related to working with graphs.

The file with this example is located in library *"Examples/3D Modeling/Graph/Waterslide.grb"*.



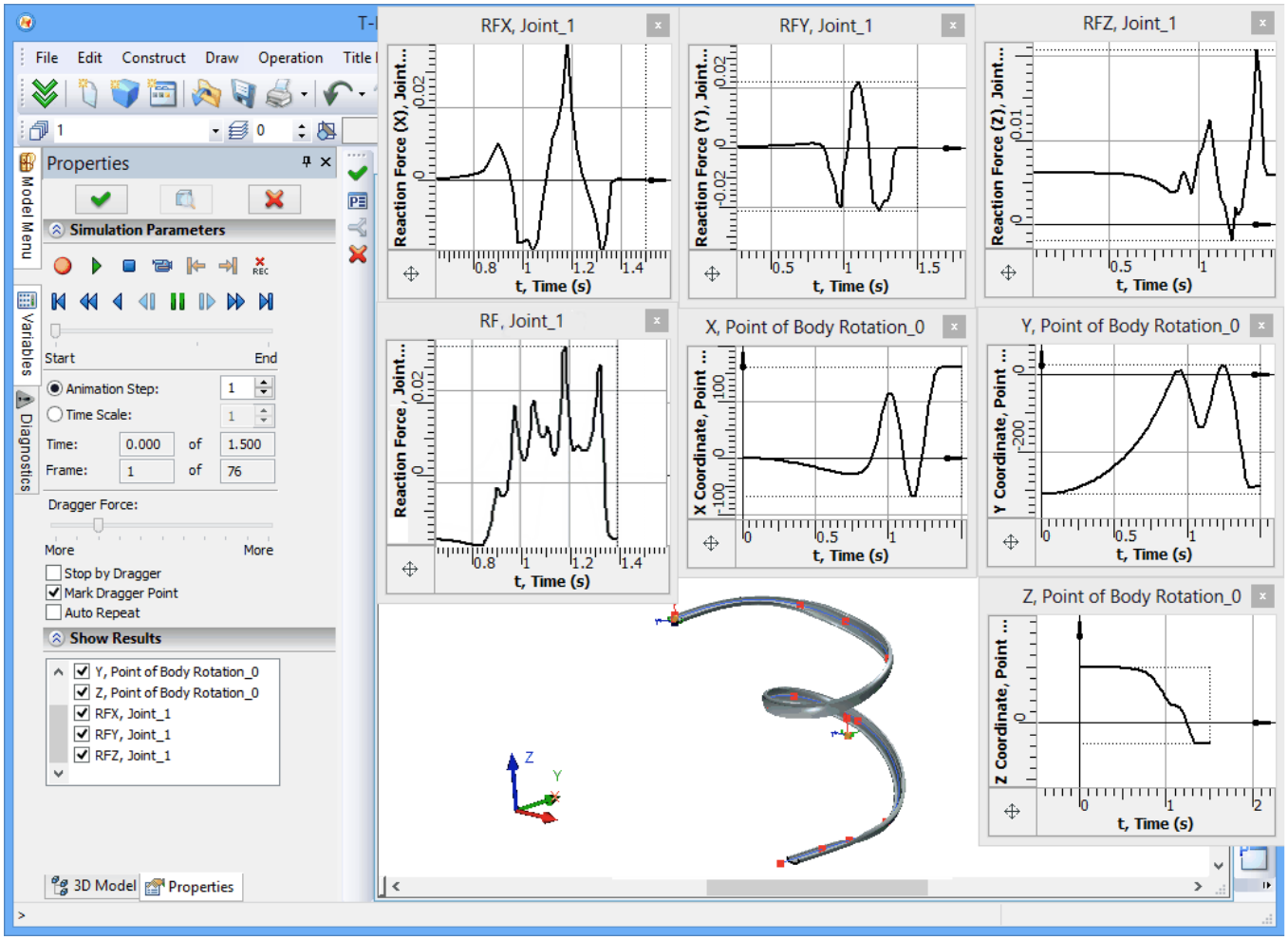
To solve this problem, you need to have the dynamic motion module installed.

The solution to this problem is obtained in several steps:

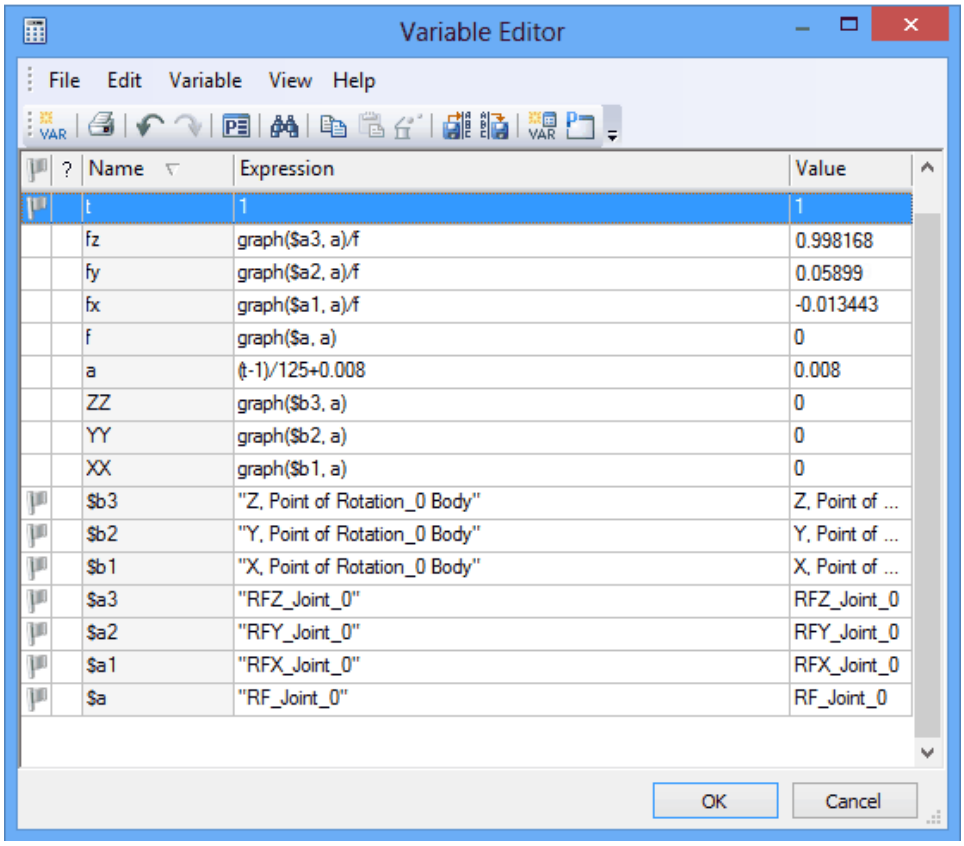
- ✓ The ball shall be made moving under the gravity force along the specified path (as if on a rail). At the same time, it is necessary to remember the extra G-forces acting on the ball. We will create the first motion analysis study for this purpose. We can collect all necessary data in the way of graphs.
- ✓ Adjusting the results of the motion analysis, creating variables, reading graphs.
- ✓ Constructing the ball path.
- ✓ Constructing the surface.
- ✓ Verifying the result. Creating the second motion analysis study.

Step 1. To create a dynamic motion study, we will need to create one "Coincident" mate, which will bind the ball (the center of gravity) to the trajectory (the 3D path). To read the results, we will need two sensors. To record the changing in time reaction force (the extra G's) we will need a joint sensor (joins are created automatically based on mates). To record the coordinates of the ball position, we will need a "point" sensor (at the ball's center of gravity). As the result of the motion analysis, we will record in the way of graphs the following values:

- the graph "X, Point of Rotation_0 Body" - records the X-coordinate of the ball position X;
- the graph "Y, Point of Rotation_0 Body" - records the Y-coordinate of the ball position;
- the graph "Z, Point of Rotation_0 Body" - records the Z-coordinate of the ball position;
- the graph "RF_Joint_0" - records the magnitude (vector length) of the reaction force in the joint_0;
- the graph "RFX_Joint_0" - records the X-component of the reaction force in the joint_0;
- the graph "RFY_Joint_0" - records the Y-component of the reaction force in the joint_0;
- the graph "RFZ_Joint_0" - records the Z-component of the reaction force in the joint_0.



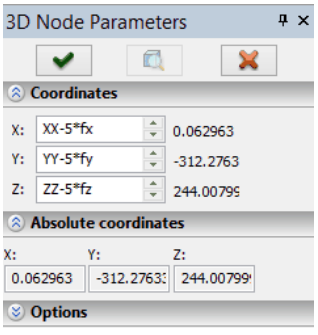
Step 2. Create several variables to read the values from graphs in the variables editor. The function `graph()` will be used in the expressions. The {a} variable will be used as the time counter. All events of interest occur in the time period from 0.008 to 1.4 seconds with the step of 1/125. The variables are recorded as shown on the following figure. The function `graph()` can read values from graphs using the time counter {a}.



Step 3. By using the variables {XX}, {YY}, {ZZ} we can re-create the ball's center of gravity position at any time instant by varying the {a} variable.

> The tools that we use (a 3D path with the parametrically changing 3D node, a parametric sweep), can only use an integer counter and vary it with the step equal to 1. Therefore, the variable {a} will be varied with the help of another counter {t}.

The variables {fx}, {fy}, {fz} will be assigned the coordinates of the reaction force unit vector, therefore those can be used to define the direction of the extra G-force action. Let's create a 3D node and enter these variables instead of the coordinates. Additionally, we need to account for the ball radius, therefore we introduce a correction to the coordinates as 5*(fx,fy,fz). This node will be changing its position along with the varying variable {t}



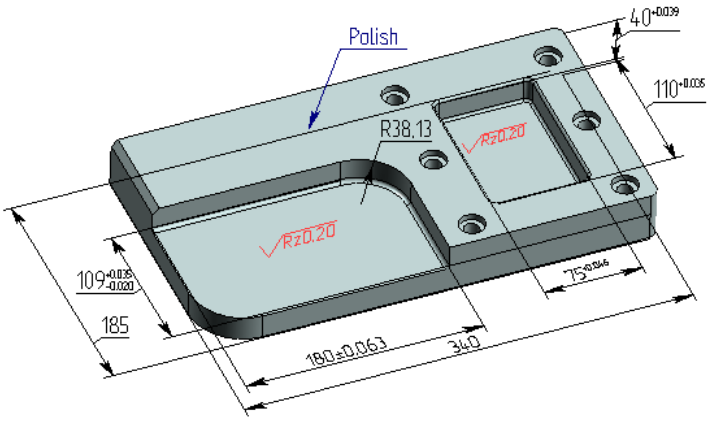
Next, based on the new 3D node, construct a parametric 3D path by varying the counter {t} in the range from 1 to 175 (at the same time, the time counter {a} will sequentially read all values from graphs). This new path (3D path_1) renders the line of contact between the ball and the surface which we need to construct. It will also be needed to correctly position the “movable” profile for the prospective surface.

Step 4. To construct the surface using the "By Parameters" operation, we need a "movable" profile. We can bind it to our "movable" 3D node by creating in it a local coordinate system (LCS). To define the direction of one of the LCS axes, we will create an offset 3D node in the force vector direction (by defining the coordinates (fx,fy,fz) relative to the source 3D node). The second LCS axis will be set tangent to the 3D path_2. By having a "movable" profile, we can create a parametric sweep by varying the {t}-counter operation parameter in the range from 1 to 175.

Step 5. Result verification. Create a new motion analysis study and release the ball on the created surface; it will exactly follow the claimed path, meaning that the created surface fully satisfies the problem requirements.

3D Annotations

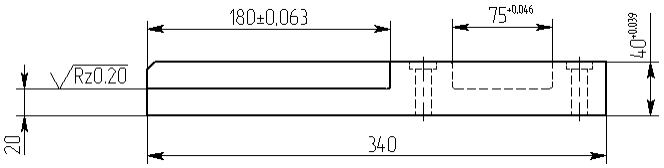
T-FLEX CAD allows creating drawing detailing elements (dimensions, leader notes, roughnesses) directly on faces of a 3D model.



General Information

The capability of creating 3D annotations allows instantly introducing in the 3D model not only geometrical, but also technological process and other information that can later be used when creating drawings by projections, as well as in other applications, for example, in the process design modules or when creating CNC control sequences.

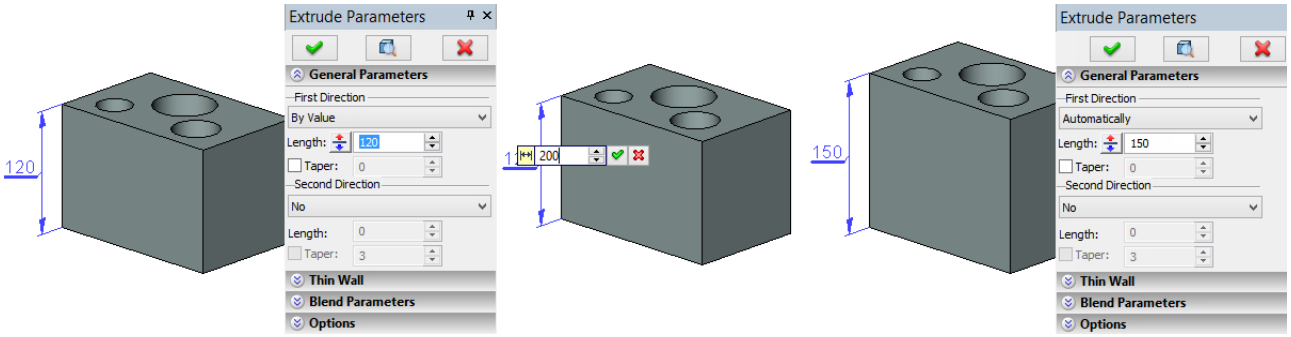
For example, if a dimension between faces or a dimension on a cylindrical face (radius or diameter) uses a tolerance or additional text lines, then, when creating the dimension in the drawing you select the respective lines on a projection - this information is automatically passed on to the dimension on the 2D projection. The same happens when creating a roughness symbol on a face – if you create a roughness symbol on the respective lines of the 2D projection image, then the roughness parameters will be copied from the 3D model.



Data transfer from 3D dimensions to a 2D projection can be done also by a direct “projection” (mapping) of dimensions from the 3D space onto the 2D projection. In this case, the system automatically determines, which of the existing 3D model dimensions can be “transferred” to the current 2D projection. The user just needs to select, which out to the system-offered dimensions one would like to put on this 2D projection. This functionality is described in detail in the chapter “2D Projections. Creating Drawings from 3D Models” (see the section “Creating Drawing Detailing Elements on 2D Projections”).

3D dimensions can be used to manage dimensions of the 3D model. When creating various 3D operations, the systems allows for an automatic creation of "driving" 3D dimensions. A driving dimension is directly related with the operation that created it, or, more precisely, with its geometrical parameter. As soon as its nominal value is modified in the command **Parameters** > **Dimensions** > **Set Value**, the respective parameter of the dimension-producing operation will change.

You need to select opposite body faces when specifying dimension.



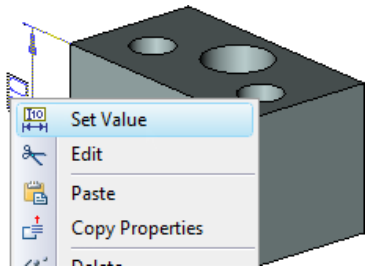
The driving 3D dimensions can also be created by the user manually. To do this manual dimension creation, select the faces, whose positions are determined by the operation's geometrical parameter. For example, you can use the opposite faces of a body created by extruding, or a cylindrical face created by blending. By using such dimensions, you can also modify the operation parameters and, therefore, the entire 3D model.

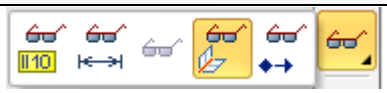
2D dimensions can also be used to modify an operation's parameter values, which are created between the lines of 2D projections that correspond to the body faces driven by that operation's geometrical parameter. The system finds this corespondence automatically.

Besides that, the system can in some cases automatically find correspondences between various 3D dimensions, which are not the control dimensions (the ones not related with the operation parameters), and 2D constructions used for the body creation, on whose faces those dimensions are based. This also allows managing the 3D model by modifying the nominal values of such 3D dimensions. When the nominal value of such a dimension is modified, the system will attempt to automatically find the 2D constructions, which shall be modified in order to accommodate for the new nominal value of the given 3D dimension. For example, if a 3D dimension is created between two side faces of a 3D profile extrusion operation, then the system is able in a number of cases to restore the constructions train: 3D profile → Profile contour on the workplane → 2D constructions that determined the 3D profile contour. In this case, as soon as the dimension nominal value is modified, the system will adjust the corresponding 2D constructions, the 3D profile based on them and the body itself, which resulted from the extrusion operation.

A 3D dimension nominal value is modified, just like that of a 2D dimension, by the command **PE: Set Dimension Values**. Note that this command does not work on 3D dimensions in the transparent mode. It can be accessed, however, from any 3D dimension's context menu (on right-clicking it).

3D annotations can be hidden or displayed in each particular window using the new command **SN: Hide 3D Annotations**:



Icon	Ribbon
	
Keyboard	Textual Menu
<SN>	View > Hide 3D Annotations

CREATING 3D ANNOTATIONS

3D annotations can be created as follows:

In the 3D window, using the common commands for creating dimensions, leader notes and roughness symbols (**D: Create Dimension**, **IN: Create Leader Note**, **RO: Create Roughness Symbol**);

As 2D detailing elements on workplanes and their consequent display in the 3D window.

Creating 3D Annotations in the 3D Window

Creating 3D Dimensions in 3D Window

To create a 3D dimension, call the command "**D: Create Dimension**" while in the 3D window.

3D dimension can be drawn:

- Between two planes. When choosing two planar faces, a linear dimension is drawn (for parallel planes) or an angular dimension - (for nonparallel planes);
- Between a plane and a line. A linear dimension is drawn if a line is parallel to a plane, and a radial one – otherwise;
- Between a plane and a point. A linear dimension is drawn between a given plane and a point;
- Between two lines. A linear dimension is drawn if the lines are parallel to each other, and a radial one – otherwise;
- Between a line and a point. A linear dimension is drawn between a given line and a point;
- Between two points. A linear dimension is drawn between selected points (vertices);
- On a single rotation surface. A radial/diametrical dimension is drawn for selected face.

The planes are specified by selecting a planar face or a planar curvilinear edge, the rotation surfaces – by selecting a face of revolution lying on the given surface. The lines are specified by selecting straight edges or the axes of the rotation surfaces. The axis of the rotation surface is specified by selecting the corresponding face of revolution. The points are specified by selecting vertices of the 3D model solid bodies and 3D nodes.

When creating 3D dimensions, the elements are selected with the help of the following auto menu options:

	<L>	Select Flat Face or Flat Edge
	<C>	Select Cylinder, sphere or torus
	<A>	Select Axis
	<N>	Select Vertex

By default, in the 3D window the user can select planar faces, faces of rotation surfaces, edges and vertices of solid bodies without turning to the corresponding auto menu options.

If upon creating a 3D dimension, the user selects first the edge-circle/circular arc, or the face of revolution of type cylinder, sphere, torus, then the system will automatically start creating a radial/diametrical dimension. In other cases, the system will wait for selection of the second object (face, edge, vertex) for creating a linear or angular dimension.

For selecting the axis of the rotation surface (cylinder, sphere, torus) as the first object for creating a linear/angular dimension, it is necessary to use the option when choosing the face of revolution. The same option is used for selecting a planar edge-circle (circular arc) as the first object of the linear/angular 3D dimension.

A dimension being created may lie in various planes. The initial position of the dimension being created is automatically determined by the system based on the 3D view. You can then modify the 3D dimension plane by the option:

	<H>	Change 3D Dimension Orientation
	<T>	Select Plane of Orientation

The option allows one to cyclically switch between various dimension positions.

The option is used for direct selection of the plane, to which the dimension being created has to be parallel. After turning on the option, in the 3D window the user has to select a face, to the plane of which a 3D dimension has to be parallel.

When creating 3D dimensions, several standard options for setting dimension parameters are also available:

	<Z>	Change leader line jog orientation (only for linear dimension)
	<Space>	Place Dimensions in absolute coordinates
	<J>	Center Dimension Text
	<Z>	Change Dimension Orientation (only for angular dimension)
	<M>	Change Dimension type (only for radial/diametrical dimensions)
	<D>	Click to toggle radius/diameter mode (only for radial/diametrical dimensions)
	<R>	Click to toggle radius/diameter mode (only for radial/diametrical dimensions)
	<K>	Break (kill) relations (only for linear and radial/diametrical dimensions)

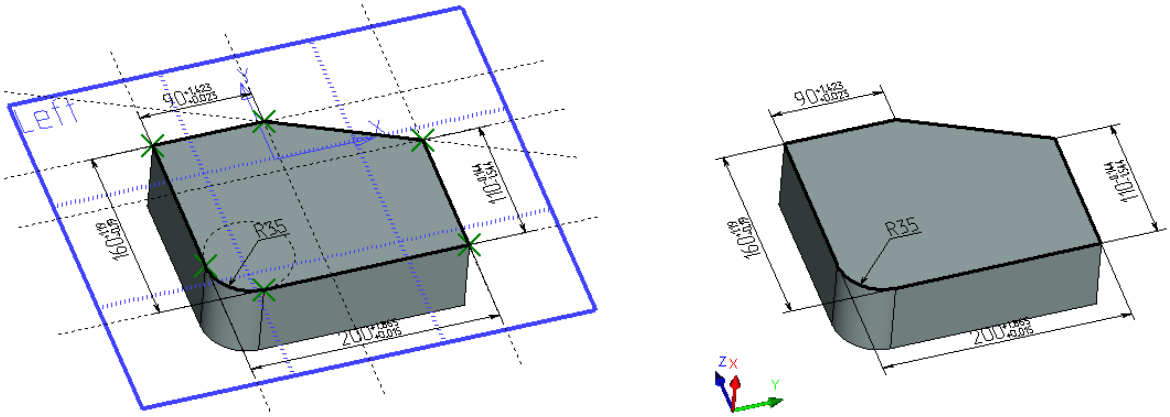
Creating 3D Leader Note and 3D Roughness in 3D Window

To create a 3D leader note or 3D roughness, you need, while in the 3D window, to call the command **IN: Create Leader Note** or **RO: Create Roughness Symbol**. When creating a leader note/roughness, you select the face, on which the element being created shall be placed. The face can be of any type. The snap point of the leader note or roughness is the point on the selected face where the cursor was positioned at the time of selecting the face. For a leader note, you need to additionally specify its shelf position in the space.

Other than that, creating 3D dimensions, 3D leader notes and 3D roughness symbols is same as creating the respective 2D elements.

Creating 3D Annotations by 2D Detailing Elements on a Workplane

To create 3D annotations by 2D elements on a workplane, you need to first create the respective 2D dimensions, leader notes, roughness symbols on the page of the workplane. To display them in the 3D window, you just need to enable the flag “Show 2D Annotations on 3D View” in the workplane parameters dialog box.



For working convenience, this option is also available in the context menu of the plane ("Show Dimensions in 3D").

Such dimensions can be used for modifying the display parameters on the workplane page and, respectively, the 3D elements created based on it (profiles, operations etc.).

Automatic Creation of 3D Dimensions

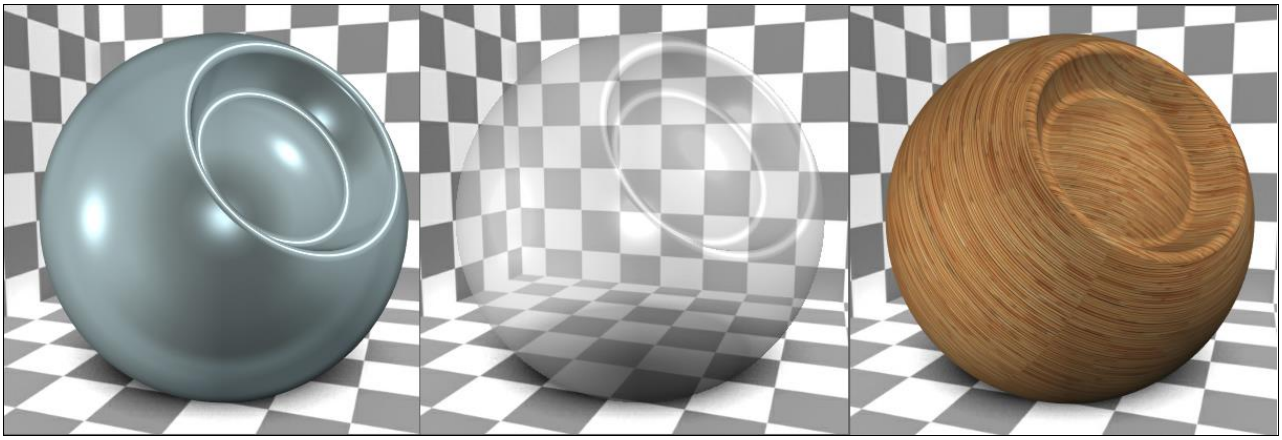
Automatic creation of driving 3D dimensions, that is the 3D dimensions related with the geometrical parameters of 3D operations, is made either directly when creating the operation, or from the context menu of an already existing operation.

When creating a 3D operations, you just need to enable the flag **Create Driving Dimensions** in the operation's properties dialog (the **Options** section). The driving 3D dimension will appear in the 3D window once this operation creation is completed.

If an operation was already created, call the context menu (by right-clicking) and select the **Create Driving Dimensions** command in it. This will also result in creating a 3D dimension that controls the geometrical parameter of the given operation. If several operations are selected, then you can still access the control 3D dimension creation command in the context menu. This command is also available in the Body's context menu. In the latter case, the control 3D dimensions will be created in all operations used for creating the Body, wherever possible.

MATERIALS

A material is a T-FLEX CAD element. It contains a list of characteristics of a real material with which we deal in reality. A material is used as a parameter for all 3D operations, and, additionally, by a special command of applying material to individual faces.



Material characteristics can be roughly divided into two types. Characteristics of the first type affect the appearance of three-dimensional objects in the 3D window, and, additionally, define the parameters of a material as part of generating a photorealistic image in the POV-Ray system. Characteristics of the second type are various physical-mechanical parameters of a material, such as density, elasticity coefficient, etc. Those parameters are taken into the account when calculating mass-inertia properties, as well as conducting structural computations in the analysis module. Besides that, materials are accounted for when creating hatches on sections of 3D models. You can define parameters of such hatch for a particular material.

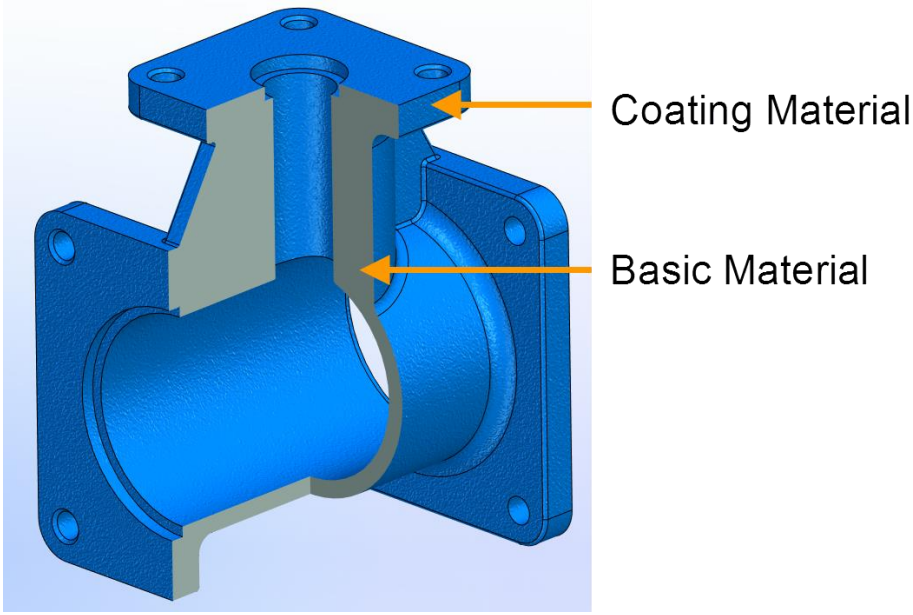
GENERAL INFORMATION

For each model we can associate a set of materials that were assigned to Bodies, operations or individual faces. These materials are called the materials of the current model. Material – is a 3D element containing a set of characteristics of the real material with which we are dealing in reality. The element "material" has parameters such as density, reflection power, absorbing capacity, etc.

For any Body of the model it is possible to specify the *basic material* and *cover material*.

The basic material is used for calculation of mass-inertial characteristics and engineering analysis, and also for displaying the Body in 3D scene in the visualization mode "Shading with materials" (if the additional material of the cover is not specified).

The cover material – is an additional material for defining parameters of the model display in the 3D window. It allows us to specify the material different from the basic one to be used for visualization purposes, creating the effect of the Body "painting". At the same time for engineering calculations the basic material will still be used.

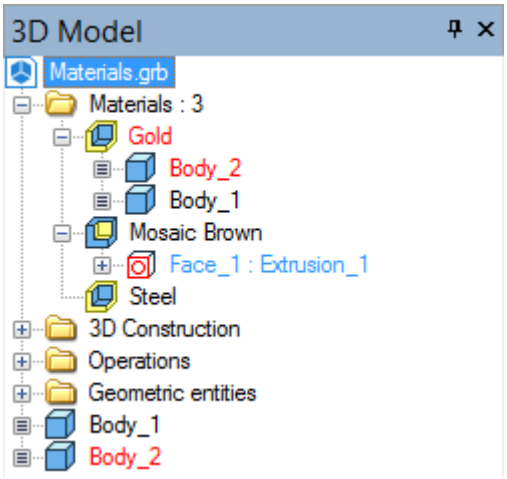


The material can be applied on individual faces of the body. When the face's geometry is changed as a result of further transformations of the model the material that was applied onto the face is retained. When applying the material on individual faces, the body itself does not change its own material. You can apply material on individual faces only when the flag **Apply as coating material** is enabled.

Materials used in the current model are displayed in the tree of the 3D model, in the operations properties dialogs, and in the dialog of the **3MT: Edit materials** command.

MATERIALS IN TREE OF 3D MODEL

Materials used in the current model are displayed in the tree of a 3D model in the special folder – **Materials**.

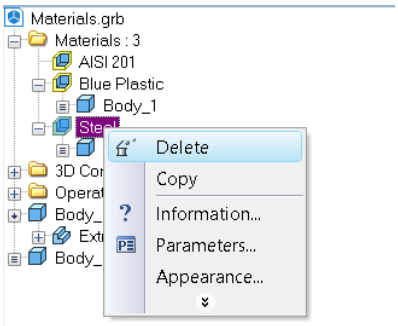


Different icons are used in 3D model tree to identify coating and basic materials.

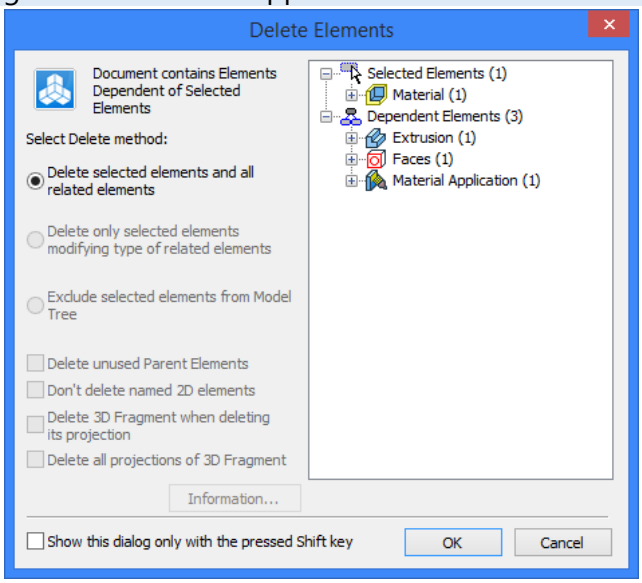
The icon is used to identify coating material. The icon is used to identify basic material.

If the material is applied to any body in scene, the icon ⊞ appears to the left of the materials name. If you press this icon, the list of objects to which the material is applied will appear. Icon do not appear near not applied materials.

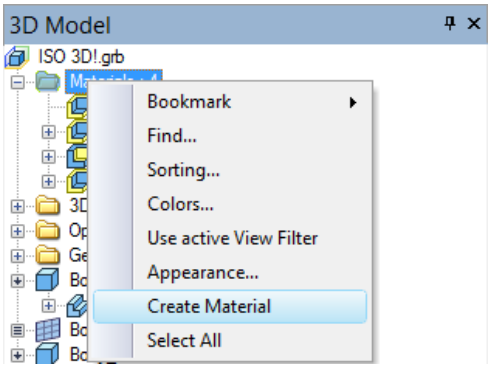
It is possible to edit material properties or delete the material from the current model using 3D model tree. The commands for deleting and editing the material properties are available in the context menu of the 3D model tree.

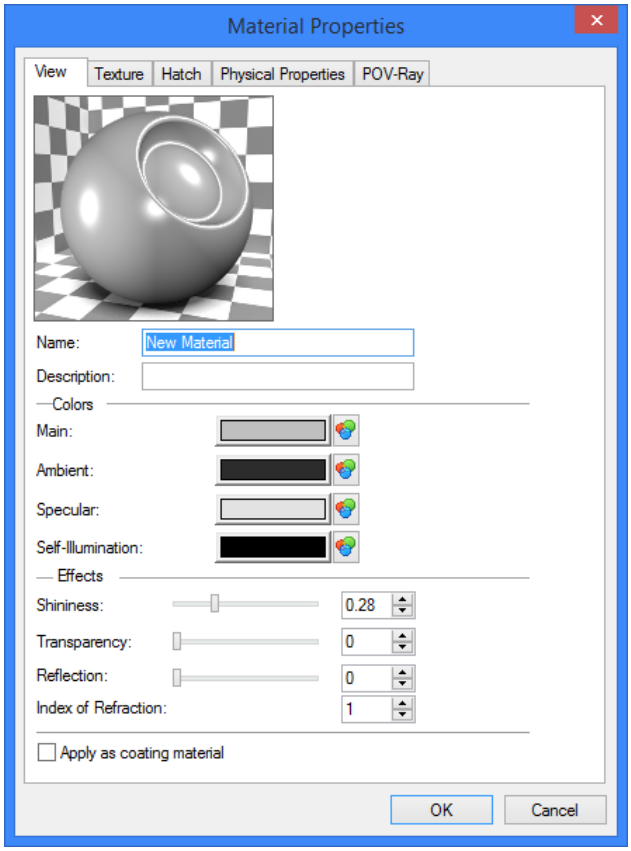


Note that upon deletion of the applied material, the system will also prompt you to delete the operations to which the given material was applied.



If necessary, the user can create the new materials directly in the tree of the 3D model by activating the command of the context menu **Create Material**. After that the material properties dialog appears on the screen.





SERVICE WINDOW "MATERIALS"

Special service window – **Materials** is used to work with materials. It is available even if there is no open document.

To display this window, the **following commands can be used**:

Keyboard	**Text menu**	Icon
<Alt+8>	Customize > Tool Windows > Materials Window	
<3MT>	Tools > Materials	

This command can also be invoked by pressing in the domain of automenu or of any toolbar.

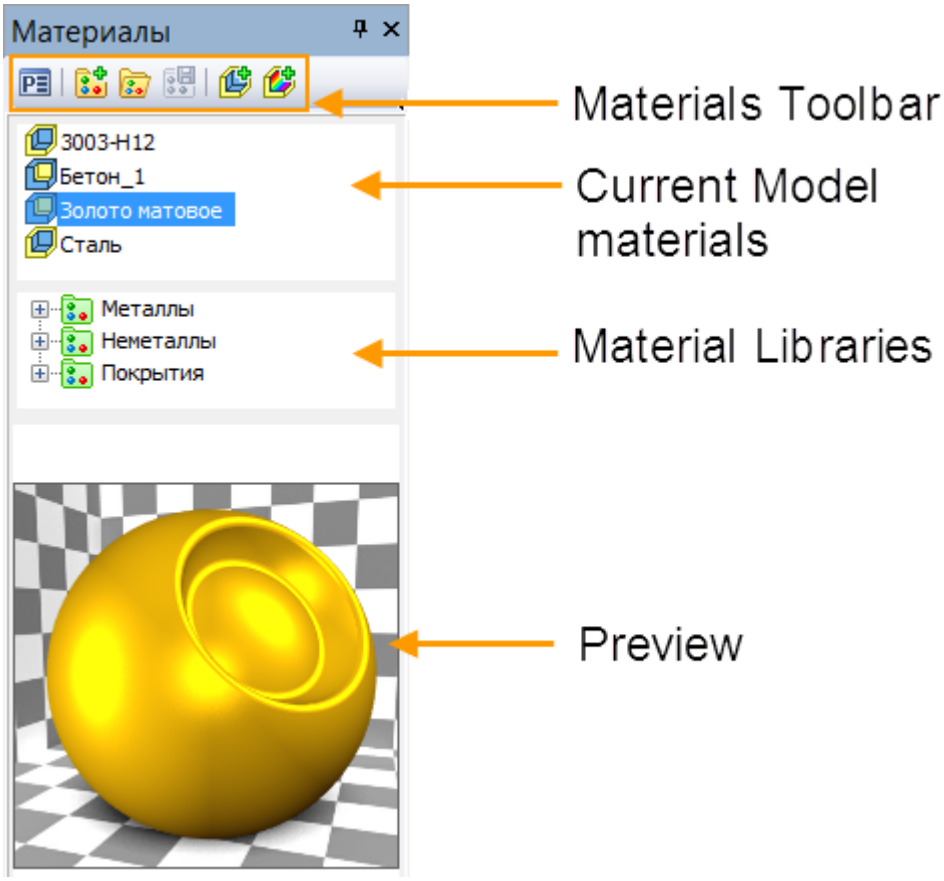
The **Materials** window is divided into three parts: **current model materials, material libraries and preview.**

The current model materials are displayed in the upper part of the window. By default, the list contains one element – "Steel".

Material libraries are displayed in the middle part of the window. All opened libraries are listed here. There are three libraries that are opened by the default: **Metals, Non-metals and Coatings.**

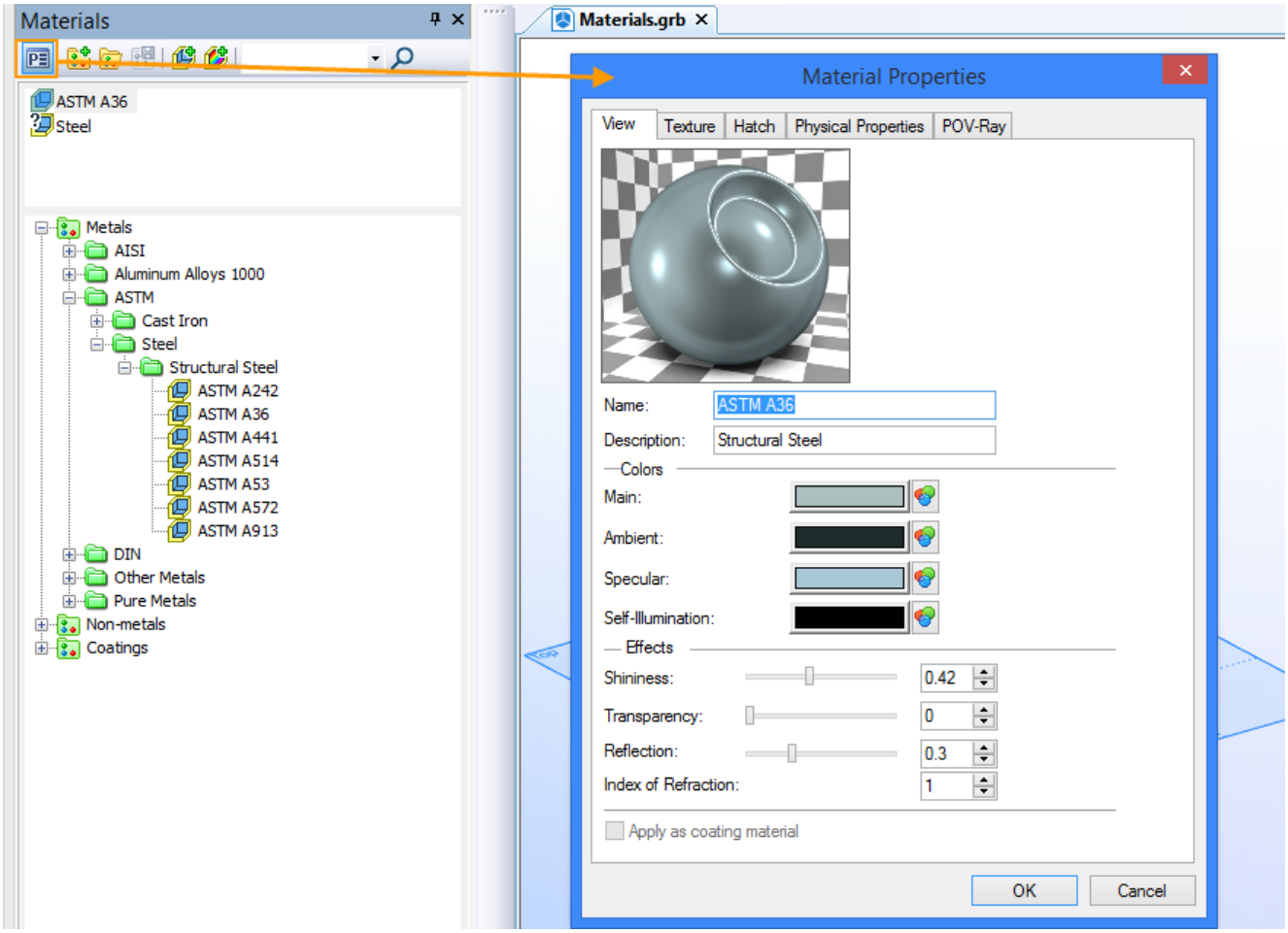
Preview window is displayed in the lower part.

On the top of **Materials** window there is a special toolbar. It is used to work with materials and material libraries.

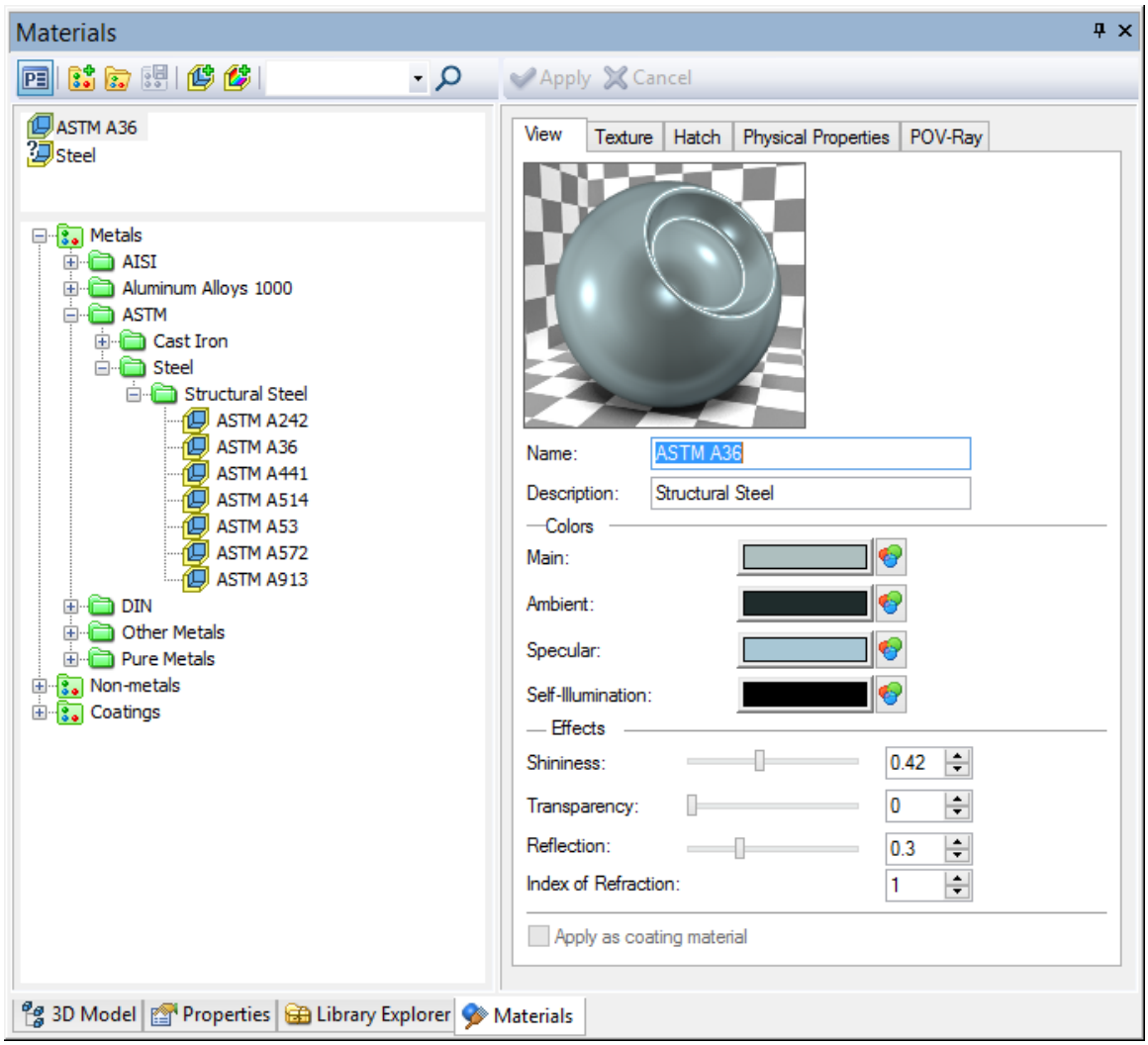


Materials Toolbar Options

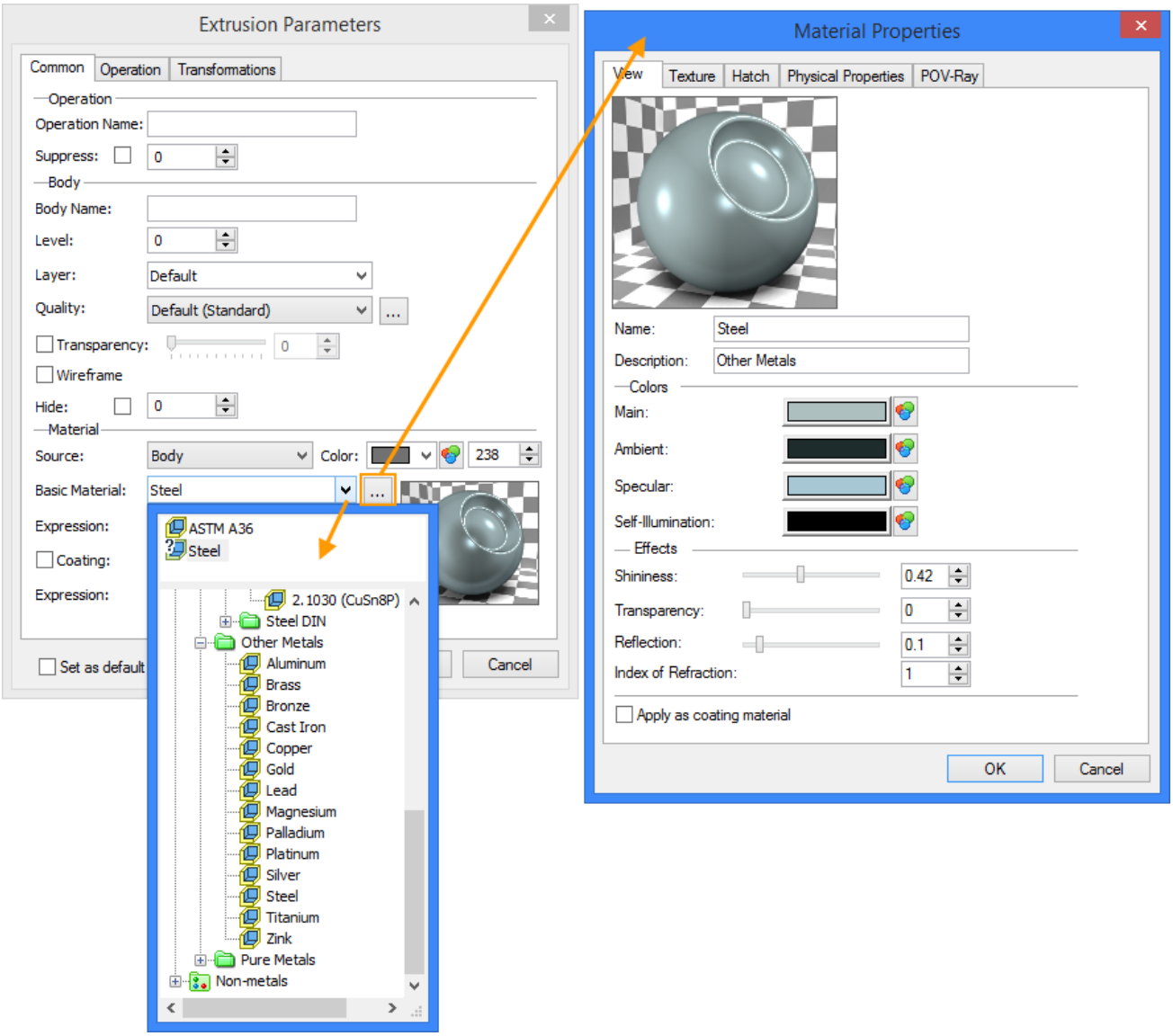
To view/edit the material's properties it is required to select the material (in the list of materials of the current model or in the library) and press the button in the upper part of the window or press . If the window is "fixed", a separate dialog window of parameters of the selected material will appear.



When the **Materials** window is in "floating" mode, the dialog of parameters of the selected material will appear in the right part of the "Materials" window itself. The dialog of parameters can be hidden by pressing again.

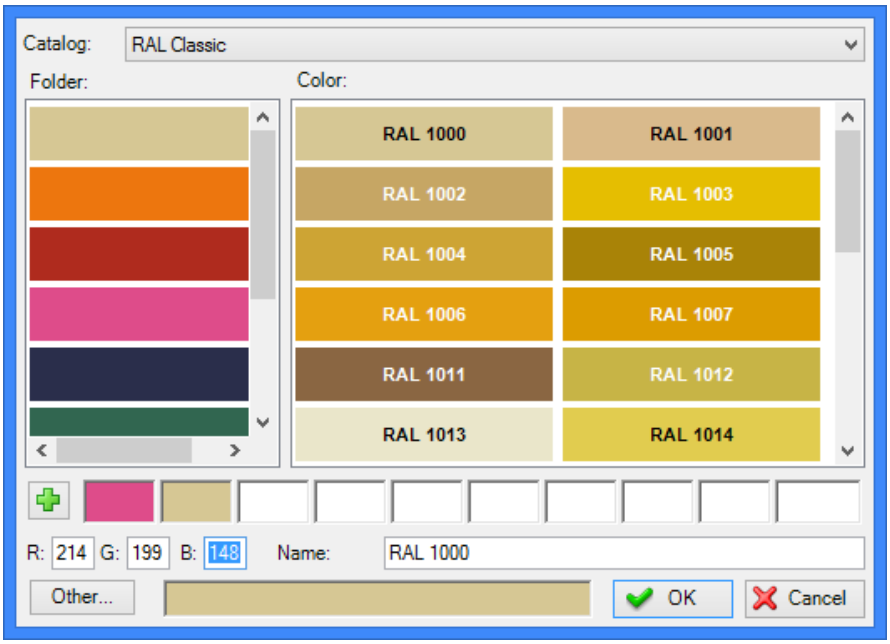


The same dialog window can be invoked from parameters of any 3D operation or Body with the help of the ... button. This button is located on the «Common» tab.



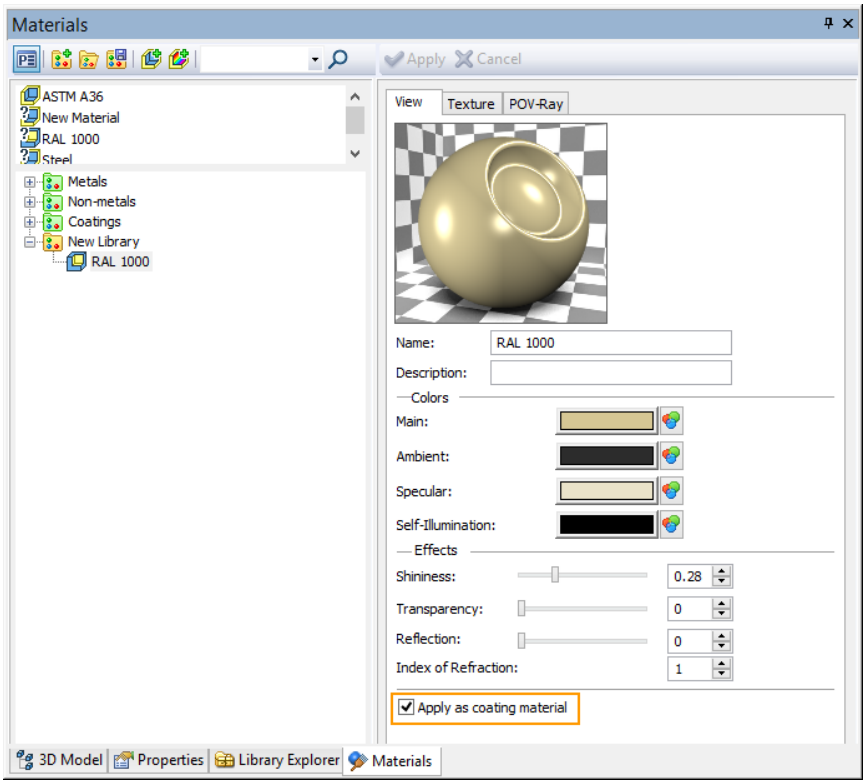
The option «**Create material**» allows you to create a new material. Material is created either in the current model or in the library, according to the currently selected part of the window.

The «**Create color-based material**» option allows us to create the cover material based on color. After activation, the dialog for selection from the colors catalog appears.

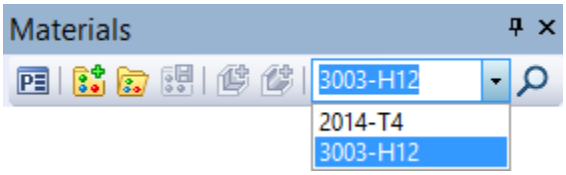


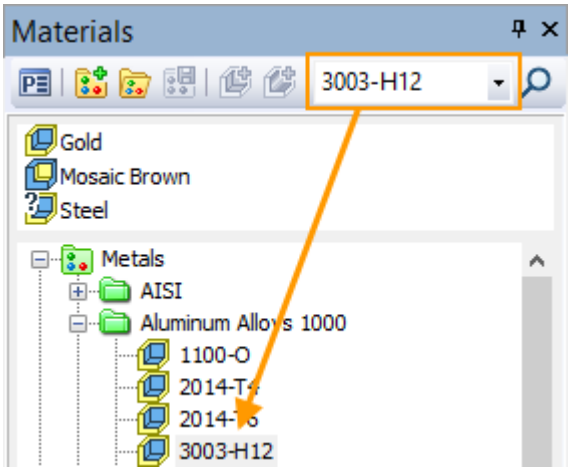
After selection of the color the new material is assigned the same name as the selected color (for example, "RAL 1000"). After confirming color selection, in the current window of the materials library or of the model the cover material is created with the corresponding name and properties.

The flag **Apply as coating material** is automatically set for a created coating material.



To navigate across the material libraries you can use search toolbar. With its help, the material can be found by the name or by part of the name.





For searching, you need to enter the name of material or part of its name into the text field or select the material from the drop-down list, if it already exists in the search field. After pressing <***Enter***> or the focus of the model tree will be placed on the desired material. Repeated pressing of <***Enter***> or initializes the transition to the next search result in the library. The entered name of the material if it does not yet exist must be added to the list in the text field.

The search among materials of the model is not carried out.

The remaining icons in the upper part of the **Materials** window are designed for working with the materials libraries.

The option allows us to create the new user-defined library of materials. The library is highlighted with green color , if it is located in system readonly folder. If library is located in any non-system folder and can be edited, it is highlighted with yellow icon .

The option "**open library**" serves for opening already existing materials library. After activation, you need to specify a path to the library file.

The option "**Save changes in libraries**" allows us to save the changes introduced into the materials library.

More information about working with libraries of materials can be read below.

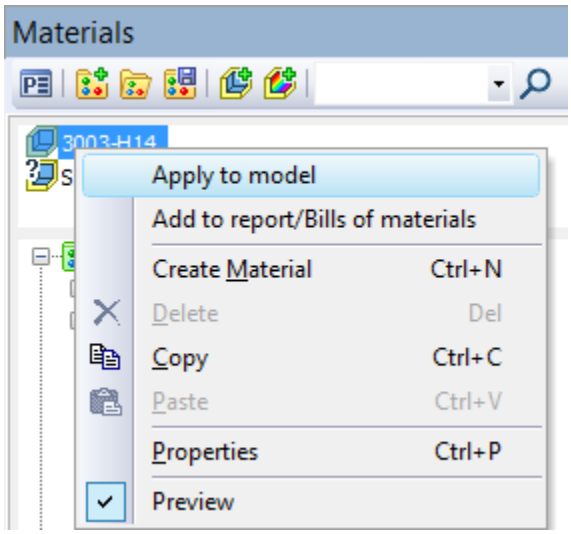
Materials of the Current Model

The list of the current models materials provides quick access to the most frequently used materials without searching through the libraries. It contains all materials that were used in the current document.

To identify coating materials icon is used. To identify basic materials icon is used. To identify materials that were applied previously but not in use, there is icon .

The following actions can be executed with the list of materials of the current model:

- Edit materials (The "Properties" option in the context menu);
- Add information about material into report and BOMs (The "Add to reports/Bills of materials" option in the context menu);
- Add new materials (the **Create Material** option in the context menu);
- Remove materials from the model (the **Delete** option in the context menu);
- Copy materials from the library to the list of materials of the model, from one T-FLEX CAD document to another (the **Copy**, **Insert** options in the context menu):
- Enable/disable preview option.



Materials libraries

The list of material libraries is an archive with basic and coating materials that are most frequently used in design. These materials are sorted by categories. There are three libraries that are opened by the default: **Metals, Non-metals and Coatings.** They are identified by icons.

> Materials from these libraries can't be changed inside the libraries. For this purpose, they should be moved into **Current model materials** window or into created library.

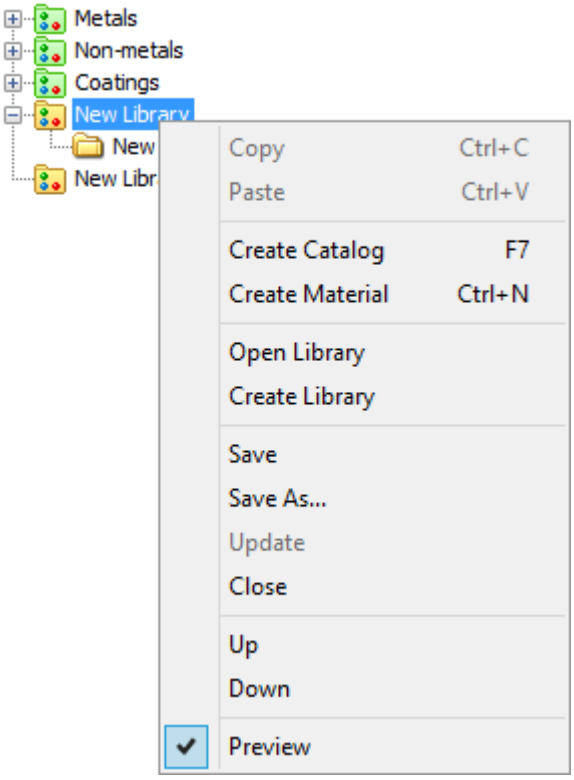
The obsolete libraries of previous T-FLEX CAD versions are also available and can be opened if necessary.

For various operations with libraries and materials you can use commands from right button context menu of "Materials" window or materials toolbar.

Material Libraries Creation

You can create your own libraries with customized set of materials.

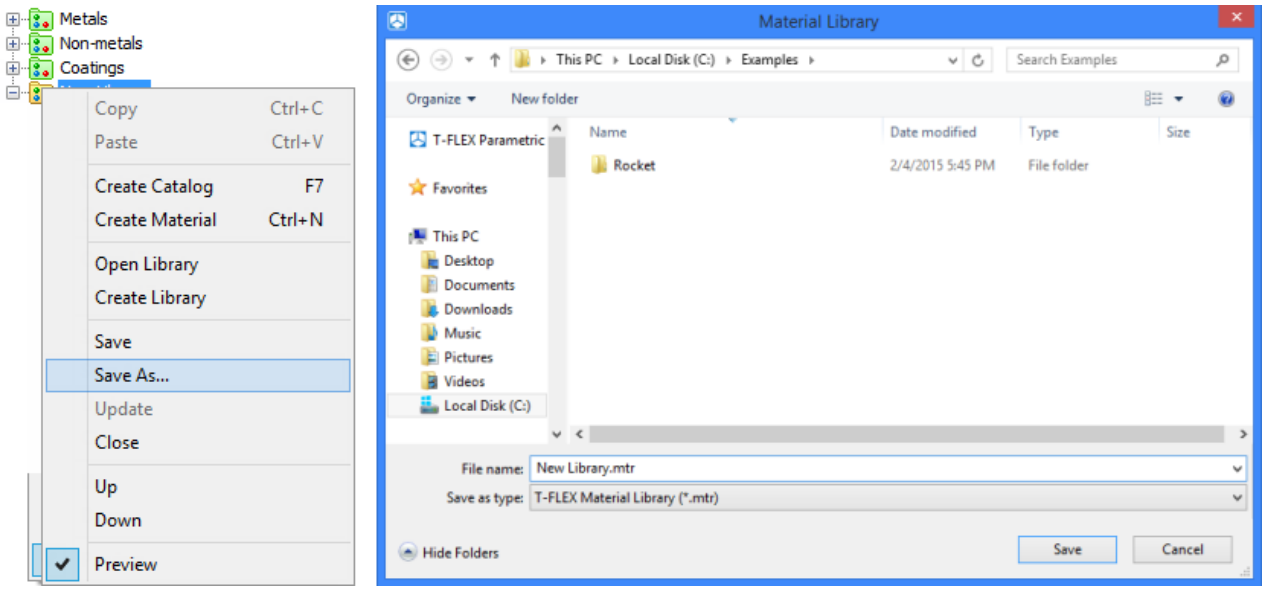
For this purpose, use option on the materials toolbar or item “Create Library” in context menu. The user-created library is identified by icon . The newly created library appears in the list of available libraries and has the default name "New Library".



Material can be created only in custom library open for editing. You cannot create material in system read-only folders identified with icons.

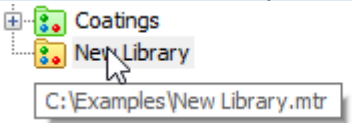
The newly created library is not saved by default. In order to make the library available in future, use option on the materials toolbar or item “Save as...” from the context menu. You can rename library and set new folder location before saving. Files in T-FLEX CAD are saved in .mtr format.

File in .mtr format can be opened in any text editor.

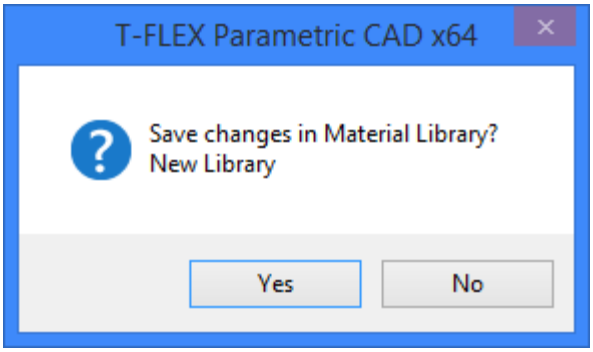


To open a library use option at the materials toolbar or item "Open Library" from the context menu.

All material libraries are located in folder *"Program\MaterialLibrary"* by default. The path is displayed when you put the cursor on the library name.



In a case, when changes in library were not saved before closing, system will offer you to save these changes.

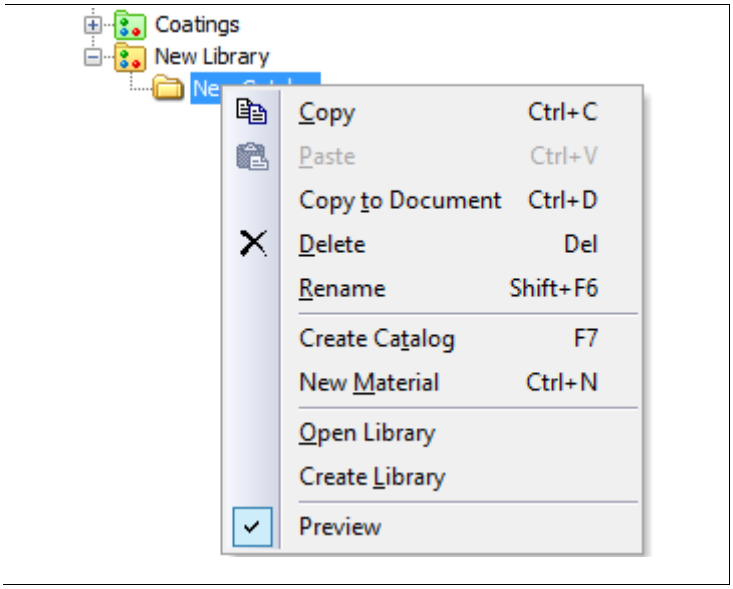


Catalogs of Materials Creation

For convenience, you can combine materials within the separate catalogs for each group of materials.

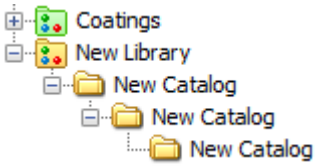
To create a new catalog you need to select library, in which it will be created, press and select item "Create catalog" from the context menu. Created category will be called "New Catalog". You can rename catalog directly after creation or rename it later using item "Rename" from the context menu.

Created category can be deleted using item "Delete" from the context menu.



To copy all materials from a catalog to the list of materials of the current model, use command **"Copy to Document"** from the context menu or drag the category manually.

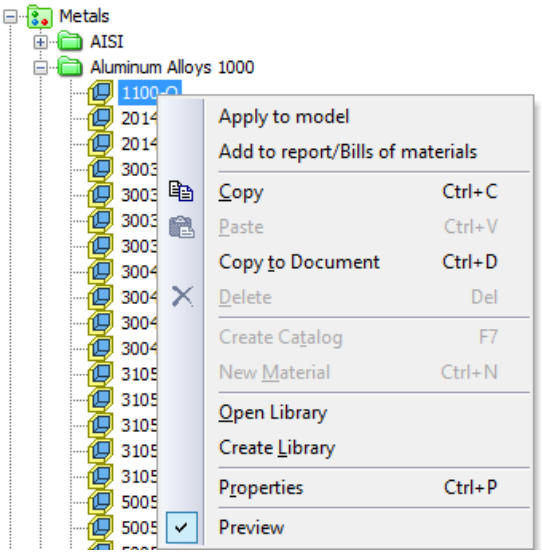
Within one library, you can create any number of catalogs. In each catalog, you can create additional folders with materials (sub-catalogs).

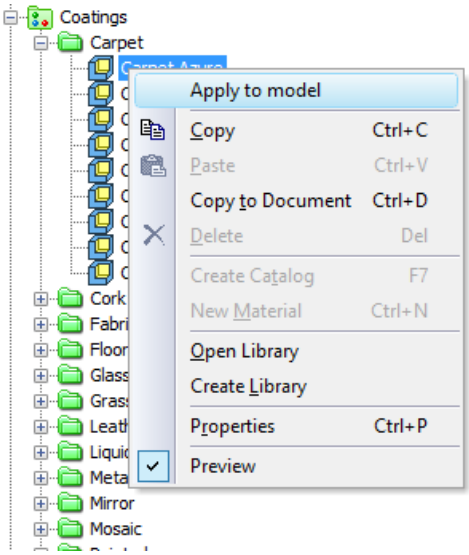


The catalog can be copied and transferred to another library or catalog.

Materials Creation and Applying

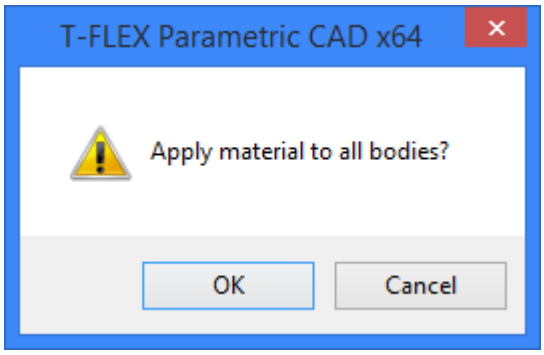
To create a material in the new libraries and catalogs, use command **New Material** or options and on the materials toolbar.



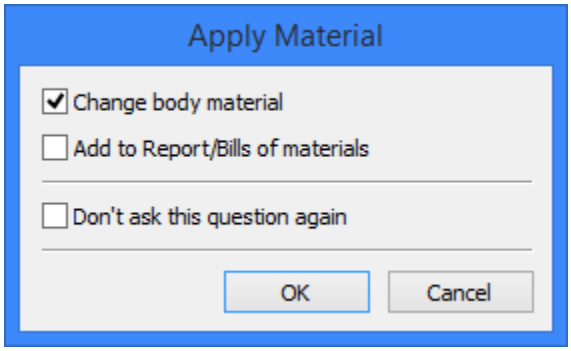


To apply a material to objects in the scene, you can use item **Apply to model** from the context menu or "drag&drop" method.

If there are several bodies in the scene, the following dialog window appears:



If you press **OK** or if there is just one body in scene, the following window appears:



Add to Report/Bill of materials allows to add information about material into reports and bills of materials.

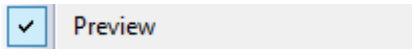
In case, when changes in one or several materials from library were not saved before closing, system will offer you to save these changes.

To move materials within a library or from one library to another or between catalogs and lists of materials of the current model you can also use "drag&drop". The individual materials can be copied, using the context menu commands **Copy** and **Paste**.

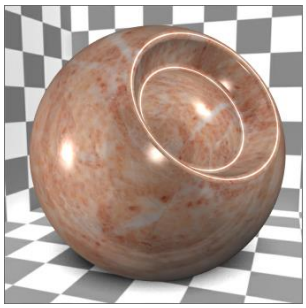
> If the material or catalog is copied to a material library or catalog that already contains an element with the same name, the copied item will be automatically renamed and numbered with (1), (2) and so on suffix.

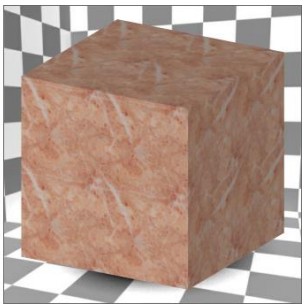
Preview

The preview window allows to estimate material visually, without applying it to the objects in the scene. The mode is activated by item **Preview** from the context menu.



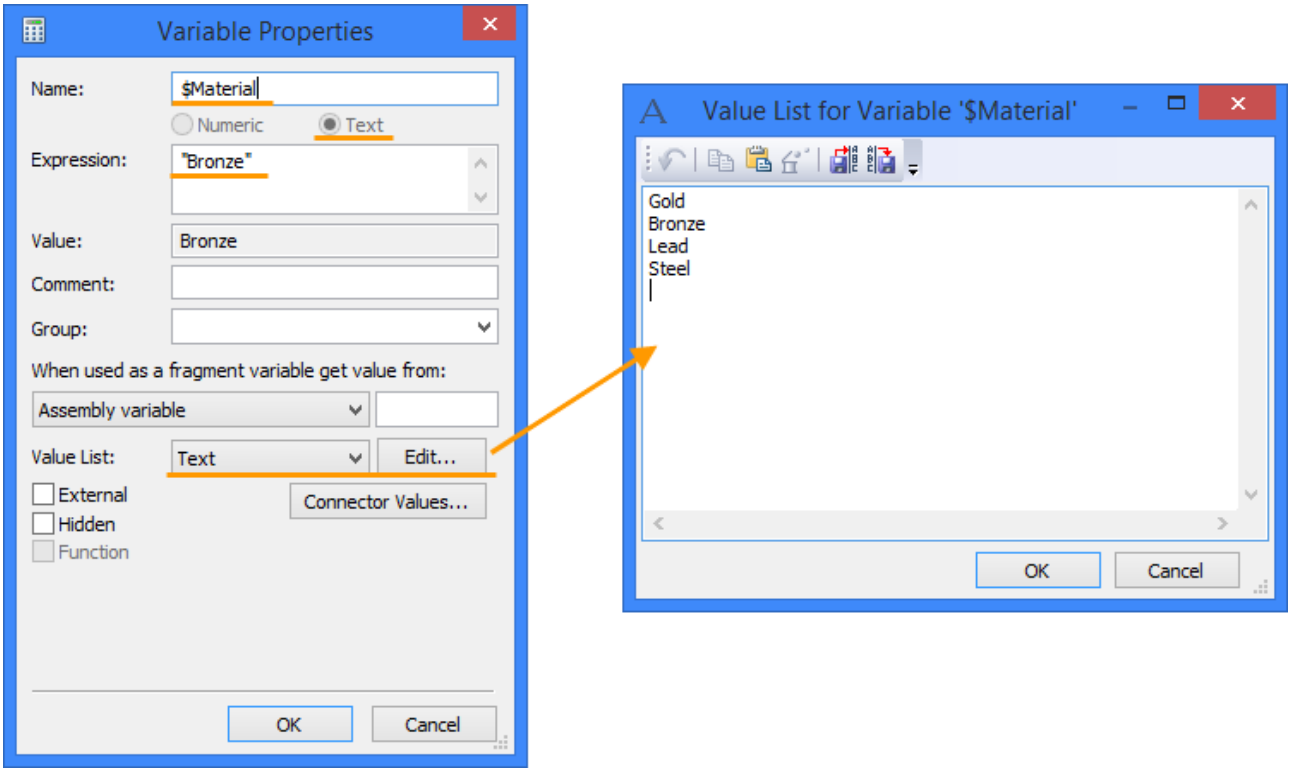
Selected material can be displayed on the cubic or spherical surface. To switch between these modes click on the picture.





CHANGE MATERIAL USING VARIABLES

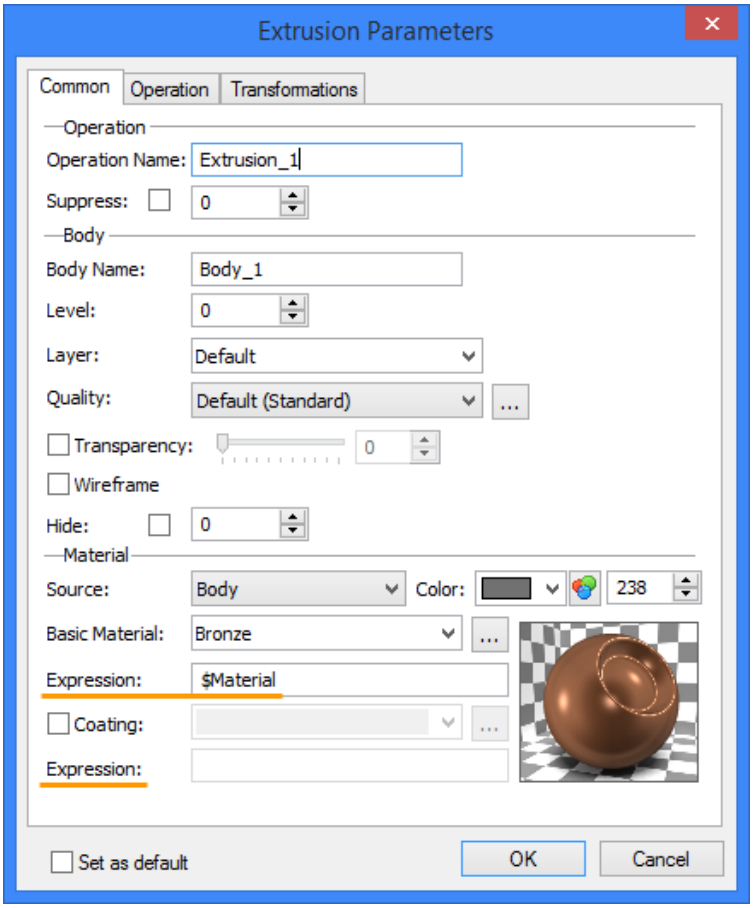
Variable can be used to assign material names. For that purpose you should use existing or create a new text variable using command: **V:Edit Variables.**



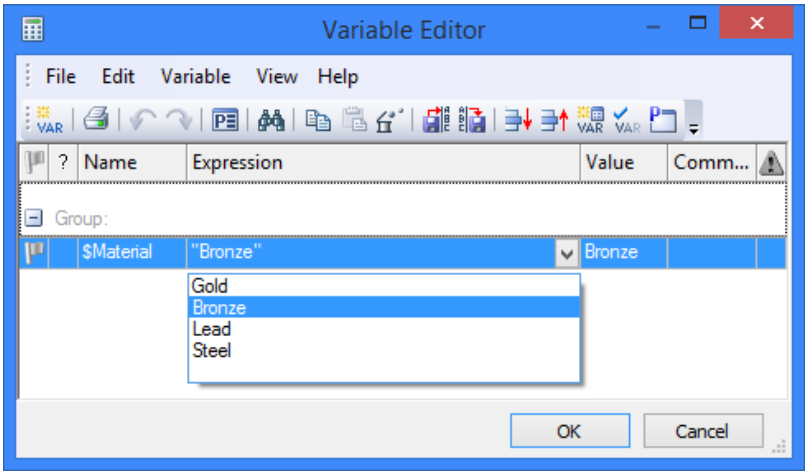
In the expression field, you should enter or copy from the library the name of the material that will be used for the model.

In order to generate a list of several items you need to use the [Edit...] button. In the window "Value List for Variable" you need to specify all materials that can be used for this model. It is important that the names of materials must match similar names in the library.

After creation of the variable you need to enter the properties of the body, and set the name of the created variable in the field **"Expression"**. Thus you can set main material and coating material.



After completing these steps, you can change the material of one or more bodies with the drop-down list in the variables window.



To display the new material in the scene, you need to update the 3D model using the option .

MATERIAL'S PROPERTIES DIALOG

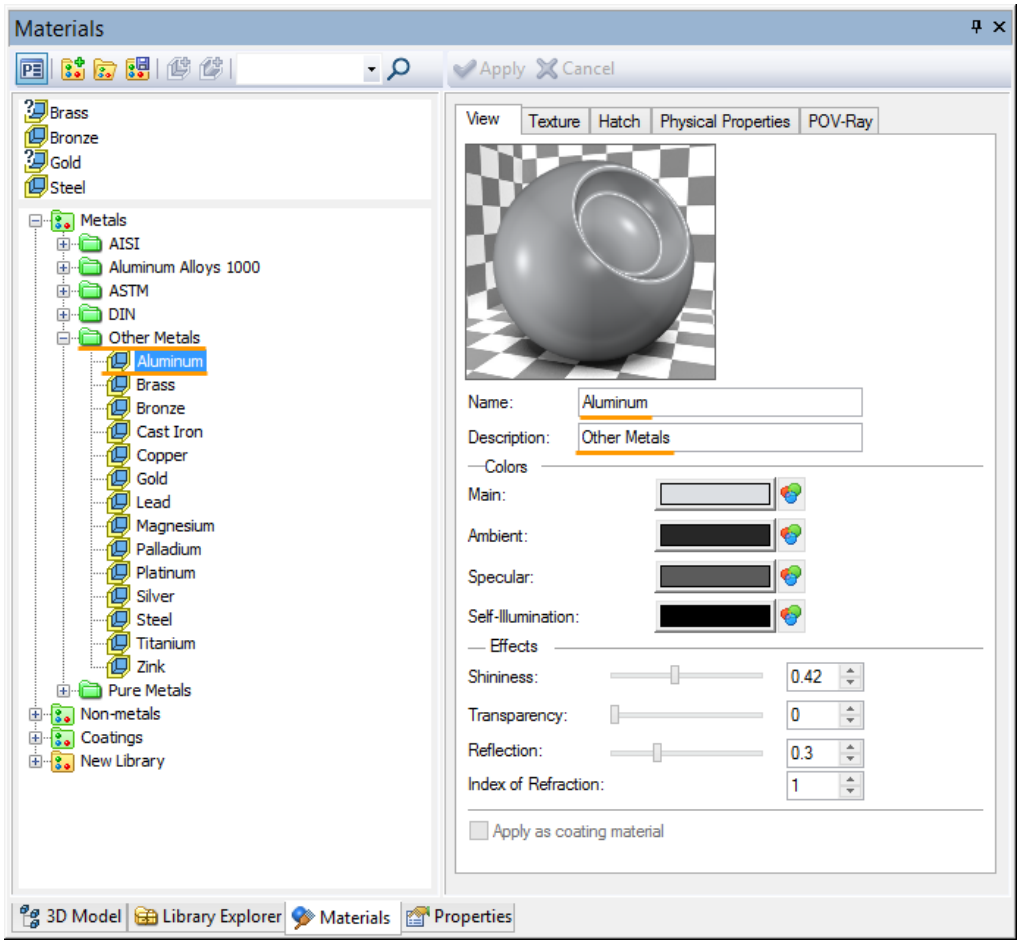
The material's properties dialog contains all physical-mechanical properties of the material, displays parameters of the material in 3D scene and on projections, and parameters for creation of photorealistic image (see the «Photorealistic view» chapter).

To call properties dialog window you need to select material (in the current model materials list or in the library) and perform one of the following actions:

- press icon on the materials toolbar
- press on the material name
- select item **Properties** from the context menu.

«View» Tab

This dialog combines parameters controlling characteristics of color, reflection, transparency and material's light emission.



The image of the material on this tab is available in two versions: in the form of a sphere with a hollow or in the form of a cube, for a more complete display of the visual characteristics of the material.

Name. In this field, material name can be edited.

Description.. This field displays the name of folder containing material.

For the model material this field can be edited so you may enter your own description.

When copying material to a library, description will be named by folder name where it is placed.

«Colors» Group

This group combines parameters defining characteristics of material's color. To specify the color for one of the parameters, it is required to click into the color rectangle and use the standard mechanism for color selection.

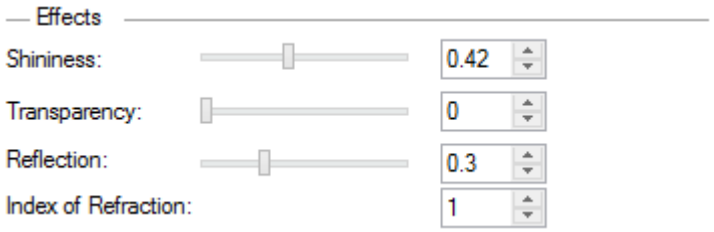
Main. This parameter controls the color of the material lighted up by direct light. This is a base color of the material.

Ambient. This parameter controls the color of the material lighted up by dispersive light. It significantly affects the color of the material's surface since at the given moment of time only a small portion of object is subject to the action of direct light.

Reflecting (blink). This parameter controls the color of blinks that appear on the surface of object. The brighter the color, the more intensive the blink is.

Light Emission. This parameter controls the intensity of the light radiated by the material. If the material does not radiate the light, the black color is specified.

«Effects» Group



This group of parameters allows us to customize the following optical characteristics of materials:

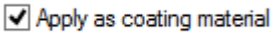
Shininess. This parameter controls the power of reflection of light sources on object's surface. It is specified as a single number that can range between 0 (minimum) and 1 (maximum). Large values create the appearance of smoother and shinier material whereas the decrease in their values imitates dull surfaces.

Transparency. This parameter controls the value of transparency of the object's surface. It changes between 0 (nontransparent surface) and 1 (totally transparent surface).

Reflection. This parameter controls the value of the specular reflection of the material. It is specified as a number in the range between 0 (reflections of surrounding objects will not be shown on the material's surface) and 1(entire reflectivity of the material is provided).

Index of refraction. This parameter characterizes the deflection of light rays as they pass through the transparent material. Using the index of refraction you can set the value of this parameter for a transparent material. This option is taken into account when generating photorealistic images.

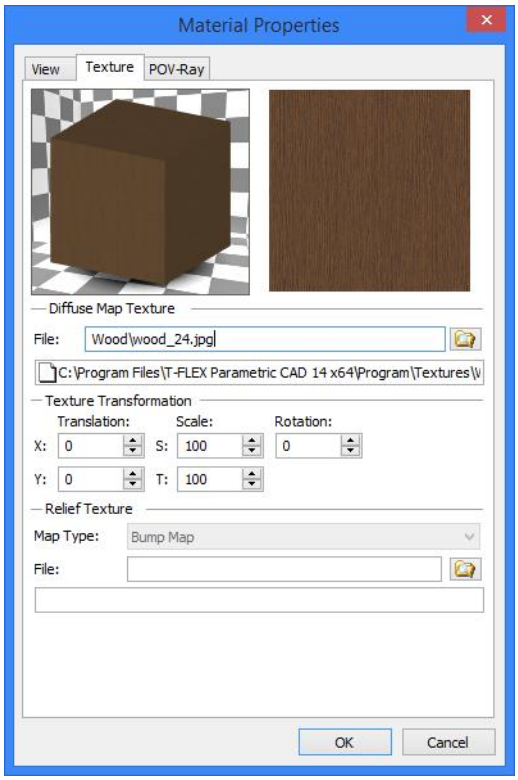
If the «**Apply as cover's material**» parameter is enabled, the material is applied as a cover's material and not as the basic material.



«Texture» Tab

This tab combines parameters that control the material's texture specification. Texture – is an image that is stored in the file and helps us to display special features of material's surface coloring.

File. File of the image containing the picture of the texture. To specify the texture, the files of the formats bmp, gif, jpg, jpeg, tga, tif, tiff and png can be used.



The work with texture files is realized according to the general algorithm of working with references to external files (similar to fragments, pictures, files of databases by reference etc.). This means that:

Texture files can be located in the same folder as the model's file that uses this texture;

Color texture and relief texture can be specified by different files;

2. Texture files can be moved with the "Move assembly" command similar to other types of files;

3. Texture files can be included into the file of the model that uses the texture (embedded textures);

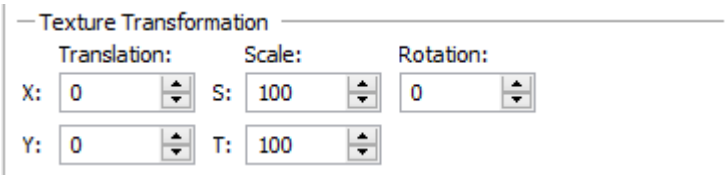
4. To specify the path to the texture file the library's name can be used;

5. To specify the path to the texture file, the relative paths that includes the folders names or transition to the upper level (..\) can be used.

When you choose texture file folder "Program\Textures" is opened by default.

«Texture transformation» group

This group of parameters serves for customizing the appearance of the texture, translation of the texture, changing its scale and for rotation.



Shift X. This parameter defines the shift of the texture's start point along the horizontal axis.

Shift Y. This parameter defines the shift of the texture's start point along the vertical axis.

Scale S. Texture's scale along the horizontal axis. The scale's value can be negative. If, for example, we specify the value «–100», the texture will be symmetrically reflected with respect to the horizontal axis.

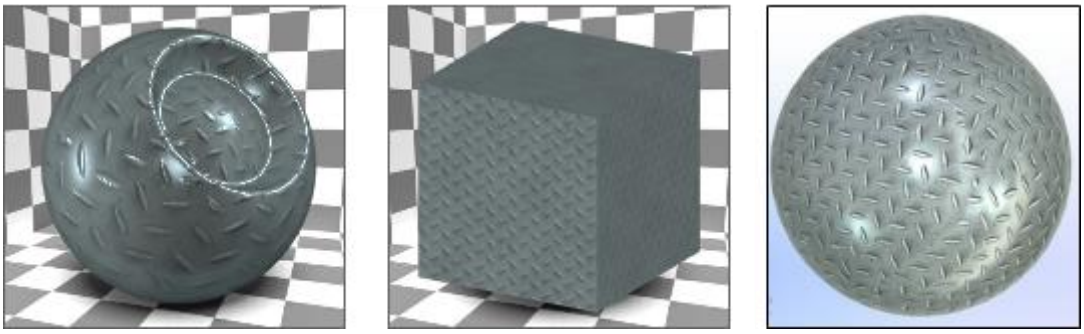
Scale T. Texture's scale along the vertical axis. The scale's value can be negative.

Rotation. Texture's rotation angle. The texture is rotated around the center located in the plane perpendicular to the direction of texture's projection. The coordinates of the rotation center in the plane are specified by the parameters «Center S» and «Center T».

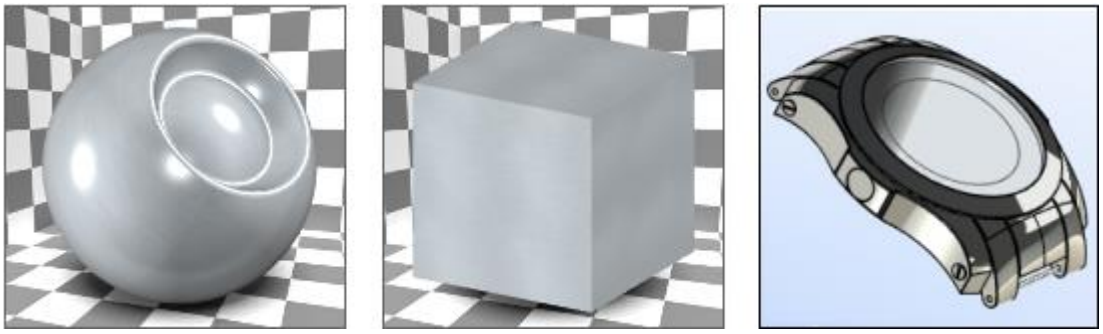
«Relief texture» group

This parameter group is responsible for creating the visual relief of surface material. The essence of the feature is that by adding shadows and highlights we get effect of the surface material relief.
There are three main ways for creating effect of relief surface corresponding to texture type:

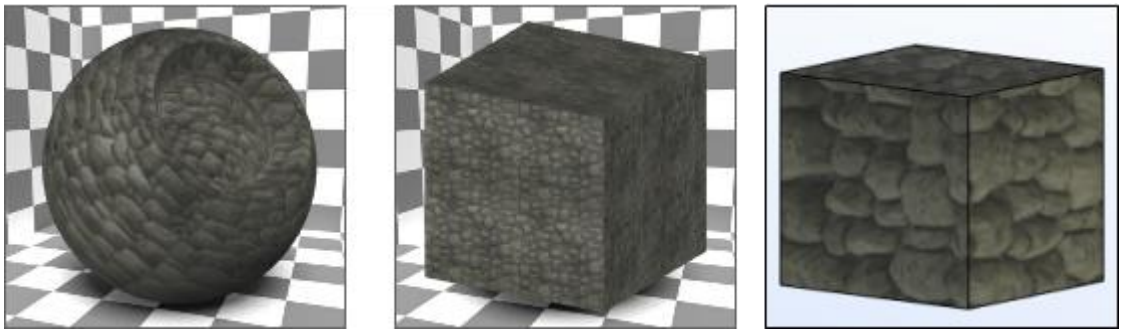
Normal Map. This texture like bump map allows to simulate bumps and roughness of the surface. This texture more accurately renders surface relief compared to bump map texture. It is set by default.

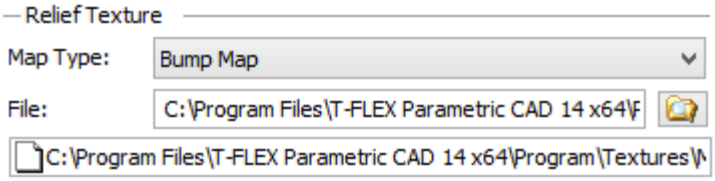


Bump Map. This texture allows you to create the illuminated and shaded areas to simulate simple bumpy surfaces, flat protrusions or dents.



Parallax Map. If you use this texture surface relief becomes three-dimensional and looks differently from various angles.

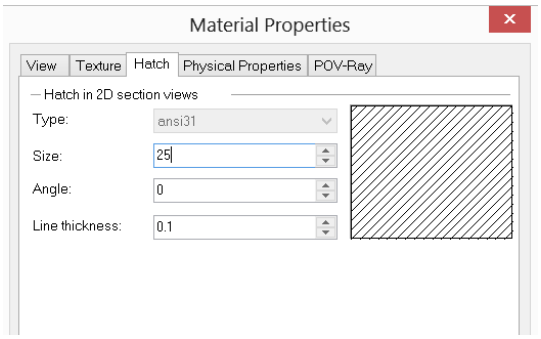




File. File of the image containing the picture of the texture's relief. To specify the texture it is possible to use the files of bmp, gif, jpg, jpeg, tga, tif, tiff and png formats.

«Hatch» tab

When creating sections and cut views in the **«3J: Create 2D projection»** command there is a capability of automatic hatching of the section. Hatching used in the section is defined with the following parameters:



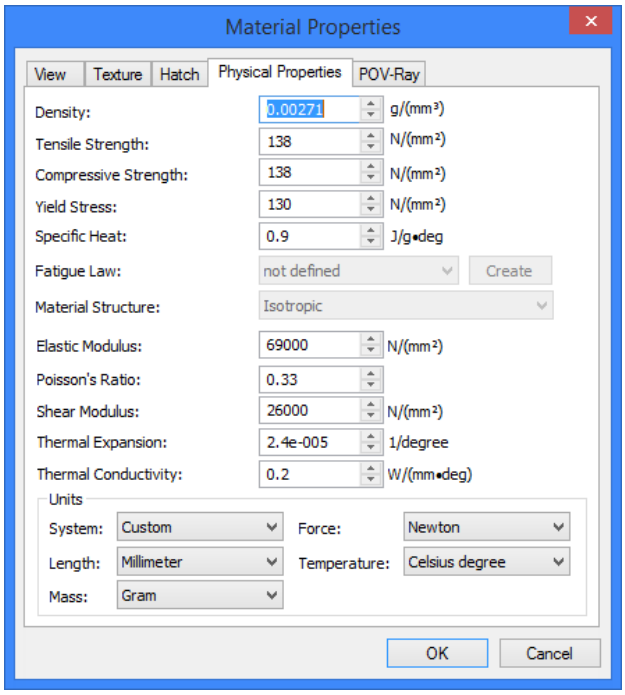
Type. Hatching type.

Size. Distance between the neighboring hatching lines.

Rotation angle. Angle of rotation of hatching lines. When selecting the hatching, the tilt angle of 0 degrees signifies the hatching with horizontal lines

Line thickness. . Parameter allows to specify the thickness of lines.

«Physical properties» tab



In the window of this dialog, you can specify physical-mechanical properties of the material. These parameters are used in calculation of mass-inertial characteristics and also when performing structural calculations in the analysis module.

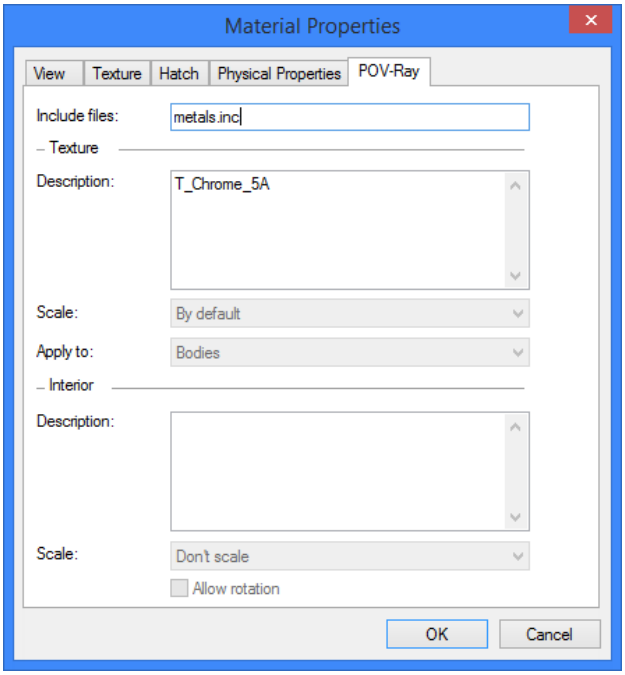
In the upper part of this window, the physical-mechanical properties can be specified. In the lower part of the window, the units of measurement can be selected.

When changing the parameters of the units of measurement, the values of physical-mechanical properties are recalculated automatically.

You can specify the physical-mechanical properties with variables.

For the material that is already used it the model, you can specify properties using graphs. Special icon appears near each of the properties.

«POV-Ray» tab



In the window of this dialog, you can specify parameters that are used upon creation of photorealistic view (see the **Photorealistic view** chapter).

In POV-Ray tab, a large number of characteristics of surfaces and inner space of bodies are used for description of the 3D scene. All materials from the materials libraries supplied with the system already have the POV substitutions (libraries from POV-Ray, for example, textures.inc, are used in them).

In the **Material POV** window, the following parameters can be specified:

Include files. The field of the given parameter shows the files in which available characteristics for the specified material are stored.

"Texture" group

Properties of the body surfaces are specified in this group.

Description. Name of texture that is specified in one of the included files is displayed in this field. Here you can specify custom properties if you know POV-Ray programming language.

Scale. Defines way of displaying for the specified texture, when applied to a body.

Don't scale. The specified texture will repeat without scale changes.

By default. The texture will be stretched on the whole surface.

Use minimum/maximum size. The scale will be maintained by the minimum/maximum size of surface to which the texture is applied.

Apply to:

Bodies. The texture display will look more real. Most of all it refers to wood textures. In this case, slice of the wood fibers will be displayed on one of the body's faces.

Individual faces. All body faces will have similar texture display.

"Interior" group

Interior properties are specified in this group: index of reflection, light dispersion in volume, etc.

Parameters **Description** and **Scale** are similar to the parameters above.

Allow rotation. If this parameter is set, texture, applied to the interior space of a body will be rotated together with the body.

APPLYING MATERIAL ON INDIVIDUAL FACE (FACES)

The command **3AM: Apply Material** allows applying a certain material on one or several model's faces. The applied material will be maintained through the future model modifications despite changes in the face's geometry. The command does not change the model geometry, but it is displayed in the model tree, standing as an independent operation.

The command is convenient for accurate texture positioning using different applying laws and in situations, when you need to dye only selected faces.

The command **3AM: Apply Material** is called via:

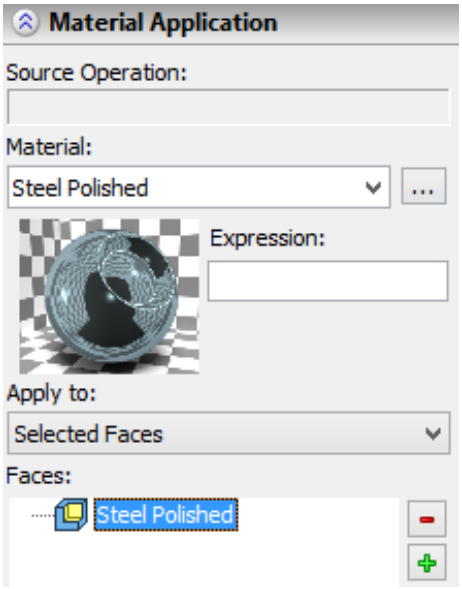
Icon	Ribbon
	3D Model → Modify → Apply Material
Keyboard	Textual Menu
<3AM>	Operation > Apply Material

You can apply material using "drag'n'drop". In this case "Apply Material" operation is automatically created.

Upon entering the command, the automenu provides options that allow selecting of body faces. The following option is automatically enabled:

	<F>	Select Face

You select faces using this option. At the same time, you can only select the faces belonging to a single operation body. The selected faces are entered in the "**Faces**" list in the command's properties window.



Using the **Material Application** parameter, you can specify on what faces the material will be applied: the **Selected Faces** or the **All Except Selected Faces**.

If you plan to assign the same material to several faces of one body, it is recommended to select all of them within one material application operation. This will help avoiding the unnecessary complications to the model structure.

To select all faces of the operation body, you can use the option:

	<O>	Select Operation

When using this option the material is applied to all faces of the selected operation body.

To select a material to apply, you can use field **Material** in the properties window. Material of the current model can be selected from the drop-down list. To open properties window use ... button.

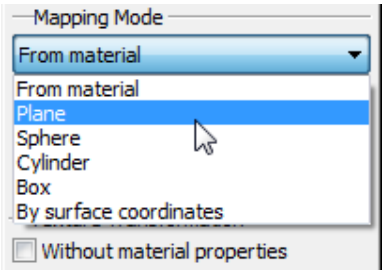
To call the operation parameters use option:

	<P>	Set entity parameters

To cancel selection of all faces, use the option:

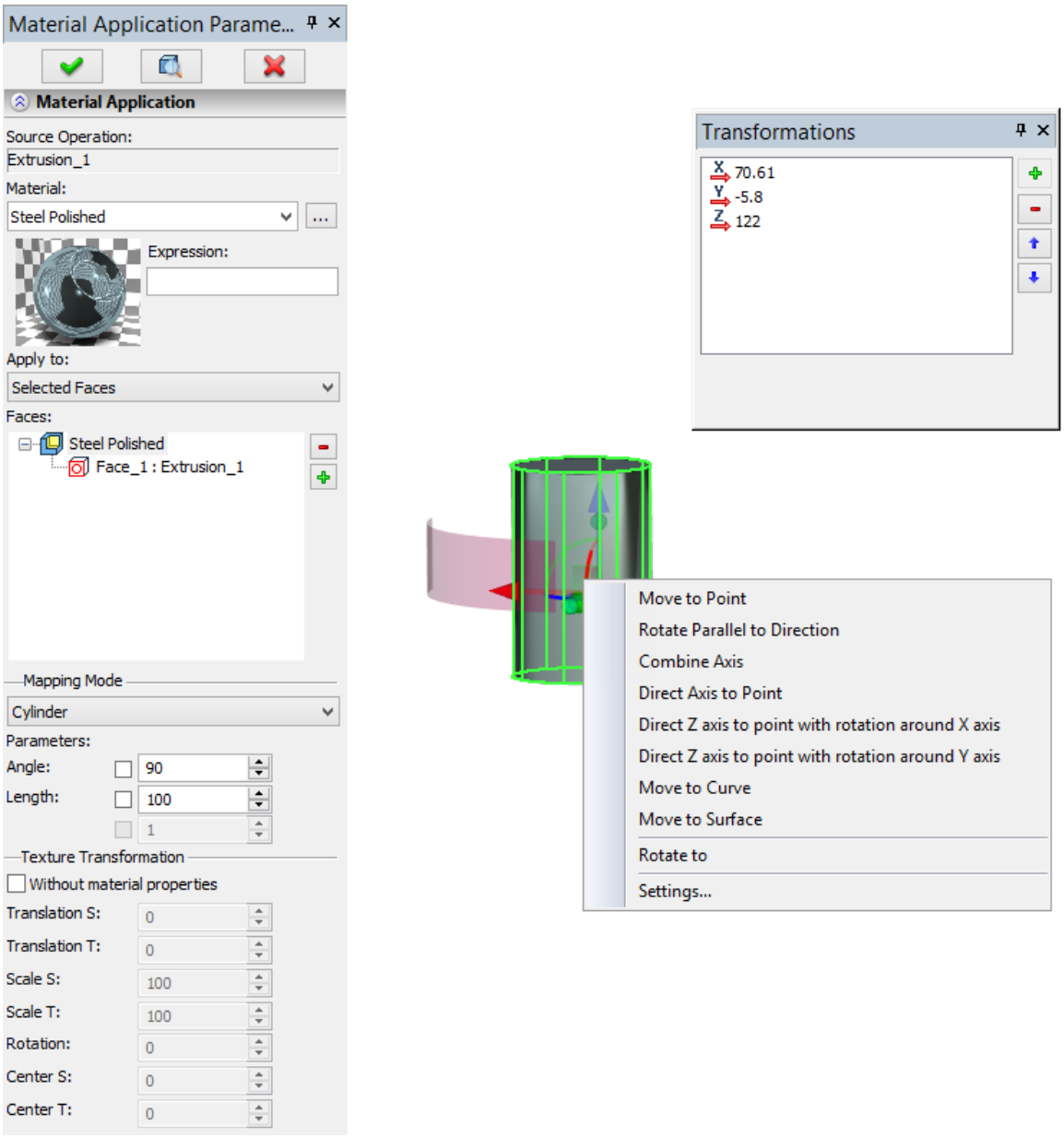
	<K>	Cancel selection of all faces

In the command's properties window, additional parameters of the texture mapping can be specified:



The **Mapping mode** group allows us to select the material mapping mode and the scale of the texture along different axes. In the drop-down list of mapping modes the following options are available:

- ✓ **From material** – all parameters are taken from the properties of the material.
- ✓ **Plane** – mapping is carried out as if the texture is projected along the normal to the plane on which the texture is located, i.e., on the XY-plane of the manipulator.
- ✓ **Sphere** — mapping is carried out as if the texture is located on a segment of the sphere. This option is convenient for mapping textures on spherical and close to spherical surfaces. Parameters of the sphere determine the scale of the texture of the material and are controlled by the manipulator.
- ✓ **Cylinder** – similar to the sphere but the type of the surface is different.
- ✓ **Box** – it is assumed that the textures are mapped on the faces of a parallelepiped and are projected from these faces. For this mapping mode, additional scale along the third coordinate is used. The scales thus define the dimensions of the «template surface».
- ✓ **By surface coordinates** – this mapping is useful for curvilinear surfaces. For this mapping mode, the texture coordinates are associated with internal coordinates of the surface. It can be considered as a generalization of the texture mapping on a sphere or a cylinder. The texture is always displayed on the surface in a unique way, therefore, the manipulators are not used.



To control the mapping of a texture and its accurate positioning, the intuitively clear manipulators are used. The view of a manipulator depends on the selected mode of material mapping. All parts of the manipulator are functional. For each element of the manipulator there is the context menu with various fixings and commands.

More information about manipulators can be found in the chapter **3D Assemblies Creation**.

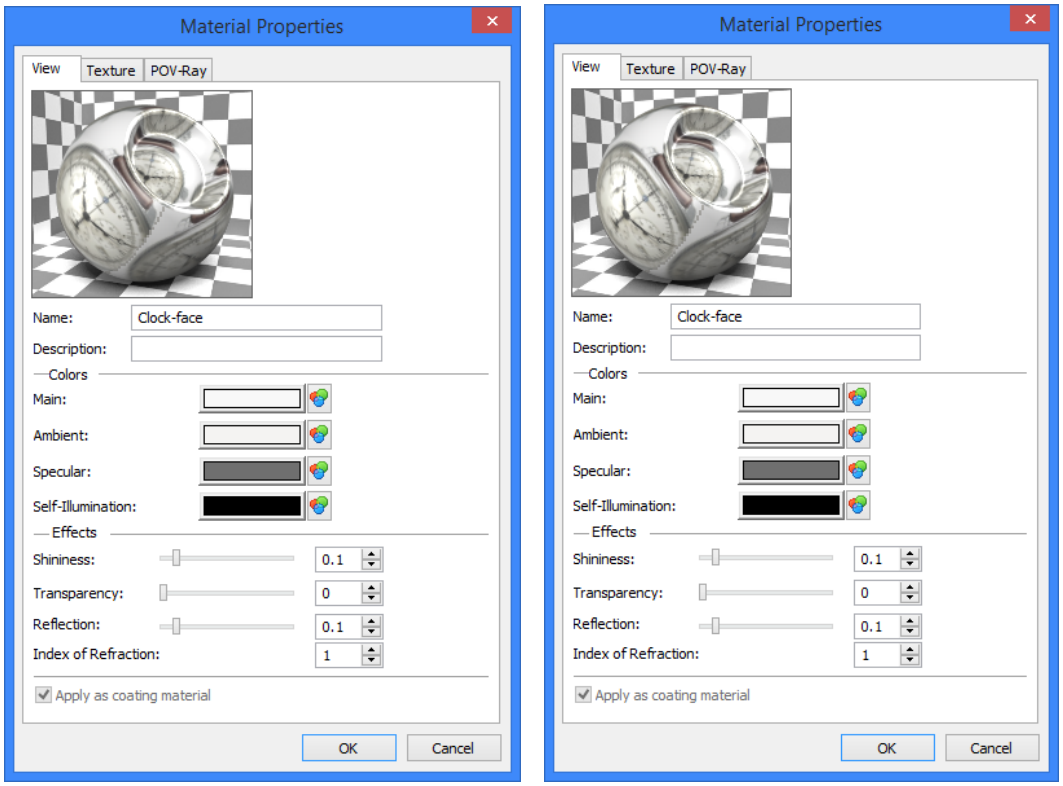
The **Texture transformations** group allows us to specify for the texture the mapping parameters that are different from those specified in the properties of the material itself. The **Without properties of material** flag determines the source for the parameters of the mapping. By default it is disabled and the mapping parameters are taken from the properties of the material. When

the flag is enabled, the user can specify new values of the parameters (**Direction S**, **Direction T** and so on.)

Example of Applying Image to a Face

An image of a wristwatch dial will be applied to the face of a watch body as an example of using "Apply Material" command.

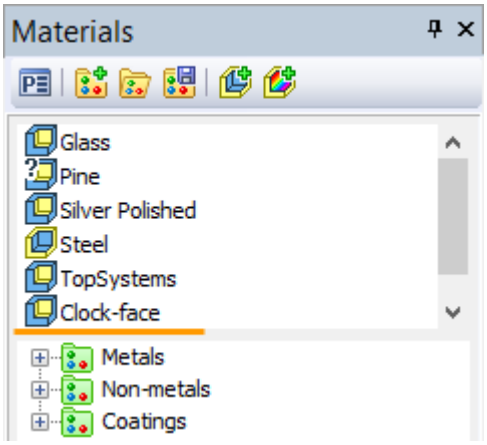
First, you need to create the coating material. Color texture of the material will be specified by image file. In this case coating material named "Clock-face" was created. The image file "Clock-face.jpg" used as the color texture. It is located in the folder: "...\Examples\3D modeling\Apply material"



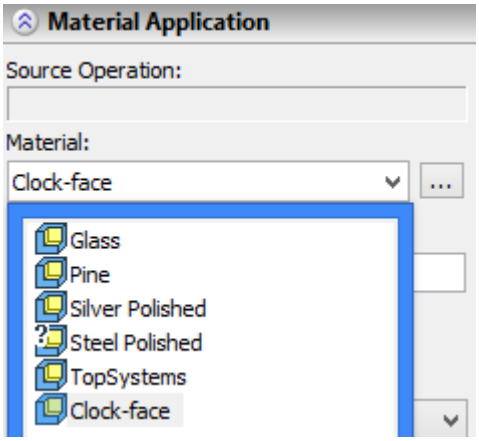
The example is located in library *"Examples\3D modeling\Apply Material\Watch.grb"*.

In order to apply the material "Clock-face" on the selected face of the watch base you should:

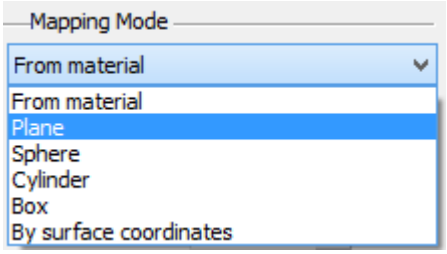
- ✓ Open the file with watch body on which the material will be applied;
- ✓ Add **Clock-face** material to the list of materials of the current model.



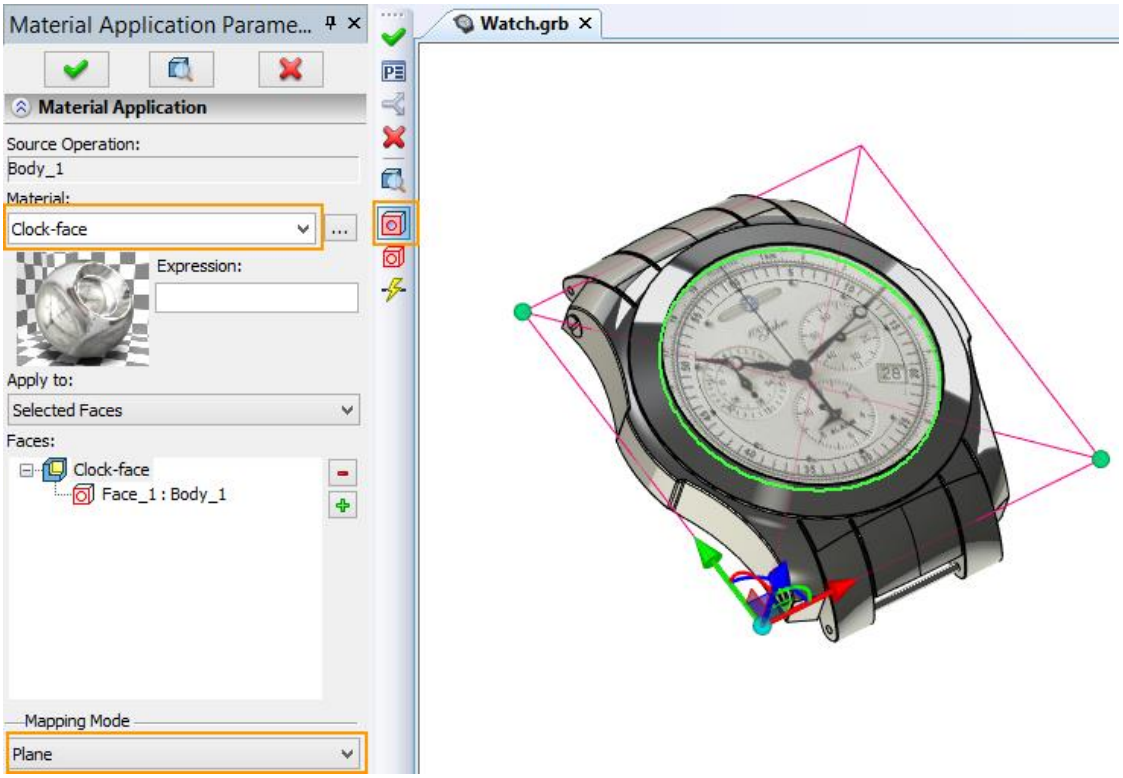
- ✓ Activate command **3AM: Apply Material** ;
- ✓ Select required watch face with the help of option **Select face** ;
- ✓ Select "Clock-face" from drop-down list in **Material Application** window



✓ Select **plane** in **mapping mode** drop-down list.



✓ Specify positioning of material on the face with the help of special manipulators in the 3D scene and the **Texture Transformation** group in properties window.



✓ Press .to finish input.

USAGE OF DRAG'N'DROP AND "COPY/PASTE" COMMANDS TO APPLY MATERIALS

You can edit body, operation and face materials not only from the clipboard with the help of **Copy** and **Paste** commands, but also using drag'n'drop: moving the selected material and "dropping" it to a face or a body .

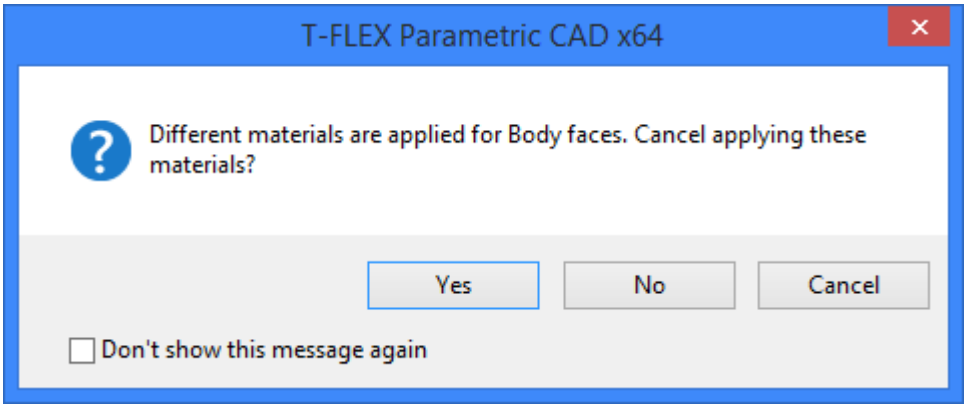
If there are several program instances running on the computer, you can apply materials from one instance to the model of the other instance using drag'n'drop. T-FLEX CAD can use materials from datasets of T-FLEX DOCs application running on the same computer including integration mode.

Copying into clipboard is performed using option **Copy** from the context menu. In **Materials** window materials from the list of the current model material and from the libraries list are copied using item **Copy** from context menu or by pressing <Ctrl>+<C>. The **Copy to Document** command (<Ctrl>+<D>) copies material from the library to the list of the current model material and to the clipboard simultaneously.

When you copy material to the clipboard, name of the material is available for pasting into any text editor.

The material from the clipboard will be applied to the selected objects: bodies and faces, when you use command **Paste**. If there are no objects selected, a dialog window appears. In this window, you can choose applying of the material to all bodies in the scene. If you refuse, the material will be placed in the list of the current model materials.

The system checks existence of coating materials applied to body faces when you use **Copy/Paste** commands or drag'n'drop to apply the material to the operation. If any coating material is already applied to one or several faces, the following dialog window appears:

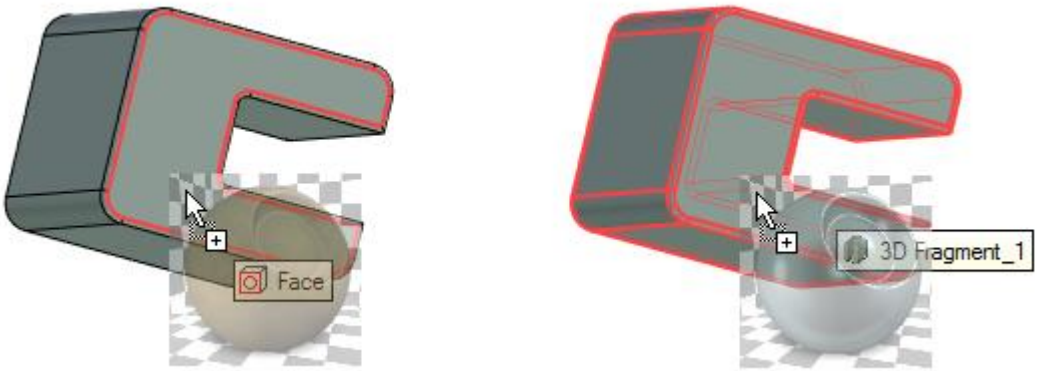


If you press "Yes", all previously applied coating materials will be deleted. If you press "No", the material will be applied only to faces without applied material. Button **Cancel** cancel applying of the coating material.

The coating material can be applied to faces and bodies, even if all filters are disabled.The appropreate filters become active when you work with the coating materials. For convenience, you can disable one of them.

The basic material is always applied to the whole body.

When you use drag'n'drop T-FLEX CAD helps you to select object for material applying. For this purpose selected objects are highlighted with red and tooltips appear near the mouse cursor. In addition, the material preview window appears in the scene.



When you use drag'n'drop, system shows the last operation, which was applied to the body.

LIGHT SOURCES

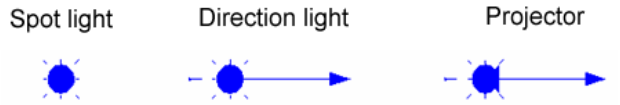
A light source is an element serving for rendering three-dimensional objects and helping controlling the lighting of the three-dimensional scene. Light sources are snapped to 3D nodes. Their directions are also defined by 3D nodes. There are three types of light sources - point, directional and projector.

Spot light source emits the light from one point in all directions over the whole 3D scene.

Directional light source emits the light in a certain direction over the whole 3D scene.

Projector light source emits a light cone from a given point in a certain direction with the specified drop-off rate.

A light source is displayed as a small "radiant" sphere.



3D window rendering uses simplified lighting calculations: the lightness (and, respectively, the color) of the surface depends only on the angle of the incident light to the surface. The effects of shadowing, reflecting, refracting and scattering of light are not present. To create more realistic scene rendering, use the command **3VY: View Ray Tracing** (see the chapter **Ray tracing**).

A light source can be hidden from display while maintaining the lighting effect. By default, lighting of the 3D scene also involves a light source (one or multiple) on the camera, oriented along the viewing direction. The parameters of those light sources are set by the command **3VP: Set View Parameters** (see the chapter **Working with the 3D View Window**).

One scene allows creation of an arbitrary number of light sources. However, the number of light sources affecting the resulting rendering of the 3D window depends on the type of a video card installed on the computer.

Lately, the number of light sources handled by video cards varies from 8 to 16.

CREATING LIGHT SOURCE

To create a light source, use the command **3L: Create Light Source**. The command is called by one of the following ways:

Icon	Ribbon
	View → 3D Model → Light source
Keyboard	Textual Menu
<3L>	Construct > Light source

After calling the command, the following options will appear in the automenu, along with the general system options:

	<1>	Select position of Light Source
	<2>	Select direction of Light Source
	<L>	Point Light Source
	<D>	Directional Light Source
	<S>	Projector

To define a light source, you need to use a special manipulator.

User may also select a 3D node or point of an object in the scene to define position for the light source. For this purpose activate the option .

Use the option to define the direction of light propagation for the directional light source or projector . You should select a 3D node or point of an object in the scene to define the direction.

The initial color of the light source is white.

Point Light Source

Point light source has no direction of light propagation. You should specify only position of the light source for its creation.

You should call the following option for the point light source creation:

	<L>	Point Light Source

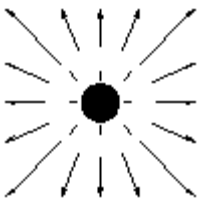
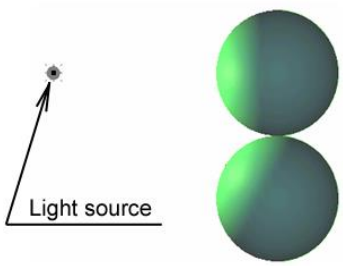


Diagram of light propagation from a spot source



Directional Light Source

To create a directional light source call the option:

	<D>	Directional Light Source

You should specify a position of the light source and direction of the light propagation for a directional light source creation. Then press .

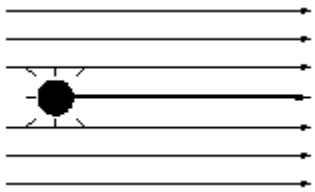
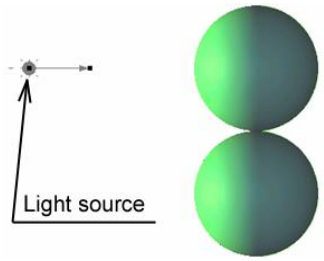


Diagram of light propagation from directional source



Projector

Select the following option for the projector creation:

	<S>	Projector

You should specify a position of the light source and direction of the light propagation for a projector creation. Then press .

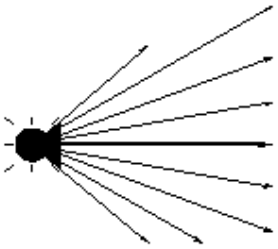
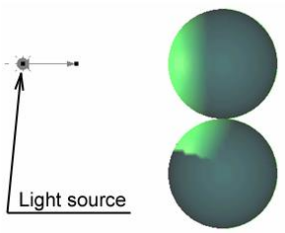


Diagram of propagation of light from projector

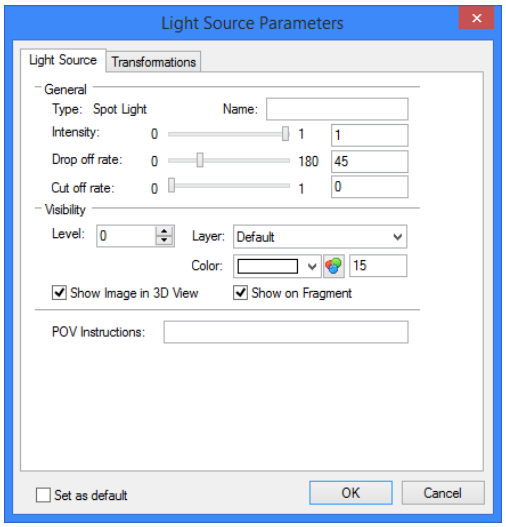


The drawings with sample constructions of all types of light sources can be found in the library *"Examples"* located in the folder: *"Additional resources/Light sources"*.

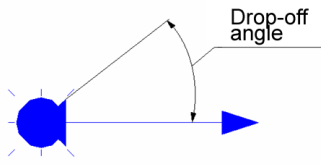
Light Source Parameters

To define parameters of a light source, bring up the dialog box. This is done by the option:

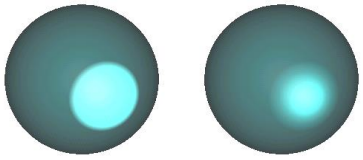
	<P>	Set entity parameters



Type. This parameter indicates the type of the light source (spot, directional, projector). It is defined automatically during the construction.



Intensity. Defines the intensity of the light source. The higher the light intensity, the more it affects the image in the 3D window, and the higher is lightness of the resulting image.**Drop off rate.** Defines the projector's drop-off angle.



Cut off rate. This rate defines the smoothness of transition from light to dark across the boundary of the projector's light spot. The diagram on the right clearly shows the effect of this parameter with the values 0 and 1 respectively.

The "**Visibility**" group of parameters is common across the system functionalities and define the modes of displaying a light source on the screen:

Color. The color of the light source. The light source is drawn in this color in the 3D window, and also emits light in this color.

Show Image in 3D View. Checking this parameter makes visible the image of a light source existing in the 3D scene. If the parameter is cleared, the light source will be hidden, but the lighting effect remain. To select, or modify properties of, a light source made invisible in the 3D window, use the model tree.

Show on Fragment. If a model is used as a 3D fragment, then the light source with this parameter set will be exported into an assembly model.

POV Instructions. You can enter instructions in this parameter's field, that will be appended to the end of the description of the light source upon exporting the given file into POV for creating a photorealistic image.

CAMERAS

A camera is necessary element for visualizing three-dimensional objects, that determines the point and direction of viewing the 3D scene. Each 3D window provides a system camera installed "by default". The user can additionally create one's own cameras and make one of them active.

A camera has its own coordinate system and a set of manipulators. The manipulators allows you to translate the existing cameras around the 3D scene and change their viewing direction. This can be used when examining inner elements of the scene and creating animation clips.

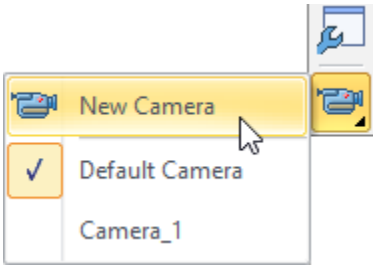
The system camera activates upon opening a new 3D window. Initially, it is located at the distance that permits watching all elements in the 3D scene.

CREATING CAMERAS

To create a camera, use the command **3CA: Construct Camera**. The command is called by one of the following means:

Icon	Ribbon
	View → 3D Model → Select Camera → New Camera
Keyboard	**Textual Menu**
<3CA>	Construct > Camera

Or you can create camera using View toolbar:



The camera creation steps include:

1. Set location and direction of viewing of the camera (you can use manipulators or select one of the existing local coordinate systems.
2. Setting parameters (optional).
3. Confirming camera creation.

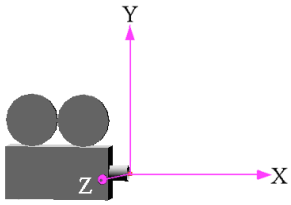
Defining Camera

Upon calling the command, the option automatically sets in the automenu:

	<L>	Select target LCS

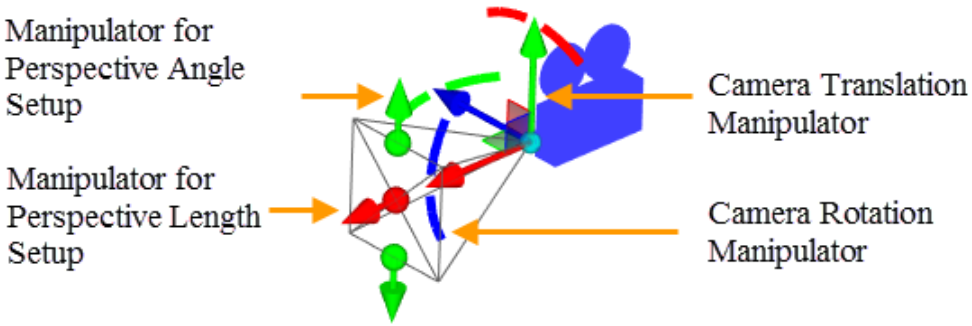
Use this option to select a local coordinate system in the 3D view window.

Once a coordinate system is selected in the 3D view window, a schematic image of the camera will be displayed. The initial camera direction is always along the X-axis, with the Y-axis always directed up, and the Z-axis to the right.

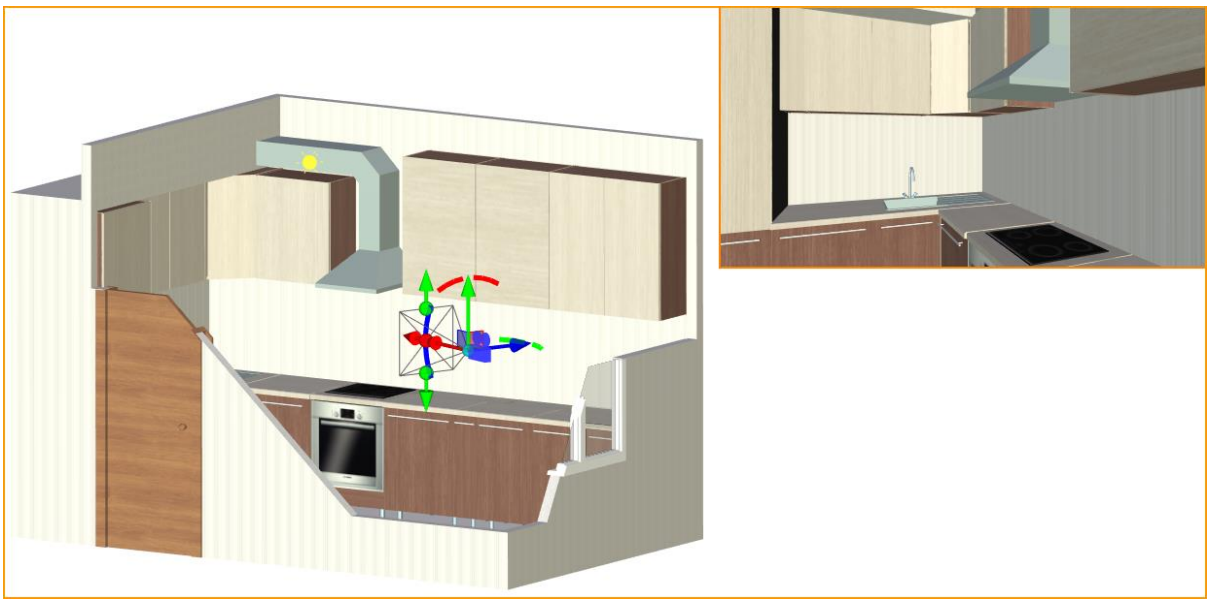


After that, you can set up parameters (if necessary), and then confirm the camera creation by pressing .

If there is no LCS existing in the scene you can specify the camera position using manipulators.



Preview window is displayed in the upper right angle of working window in camera creation mode. It shows the view from the camera in real time.

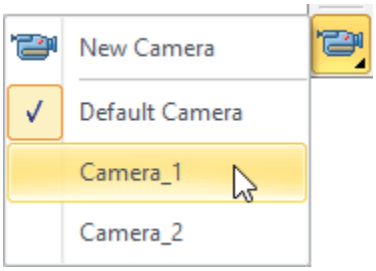


Activating Camera

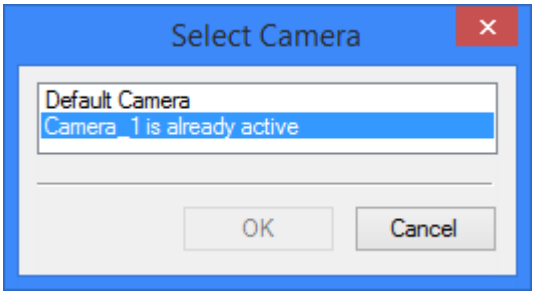
To activate a created camera, use the command **3VC: Select Camera**:

Icon	Ribbon
	View → 3D Model → Select Camera
Keyboard	Textual Menu
<3VC>	View > Camera

Or you can select camera using View toolbar.



Calling this command brings up a dialog box containing a list of all cameras existing in the 3D scene. Select the camera with which the 3D scene will be viewed.



Another way of selecting the active camera is, while in the command waiting mode, moving the pointer over a 3D camera in the 3D window and right clicking, and, after that, in the coming up menu picking the item "Select".

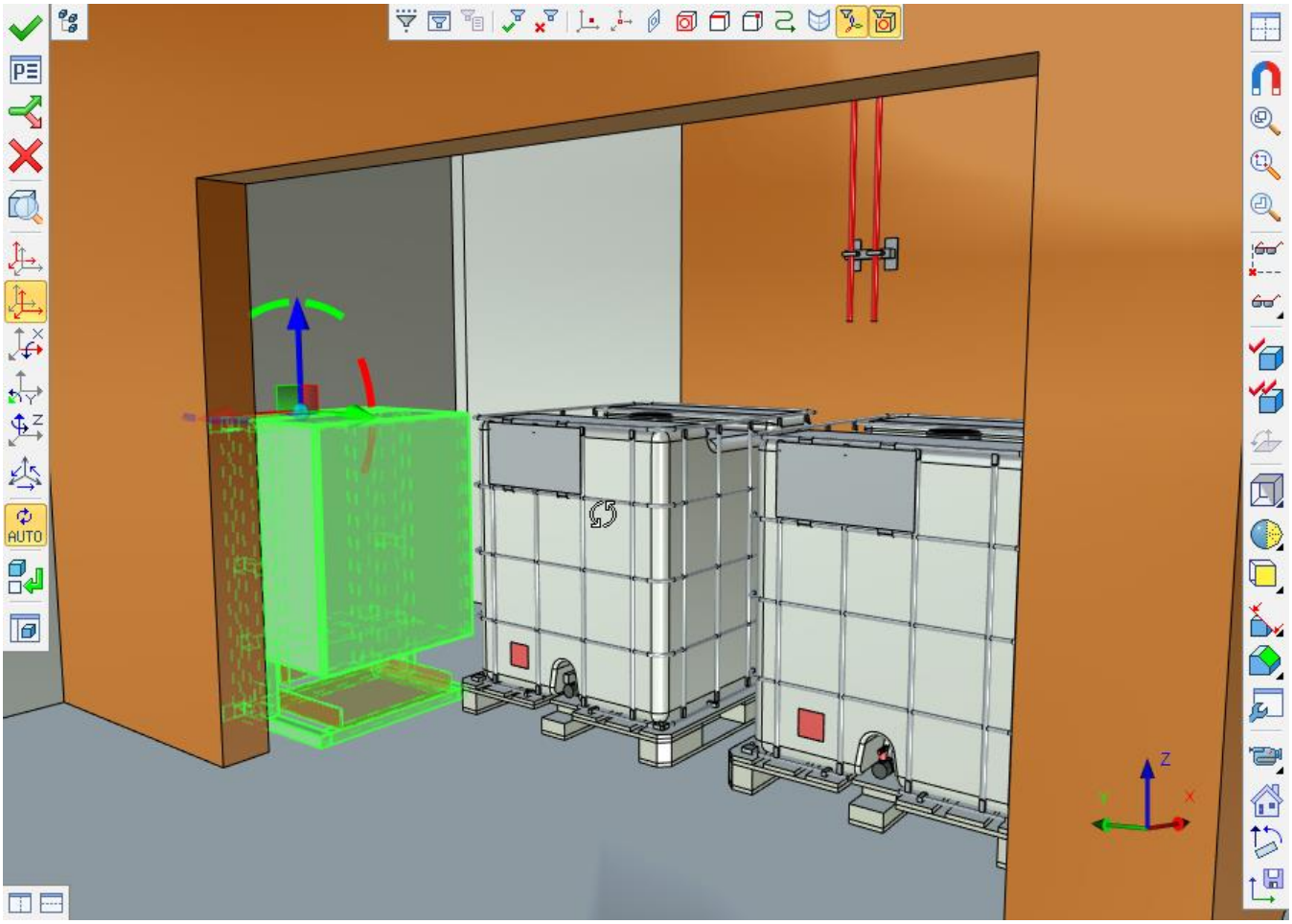
A camera can be selected from the context menu accessed by right clicking over a blank area of the 3D window. Yet another way is to do so on the 3D model tree.

Moving Camera

To watch the 3D scene via a moving camera, you need to make that camera active.

Selected camera mode enables "walking" around the current 3D scene. It is especially convenient when creating floor plans of buildings.

A separate mechanism for cameras movement is used. When camera is active you can move inside designed areas, rotate camera and perform operations as usually in T-FLEX CAD.



If camera is moved inside the solid body in the scene, a wireframe mode is activated for the body.

Controls in active camera mode:

> Note! Controls for the perspective camera are described below. Result of the key combinations usage may differ for the parallel camera, because of the parallel projection properties.

LMB[1] - camera rotation.

> To eliminate rotation around view axis you need to activate **Panorama** mode.

Arrow keys allow to move camera forward/backward.

Page up/Page down keys allow to move camera up/down.

[1] Pressed left mouse button

It is recommended to disable object snap on **View** toolbar upon movement in selected camera mode.

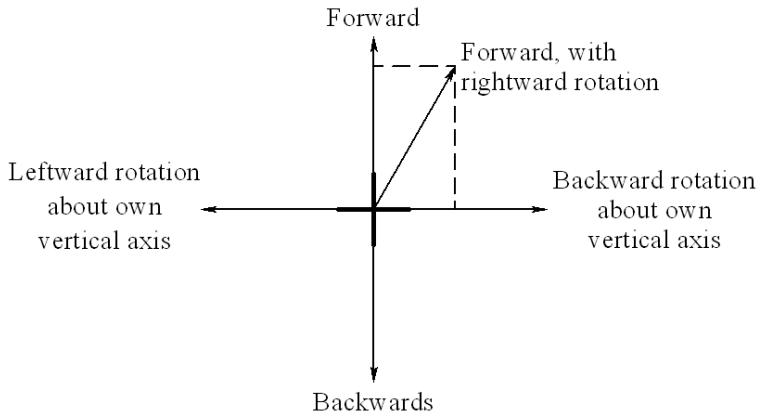
LMB + CTRL – movement in the screen plane.

LMB + SHIFT – zoom.

LMB + ALT – rotation around vertical axis.

Pressed mouse wheel + CTRL – mode of smooth move forward/backward. Movement speed changes according to the cursor position.

A cross will appear on the screen that will become the reference center for the direction vector (the vectors are not displayed). The motion occurs in the following way: moving the pointer by the mouse with the button depressed upward from the cross moves the camera forward. Moving the pointer downward from the cross will move the camera backwards. Moving the pointer in the horizontal direction right or left of the cross will make the camera rotate about its vertical axis right or left, accordingly. Moving the pointer along a diagonal with horizontal and vertical components (see the diagram) will make the camera both move in the respective direction and rotate. Note that, depending on the pointer's bias to one or the other main vector, the translation or else the rotation will dominate to some degree. The farther the pointer is moved away from the cross, the faster the camera will move.

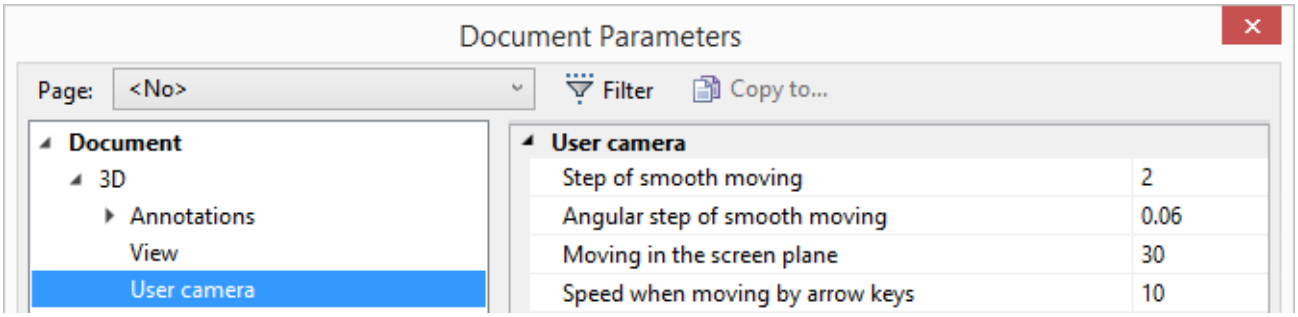


Mouse wheel rotation – zoom: move the camera forward/backward.

Mouse wheel rotation + CTRL – zoom: perspective angle changed without camera movement.

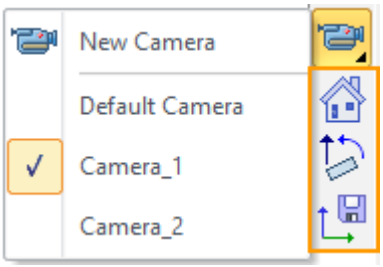
Long press left mouse button (until the cross symbol appears) – orbital rotation around the selected center. The center is selected using the existing in the scene geometry.

Special tab in the **Document Parameters** dialog is used for camera management.



More information about camera management can be found in "T-FLEX CAD User Manual. Fundamentals. 2D Design" book in "Drawing setup" chapter.

Additional options for camera are available on the **View** toolbar after selecting camera as active.



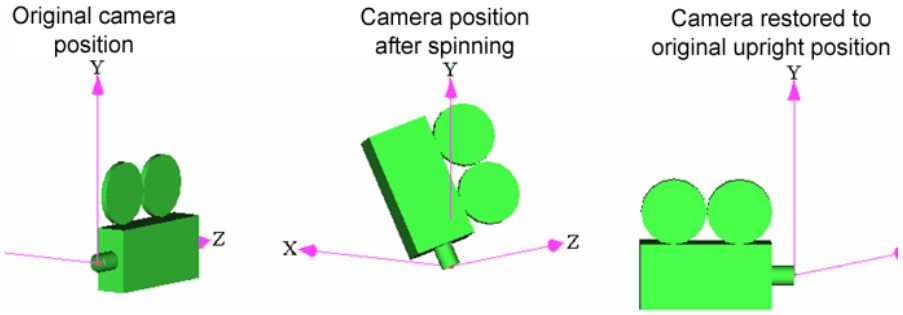
Active Camera Options

Options available when camera is active:

Option allows to restore original/last saved position of the camera.

Option allows to reset vertical orientation of the camera.

This option performs a one-time action. In this way, the values of the coordinates resulting from spinning the camera and setting its vertical position reset to the original values, which are those recorded as a result of the last call of the command.



Option allows to save current position and perspective of the camera. If you didn't activate the option after movement/rotation of the camera, the camera will return to the original position after exiting selected camera mode.

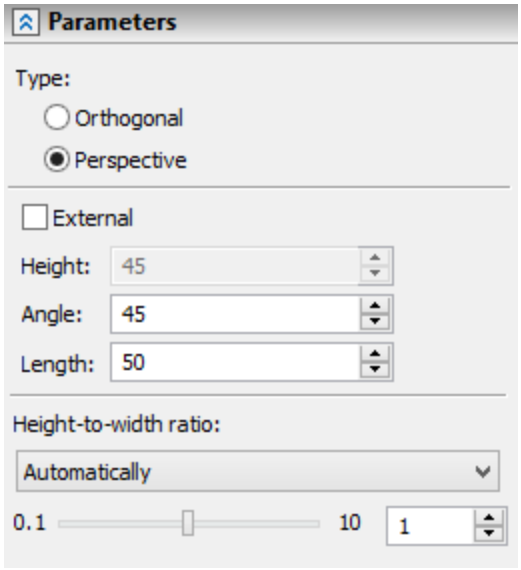
CAMERA PARAMETERS

Parameters section:

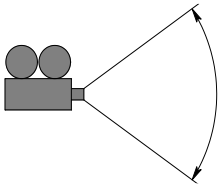
The "Type" parameter defines the method of projecting the objects in the 3D scene when visualizing with the help of the given camera.

The projection can be "Orthogonal" – defining parallel mapping without respect to the distance from the viewpoint, or "Perspective" – defining perspective mapping that accounts for the distortion as the distance between the camera and the watched object grows.

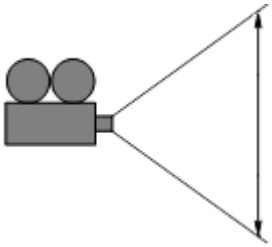
External. If a model is used as a 3D fragment, then the model's cameras with this parameter set will be also available in an assembly. The position of such camera will be defined by the position of the 3D fragment. Parameters of such camera cannot be modified in an assembly.

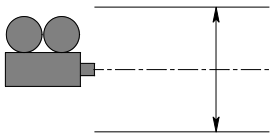


Angle. Defines the watching angle of the perspective projection. The height of the rectangle adjusts with changes in the view angle.

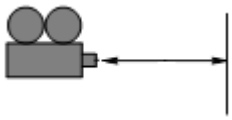


Height. Defines the watching height of the projection. The view angle adjusts with changes in the height.





Length. Defines distance to the projection.



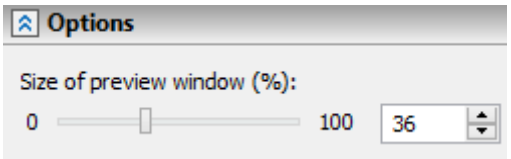
Height-to-width ratio defines projection height to width ratio. You can select a value from the drop-down list or manually.

Height-to-width ratio is applied when you record explode animation.

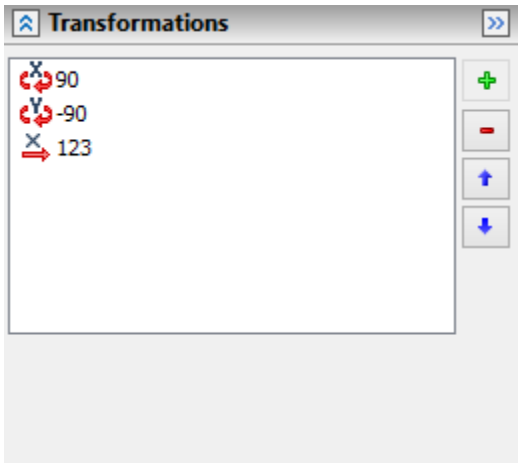
More information about exploded view can be found in "Auxiliary Tools for 3D Assemblies Modeling" chapter.

Options section

Size of preview window. Specifies size of the preview window in the scene.



All camera translations are displayed in Transformations section.



To define additional camera parameters, call the respective dialog box. This is done by the option:

	<P>	Set entity parameters

The “Visibility” group of parameters is system-wide and defines the modes of displaying cameras on the screen:

Show Image in 3D View. Setting this parameter makes the image of an existing camera visible in the 3D scene. If the parameter is not set, the camera will be hidden, but the effect of its presence remain.

PHOTOREALISTIC IMAGES

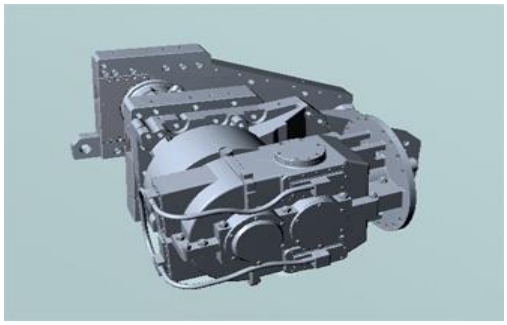
GENERAL CONCEPTS

Ray tracing of the 3D scene is a special way of rendering the scene that accounts for shadows cast by the objects, as well as for such phenomena as light reflection and refraction.

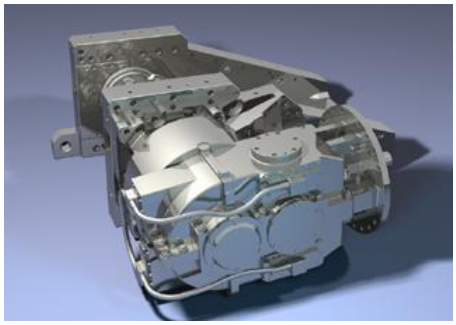
There are three different mechanisms of photorealistic images generation. First of them uses POV-Ray application; the second uses integrated NVIDIA OptiX technology; the third uses Embree – ray tracing kernel developed by Intel.

RAY TRACING

For the first mechanism uses POV-Ray, a ray tracing program which generates images from a text-based scene description exported from T-FLEX CAD. POV-Ray installation is included in T-FLEX CAD installation package or can be downloaded from appropriate web site.



T-FLEX CAD rendering



Ray tracing (POV-Ray) rendering

Note that the POV-Ray application requires separate installation. To install it, select the file "**povwin36.exe**" on the distribution CD, in the folder "**POV-Ray**". POV-Ray installation runs in English. The users who do not know the English language are advised to follow all pre-selected items (such buttons as **[Next]**, **[Yes]** or **[I Agree]**) in the subsequently coming windows of the wizard.

When creating a ray-traced image, the 3D scene is exported into POV format, using the settings of the current 3D window. Next, the POV-Ray application is automatically launched for generating the photorendering. Once generated, the resulting image can be viewed in the View window and, if desired, saved in a file.

When exporting into POV-Ray, the textures are mapped onto the objects in the same way as they are displayed in the 3D window of T-FLEX CAD. Moreover, along with POV-Ray it is possible to use the textures of all formats supported by POV-Ray (gif, tga, iff, ppm, pgm, png, jpeg, tiff, sys).

POV-Ray processing runs parallel to other systems; therefore, you can continue working with T-FLEX CAD after launching this application. However, depending on the complexity of the image being generated, POV-Ray may consume more resources, and so T-FLEX CAD operation could be delayed.

Command Operation

To create a ray-traced image, use the command **3VY: Ray Tracing**. This command will be available if the 3D window is active. Before calling the command, set the 3D scene in the desired orientation, apply the desired materials to the operations, place light sources (one can use the light sources on the camera). Prefer the perspective projection when creating a ray-traced image.

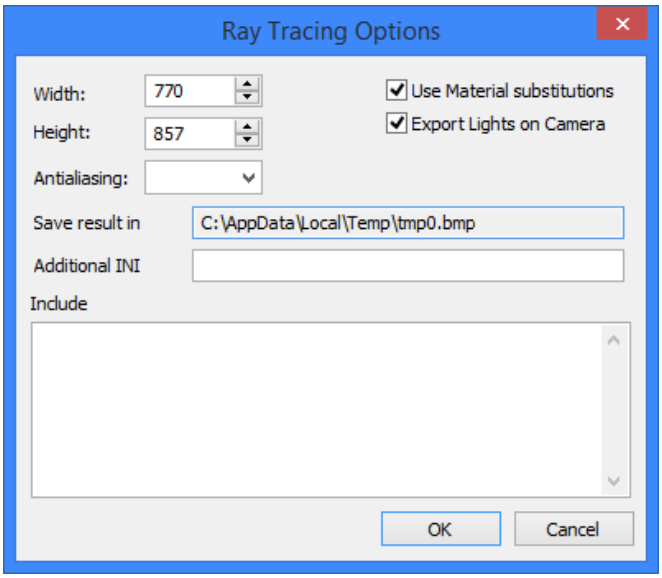
The command is called as follows:

Icon	Ribbon
	Tools → Title block → Photorealistic view → Ray Tracing (POV-Ray)
Keyboard	Textual Menu
<3VY >	Tools > Ray Tracing (POV-Ray)

T-FLEX CAD saves the information about the location of POV-Ray application and checks its availability upon each call to this application.

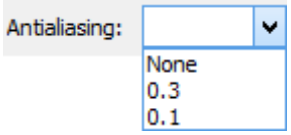
The first time POV-Ray is called, as well as in the cases, when the system cannot find this application, T-FLEX CAD requests the path to the application. In such a case, a dialog box is displayed that allows defining the path to the POV-Ray application. Normally, the path to this application is: "Program Files\POV-Ray for Windows v3.6\bin". If this folder is missing, the application may not be installed (refer to the topic "General concepts").

Upon calling the command, the following dialog box will be displayed.



Width and Height. These define the width and height of the ray-traced image being created, in pixels. The default size is that of the current 3D window.

Antialiasing. Provides smooth color transition in the generated image. The value of this parameter must be greater than 0.



The smaller is this value, the softer will appear the transition from one color to another; however, the rendering (image generation) will take more time in this case. The value of this parameter can be selected from the list or specified manually.

Use Material substitutions. POV-Ray uses a special language for describing the 3D scene. It allows specifying a large variety of properties for material surfaces and insides. Due to this, T-FLEX CAD uses special instructions for materials, determining, how a material would appear when visualized in POV-Ray (the command **3MT: Edited Materials**, the button **[POV Material])**. When the **Use Material substitutions** flag is set, those instructions are passed to POV-Ray. All materials included in the system distribution possess special instructions for POV-Ray. Besides the materials, the light sources and additional instructions for light sources will also be exported in POV (see Light source parameters, the item "POV Instructions").

If the **Use Material substitutions** flag is off, then the instructions will be passed on to POV-Ray, that are automatically generated by T-FLEX CAD based on such material properties as color and specularity.

Export Lights on Camera. By default, one or more light sources are attached to a camera in the 3D window. These light sources have specified orientation with respect to the camera and move together with it (see the manual on "3D View Parameters"). If the flag **Export Light on Camera** is on, those light sources are passed to POV-Ray.

Save result in. This item displays the path to the temporarily created output file that used by POV-Ray for setting the resulting image in the .bmp format, and by T-FLEX CAD for reading it. Therefore, if T-FLEX CAD is closed before the result is created, the image in this file can be viewed later, using any other program for viewing images.

> All temporary files created during the image generation are stored in the folder pointed at by the system variable TEMP. Upon completing the image, all files except the output one are deleted. The output file itself will remain in this folder up until creation of another ray tracing.

Information for advanced users of POV-Ray

Additional INI file: When the POV-Ray application is started, a file is created with the extension ini, recording the exported settings. If necessary, different settings can be specified in this file, even redefining those generated in T-FLEX CAD. Should this be the case, enter the desired filename in this input box of the dialog.

Include statements: You can type expressions in this pane, in the POV format, that will be inserted in the exported file.

Clarification: When launching the command, a file in the POV format is created that has the following structure:

<generated variables>

<included expressions>

*<exported 3D scene>***.**

Generated variables.

The following variables are included in the exported file:

fAspectRatio – Width / height screen ratio. When redefining the Width and Height settings in the additional INI file, you need to also redefine this variable by using <included expressions>.

vSceneMin and vSceneMax – the corners of the box outlining the 3D scene in the 3D space.

vSceneCenter – the center of the box.

fSceneSize – the length of the box's diagonal.

vCameraPos – the camera's position.

vCamera2Scene – the vector from vCameraPos to the box center.

fCamera2Scene – the vCamera2Scene vector length.

cBackColor – the background color.

These variables can be redefined or used in <included statements>. Example:

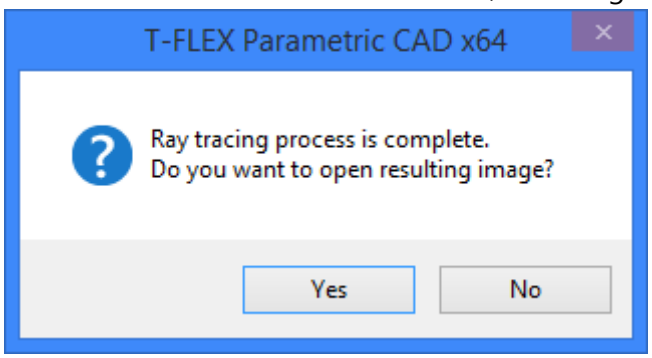
```
#declare cBackColor <0.1, 0.1, 0.1>
fog {
        fog_type 2
        distance fCamera2Scene / 2
        rgb <0, 0, 1>
        fog_offset vSceneMin . z
        fog_alt (vSceneMax . z -  vSceneMin . z) / 4
        up <0, 0, 1>
}
```

redefines the background color and sets blue fog that depends on the orientation and size of the 3D scene.

Upon defining all necessary parameters for creating a ray tracing, press the button **[OK].** Sometimes, when running POV-Ray, the "About POV-Ray(tm) for Windows" dialog box may pop up. In such a case, to run the application, simply press the button **[OK]**.

When creating an animation with photo rendering turned on by using the command **AN: Animate Model**, you are advised to wait until the first frame visualization starts in POV-Ray, to make sure that the "About POV-Ray(tm) for Windows" window does not pop up and does not interfere with the animation creation.

Once POV-Ray is launched, control returns to T-FLEX CAD so that you can continue working with it. Upon completing the image generation or in the case of its termination, a message is displayed on the screen:



To view the resulting image, press the button **[Yes]**. As a result, the view window opens allowing to save the image in a file. If viewing and saving the resulting image is not necessary, press the button **[No]**. In this case, the resulting ray tracing will remain in the system folder TEMP (until the creation of another ray tracing).

> POV-Ray can be started again before completing an image generation (the number of starts is not limited). In such a case, T-FLEX CAD exports data in POV, and then, once the process of the previous image generation finishes, will launch another POV-Ray application. This creates a queue of image generation tasks, with a new task launched upon completing the previous one.

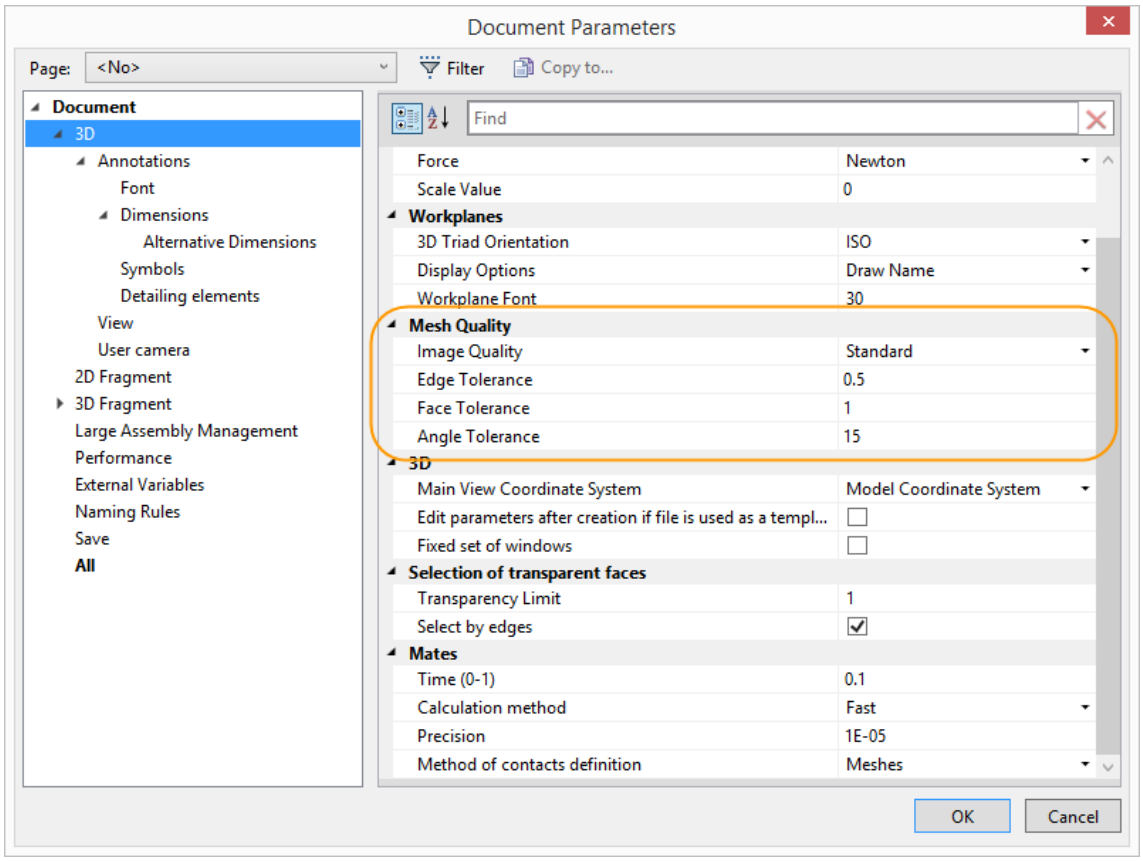
Selection and Settings of Image Quality

It is rather unlikely that a satisfactory photorealistic image will be created on the first try. Usually, several test photo renderings are required that help adjusting the camera position, brightness and position of the light sources, and checking the animation correctness. After that, the final rendering is run.

However, photo rendering could take unspecified time, depending on the scene complexity and the parameters defining the image quality. Understanding these parameters helps avoiding excessive time spending on test renderings, on one hand, and helps achieving higher quality of the final image, on the other hand.

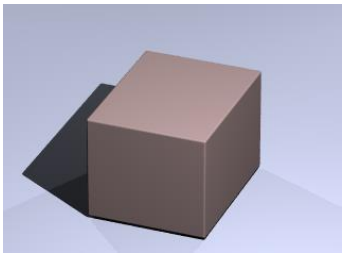
There are various parameters for adjusting the quality of the resulting ray-traced image.

Mesh density. This parameter is defined in the document parameters (the command "**ST: Set Document parameters**"), and, besides the photo rendering, affects the quality of rendering objects in the 3D window.

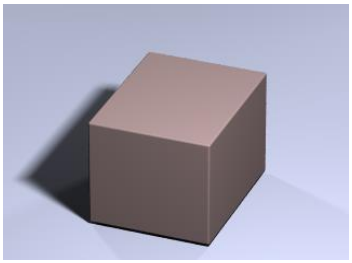


The greater this parameter is, the longer time it takes for exporting the scene by POV, the more RAM is used by POV-Ray and the longer POV-Ray runs the preprocessing of the scene before the rendering (Parsing). Due to this fact, we recommend decreasing the mesh density when running a test rendering, sometimes even up to the minimum setting. When running the final rendering, best is to use the maximum mesh density.

Light source parameters. When creating a photorealistic image by using conventional light sources, the objects cast well-defined shadows because the light sources are infinitely small. In the reality, this happens very seldom; therefore, shadows are often blurred. The use of the diffused light sources allows making shadows smoother and enhances the image quality and realism. The diffused light sources use multiple concentrated light sources offset from each other, instead of one such light source. The more those are offset, the less sharp will be the shadow. The more concentrated light sources are included in a diffused light source, the more the shadow is blurred and the longer time it takes for the rendering.

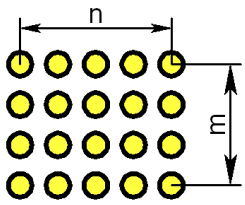


Conventional light



Diffused light

A diffused light source in POV-Ray is made of multiple concentrated light sources.



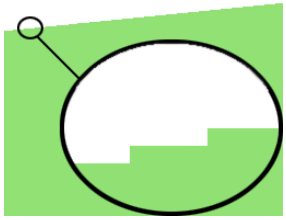
These light sources are placed in a rectangular array oriented in a certain way with respect to the specified center. The number of light sources along each of the rectangle's sides can be different.

To make a light source created in T-FLEX CAD a diffused light source in POV- Ray, enter the following in the "POV Instructions" input box for the light source properties:

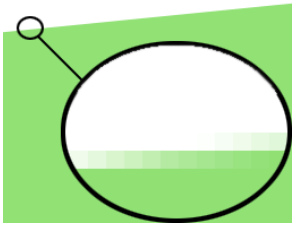
area_light <0.035, 0, 0>, <0, 0.035, 0.035>, 5, 5 adaptive 1 jitter

Above, the coordinates of the opposite corners of the rectangle are entered in the angle brackets with respect to the origin point (the point where the diffused light source is positioned). "5, 5" - is the number of light sources in each direction. In this way, the total number of the concentrated light sources is equal to 5x5=25. "adaptive 1 jitter" - are the additional parameters directing optimization of shadow calculation.

Antialiasing. In conventional rendering, the phenomena of jagging and dashing of thin lines could be encountered. Smoothing by additional calculations may reduce the negative impact of such phenomena.



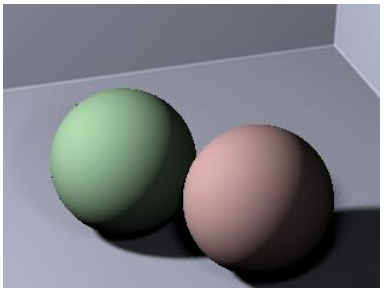
Jagged edges



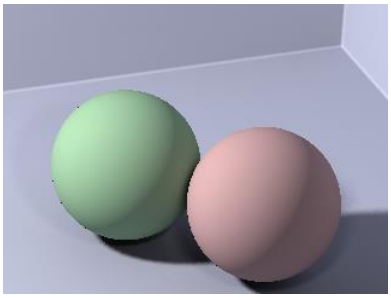
Antialiasing is on

Antialiasing involves increasing the resolution when visualizing some parts of the scene. This slows down the scene rendering. Therefore, avoid using antialiasing in test renderings. However, antialiasing is preferable in the final rendering.

Radiosity. In conventional renderings, only the direct light is accounted for, that illuminates only the objects directly lightened by the light source. However, in the real world, the light originates not only from the sources. It is also reflected from the objects illuminated by the direct light. POV-Ray provides an option of turning on the method of calculating radiosity, that in certain cases enhances the image realism.



Conventional light



Radiosity

Due to bulky additional calculations, use of radiosity mechanism may cause significant slowdown of rendering. Therefore, use low resolution when including radiosity in test renderings.

To turn on the radiosity mechanism, enter the following expression in the **"Include statements"** input box of the **"Ray Tracing Options"** window:

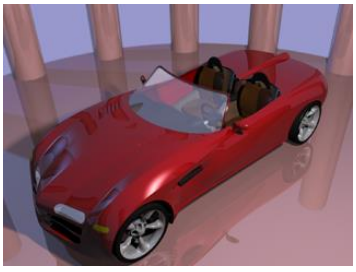
global_settings {

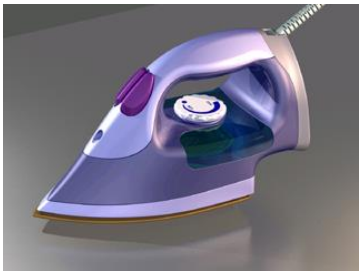
radiosity { count 500 minimum_reuse 0.018 brightness 0.8}}

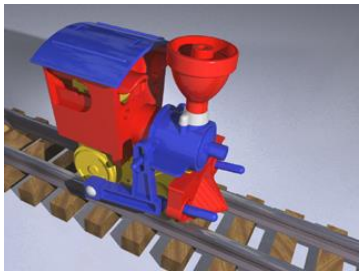
Refer to the POV-Ray application manual for the meaning of the quoted instructions, as well as for additional information regarding the radiosity mechanism.

Image resolution. This parameter noticeably affects the time used for rendering. With the image quality defined, the speed of rendering is directly proportional to the size of the resulting image. You can restrict test renderings to small resolution, for example, 320*240.

Examples of Ray-Traced Images of T-FLEX CAD Models







PHOTOREALISTIC VIEW

The second mechanism for generation of the photorealistic images uses NVIDIA OptiX technology. It is used for creating of high quality images based on lighting and material properties such as transparency, refractive index, surface properties, etc.

The mechanism allows obtaining photorealistic images directly from T-FLEX CAD environment, providing convenient interface of 3D scene parameters management, control over the quality of result and possibility to print result or save it into a file.

With this mechanism, photorealistic image can be generated not only from three-dimensional solid models, but also from imported 3D models.

> The NVIDIA OptiX technology is used for creation of photorealistic videos when you record 3D model exploded view animation in command **3VX: Exploded View**.

The third mechanism for generation of the photorealistic images uses Embree – ray tracing kernel developed by Intel.

Embree uses CPU for its calculations and provides high performance and image quality.

> User interface for NVIDIA OptiX is similar to the Embree. They will be described together in the following chapters

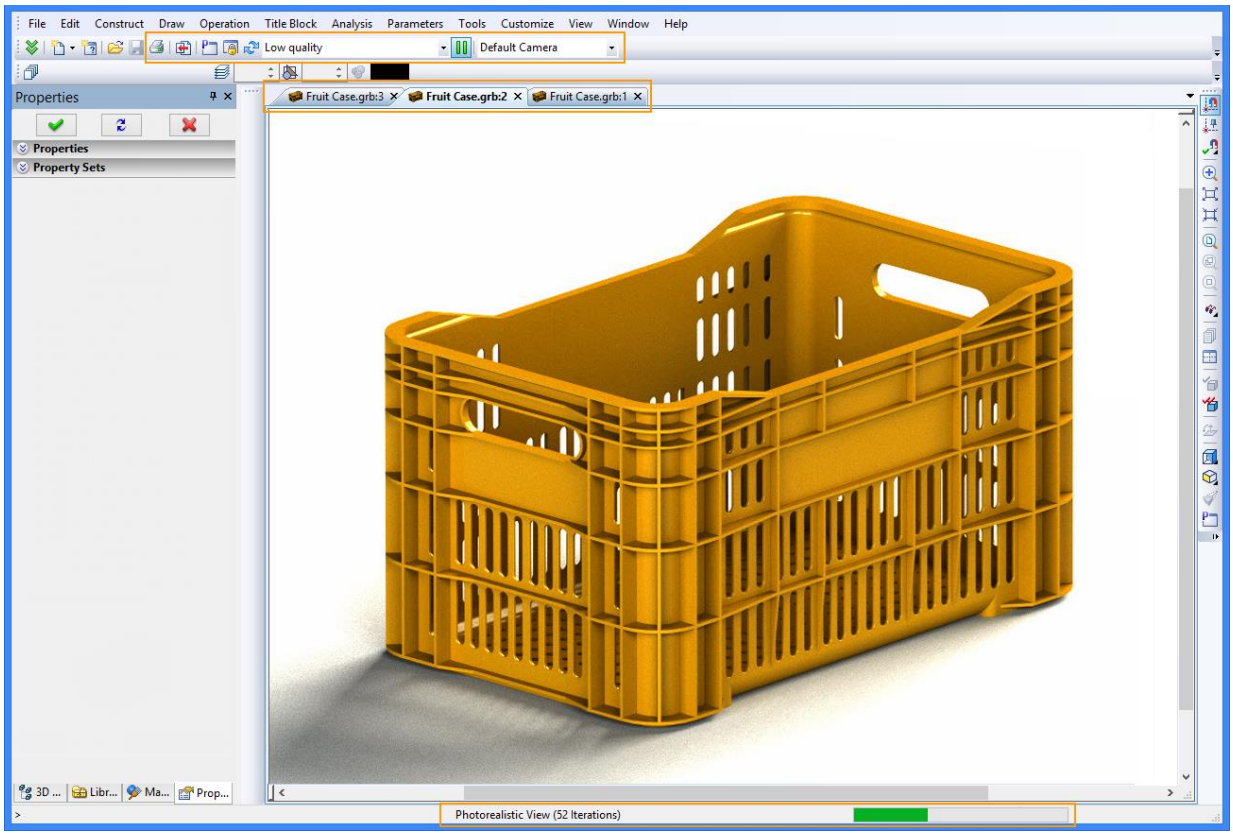
Command of Photorealistic View

To activate the command use:

Icon	Ribbon
	Tools → Title block → Photorealistic view → Photorealistic view (GPU NVIDIA)
Keyboard	Textual Menu
<3RV>	Tools > Photorealistic view (GPU NVIDIA)

Icon	Ribbon
	Tools → Title block → Photorealistic view → Photorealistic view (CPU Intel)
Keyboard	Textual Menu
	Tools > Photorealistic view (CPU Intel)

After activation new window with generated image appears.



Quality of the generated image depends on the number of iterations. The iteration is evaluation of image pixels color. The evaluation process depends on image size, mesh quality and number of objects.

The iteration number is displayed in the lower part of the window.

Process of the image generation can take from several minutes to several hours according to your computer power, model complexity and image quality.

Options for the operation are displayed on toolbar.

Print image. Allows to print generated image.

Save image. Allows to save generated image in *.bmp, *.jpg, *gif, *tiff, *tif, *.png, *.tga formats. You can set the file name and specify folder.

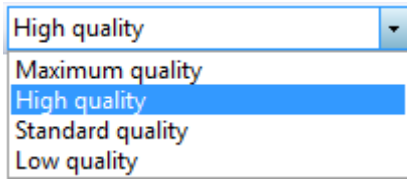
View parameters. Allows defining the image generation parameters. More information about it can be found below.

Fix generation parameters. Allows fixing the image view and scale. Model rotation becomes unavailable.

Update generation of photorealistic view. Restarts photorealistic view generation. All current results are cancelled.

Pause photorealistic view generation. Allows to pause generation of the image. This option stops calculations and releases computer resources.

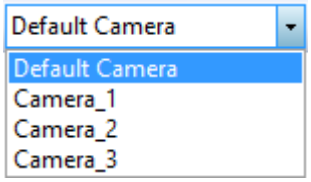
Image quality. You can set one of four kinds of quality from the drop-down list.



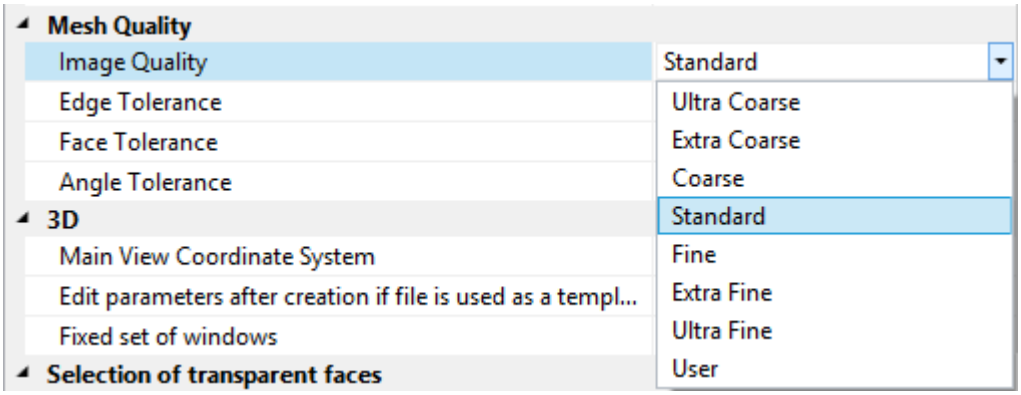
The **low** and **standard** qualities are used for rough drafts of images. When one of them is set, the minimum quantity of iterations for generation of the image with certain level of "noise" is evaluated by the system.

To generate the most realistic images you need to choose **high** or **maximum** quality. When the quality set to **maximum**, the number of iterations is not limited.

Select of current active camera. Allows to select one of the cameras existing in scene. Image will be generated according to the camera location.



An important role in the photorealistic image creation plays **Mesh Quality** configuration. You can change it with the drop-down list on the **3D** tab in the **ST: Set Document Parameters** window.



The higher the quality the higher is the density of the mesh. To create the most realistic images, it is recommended to set "**Fine**" or higher quality.

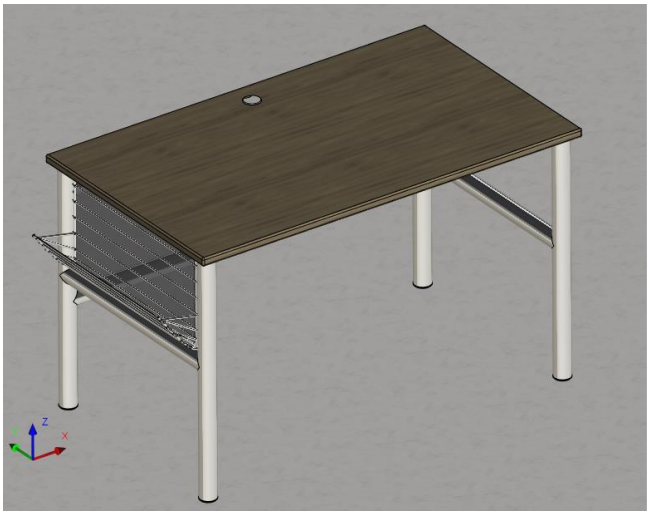
This configuration is especially important for models with rounded faces.

Difference between images with different quality.

Ultra Coarse | Standard | Ultra Fine

Generation of photorealistic images requires high computer characteristics. More information about requirements can be found on our website or in the **Getting started** chapter.

The image generation can be paused anytime. Its result can be saved to disk with option or printed with option .

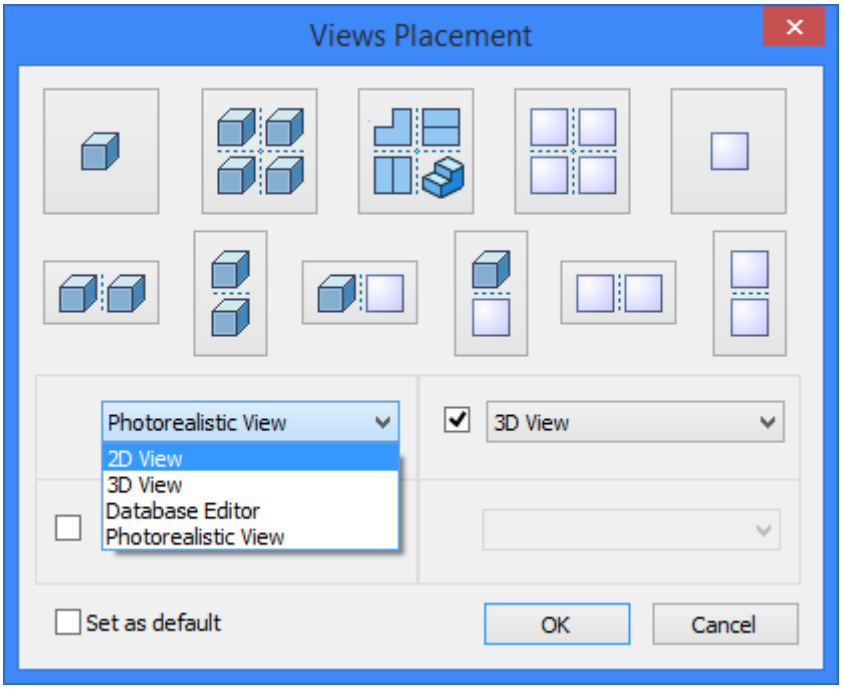


Source model

Photorealistic image

Examples of photorealistic image generation can be found in library "Examples/Additional resources\materials and photorealistic view.

The photorealistic image and 3D model windows can be displayed simultaneously. For this purpose use command **WO: New Window.**



In the dialog window select **Photorealistic View** from the drop-down list. You can set convenient windows placement with the help of four drop-down lists.

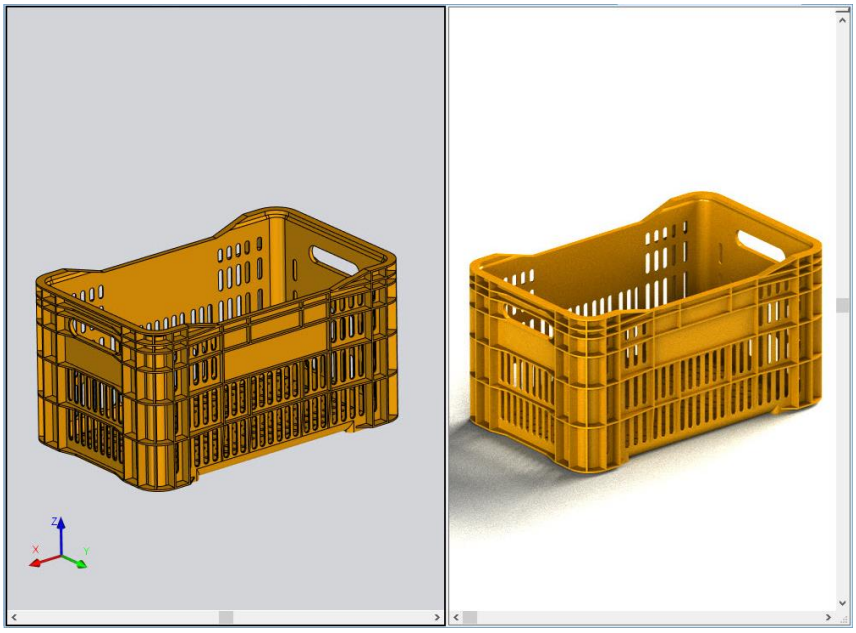


Image Parameters

Fit to window. The option is enabled only when the flag “Use current view size” is set. When the option is active, image of the current size is fully displayed.

Use current view size. When active, the option allows to set the generated image size. After activation, option **Fix generation parameters** on the toolbar automatically activates.

Size of the image is specified in pixels. The image of specified size will be generated completely, even if it does not fit the window. To generate images of the highest quality it is recommended to set the largest current view size.

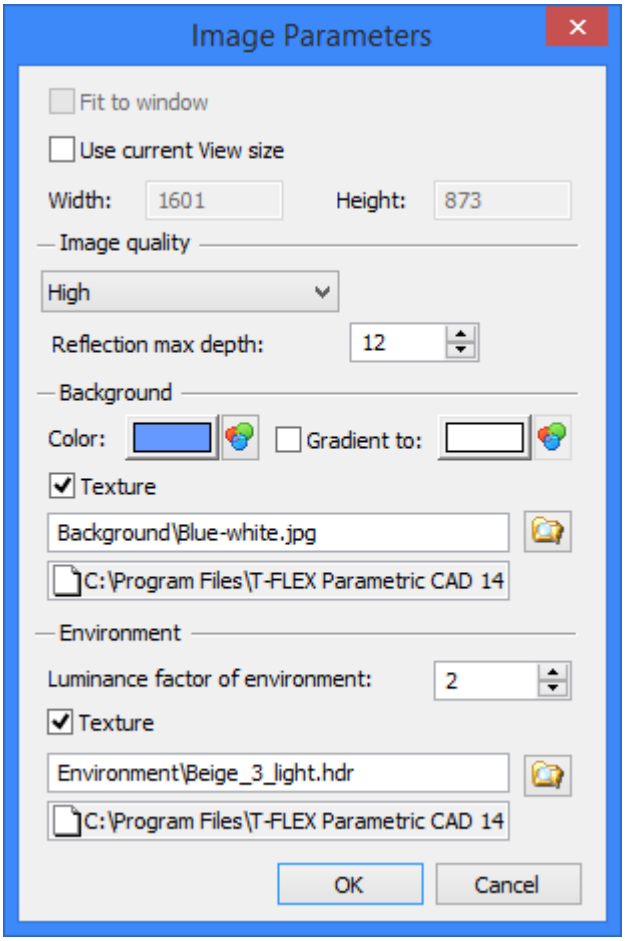
Image quality. The option duplicates the drop-down list of settings from the toolbar. The only difference is the possibility to set the number of iterations manualy. You need to select user quality and specify the number of iterations.

Reflection max depth. The option is important for generation of refractions and reflections.

All texture parameters are similar to the standard parameters of 3D view.

> More information about them can be found in chapter “Working with the 3D view window”.

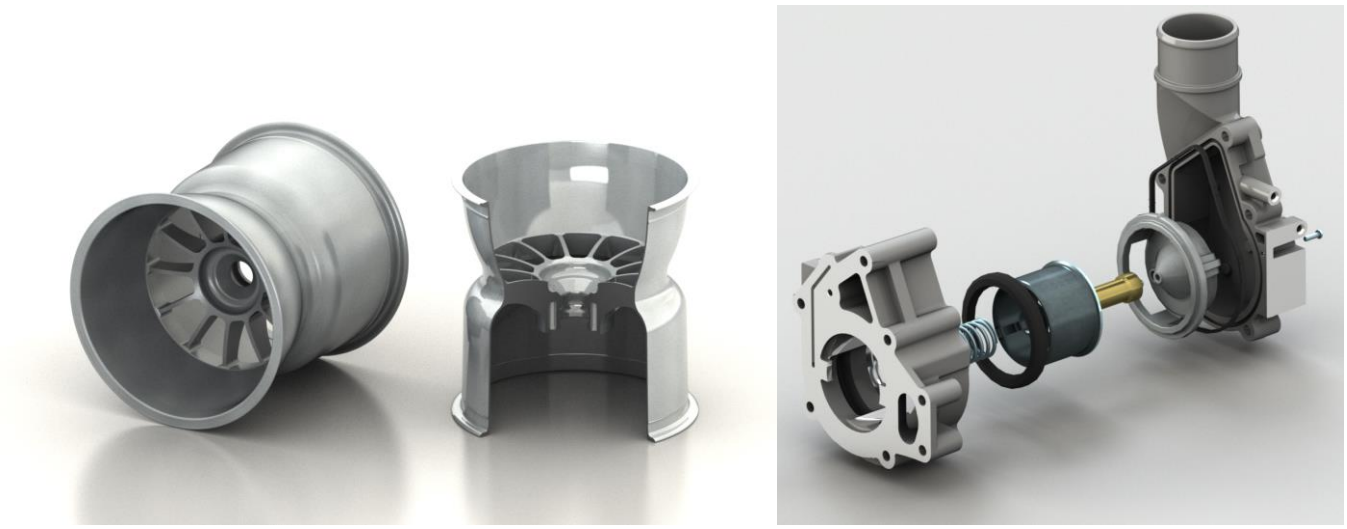
Luminance factor of environment. Allows to set luminance by adjusting the amount of light coming to objects.



Photorealistic Images Examples

NVIDIA Optix:

Embree:



PHOTORENDERING PROTOTYPES

The standard distribution includes the prototypes designed specifically for quick creation of ray tracings. To create documents based on such prototypes, call the command **FP: Create New Document Based on Prototype**, and on the tab **Photorendering** select one of the two prototypes: **Room** or **Flying around object**.

Each of these prototypes has several predefined light sources, a camera and a coordinate system for fixing a 3D fragment. The placement of these elements can be modified as desired by moving the respective elements in the drawing window. Additionally, the 2D window provides a brief instruction on using the prototype.

Normally, you work with these prototypes as follows. Create a new document based on one of the prototypes. Insert a 3D fragment in this document as the 3D fragment or 3D picture (using the appropriate scale), whose ray tracing needs to be generated. Next, do several test renderings to determine the appropriate placement of the light sources and the camera. After that, run the final rendering.

The settings required for test and final renderings will be described below. Before that, let us mention distinct features of each of the prototypes.

The **Room** prototype is intended for creating a static image. The scene in this prototype appears as a "room", with two light sources and a camera. Additionally, a coordinate system is created in advance for convenience of fixing the 3D fragment. By default, the two walls and the ceiling of the "room" are not visible; however, those can be made visible by clearing the "Hide ceiling" flag in the 2D window.

The **Fly around object** prototype is intended for creating either a static image or a photorealistic animation in which the camera moves around the object. The scene appears as a large round platform, with three light sources and a camera. A coordinate system is provided in advance with the scene for fixing a 3D fragment. Additionally, the camera position is driven by an expression and depends on the particular frame of showing the scene. The animation duration needs to be defined in the 2D window (this is the time, in which the camera completes the flight around the object and returns to the original position). The scene in animation needs to be related to the variable "frame", considering the number of frames per second equal to 25.

> You may find the example on using "Flying Around Object" template in the library *"Examples"*, folder *"Additional resources\Ray Tracing\Flying around object"*. After opening "Scene Based on Template.grb" file activate camera "Camera" in 3D scene using **3VC: Select Camera…**. Then use **AN: Animate Model** command and apply animation by variable "frame" from 0 to 250 with step 1.

Creating Custom Elements For Sheet Metal and Holes Libraries

This chapter describes the rules of creating custom elements for sheet metal and holes libraries.

The libraries of typical sheet-metal features, "*Sheet Metal Features*" and "*Sheet Metal Features Inch*", provided with the system, are used when creating the "Forming Feature" operation. The library elements are 3D models created in a special way. If necessary, the user can append the standard libraries with one's own elements. The first section of this chapter describes the sequence of steps to create such elements.

The standard holes libraries "*Hole Features*" and "*Hole Features Inch*" are used in the "Hole" operation. Each element of a hole library is a parametric 3D model, whose geometrical parameters are bound to special reserved variables. Hole libraries can be appended by the user. The second section of this chapter describes methods of creating a 3D model that can be used as a 3D fragment-hole.

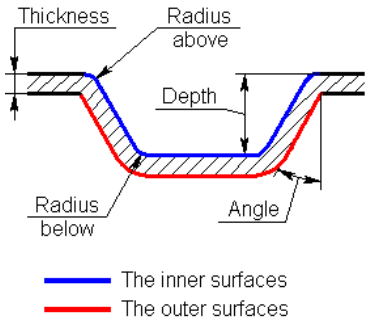
Example of Creating a New Sheet Metal Features Library Element

Sheet-metal features library elements are 3D models created by following certain rules. The base of each element's 3D model is the base 3D profile. It is an "external parameter" of the sheet metal feature, similar to external geometrical parameters of adaptive 3D fragments.

Sheet metal feature elements are applied to a part by using the command **SMF: Create Sheet Metal Feature**. At that time, a 3D profile is selected on the part that defines the desired shape of the sheet metal feature. That profile is passed by the command into the 3D model of the sheet metal feature and is used to substitute its base profile. The 3D model of the sheet metal feature element is recalculated according to the new shape of the profile and is inserted in the part's 3D model.

Let's review the creation of a new element using the example of the "Emboss" element included in the standard T-FLEX CAD distribution.

The diagram shows a draft of the elements being created. The inner and outer surfaces of the prospective element are marked by different colors. The dimensions indicate the parameters whose values can be defined by the user. Besides those, the user will be able to select the blending radius of the corners of the original profile.



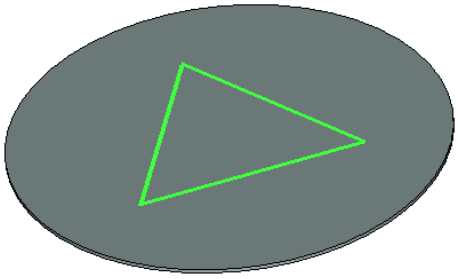
The new sheet metal feature element shall be created as a 3D fragment in the assembly context. Therefore, first you need to create an auxiliary file with the "workpiece" part, to which the forming feature will be applied. This is exactly the document, in which the 3D fragment-forming feature will be created in the assembly context.

> An auxiliary file is required only for the process of creating a new sheet metal feature element. You can delete it once the library element creation is complete.

Creating Auxiliary Document

Let's create an auxiliary document to create a sheet metal feature element. With the help of command you need to create a new 3D model.

In the created document, let's build the "workpiece" part for the forming feature (by any familiar means). The workpiece should have a simple shape and as few edges and surfaces as possible. The forming feature element file size will depend on how simple the workpiece is, and so will the time to open and recalculate the element. Best of all is to make it round and flat.



Let's create the 3D profile on the top faces of the workpiece, that will be the base profile of the forming feature element. The profile should also be made as simple as possible. In this example, let's create a triangular profile.

Save the file under the name *"Part for emboss"*. Now we can proceed with creation of the forming feature element itself.

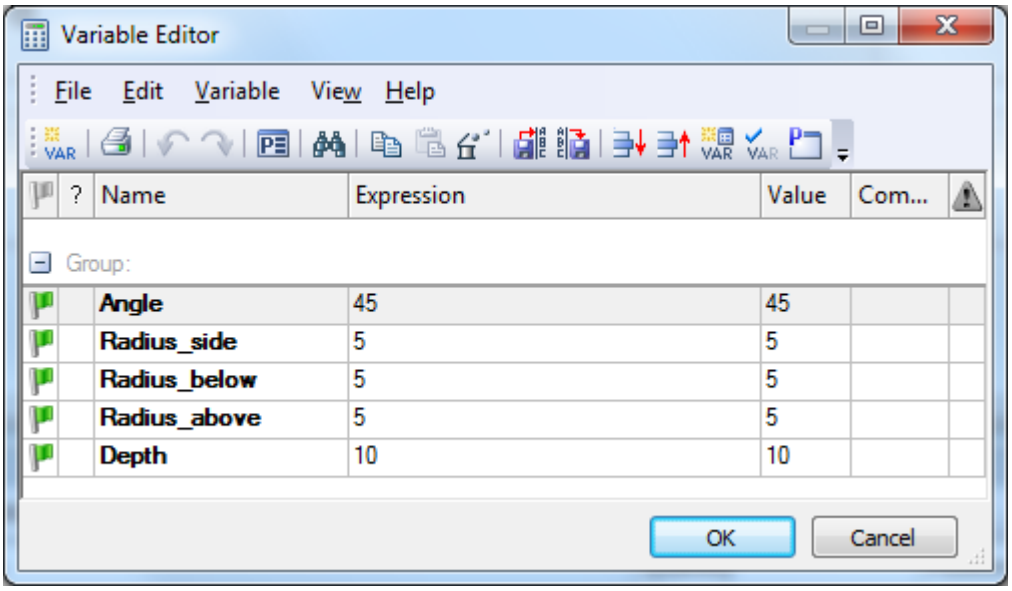
Creating the 3D Model of a Sheet Metal Forming Feature Element

1. Create a new 3D fragment by calling the command **FM: Create Fragment in Assembly** (**File** > **Fragment** > **Create**) and specify an arbitrary name for the prospective element, for example,

"Emboss new". After clicking **[OK]**, the assembly context mode activates automatically (the source body will become transparent).

Make sure that the associative snapping mode is On. To do this, click the right mouse button on an empty space in the 3D window and select the context menu item "Snap Modes".

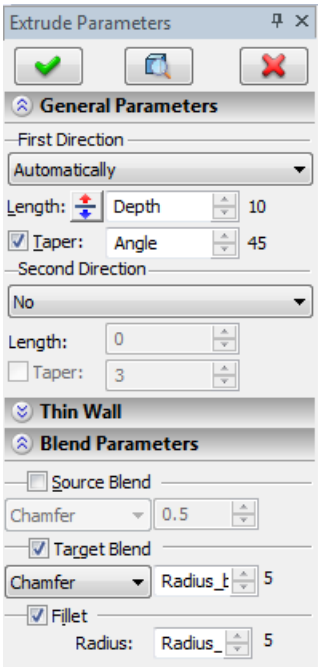
2. In the variable editor, create required variables that will define the element parameters: "Depth", "Angle", "Radius_side", "Radius_below", "Radius_above". All variables must be external.



3. To correctly create the forming feature, you need to know the thickness of the source workpiece. To have this thickness defined automatically, you can use the top and bottom faces of the workpiece. Call the command **Parameters** > **Measure**, select the option "Measure relation between two elements". Specify the top and bottom faces of the workpiece, and click . In the list of relations, select "Distance", and then create the "Thickness" variable based on it. The value of the created variable will be equal to the distance between the selected faces (the thickness of the source body).

4. Extrude (the **3X: Create Extrusion** command) the triangular profile lying on the top face of the source body. When creating the extrusion, the top face shall be selected for the direction; click the button in the property window to flip the extrusion vector. The end blending shall be defined by the "Radius_below", the side edges blending – by the "Radius_side" variable.

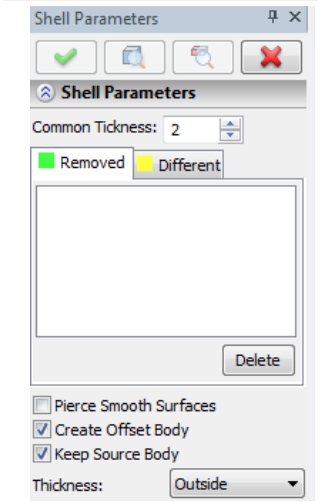
   The extrusion we have created ("Extrusion_1") will be used to obtain the **inner** surface of the forming feature.



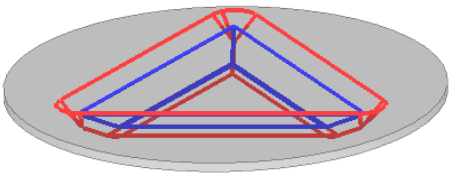
When creating an ordinary model, it is possible to skip using additional blending options for the extrusion and instead use common blend operations. However, in the present case that is not an option, since the model must correctly recalculate with any admissible 3D profile. If, for example, we would change the number of joint points of the top 3D profile in the "Part for emboss" file, then the 3D model in the "Emboss new" file would still recalculate correctly. If using an ordinary blend, then this may not work if it fails to account for the increased number of profile edges.

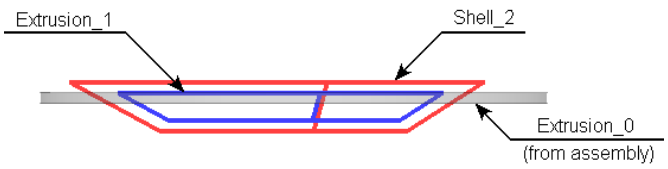
5. To obtain the **external** surface, we will use the "Shell" operation. Call the command **3SH: Create Shell**, select the option "Select a body" and specify the extrusion created at the previous step. Enter the name of the "Thickness" variable in the "Common Thickness" field. Set the shell parameters as shown on the figure on the right: the "Create offset body" and "Keep source body" flags shall be made active.

   The created body will be named "Shell_2".

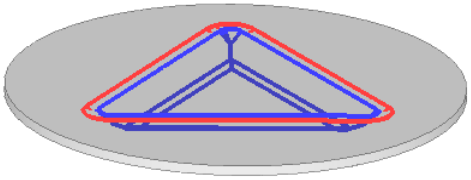


6. By now we created the inner portion of the "Emboss new" prospective element – "Extrusion_1", its outer portion – "Shell_2" and the source workpiece – "Extrusion_0 (from assembly)". But "Shell_2" comes out above the source body. This outcoming portion needs to be deleted.





To cut the extra portion, we will use the **3CU: Cut By Section** operation. Select the top face of the source workpiece body as the cutting plane. The resulting body will be called "Cut operation_3".

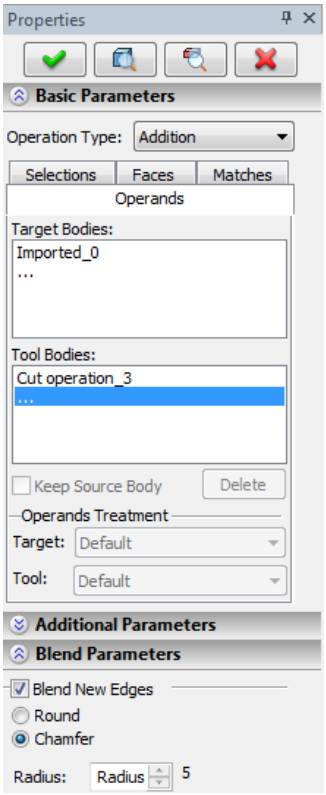


7. It is now possible to add together "Cut operation_3" and the source workpiece (the latter being no longer needed) using the Boolean operation (the **3B: Create Boolean** command).

   In the Boolean operation properties set the flags "Blend New Edges" and "Chamfer". The "Radius" parameter value shall be defined by the following expression:

   Radius_above-Thickness>=0? Radius_above-Thickness:0.

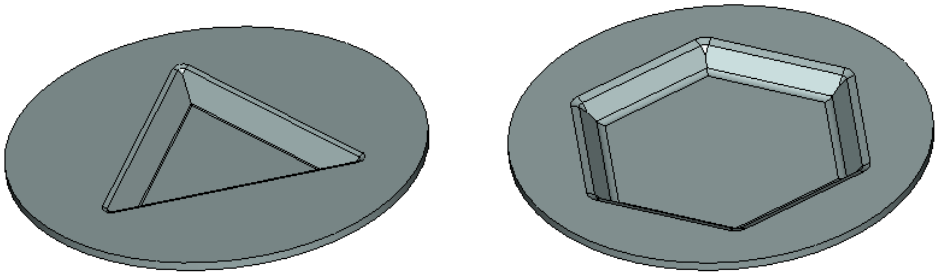
   When creating the Boolean operation, chamfers will be created on new edges. The chamfer offsets shall be less than the "Radius_above" defined by the user, by the "Thickness" amount, and cannot be less than zero. This requirement conditions the above-written expression.



8. What is left now is to generate the inner surface of the "Emboss new" element. To do this, we shall use a Boolean operation again. We will subtract "Extrusion_1" from the "Boolean_4" body obtained at the previous step. Just like in the previous step, the option of chamfer creation on new edges shall be enabled in the Boolean operation properties. The chamfer size shall be defined by the "Radius_above" variable.

At this point, the new forming feature model element can be assumed completed. Next, right-click on an empty space in the 3D window and select the context menu item **Save Fragment and return**. The *"Emboss new"* file, which contains the created 3D fragment, will become a new element in the sheet metal features library.

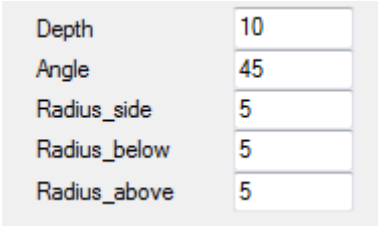
What to do with the obtained sheet metal forming feature element will be described a bit later. For now, let's verify the function of the created forming feature element. Upon finishing working in the assembly context, the *"Part for emboss"* file will open again. You do not have to save it – we shall not need it anymore. But, before the file is closed, change the number of segments in the source profile on the "workpiece" of the part – the forming feature shall adjust accordingly.



Creating a Dialog for New Library Element

Besides the 3D model proper, a new sheet-metal library element requires creation of a dialog to input parameters of the given forming feature element (which is the external fragment element parameters). This dialog will be coming up in the command **SMF: Create Sheet Metal Feature** when inserting our forming feature element.

To create the dialog, open the file *"Emboss new"*. Add a new 2D page to it to hold control elements (the **TR: Create Control** command). On this page, position the elements of the "Static Text" type with the appropriate parameter names and the value input fields ("Edit Box" elements) for all external variables of the created fragment using the **TR: Create Control** command. As a result, you will get a simple dialog that will serve to help the user defining parameters of the forming feature being created. If desired, a more complex dialog can be created, by introducing in it a schematic image of the element as in standard sheet metal library element dialogs.



> The forming feature dialog is automatically scaled in the **"SMF: Create Sheet Metal Feature"** command property window so as to have the font size in it matching the Windows font size. Therefore, when creating a dialog, we recommend you not to change the font size set by default for the dialog page (it is set to be close to a standard Windows font). This will help avoid distortions when displaying the dialog.
>
> Besides that, do not create too wide dialogs that do not fit within the standard property window.

After creating the dialog, save the file again.

Defining Limitations on the Source 3D Profile Type

Limitations can be imposed on the type of a 3D profile that will be used for creating the new forming feature element in the **SMF: Create Sheet Metal Feature** command. To impose limitations, you need to create two external variables named "contour_type" and "profile_type". The values assigned to those variables will define the 3D profile types allowed for the given forming feature element:

profile_type: defines the geometrical appearance of the 3D profile contour	contour_type: defines the nesting of the 3D profile contour
0 - any 1 - closed 2 - open 3 - line segment 4 - from polylines	0 - any 1 – simple only (no nested contours and no multiple contours) 2 - no nested contours

When these variables are created, then, upon an attempt to select a 3D profile of an inappropriate type in the forming feature creation command, the system will output an appropriate warning in the Diagnostics window. As a result, the profile will not get selected. On the other hand, if the variables were not created in the forming feature fragment, then, when applying it, any 3D profile types will be allowed for selection. The fact that an incorrect profile was selected will then only be hinted by the errors occurring when recalculating such a forming feature.

In our example, the "profile_type" variable must be equal to 1, and the "contour_type" variable must be equal 0.

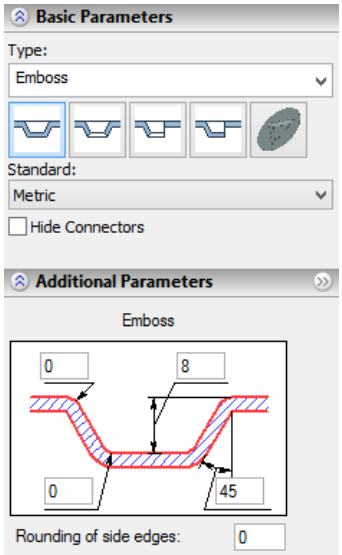
Adding a Created Fragment to the Sheet Metal Features Library

To use a created element in the **SMF: Create Sheet Metal Feature** command, you need to copy the "Emboss new" file into the folder *"...\System\Sheet Metal Features"*. After that, when calling the forming feature creation command, the created element will show up in the list.

Element Name and Icon

Our new forming feature element will be referred to in the **SMF: Create Sheet Metal Feature** command by its filename and the standard icon of a T-FLEX CAD document. If desired, it is possible to define for it another, more intuitive and complete, name. To do that, call the **PS: Show Model Properties** command in the element file (in our example that's "Emboss new"), and fill in the "Title" field on the "Summary" tab.

Icon may be changed using command **IC: Create/Edit document icon**.



More information about icons creation can be found in "Preview-slide" chapter.

EXAMPLE OF CREATING A NEW HOLES LIBRARY FRAGMENT

Let's review another example of creating a new hole fragment. Open a standard hole – "Center Hole with metric thread":

1. Since there are different implementation versions of the hole, an internal database was created within the fragment document, that contains standard dimension sets.

b - Center hole (shape F).grb | Center hole (shape F).grb:1 | NONAME1.g

b

№	dt	D	d	p	c	d1	d2	l	l1
1	3	8	3	0.5	0.5	3.2	5	2.8	1.56
2	4	10	4	0.7	0.5	4.3	6.5	3.5	1.9
3	5	12.5	5	0.8	1	5.3	8	4.5	2.3
4	6	16	6	1	1	6.4	10	5.5	3
5	8	20	8	1.25	1.6	8.4	12.5	7	3.5
6	10	25	10	1.5	1.6	11	15.6	9	4
7	12	32	12	1.75	1.6	13	18	10	4.3
8	16	40	16	2	2	17	22.8	11	5
9	20	63	20	2.5	2.5	21	28	12.5	6
10	24	100	24	3	2.5	25	36	14	9.5
11	30	160	30	3.5	2.5	31	44.8	18	12
12	36	250	36	4	3	37.5	53	20	13.5
13	42	400	42	4.5	3	43.5	59.7	22	14
14	48	630	48	5	4	49.5	74	24	16
15	56	900	56	5.5	4	58	85.6	27	18
16	64	1200	64	6	4	66	95	29	19
17	72	1200	72	6	4	74	704.7	31	20
18	80	1200	80	6	4	82	115.7	34	22
19	100	1200	100	6	4	102	140	36	24

2. Special **external** variables were created in the fragment document that are required for correct handling of the fragment in the command **3H: Create Hole**:
   - **IsThrough** – this is the variable that defines the hole type (0 – blind, 1 – through).

     The 3D model of any hole fragment shall be created in such a way as to have the hole type (blind, through) change depending on this variable value. The variable value will be determined by the state of the option in the command "**3H: Create Hole**" when inserting the hole. This external variable must necessarily be present in the hole fragment.
   - **length** – this is the variable responsible for the hole depth (when creating a fragment, this variable value can be arbitrary).

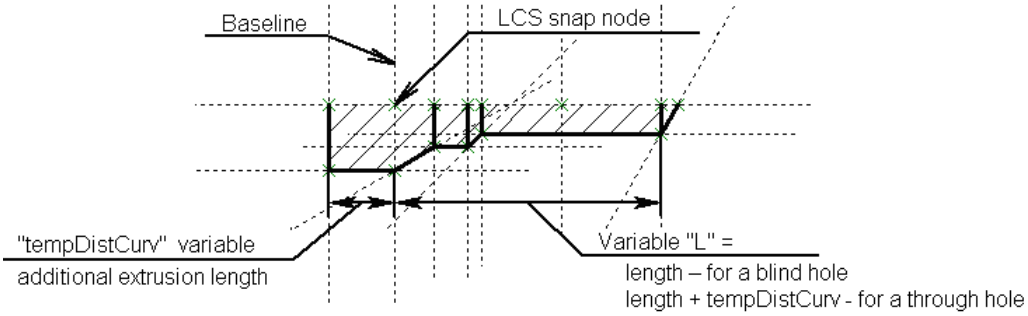
     When applying a fragment, this variable value will be determined by the system (for a through hole) or by the user in the fragment's external variables dialog (for a blind hole).

     You do not have to create this variable (or make it external), if the hole being created can only be blind and its depth cannot be arbitrary, but is rather determined by the

standards. In such a case, the depth is uniquely determined by other hole parameters (for example, the diameter, as found in the holes for the positioning screw tips). When applying a hole without the "length" external variable, the hole type selection options will be inaccessible in the **3H: Create Hole** command automenu.

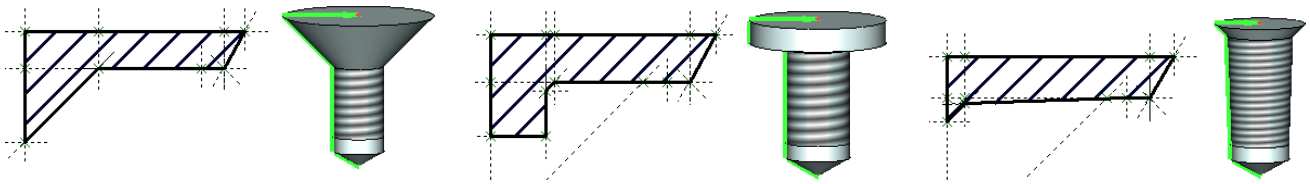
Other necessary variables can be created and named by the user as desired. For example, in the described hole fragment, an additional external variable "IsThrough" is created. Depending on its value, the hole length can be increased by the amount defined by another external variable, "tempDistCurv". This is necessary in the case when the hole is applied on a curved face (convex or concave) or is directed at an angle to a flat face. When the "IsThrough" variable value is equal 0 – no length extension is required, 1 – the length needs to be extended. A blind hole will be extended by this amount on one side, a through one - on both sides.

3. The 3D profile of the hole is created based on 2D constructions on a workplane. 2D constructions are made in such a way as to describe all possibilities for creating a given hole (the change of the hole type is defined by the values of the above-described variables).



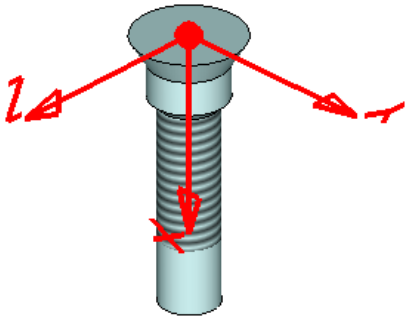
4. A rotation operation is created based on the 3D profile.

   When creating a hole it is also possible to use the "Extrusion" command. An example is a square hole in the standard library. A general rule while creating any hole fragment is to create only a single body in it. If a hole has several versions, its shape should be possible to change using construction lines.

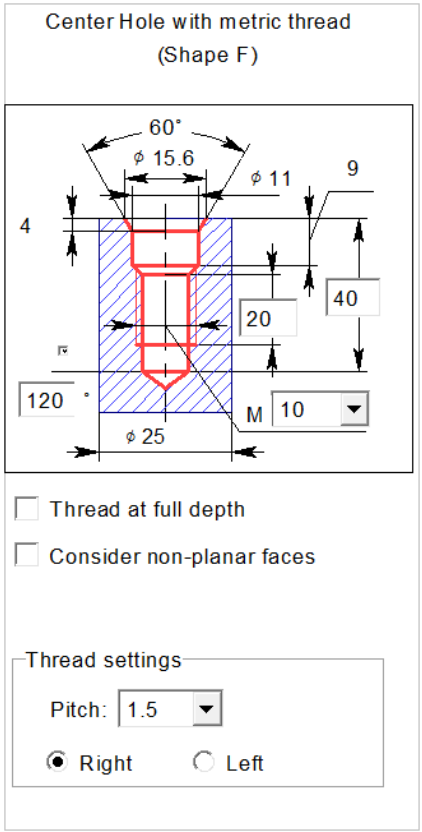


5. Since the considered hole is threaded, there is a thread applied to the cylindrical face (the **"3AT: Create Thread"** command). The cylindrical face diameter corresponds to the inner thread diameter. The thread parameters (type, diameter, pitch, direction) are bound to variables. To make sure that the thread in the hole appears correctly on a projection, the "Thread" operation parameters include the "Side/Inner" property.
6. To snap a hole when using it in the **3H: Create Hole** command, a local coordinate system is created. An LCS for snapping any hole is created by the following rules: the coordinate system

origin point must be on the hole axis, the X-axis must be directed inward of the hole along its axis. In the coordinate system parameters, have the "Use for Fragment Fixing" flag set.



7. To manage hole parameters, a dialog of control elements is created.



8. The file of the described hole fragment is stored in the folder "...*System**Hole Features*". This is how it is made accessible in the list of holes in the **3H: Create Hole** command.

The mentioned folder is for the metric library of holes. Save inch-base hole files in the folder "...*System**Hole Features Inch*".

9. Additionally, the full name of this hole is specified among the properties of the described fragment file. Besides that, the file of the whole icon is created in the same folder, that contains the images in two sizes (16x16 and 32x32). The icon file name matches the name of

the hole fragment file. The hole name and its icon will be displayed in the list within the **3H: Create Hole** command.

The icon creation and the hole name definition are optional. Nevertheless, those help make the hole handling more intuitive in the **3H: Create Hole** command.

Model Geometry Analysis

MODEL GEOMETRY ANALYSIS

A set of commands united in the **Geometry Analysis** group serves for conducting various checks on the model. These commands neither modify the model nor create new bodies or operations.

The set of the geometry analysis commands includes the following:

- ✓ Mass-Inertia Properties;
- ✓ Model check;
- ✓ Intersection check;
- ✓ Curve curvature;
- ✓ Surface curvature;
- ✓ Surface deviation;
- ✓ Surface gap;
- ✓ Normal deviation;
- ✓ Surface smoothness;
- ✓ Model separation check;
- ✓ Model Separation Check.

The commands can be called from the textual menu **Tools** > **Geometry Analysis**.

MASS-INERTIA PROPERTIES

This command allows calculating mass-inertia properties of the selected operations. The command **3MP: Mass-Inertial Properties** is called via:

Icon	Ribbon
	Measure→ Measure → Mass Properties
Keyboard	**Textual Menu**
<3MP>	Tools > Geometry Analysis > Mass Properties

Upon entering the command, the following option is active by default:

	<B>	Select source 3D operation

It allows selecting the operation, whose properties need to be calculated.

To calculate the properties that depend on all existing operations, do not select any operations or use option:

	<A>	Select all solids

This is the case if you need to calculate properties of an assembly model composed of several bodies. The bodies may even have different density. The system will account for that.

To cancel selection of all operations, use the option:

	<F>	Cancel Operation selection

If necessary, calculations can be done relative to a local coordinate system that can be selected by the option:

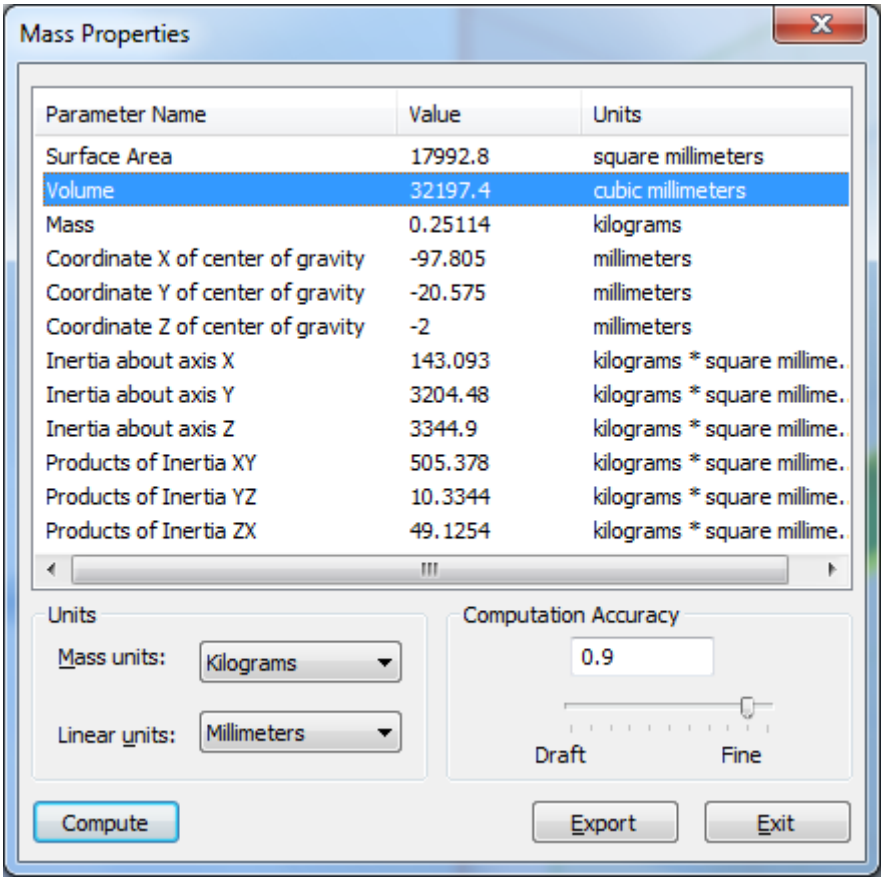
	<L>	Select target LCS

To cancel coordinate system selection, use the option:

	<K>	Cancel LCS selection

If no target coordinate system is specified, the calculation will be done in the global coordinates.

Upon selecting the operations and the coordinate system, finish the input. Upon finishing the input, the "**Mass properties**" window will be displayed.



The parameters "**Mass units**" and "**Linear units**" allow setting up the desired measurement units.

The computational accuracy can be defined in the respective input box or set by the provided slider.

If you press the button **[Compute]**, the dialog will display the calculated values of the parameters that are represented in this dialog box.

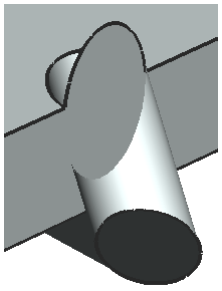
The button **[Export]** allows saving the calculation results in a text file.

The button **[Exit]** closes the parameters calculation window and brings you back to the mode of selecting bodies and coordinate systems.

MODEL CHECK

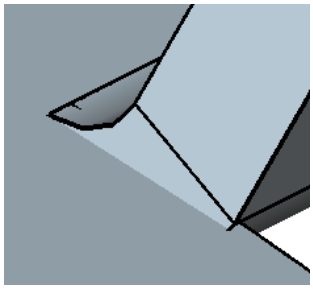
This command serves for analyzing the selected body in order to expose errors in its geometry. Errors may become an obstacle on the way of future manipulations with the model. Errors in geometry may occur, for example, due to deficiencies of importing 3D models from other systems. Sometimes, however, errors may be introduced at the time of creating a 3D model in T-FLEX, since only a local model check is automatically performed when creating a 3D operation. Meanwhile, the full check may take some time, depending on the model complexity.

Examples of some typical errors are shown below.



Mismatching loops

Self-intersection



Mismatching faces

To call the command **QM: Check Model**, use the textual menu:

Icon	Ribbon
	Measure→ Geometry analysis → Model Check
Keyboard	Textual Menu
<QM>	Tools > Geometry Analysis > Model Check

Once in the command, you can select elements to check. Those could be edges, faces or whole operations (in the latter case, all edges and faces belonging to this operation will be checked simultaneously). Use the following options for selection:

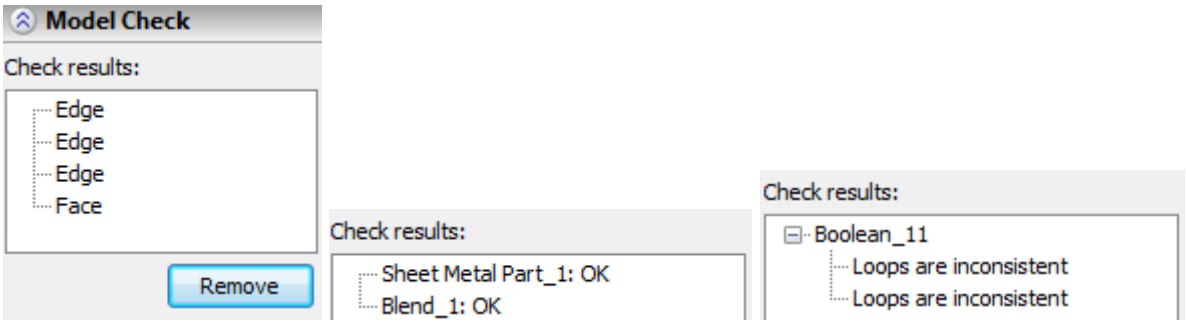
	<E>	Select Edge or cancel Edge selection
	<F>	Add/Remove selected Face

	<B>	Add/Remove all Faces of Solid

To select all operations at once, you can use the option:

	<A>	Select all operations

Once selected, the element is put in the list **Check results**. This list displays the type of the element and whether it was already checked or not. If it was, then the results of the analysis are displayed as well.



To cancel selection of one of the elements, click it again in the 3D window. To cancel selection of all elements, use the option:

	<R>	Cancel selection

To run the check on the selected elements, use the option:

	<Enter>	Finish input

After picking this option, the system will check the selected elements and display the results in the **Check results** pane. If there are no errors, **[OK]** will be put next to the element.

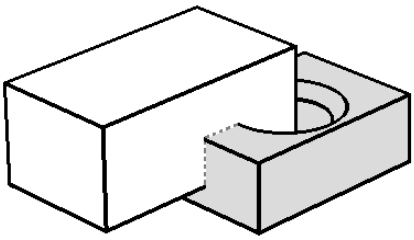
If, on the contrary, an error was found, then the list of the detected errors will be displayed next to the element name.

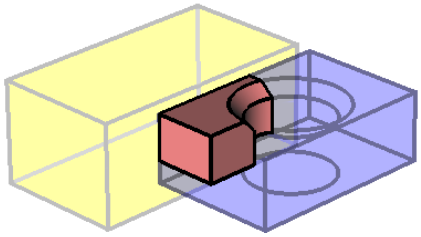
If you quake a particular error, then the erroneous part of the model will be highlighted red in the 3D window.

You can immediately select more elements to check, without leaving the command.

INTERSECTION CHECK

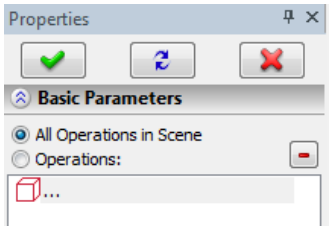
This command allows checking the model for intersections and contacts of the selected bodies. The command is particularly helpful when working with assemblies.





To call the command **QI: Check Model Intersections**, use the textual menu:

Icon	Ribbon
	Measure→ Geometry analysis → Intersection Check
Keyboard	Textual Menu
<QI>	Tools > Geometry Analysis > Intersection Check

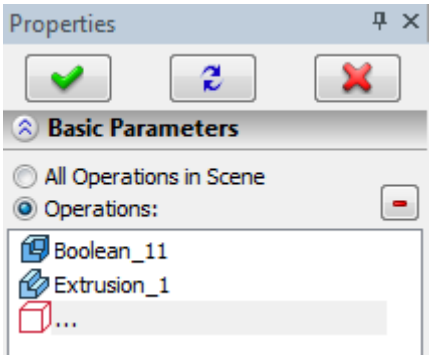


The command offers two checking modes: checking intersections of all bodies in the scene, and checking intersection of the selected operations. The desired mode is selected in the property window.

Upon entering the command, the **All Operations in Scene** button is pushed in the property window. Therefore, you can finish input immediately and run the check for intersection of all operations in the scene.

If the scene is rather complex, contains numerous bodies, and the bodies have complicated geometry, then checking intersection of all bodies in the scene can take long time. In such cases, you are advised to directly specify the bodies whose intersection needs to be checked. To do this, use the option:

	<F>	Select source 3D operation



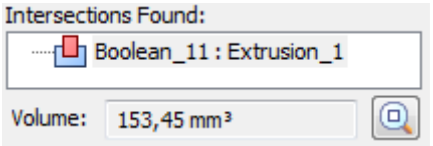
Once an operation is selected, it will be added to the checklist, and the "**Operations**" button will be pushed automatically in the properties window. Upon finishing the input, the check for intersections will be run on the listed operations only.

To delete a specific operation from the checklist, use the button . To cancel selection of all bodies, you can simply switch to the **All Operations in Scene** mode, and the list of the operations to check will be cleared.

To run the intersection check, use the option:

	<Enter>	Finish input

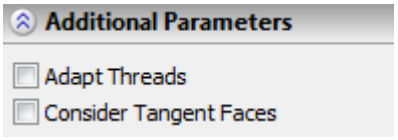
Upon finishing the check, the results will be displayed in the "Intersections Found" pane. If you click an intersection report, the intersecting bodies will become semitransparent in the 3D window and change the color, while the intersection region will be brightly highlighted. The volume of the bodies' intersecting region will be displayed in the **Volume** pane.



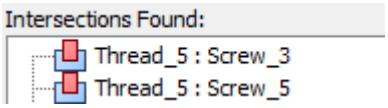
If the intersecting region is too small, you can use the button **Zoom in**, and the image of the intersection region will be magnified.

Depending on the specified additional parameters, the analysis results will be displayed differently in the "Intersections Found" pane.

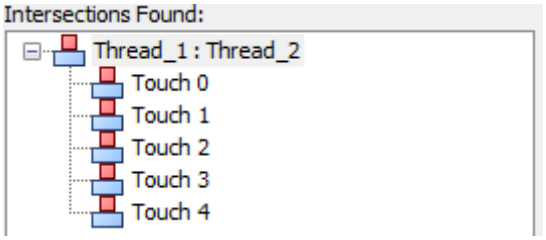
The command provides two additional parameters, **Adapt Threads** and **Consider Tangent Faces**.



Often, one part is connected with another one by a thread. In this case, with the **Adapt Threads** parameter set, the system will be taking into the account thread parameters of the intersecting operations and, in the case of finding a mismatch in some thread parameters (for example, the pitch), the message "Face geometry doesn't match thread standard" will be displayed in the "Intersections Found" pane. If you click this message, the respective faces with the thread will be marked in the 3D window by dashed lines.



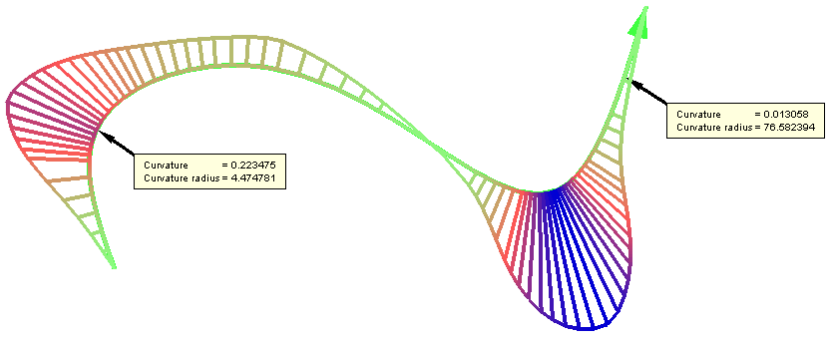
By default, the system accounts only for intersections of the bodies penetrating each other. If the **Consider Tangent Faces** option is set, the system will be also accounting for touching contacts, besides intersections of operations. Therefore, the touching faces of the operations will be mentioned in the "Intersections Found" pane, besides intersections and mismatching thread parameters.



If you click a report about a touch, the respective faces will be marked by dotted lines in the 3D window.

Measuring Curvature of Curves

This command allows measuring curvature and the curvature radius of the selected curves. You can select edges and 3D Paths to measure curvature.



The command **QC: Show Curve Curvature** can be called via:

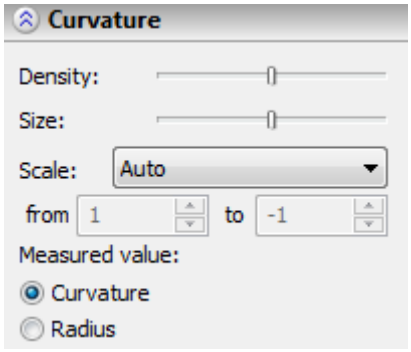
Icon	Ribbon
	Measure→ Geometry analysis → Curve Curvature
Keyboard	Textual Menu
<QC>	Tools > Geometry Analysis > Curve Curvature

Upon entering the command, you need to select one or several elements, whose curvature needs to be measured. To select, use the options:

	<E>	Select Edge or cancel Edge selection
	<T>	Select 3D Path

To cancel selection of a curve, you need to click it again. To cancel selection of all curves, use the option:

	<U>	Cancel selection

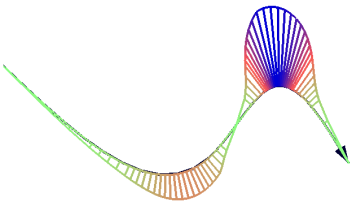


Right after selecting an element, its curvature will be measured and displayed in the 3D window as a multiplicity of colored lines stemming from the curve, called curvature porcupine quill.

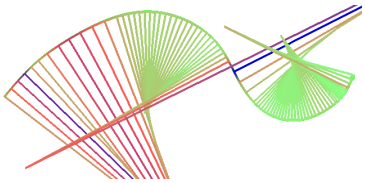
You can modify two parameters: **Curvature** and **Radius**. Depending on the selected parameter, rendering in the 3D window will appear differently.

If **Curvature** is selected, the colored quill lines will be pointing in the direction away from the center of curvature. The lines originate on the curve. Their length corresponds (to a certain scale) to the curvature at the point, from which the line originates.

The scale coefficient that relates the curvature of the curve and the length of the colored lines is defined in the properties window by the parameter **Size**.



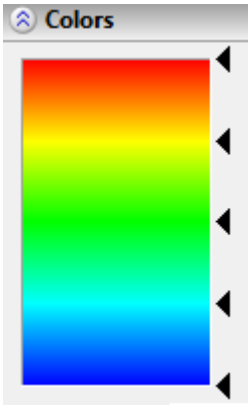
If the **Radius** parameter is evaluated, the colored lines will be connecting points on the curve with their respective centers of curvature. The length of the colored lines corresponds to the radius of the curve's curvature.



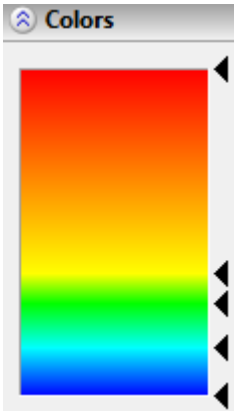
The density of the colored lines is controlled by the **Density** parameter both in the case of curvature evaluation and in the case of the curvature radius evaluation.

Working with the tab "Colors"

The quill lines are painted in different colors, depending on the value of the curvature at the point. Color distribution within the range of the measured parameter is set up on the tab **"Colors"**, located in the properties window.



A color scale is displayed on the tab. The values at the left of the scale correspond to certain colors on the scale. The black triangular markers at the right of the scale are used for adjusting the scale.
Each marker relates to a certain color. The colors of the scale change smoothly from one marker to another. You can move the markers along the color scale. To do this, you need to "grab" a marker with the mouse and drag it to the new position. As you do this, the color scale will change its appearance, and so will the image in the 3D window.



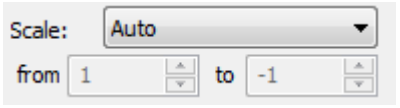
To define the color of a certain marker, right click this marker, and in the coming up menu select "Color". After that, you can select the color in the standard color selection window.

The scale can have two markers (for the maximum and the minimum values), or more. To add a marker, right click where you want to add it, and in the coming up menu select **"Add"**.

To delete a marker, right click this marker, and in the coming up menu select **"Delete"**.

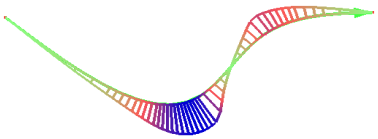
The marker placement scheme and the colors can be saved and unloaded to a file (the file extension is "*.col"). To do this, use the menu items (**Save** and **Load**), coming up upon right clicking anywhere on the **Colors** tab.

The range of the color scale for the measured parameter can be modified as well. To do this, use the parameter **"Scale"** at the top of the properties window. By default, the parameter is set to **"Auto"**. This means that the minimum and maximum values on the scale correspond to the minimum and maximum parameter values of the selected elements.

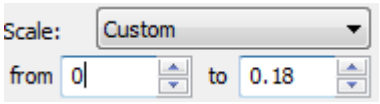


In this case, the entries "from" and "to" cannot be edited, since these parameters are defined automatically.

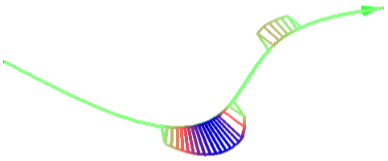
Note that the quill lines defining the amount of curvature are displayed only on the portions of the curve whose curvature falls within the specified range. Thus, if using the automatic scale, the colored quill will be displayed along the whole curve.



If the parameter is set to **"Custom"**, then the values in the entries "from" and "to" can be specified manually. Note that the quill will be shown only along the portions whose curvature falls within the specified range.



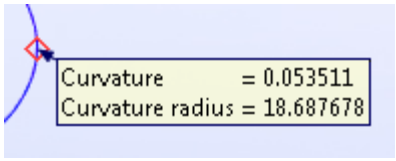
Thus, for example, the minimum curve of the curve shown on the diagram is equal to zero. Therefore, setting a positive value in the "from" input box (in the example, set to 0.18), will make the quill disappear on some parts of the curve. The quill coloring will also be different.



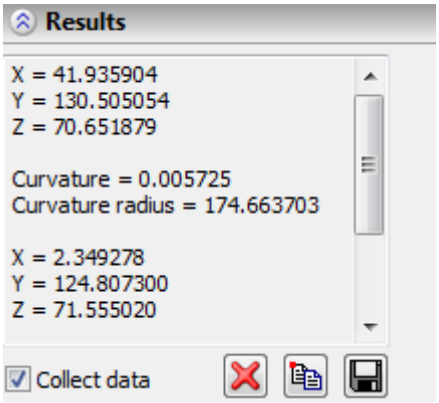
Besides the color-coded curvature display, a provision is made for reading the curvature at a specific point. To do this, use the option:

	<C>	Measure curvature at arbitrary point

When this option is active, the pointer changes its shape. As the pointer approaches the curve, the position on the curve where the curvature is measured is marked by the red square. The value of the curvature and the radius of curvature at the point appear in a tooltip next to the pointer.



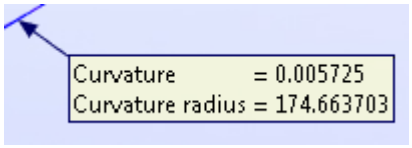
If the option **Collect data** is set on the tab **Results**, then, upon clicking a point on the curve, the point coordinates, curvature and the radius of curvature will be written in the curvature measurement log.



The created log can be copied to the clipboard (the button) or saved in a text file (the button).

To clear the log, use the button .

Besides collecting data into the log, you can create 3D labels when clicking a point on the curve. A 3D label appears as a semi transparent text in the 3D window, containing the results of the conducted measurements and an arrow marking the point where the measurement was done.



3D labels are handled by using the parameters located on the tab **3D Labels**.

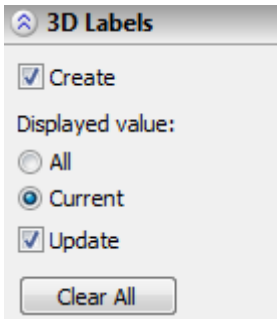
The option **Create** determines whether to create new 3D labels upon clicking a point on the curve.

If the option is set **Displayed value: All**, both values will appear by the newly created 3D label: the curvature and the radius of curvature.

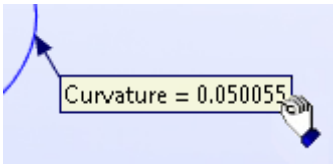
If, instead, the option **Displayed value: Current** is set, then the newly created label will show only the parameter that is currently being measured.

If the **Update** option is set, then the 3D labels that show the current value are updated automatically when switching to a difference type of the measured value.

To delete all 3D labels, use the button **[Clear All]**.



3D labels are displaced at some distance (in the screen coordinates of the 3D window) from the point they corresponds to, and participate in the motion when the screen is spun. This relative position can be modified. To do this, move the pointer to the 3D label (in this case, the pointer will change its appearance) and drag the 3D label to the new position.



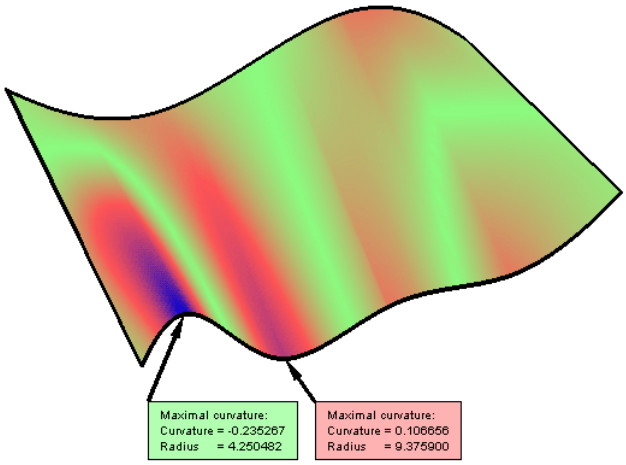
To find the point of maximum curvature, use the option:

	<M>	Find point with maximal curvature

When using this option, the system will automatically create a 3D label indicating the point on the curve with the maximum curvature. A distinctive feature of such 3D label is that it is red.

MEASURING CURVATURE OF SURFACES

This command allows measuring the curvature and radius of curvature of one or several selected faces.



You can view the overall distribution of the curvature on a face (for this, the model will be colored appropriately), or find the curvature value at a specific point.

To call the command "**3VV: Show Surface Curvature**", use the textual menu:

Icon	Ribbon
	Measure→ Geometry analysis → Surface Curvature
Keyboard	**Textual Menu**
<3V V>	Tools > Geometry Analysis > Surface Curvature

When measuring the curvature, you can select either separate faces or whole operations (that is, all faces of the given operation).

	<F>	Add/Remove selected Face.
	<B>	Add/Remove all Faces of Solid

To cancel surface selection, click it again. To cancel selection of all objects, use the option:

	<U>	Cancel selection

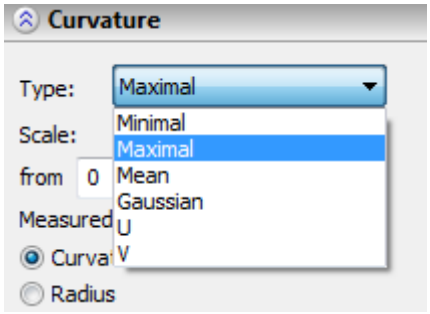
When selected, the face will change its color according to the type of curvature subject to the measurement and the settings on the "Colors" tab. Color handling for surface curvature measurements is done in the same way as in the command **QC: Show Curve Curvature.**

When additional faces are selected, the range of the curvature radius changes; therefore, the color of the earlier selected faces may change.

Types of Measured Values

As in the case of measuring curvature of curves, you can specify the value to measure in this command: **Curvature** or **Radius**. However, you need to additionally specify the type of the measured value.

The type of the measured value is defined with the help of the **Type** combo box, located at the top of the properties window.



The surface curvature at a specific point depends on the direction, in which the curvature is measured. The curvature will be maximal in one direction and minimal in another direction. Due to these facts, the system allows measuring the following types of values:

"Minimal" or "Maximal"**.** With these parameter settings, measurements of the maximum or the minimum value will be done at each point of the selected surface.

"Mean" - *is half the sum* of the minimum and the maximum measured values.

"Gaussian" - *is the product* of the minimum and the maximum measured values.

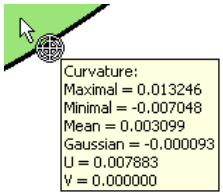
"U" and **"V"** - these parameters require the measurement to be done in just one of the directions.

Measuring Curvature at a Point

To measure curvature at a specific point, use the option:

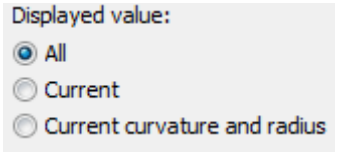
	<C>	Measure curvature at arbitrary point

Now, if you move the pointer to any point on the surface, the measured values will be displayed next to the pointer. The types of the measured values are defined by the parameter **Displayed value**, located on the tab **3D Labels**.



Clicking at some point on the face creates a 3D label.

3D label handling is done in the same way as in the command **QC: Show Curve Curvature**, except that the parameter **Displayed value**, located on the tab **3D Labels**, can take three values instead of two.



Upon clicking a face, besides 3D label creation, the measured parameters can be added to the log. Once a face is clicked, the following is logged: the point coordinates, the values of all types of curvature and the values of the curvature radii of all types. Working with the **Results"** tab is done same as in the command **QC: Show Curve Curvature"**.

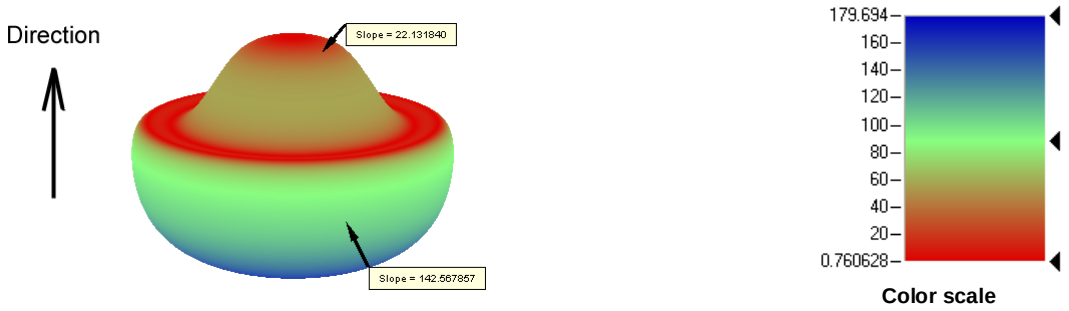
To find the points of the maximum and minimum curvature of the specified type, use the option:

	<M>	Find minimal and maximal curvatures

When using this option, two 3D labels will be created automatically, one – red, and another one - blue, indicating the positions of the minimum and the maximum curvature.

SURFACE DEVIATION

This command allows finding deviation of the normal from the specified direction for the selected face.



The colored image of the face allows viewing the deviation over all face area. It is also possible to measure deviation at a specific point.

The command **QD: Show Face Deviation** can be called via:

Icon	Ribbon
	Measure→ Geometry analysis → Surface Deviation
Keyboard	Textual Menu
<QD>	Tools > Geometry Analysis > Surface Deviation

Upon entering the command, you need to specify the direction, with respect to which the surface deviation will be determined. To select a direction-defining element, use the option:

	<D>	Click to select direction

When using this option, you can select faces, edges, 3D profiles, workplanes and coordinate systems.

To define the direction by specifying two 3D points, use the options:

	<S>	Select first Vertex or 3D Node
	<E>	Select second Vertex or 3D Node

For point selections, you can use 3D nodes, vertices, edges, coordinate systems, sphere or torus faces.

The specified direction can be flipped to the opposite by the option:

	<I>	Switch

Upon choosing the direction, you need to select faces, whose deviation needs to be evaluated. To specify a particular face, use the option:

	<F>	Add/Remove selected Face

To select all faces of some body, use the option:

	<B>	Add/Remove all Faces of Solid

Select all operations in the scene using the option:

	<A>	Select all operations

To cancel face or body selection, you need to select them once again.

To cancel selection of all faces, use the option:

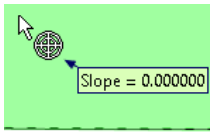
	<U>	Cancel selection

If the direction is specified, selected faces will change color according to the scale located on the tab **Colors**". Working with this tab is exactly same as in the command **QC: Show Curve Curvature".**

To measure deviation at a specific point, use the option:

	<C>	Measure slope at point

If you then move the pointer to a certain point on the face with this option active, the value of the face deviation from the specified direction will be displayed next to the pointer (at the given point).



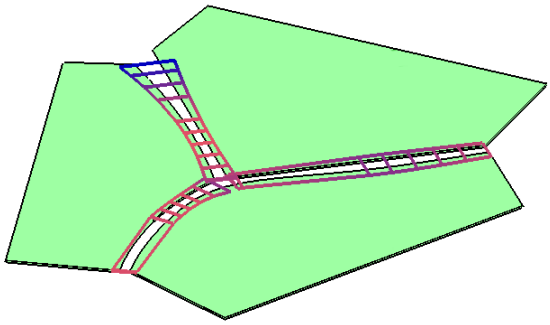
By default, if you click at a point on a face, a label will be created with the value of the face deviation in this point. Additionally, the values of the point coordinates and the slant angle will be recorded in the measurements log.

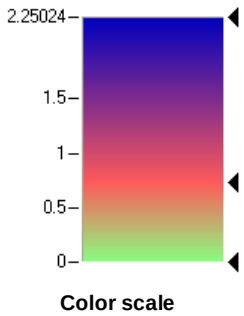
Working with the tabs **Results** and **3D Labels** is same as in the command **QC: Show Curve Curvature.**

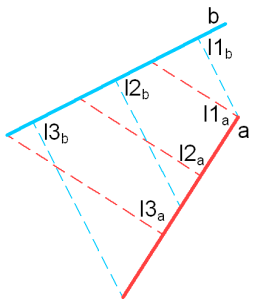
SURFACE GAP

This command allows evaluating the gap between two or several selected faces or edges. The system evaluates the distance between edges. In this case, the command is used for evaluating the distance between edges throughout their extent, rather than the direct measurement of the distances between two points on the edges. To measure the exact distances, use the command **PM: Measure Element or relation between two Elements**.

The command is mostly used for analyzing models resulting from deficiencies of an import or export.







The distances between edges are calculated in the planes perpendicular to the edges. Therefore, for example, two choices of the distance it is used for a pair of nonparallel edges: from the first edge to the second and from the second edge to the first.

The diagram on the right-hand side depicts two lines: **a** and **b**. The distances from **a** to **b** are shown by dashed lines (the segments **l1a**, **l2a**, **l3a**) and from **b** to **a** (the segments **l1b**, **l2b**, **l3b**).

These are the exact distances (the dashed lines on the diagram), whose set is displayed in the 3D window as the distances between edges.

The command **QH: Show Gap Between Faces** is called via:

Icon	Ribbon
	Measure→ Geometry analysis → Surface Gap
Keyboard	Textual Menu
<QH>	Tools > Geometry Analysis > Surface Gap

Upon entering the command, you need to select faces or edges, between which you want to measure gaps. To select the faces, use the option:

	<F>	Add/Remove selected Face

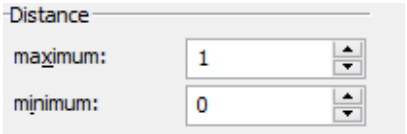
To select ribs use command:

	<E>	Select edge or cancel edge selection

To cancel specific face of rib selection, click it again. To cancel selection of all faces, use the option:

	<U>	Cancel selection

When several faces are selected, colored “quill” lines appear between the edges, indicating the gaps between the faces.



The option **Maximal distance** defines the maximum distance between two edges, for which the gap is to be measured. The greater is this parameters value, the more will be the number of edges between which the system would evaluate the gaps.

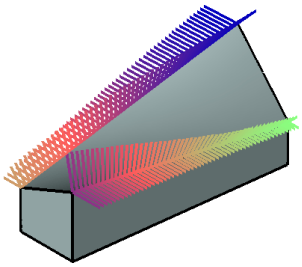
If the **Maximal distance** value is great enough, the gaps will be evaluated from each edge of one face to each edge of the other face.

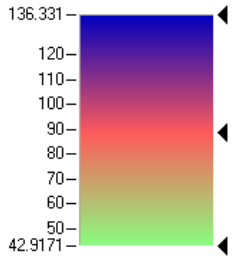
Minimum distance defines the minimum distance between two edges, for which the gap is to be measured.

Each line of the quill is painted depending on its length, in an appropriate color according to the color scale. The color scale setup is done on the tab **Colors**. Working with this tab is similar to that in the command **QC: Show Curve Curvature.**

DEVIATION OF NORMALS

This command allows measuring deviation of the normals on the adjacent faces at the selected edges.





Color scale

The quills displayed in the 3D window allow viewing deviation of normals across the extent of an edge. You can also measure the angle of normal's deviation at a specific point.

The command **QN: Show Normal Deviation"** is called via:

Icon	Ribbon
	Measure→ Geometry analysis → Normal Deviation
Keyboard	**Textual Menu**
<QN>	Tools > Geometry Analysis > Normal Deviation

Upon entering the command, you need to specify edges, at which you need to measure the deviation of normals. To select all edges belonging to a certain face, use the option:

	<F>	Add/Remove selected Face

To select a specific edge, use the option:

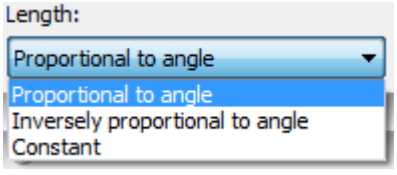
	<E>	Select Edge or cancel Edge selection

To cancel element selection, click it again. To cancel selection of all elements, use the option:

	<U>	Cancel selection

Upon selecting an edge, a quill diagram is displayed reflecting differences between the normal directions. The quill color varies according to the color scale defined on the tab "Colors". Working with this tab is same as in the command **QC: Show Curve Curvature.**

Depending on the difference in the face normals, the length of the quill lines may vary as well as the color. The way in which the length changes depending on the angle difference is defined by the parameter **Length**.



The length of a quill's lines can be proportional to the angle, inversely proportional to the angle, or constant.

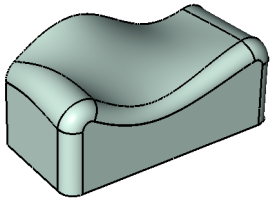
To measure normals deviation angle at a specific point on an edge, use the option:

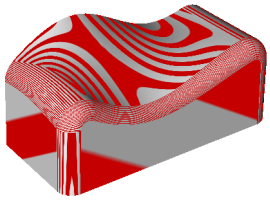
	<C>	Measure deviation at point

As the pointer approaches an edge, the angle value will be displayed next to the pointer. If you click this point on the edge, a 3D label will be created. Additionally, by default, the point coordinates and the angle value will be entered in the log on the tab **Results**. Working with the tabs **Results** and **3D Labels** is done same as in command **QC: Show Curve Curvature.**

SURFACE SMOOTHNESS

This command allows evaluating regularity of the model.



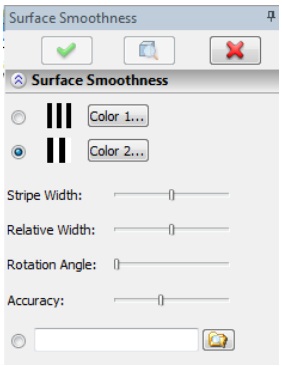


This type of rendering allows to visually determine whether the adjoining faces are connected smoothly with tangency transitions, or with continuous transition of curvature.

If all surfaces adjoin tangentially, then the texture stripes appear continuous, although may sharply change the direction. If the surfaces have smooth transition with continuous curvature, then the texture stripes are not just continuous, they also transit smoothly.

The command **QZ: Show Surface Smoothness (Zebra)** can be called via:

Icon	Ribbon
	Measure→ Geometry analysis → Surface Smoothness
Keyboard	**Textual Menu**
<QZ>	Tools > Geometry Analysis > Surface Smoothness



Right after entering the command, the model coloring changes. By default, it will be a two-color pattern. The colors used in this case are defined by the buttons **[Color 1]** and **[Color 2]**.

The width, relative width, the slant angle of the stripes and accuracy of their rendering are defined in the properties window. Use the appropriate sliders provided for this purpose.

Besides, you can specify your own file in bmp, jpeg, gif or png format, containing the pattern that you want to use for examining the model's smoothness. An input box is provided for this purpose at the bottom of the properties window.

However, you may find the default pattern most convenient. In this way, the higher is the curvature, the finer are the stripes of the pattern applied on the model. The colored stripes are widest on flat faces.

The stripes that paint the faces can be drawn in different directions. However, if the faces adjoin smoothly in terms of continuous tangency, the stripes are neither interrupted nor shifted with respect to each other, but can sharply change the direction. If the faces connect smoothly with continuous curvature, the stripes are not interrupted and smoothly change the direction.

Non-smooth connection faces

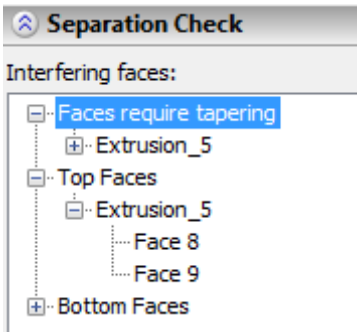
Smooth (continuous tangency) connection of faces

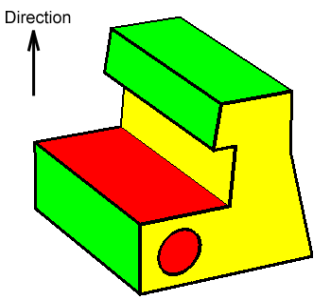
Smooth (continuous curvature) connection of faces

Rendering of the colored stripes in the 3D window is noticeably affected by the parameter **Mesh density**, defined on the tab **3D** in the document status. We recommend using highest accuracy of the mesh density when examining the model regularity.

Model Separation Check

This command serves for checking, whether an injection or die-cast mold used for making the model part is separable.





The command allows finding the faces of the regions preventing the mold from parting. Additionally, the faces are highlighted, whose taper angle with respect to the selected direction is less than specified.

The command **QS: Check Model Separation** is called via:

Icon	Ribbon
	Measure→ Geometry analysis → Model Separation Check
Keyboard	Textual Menu
<QS>	Tools > Geometry Analysis > Model Separation Check

Upon entering the command, you need to specify the direction of separating the mold. To select a direction-defining element, use the option:

	<D>	Click to select direction

When using this option, you can select faces, edges, 3D profiles, workplanes and coordinate systems.

To define the direction by specifying two 3D points, use the options:

	<S>	Select first Vertex or 3D Node
	<E>	Select second Vertex or 3D Node

For point selections, you can use 3D nodes, vertices, edges, coordinate systems, sphere or torus faces.

Upon choosing the direction, you need to select the operation, whose faces will be analyzed. To select an operation, use the option:

	<B>	Select source 3D operation

Select all operations in the scene using the option:

	<A>	Select all operations

To cancel an operation selection, you need to select it once again.

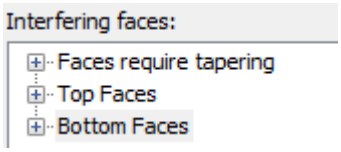
To cancel selection of all operations, use the option:

	<U>	Cancel selection

Upon completing the check, the operation's faces change in color. By default, highlighted red are the faces of the regions preventing the mold from parting. The yellow color marks the faces, whose taper angle with respect to the selected direction is less than specified. The green marks the faces that are sufficiently tapered with respect to the selected direction and do not prevent the mold from parting.

If multiple bodies are selected simultaneously, the system will run the separation check on all bodies together as if it was one part. Therefore, the results of checking several bodies separately and together will be different.

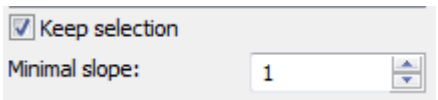
Besides the colored rendering in the 3D window, the check results are displayed in the **Interfering faces** pane. It contains overlapping faces (that prevent the part from ejecting) and faces with insufficient draft. *The core* and *the cavity* faces are also specified here.



The core and *the cavity* faces – not overlapping faces with sufficient draft that ensures that the part ejects properly from the tooling. These faces define the mold working region.

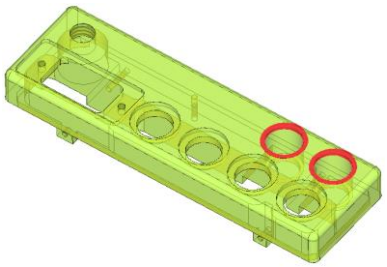
If you expand the list of reports and click the name of a problematic face, the face will be highlighted in the 3D window.

If the **Keep selection** flag is set, then, upon leaving the command, the highlighted faces will stay selected. This is done for convenience.



The parameter **Minimal slope** sets the minimum permissible taper angle of the faces with respect to the selected direction.

CHECKING MODEL CHANGES



The command for checking modifications of the model allows the user to reveal the differences between two solid bodies of the 3D model. With the help of this command, the user can compare different bodies or the same body at various stages of creating modifications. Also, upon the presence of the saved configurations in the model, you can compare between each other two versions of the same part. In addition, the command is able to calculate the volume of the modified space, which allows us to estimate the extent of differences more accurately.

The command **QA: Check Model Changes** can be invoked by the following means:

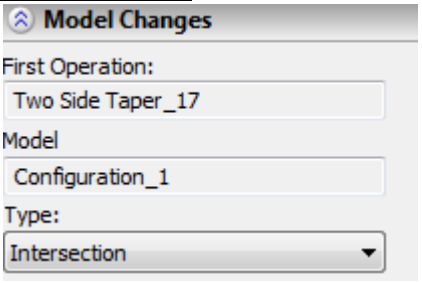
Icon	Ribbon
	Measure→ Geometry analysis → Check Changes
Keyboard	Textual Menu
<QA>	Tools > Geometry Analysis > Check Changes

After entering into the command, the user has to specify the model operations/configurations to be compared.

For comparing two bodies the following options are used:

	<F>	Select first Solid
	<S>	Select second Solid

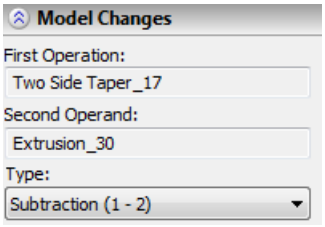
With the help of these options, the user can select different solid bodies of the same model as well as operations applied at different history levels to the same body. For selecting operations, you can use, for example, the tree of the 3D model. The selected operations are put into the fields **First operand** and **Second operand** in the command's properties window.



If in the current 3D model there are saved configurations, in the auto menu of the command the additional option will appear:

	<C>	Select Model Configuration

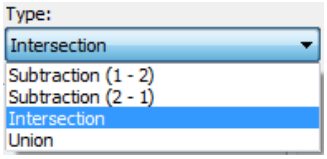
This option allows selecting one of the configurations saved in the model as the second operand for comparison.



The way of comparing the selected operands can be chosen in the dialog of the command's properties window with the help of the drop down list **Type**:

Subtraction (1-2). The result of comparison will be similar to the outcome of the boolean operation of subtraction of the first operand from the second;

Subtraction (2-1). The result of comparison will be similar to the outcome of the boolean operation of subtraction of the second operand from the first;



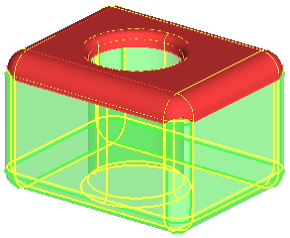
Intersection. The result of comparison will be similar to the outcome of the boolean operation of intersection of the first and second operands;

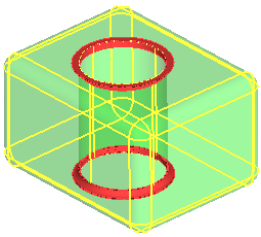
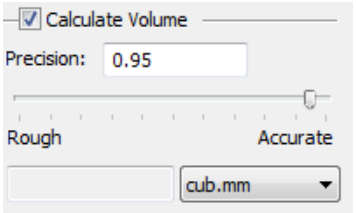
Union. The result of comparison will be similar to the outcome of the boolean operation of union of the first and second operands.

The way of comparing can be also selected in the auto menu of the command by turning on one of the following options:

	<->	Subtraction “1-2”
	<Ctrl><->	Subtraction “2-1”
	<*>	Intersection
	<+>	Union

After pressing the result of comparison of the selected operands is highlighted in the 3D window with the red color.





For calculating the volume of the modified space, you additionally need to set the flag **Calculate Volume** in the dialog of the command's properties. The precision and units for calculating the volume are set with the help of the group of parameters below the given flag.