

A large version of the TopSolid logo is centered on the page. It consists of the word "TopSolid" in a large, bold, italicized black font, with a red swoosh underline. Below the swoosh, the year "2021" is written in a smaller, grey, sans-serif font.

**TopSolid 7.15**  
**What's New**

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Version 7.15 Rev.02

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## Welcome to TopSolid 2021

Throughout this document, you will discover the main enhancements and learn about the new features and benefits of the latest **2021** release of **TopSolid 7**. The innovations described here only represent a small portion of the new functionalities.

If you are interested in finding out more about the newest **TopSolid 2021** features, please contact your local reseller.



## What's New in TopSolid'Design 7.15



This section describes the enhancements made to the **PDM, Design, Sheet Metal, Nesting, FEA, Drafting** and **Unfolding** applications in version **7.15** of **TopSolid 7**.

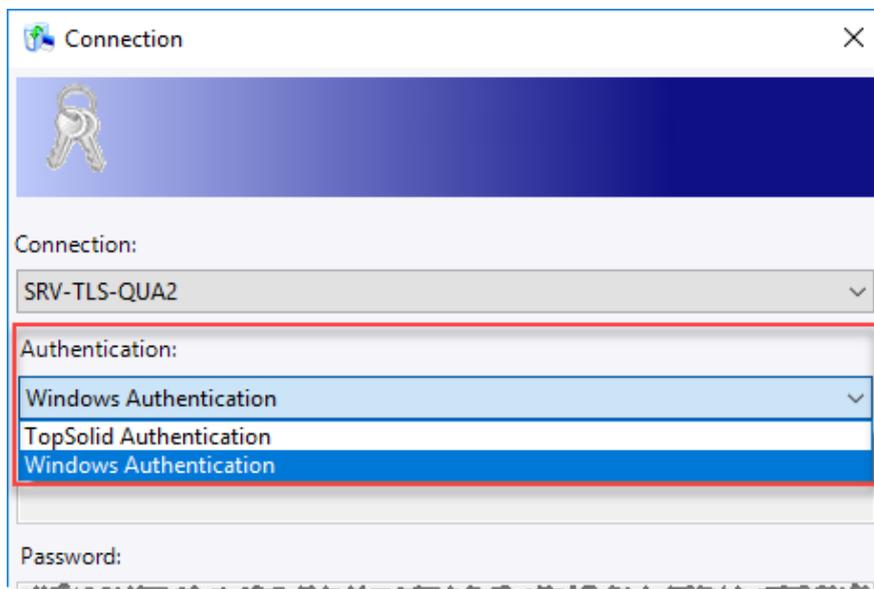
## PDM

### Multisite

**TopSolid'Pdm Server** now allows you to define a primary server and as many secondary servers as there are sites. This mode requires a specific license, a fast Internet connection (fiber optic, SDSL, VDSL, 5G, 4G, etc.), as well as the implementation of a VPN tunnel between the different sites. The data is then automatically replicated between the different servers.

### User management

The connection to the PDM server can now be established using the list of users defined in the Windows Active Directory. This simplifies the administration of **TopSolid**. All Active Directory information (users' last name, first name and e-mail address, whether or not the user belongs to a user group, etc.) is retrieved directly from the domain controller.



# Sketch

## Dialog box

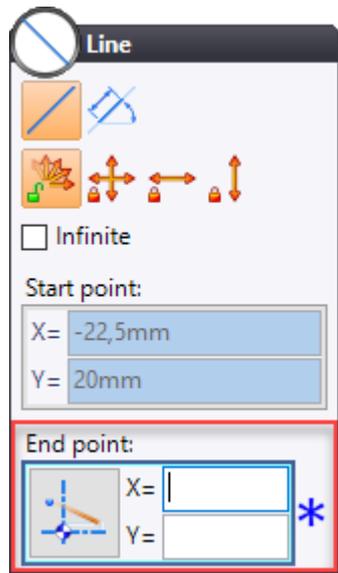
In the **Line**, **Rectangle**, **Circle** and **Arc** command's dialog boxes, the radio buttons have been replaced by icons.



## Relative coordinates

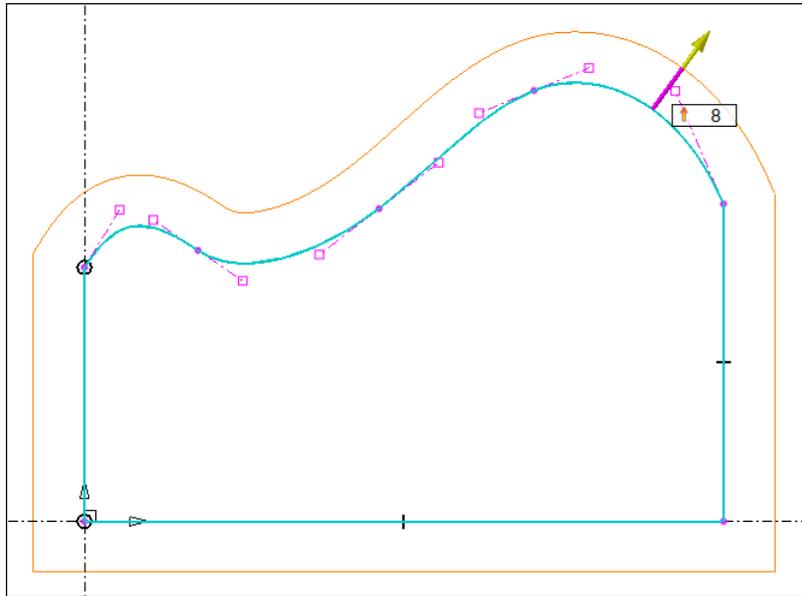
The **End point** button has been added to the **2D Sketch > Contour** and **2D Sketch > Line** command's dialog boxes, and also to the **3D Sketch > Contour** and **3D Sketch > Line** command's dialog boxes.

This option allows you to indicate the end coordinates of the segment either in relation to the sketch origin, or in relation to the last point.



### Offset

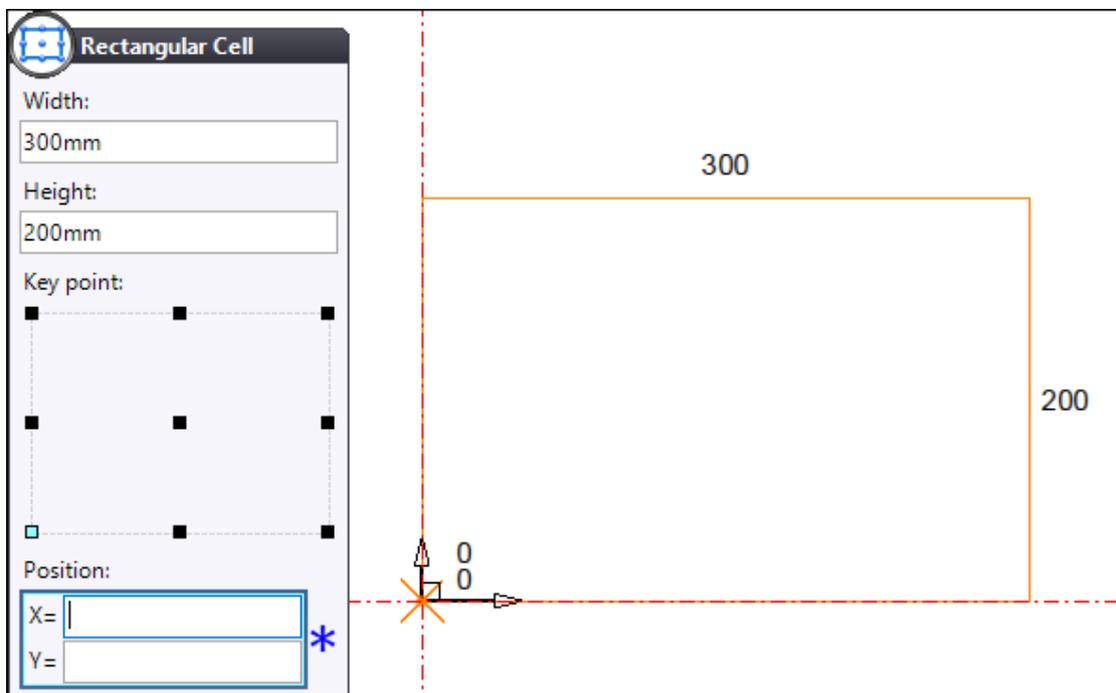
The **2D Sketch > Offset** command now supports the spline profiles. You no longer have to perform a sketch operation. However, you must check the **Solving with profiles** box in the **2D Sketch > Options** command.



Example of offsetting a profile containing a spline.

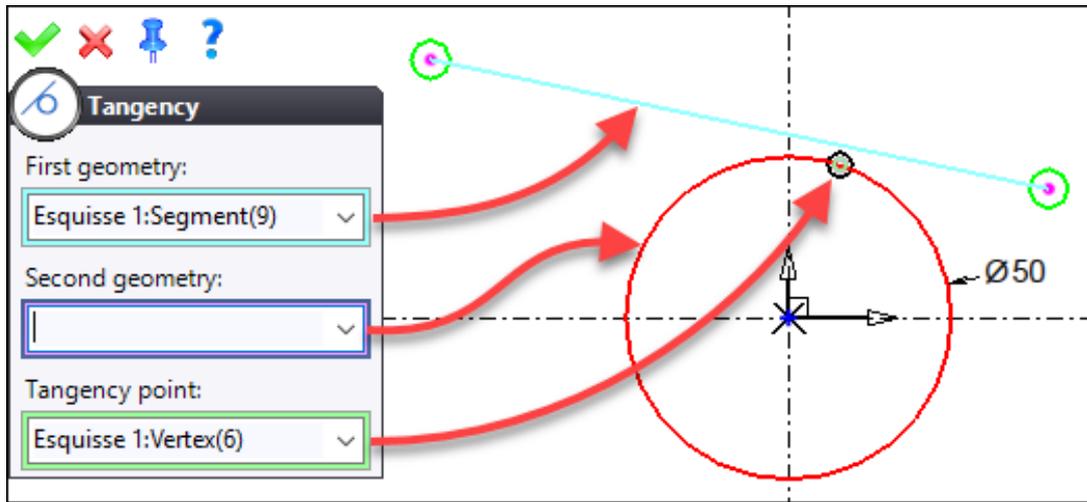
### Cells

The new **2D Sketch > Rectangular Cell** command allows you to create a rectangular profile and position it using one of its key points (top left, center left, bottom left, top right, etc.), in much the same way as for an extruded bar.



## Tangency

When creating or editing a tangency constraint, you can now indicate the tangency point. However, it must be selected before the second geometry.

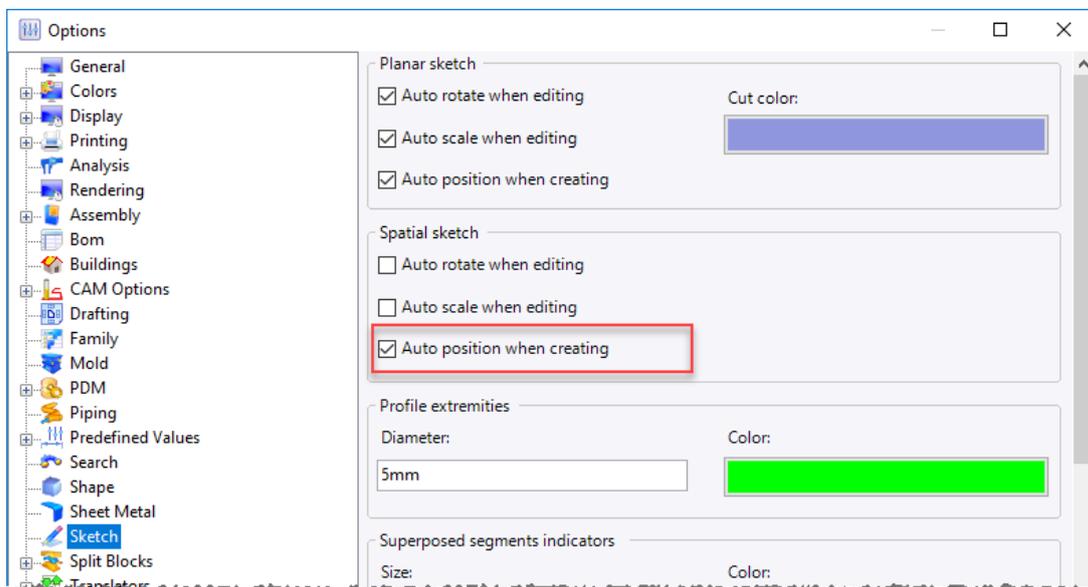


## Hypertext link

Please refer to the *Hypertext link* section of the *Drafting* chapter.

## Positioning a spatial sketch

The **Auto position when creating** box has been added to the **Tools > Options > Sketch** section. As with a planar sketch, it allows you to automatically position the 3D sketch either on the absolute frame, or on the selected face.



*3D sketch automatic positioning option.*

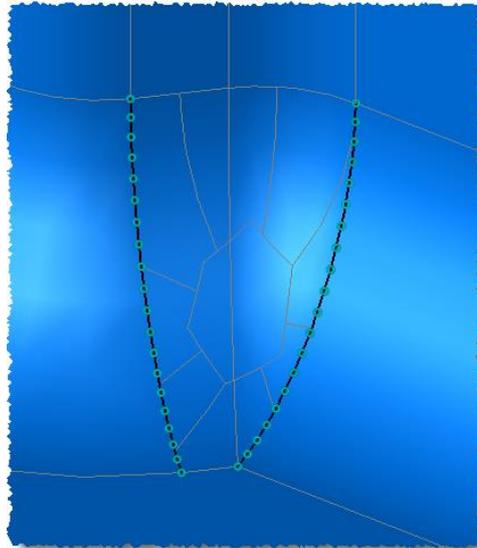
In addition, when the automatic positioning options are unchecked, the question is now asked all the time, even when creating the first sketch of the document.

## Point repetition

Points are now supported by the **2D Sketch > Operations > Repetition** and **3D Sketch > Operations > Repetition** commands.

### ***Seedling on profile***

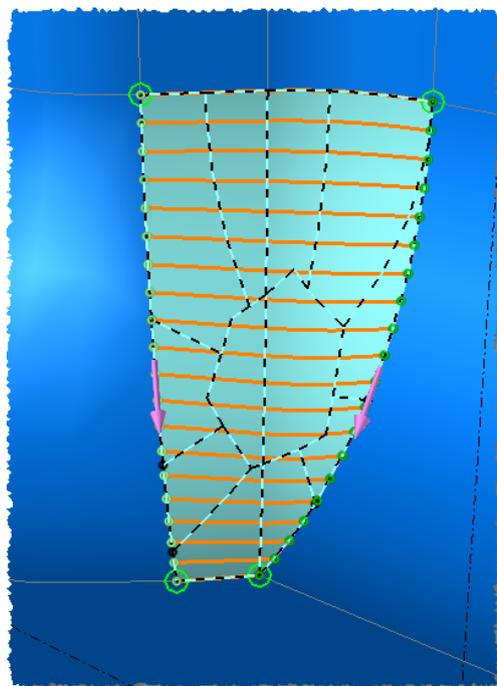
The new **3D Sketch > Operations > Seedling on Profile** command allows you to distribute points either randomly, or spaced by a curvilinear length, or along a direction on a profile. The created points can be used to create multiple intersection profiles.



*Example of creating point seedlings.*

### ***Multiple intersections***

The new **3D Sketch > Operations > Multiple Intersection** command allows you to create the intersection profiles between a shape or a set of faces and parallel planes or guide profiles. The created profiles can be used to define a shape using the **Surface > Fitting** command.

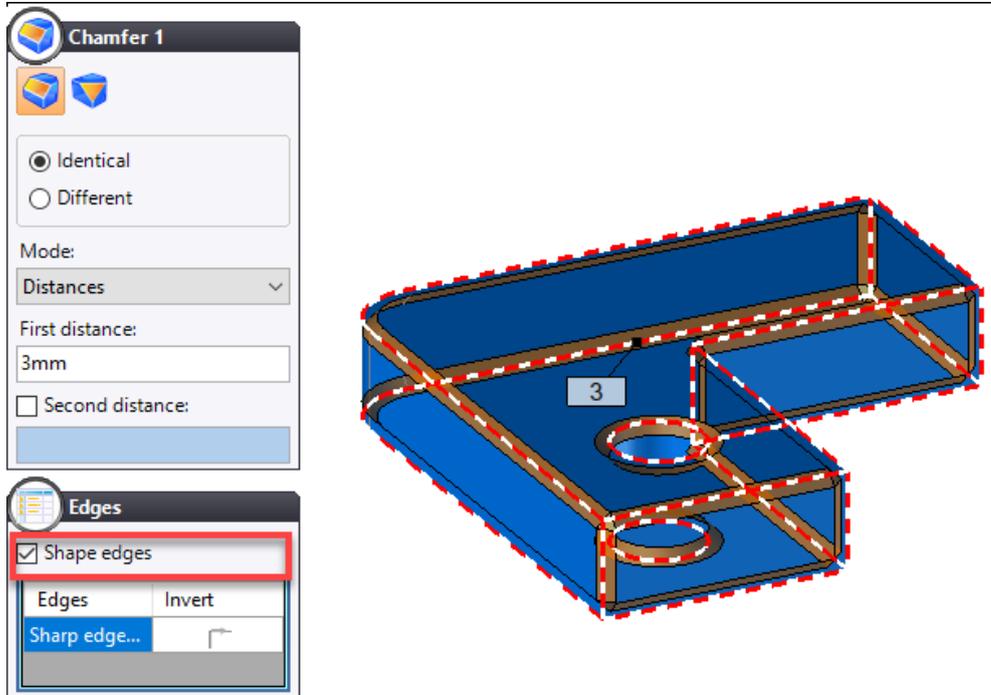


*Example of creating multiple intersections.*

# Shape

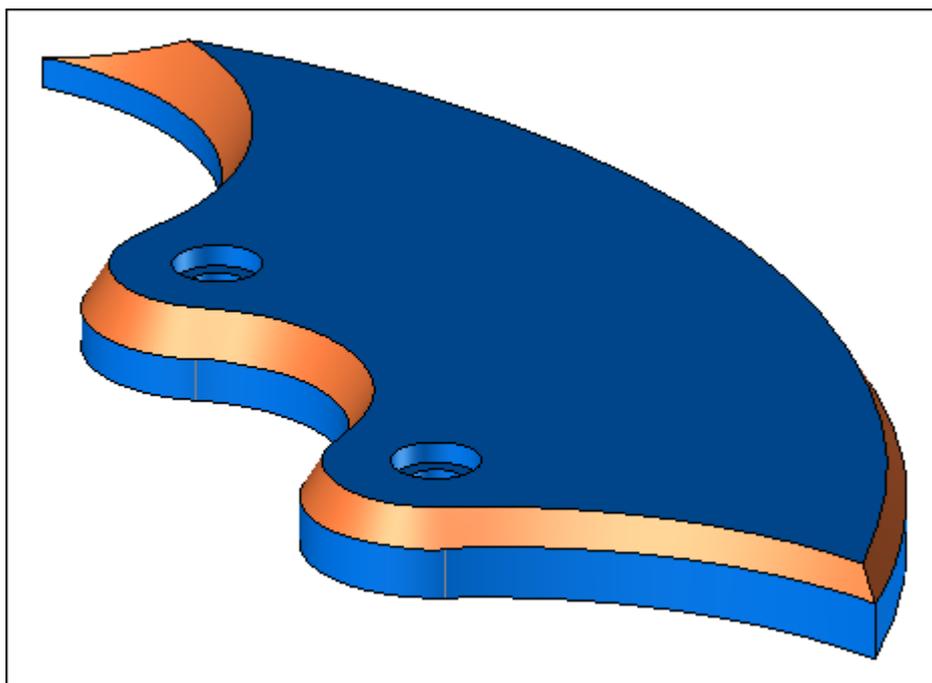
## Chamfer

In the **Edges** section of the **Chamfer** command, the **Shape edges** box allows you to select all the sharp edges of a shape.



*Example of a chamfer on all edges.*

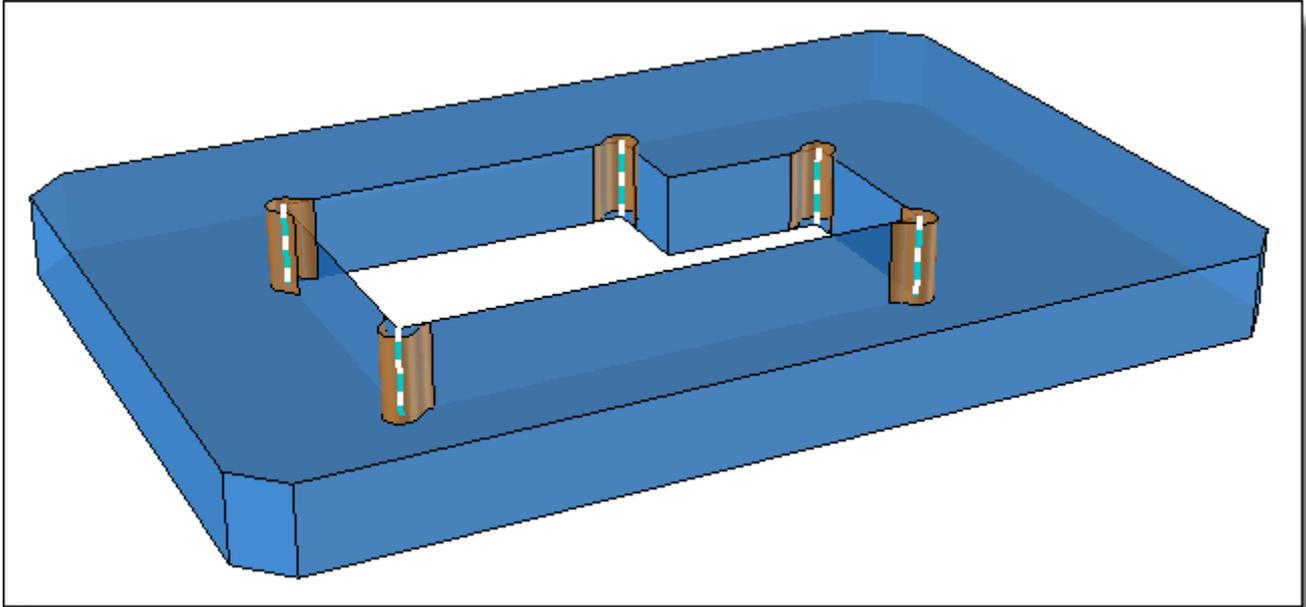
In addition, the **Variable Chamfer** command has been added to the **Shape > Other Operations** menu. It allows you to create a chamfer by indicating different values at the ends of the selected edge.



*Example of a variable chamfer.*

## ***Angle clearance***

The new **Shape > Angles Clearance** command allows you to create the angle clearances of the pockets according to four different modes: **Fillet**, **Chamfer**, **Symmetrical** and **On face**.



## ***Drilling group***

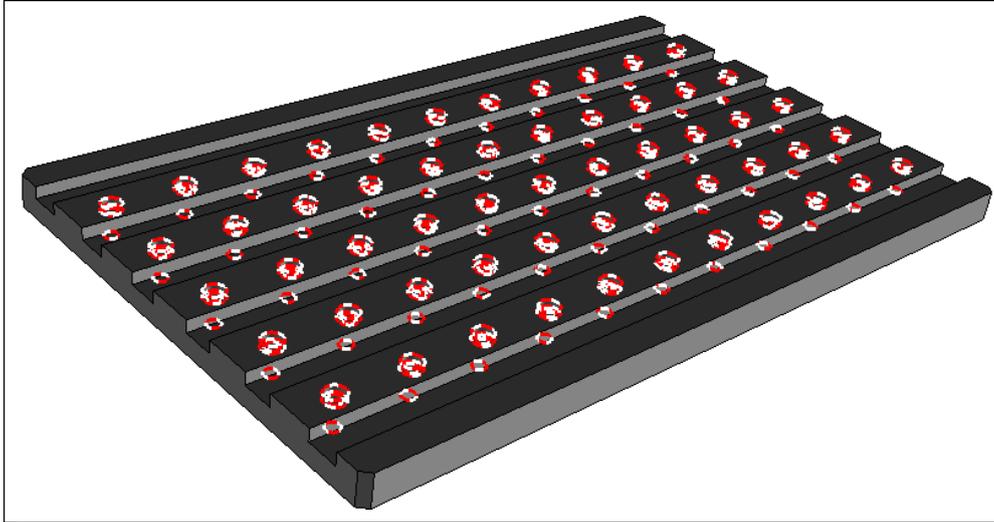
The **Shape > Drilling Group** command now supports the **manufacturing features (MF)**.

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## FreeShape

### Drillings

In the **FreeShape > Extract Drillings** command, the new **Extended search** advanced option allows you to extract groups of drillings located at the same altitude but on different faces.

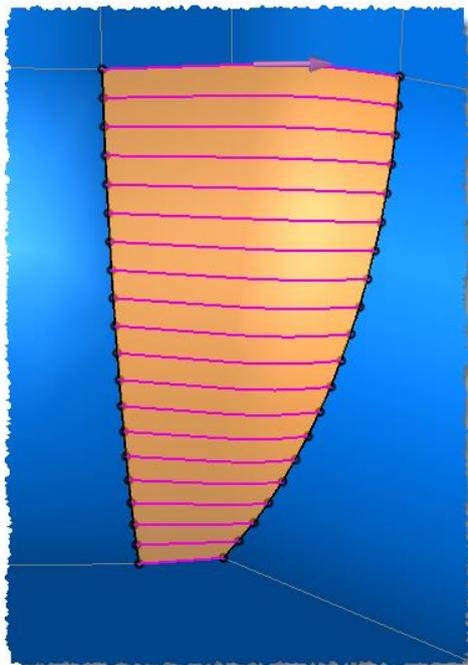


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## Surface

### Fitting

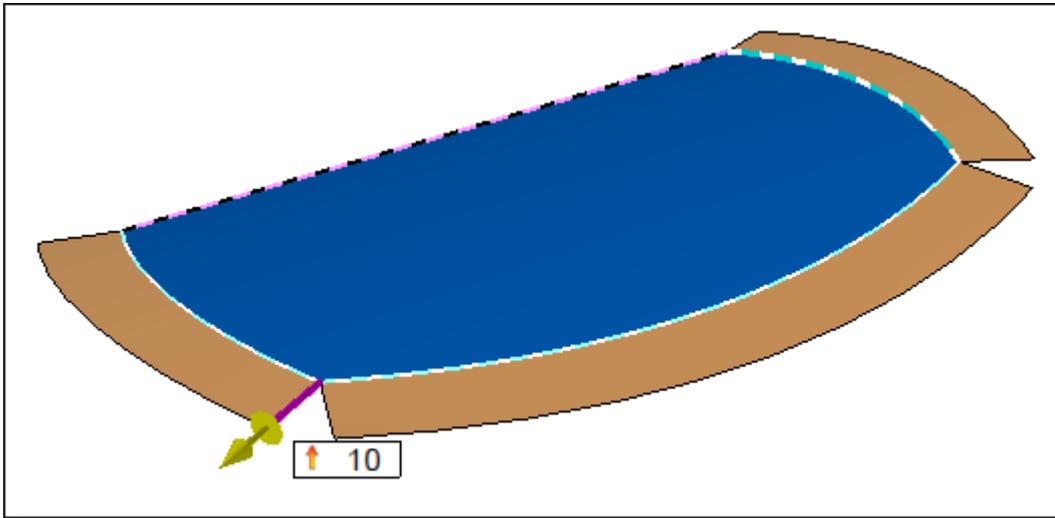
The new **Surface > Fitting** command allows you to create a surface from a set of profiles and possibly guide profiles, as well as passing points. The **Extension** section allows you to extend the resulting surface at the start and at the end.



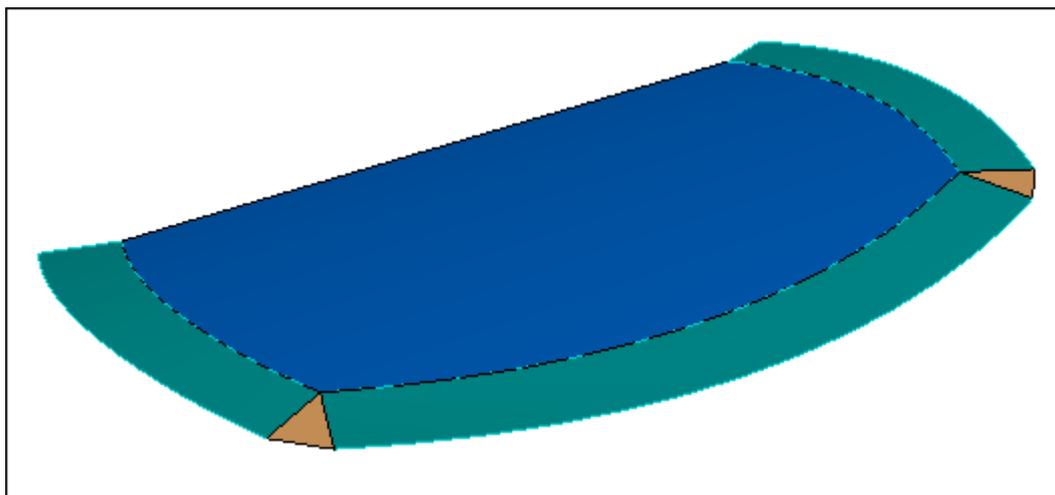
*Example of creating a surface on a set of profiles.*

### Tangent surfaces

The dialog box for the **Surface > Tangent Surfaces** command has been modified. It now takes the form of a wizard and the second stage of the operation in the dialog box provides options for reciprocal trimming and corner filling operations. The generated shapes are regrouped in a folder of the Operations tree.



*Selection of the face and the edges in the tangent surface creation wizard.*



*Result with the corners filled.*

## Sheet Metal

### Sheet metals recognition

#### Only allowed thicknesses

In the dialog box of the **Sheet Metals Recognition** command, the new **Only allowed thicknesses** option allows you to consider only the parts whose thickness is available in the list of allowed thicknesses.

You can predefine the allowed thicknesses in the **Tools > Options > Sheet Metal** command.

#### Minimum size allowed

The new **Minimum size allowed** option allows parts that could have been considered as sheet metal parts to be considered as non-convertible. For example, a washer with a constant thickness that is included in the list of thicknesses to be considered. If the minimum dimension of its enclosing box is less than the allowed one, it will be excluded from the list of parts to be converted.

#### Identify tubes and standards profiles

In addition to identifying the parts to be converted into sheet metal parts, the **Sheet Metals Recognition** command is now able to distinguish the parts that can be converted into **tubes** and/or **profiles** in the parts list of an assembly. The parts converted into tubes and profiles are automatically **ignored** when creating the unfoldings.

#### Conversion into unfoldings

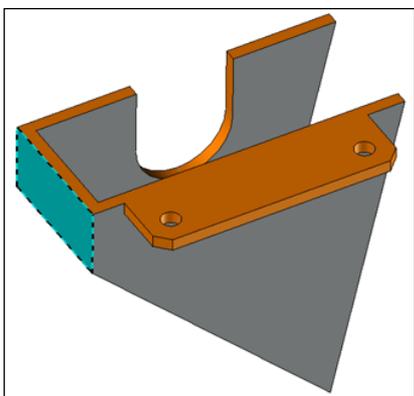
The **Conversion in Unfoldings** command allows you to manage AutoCad imports containing multiple unfoldings in order to create unfolding documents for each part.

Previously, performance was problematic on certain types of parts, especially the parts containing profiles with many segments or parts including many small, non-circular holes. The processing times have been improved in version 7.15. In addition, an option allowing you to clean the imported sketches is now available.

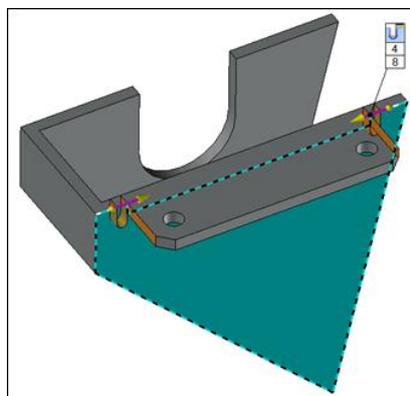
#### Notches

The new **Notches** command allows you to add notches on shapes, before or after converting a shape into a sheet metal part.

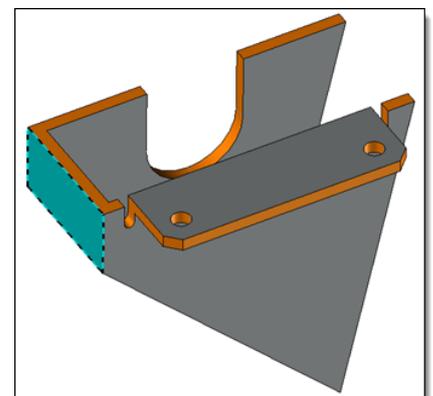
Adding notches on parts with ambiguous modeling allows a correct recognition of the side faces, and thus a good conversion into a sheet metal part.



*Ambiguous modeling: poor recognition of the side faces.*



*Creation of notches.*



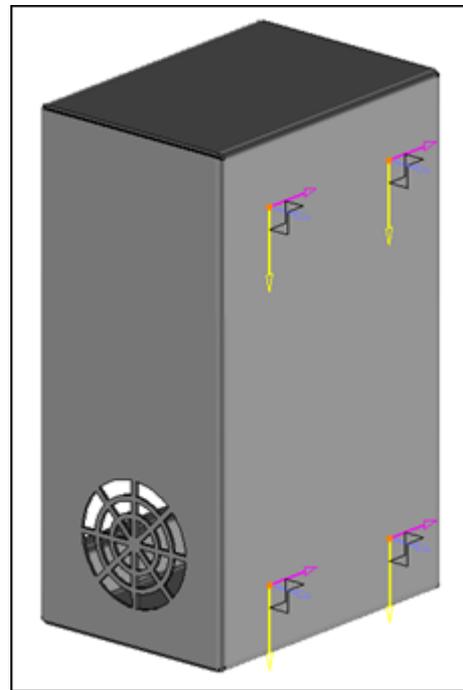
*The face recognition and the conversion into a sheet metal part are correct.*

### Internal forming removal

The new **Internal Formings Removing** command allows you to remove the internal formings on one face of a shape. In the case of **non-sheet metal** shapes, this simplification operation can thus facilitate the creation of a **sheet metal by recognition**.



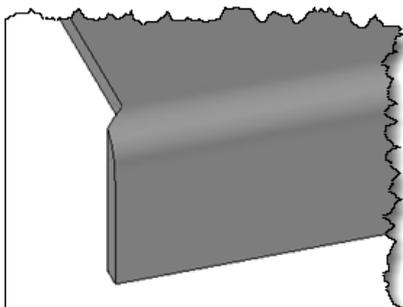
Part before removing the formings.



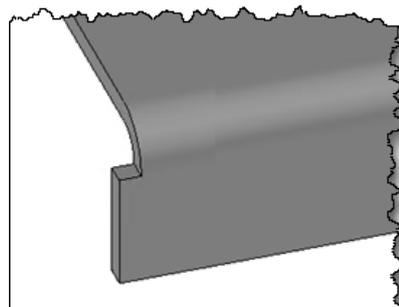
Simplified part. The formings have been removed and frames have been created on one of the faces.

### Bend replacement

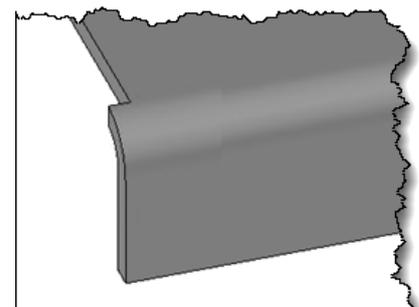
The **Bends Replacement** command allows you to **extract** or **replace** bends on parts if the modeling of these parts is not perfect. This command can be applied to a sheet metal or a **non-sheet metal** shape.



Part before replacing the bend.



Bend replacement in **Minimum** mode.



Bend replacement in **Maximum** mode.

### Ignore for unfolding

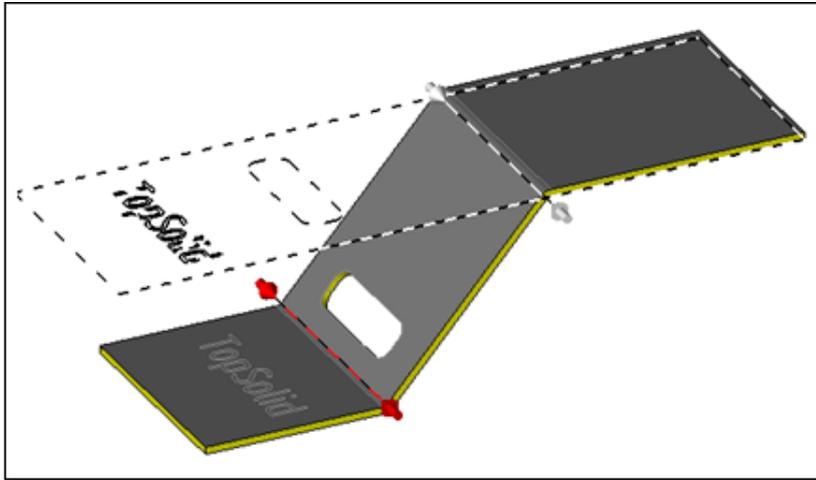
The new **Ignore for unfolding** command allows you to declare the part document as to be ignored for the unfolding process. When the **Ignore for unfolding** box is checked, a parameter is created in the part document. If a part document contains this parameter:

- the creation of the unfolding document by right-clicking on the part document's tab (contextual menu) displays a warning message and is rejected;
- the inclusion of the part document in an unfolding document displays the same warning message;
- the multiple unfoldings generated from an assembly that contains parts with this parameter will not display these parts in the list of parts to be unfold.

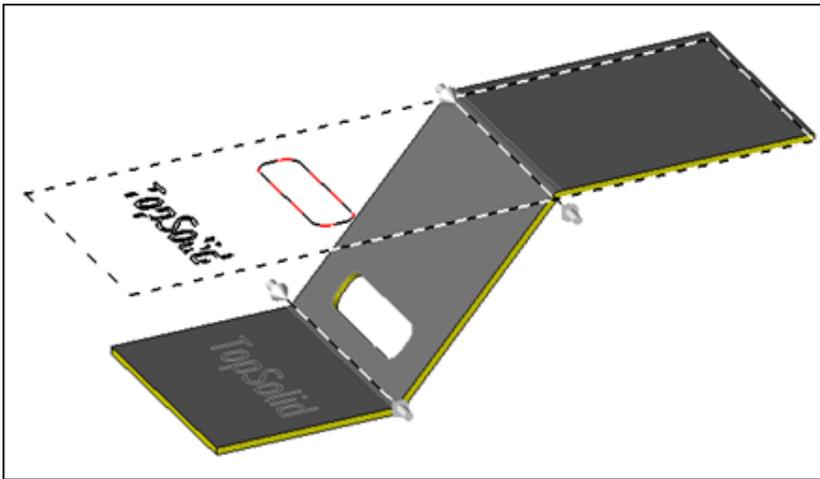
### Generation of sheet metal manufacturing features (MF)

In the **Analysis** stage, the **Sheet Metal Properties** operation now generates **manufacturing features (MF)** if the

 **Create MF** option is enabled.

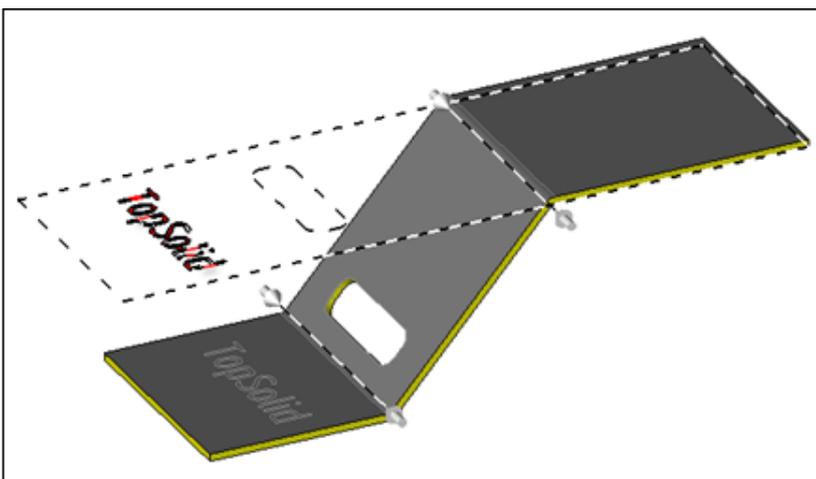


The bending MFs are represented by the bend line and arrows at the ends on the sheet metal part.



The cutting MFs are shown on the flat shape of the sheet metal part.

A MF is created for each opening and a MF is created for the external cutting contour.

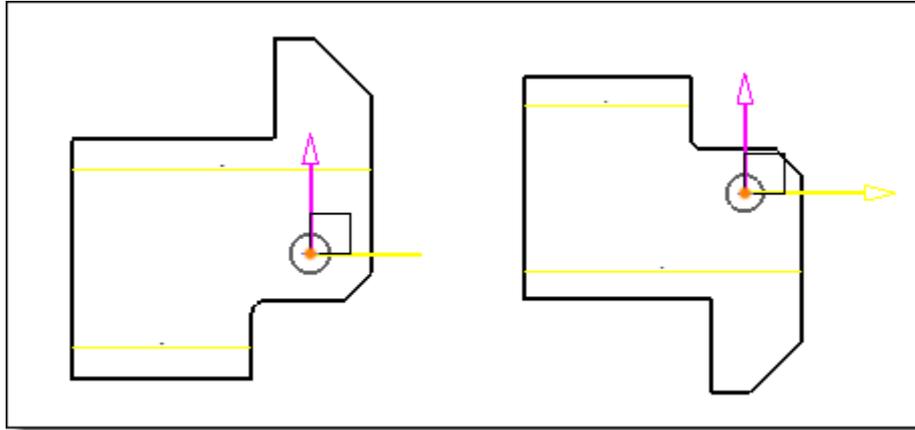


The engraving MFs are shown on the flat shape of the sheet metal part.

### ***Unfolding a mirror part***

The **Mirror Part** command has been removed from the **Tools > Derivations** menu in the unfolding document. In fact, it resulted in a part document, which was incorrect.

It has been replaced by the **Tools > Derivations > Mirror Part Unfolding** command which allows you to create the mirror part of the part included in the unfolding document and to create a new unfolding document from the new mirror part.

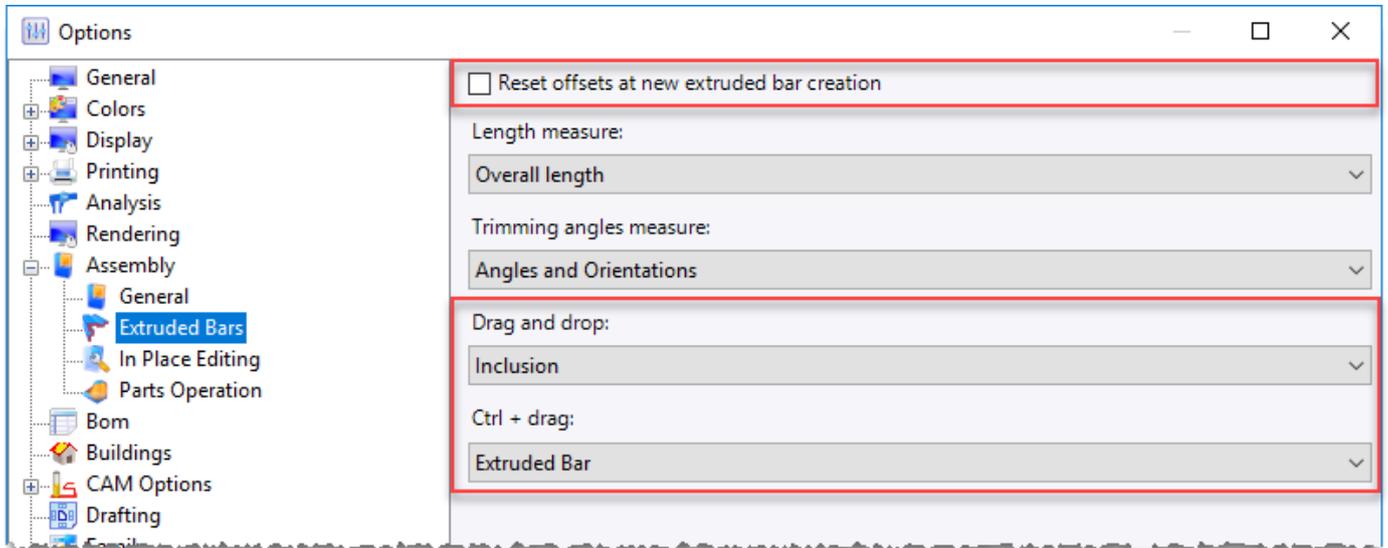


*Unfolding and mirror part unfolding.*

# Assembly

## Extruded bars

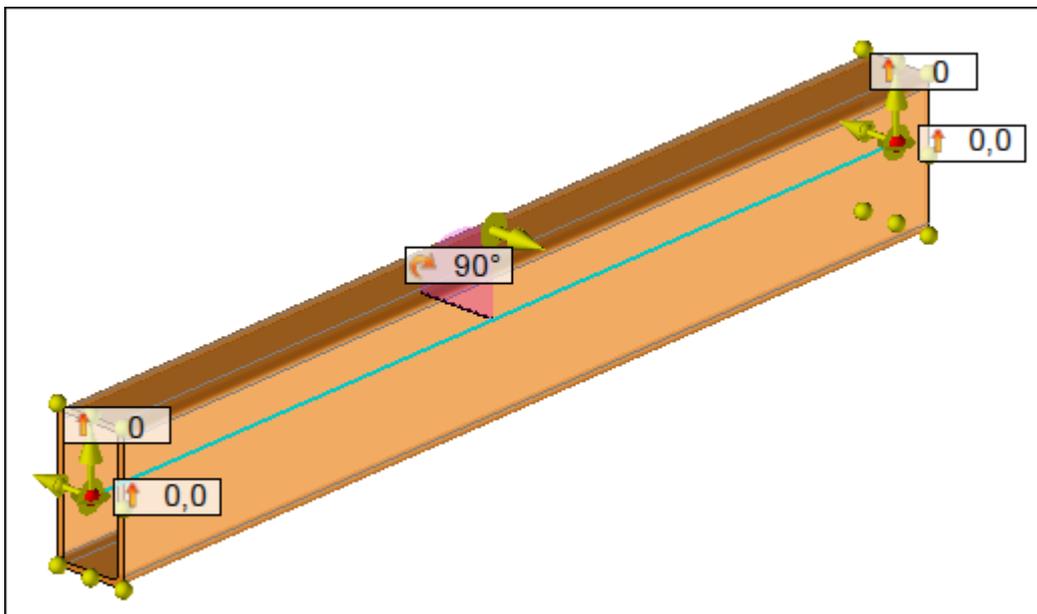
In the **Tools > Options > Assembly > Extruded Bars** command, you can now choose the command to be launched (**Assembly > Inclusion** or **Modeling > Extruded Bar**) when **dragging and dropping** an already included extruded bar from the Project tree, as well as when duplicating an extruded bar using the **Ctrl** key and **dragging** the mouse.



The positioning and shifting handles are now placed at the ends of the extruded bar being included, while the rotation handle is positioned in the middle.

By default, the shift values are kept when a new extruded bar is included.

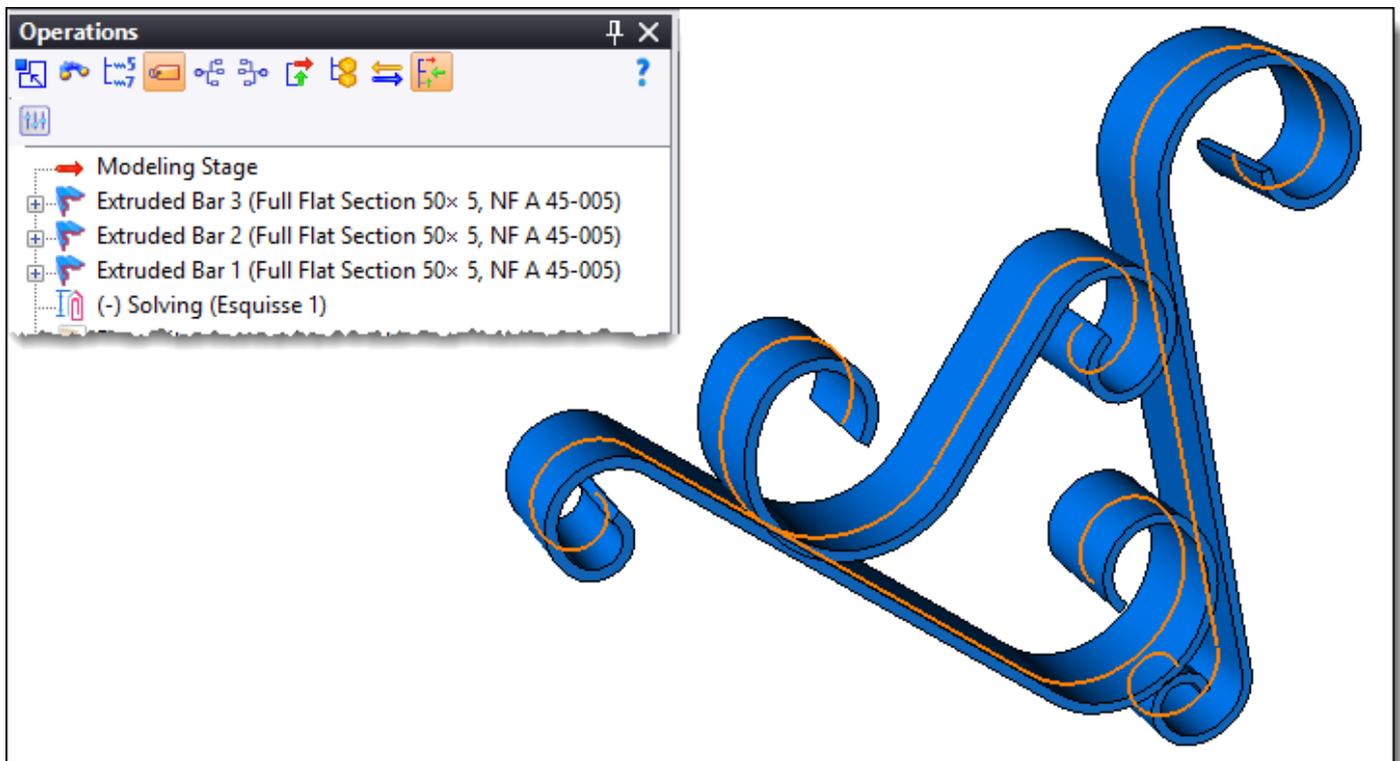
However, you can systematically reset these values to zero by checking the new **Reset offsets during the new extruded bar creation** box available in the **Tools > Options > Assembly > Extruded Bars** command.



*The new extruded bar handles are placed at each end of the extruded bar.  
The rotation handle is located in the middle of the extruded bar.*

### Curved extruded bars

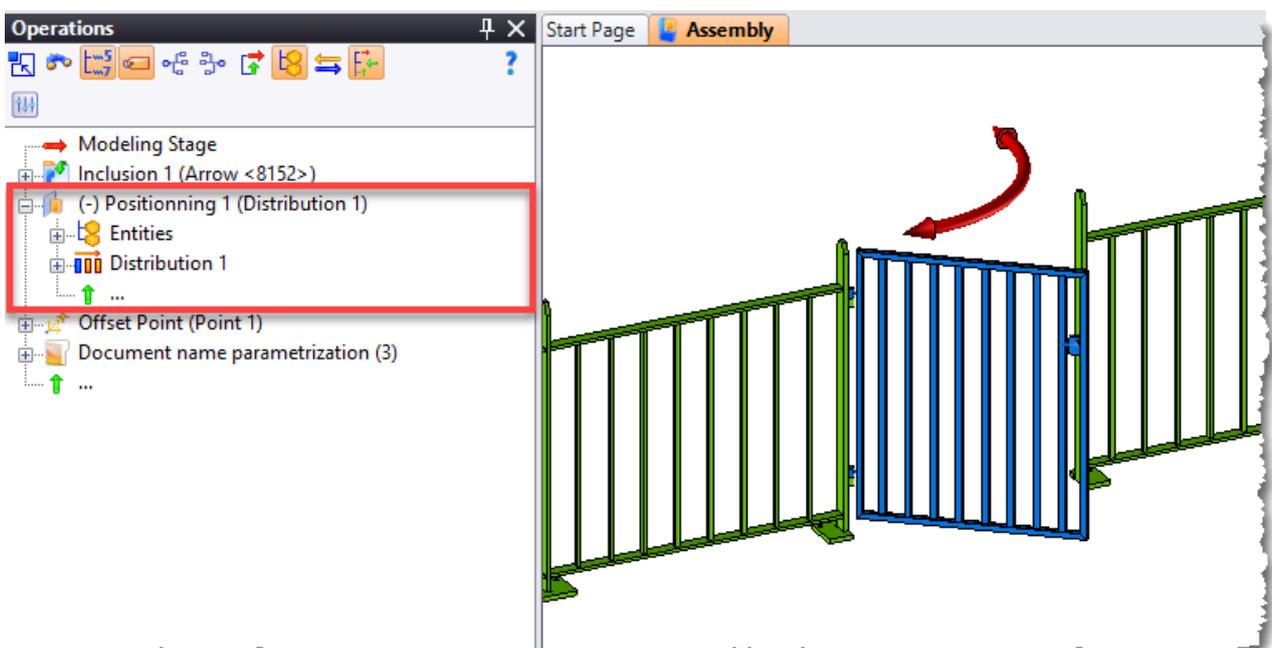
Extruded bar components can now be included on a 2D or 3D sketch containing curves. In addition, when including an extruded bar on a sketch, the tangent sections are now considered as a single curved extruded bar.



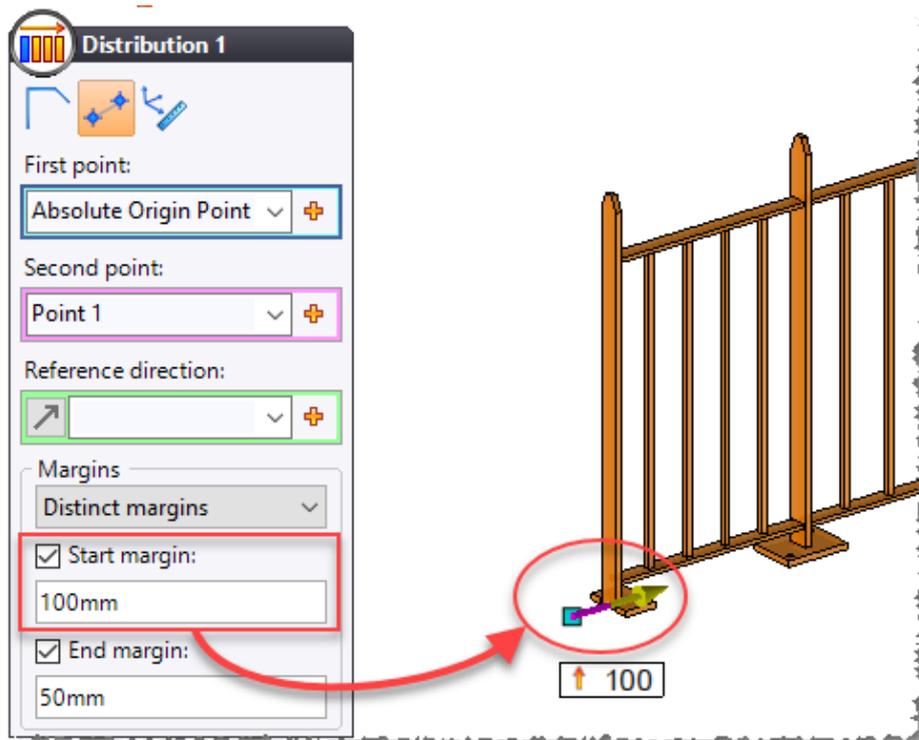
Example of including extruded bars on a sketch with tangent sections.

### Distribution

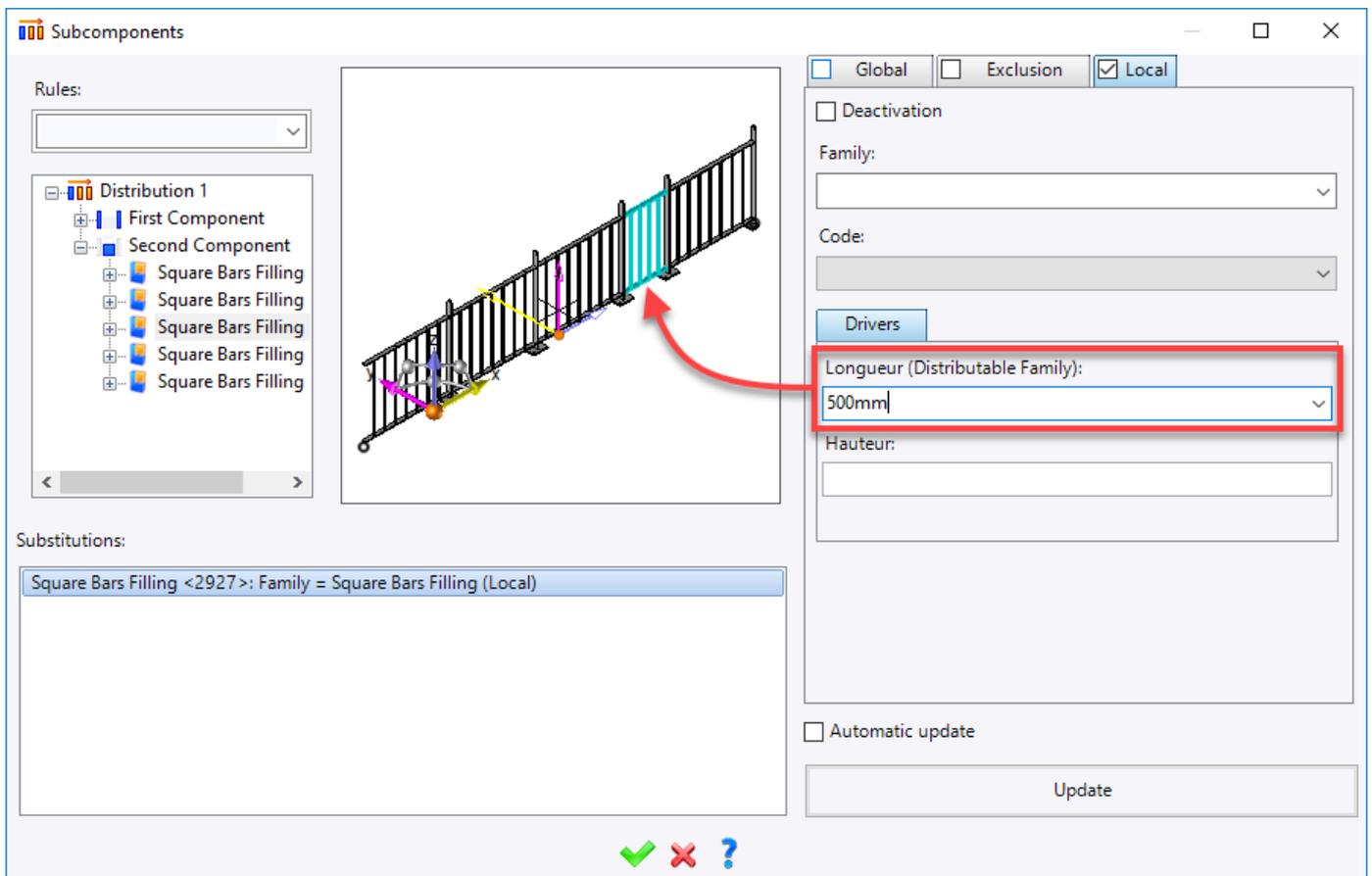
When distributing a component, if it is articulated, a positioning operation is automatically added so that the articulation can be played.



A margin management option has been added and is identical to the margin management of a constrained linear pattern.



The **Subcomponents** button available in the **First Component** and **Second Component** sections now allows you to modify the length of a distributed component.

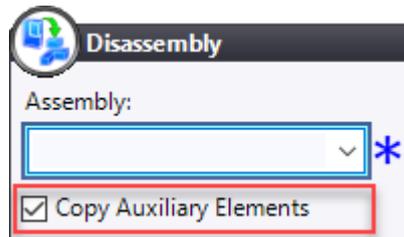


## ***Inclusion with geometric drivers***

When including an articulated component, a positioning operation is automatically added so that the articulation can be played. If the component is not articulated, only the inclusion operation is created.

## ***Disassembly***

In the **Modeling > Disassembly** command, the new **Copy Auxiliary Elements** box allows you to retrieve the auxiliary elements from the "L-1" level sub-assembly in the "L" level assembly document. These elements are then stored in the **Auxiliary Elements** folder of the Entities tree.



## ***Decorrelation***

During a decorrelation operation, the elements hooked to the neighborhood are now basified and the neighborhood is deleted.

## ***Local part repetition***

The repetition of a local part (from a serialization for example) now results in an occurrence of the local part, which allows performance improvement as well as a decrease in the size of the document.

## ***Process server***

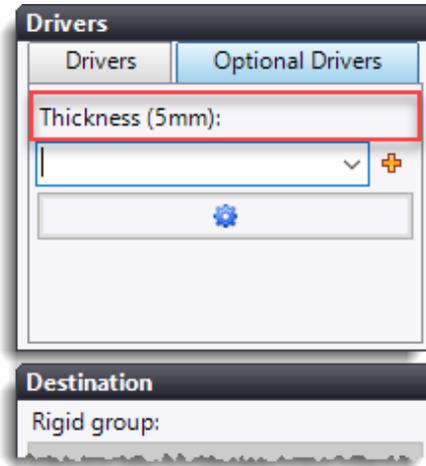
In order to improve performance when updating the document, the process operations for the identical components (same family, same code, same drivers and same processes) are performed only once.

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# Family

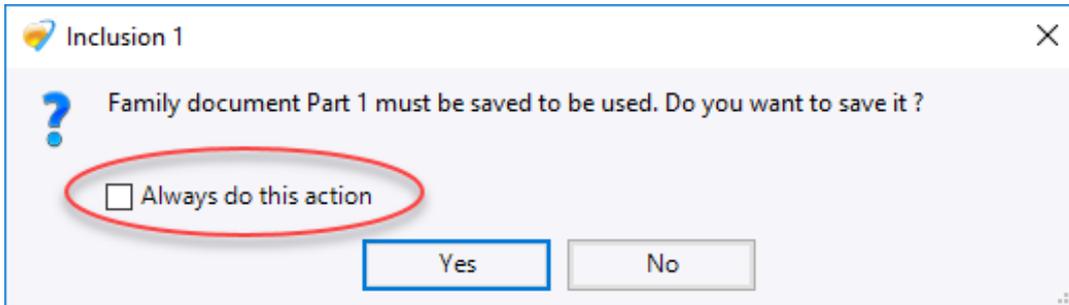
## Optional drivers

The default value of an optional driver is now displayed when including the family (or the symbol).



## Modification

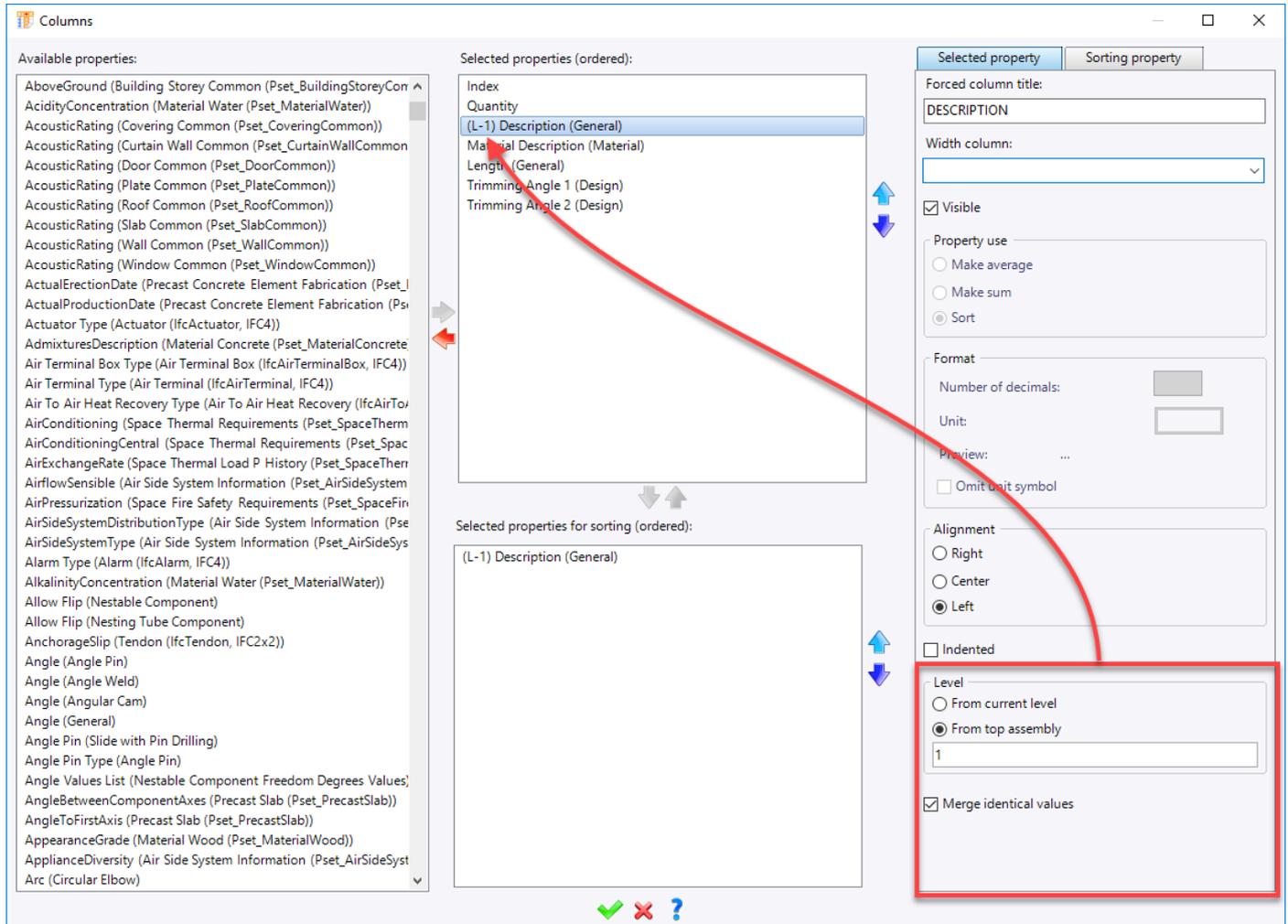
When opening an assembly document using a modified but unsaved family document, **TopSolid** displays a message asking you if you want to save the family document. A checkbox to always perform this action is now available.



# Bill of Materials

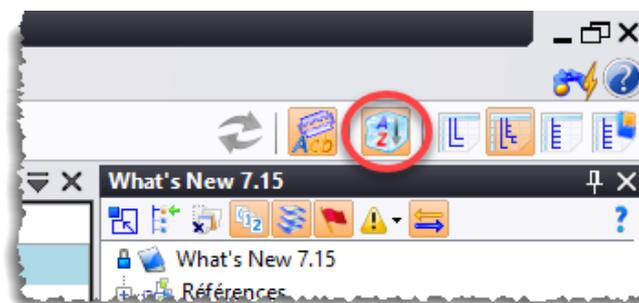
## Retrieving properties

When defining a column, the new **Level** section allows you to retrieve a property in a higher level of the assembly's tree structure (Part - Sub-assembly - Assembly).



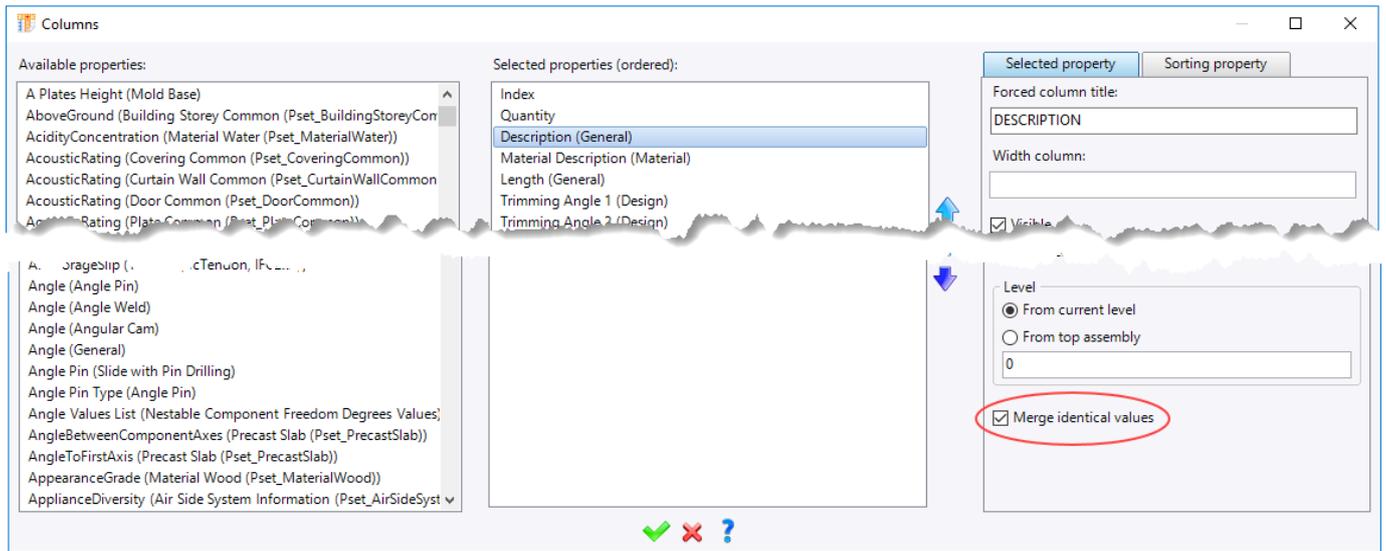
## Sorting

The new **Sort Frozen** mode available in the icon bar at the top right of the **Bill of Material** menu saves you from updating the bill of materials when modifying the part properties.



## Columns

The **Merge identical values** box has been added to the **Bill of Material > Columns** command. It allows you to obtain a BOM table with merged cells in the drafting document.



ID	QTY	DESCRIPTION	MATERIAL	LENGTH	ANGLE 1	ANGLE 2
101	1	Wood Door	Acacia	540.0mm	-	-
102	1	Top Plate	Acacia	150.0mm	-	-
103	1	Chaise	Acacia	470.0mm	-	-
104	2	Sheetmetal Screw (D07051-D70.0-05-C-Z)	Acacia	-	-	-
105	1	Right Plate	Acacia	470.0mm	-	-
106	1	Strip L4x.5x.144.3 millie+74.5 x44.3 v02	Acacia	-	-	-
107	1	Acacia	Acacia	480.0mm	45.0°	45.0°
108	1	Acacia	Acacia	440.0mm	45.0°	45.0°
109	1	Acacia	Acacia	480.0mm	45.0°	45.0°
110	1	Acacia	Acacia	480.0mm	45.0°	45.0°

ID	QTY	DESCRIPTION	MATERIAL	LENGTH	ANGLE 1	ANGLE 2
112	1	Hollow Wall	Acacia	350.0mm	0.0°	0.0°
113	1	Hollow Wall	Acacia	470.0mm	0.0°	0.0°
114	1	Hollow Wall	Acacia	470.0mm	0.0°	0.0°
115	1	Hollow Wall	Acacia	550.0mm	0.0°	0.0°
116	1	Hollow Wall	Acacia	550.0mm	0.0°	0.0°
117	1	Hollow Wall	Acacia	470.0mm	0.0°	0.0°
118	1	Hollow Wall	Acacia	470.0mm	0.0°	0.0°
119	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
120	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
121	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
122	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
123	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
124	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
125	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
126	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
127	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
128	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
129	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
130	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
131	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
132	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
133	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
134	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
135	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
136	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
137	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
138	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
139	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
140	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
141	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
142	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
143	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
144	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
145	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
146	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
147	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
148	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
149	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
150	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
151	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
152	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
153	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
154	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
155	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
156	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
157	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
158	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
159	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
160	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
161	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
162	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
163	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
164	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
165	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
166	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
167	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
168	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
169	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
170	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
171	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
172	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
173	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
174	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
175	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
176	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
177	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
178	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
179	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
180	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
181	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
182	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
183	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
184	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
185	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
186	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
187	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
188	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
189	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
190	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
191	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
192	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
193	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
194	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
195	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
196	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
197	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
198	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
199	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°
200	1	Hollow Wall	Acacia	450.0mm	0.0°	0.0°

Example of a bill of materials with merged identical values in the **Description** and **Material** columns.

In addition, when a table containing merged cells is split, the split operation can now be performed on the merged cells using the new **Split merged cells** option.

This option is also available in the **Detailing > Drillings Table** and **Detailing > Coordinates Table** commands.

## Parameter from the assembly

The **Parameter from Assembly** command has been added to the **Construction > Parameters** menu. It can be used to retrieve parameters from the assembly document, which allows you, for example, to parameterize the name of the BOM document in the same way as that of the assembly.

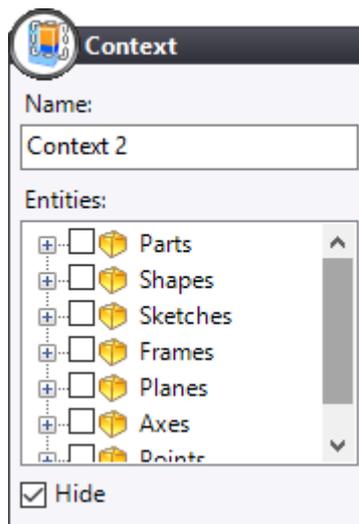
## Workspace

### Area

The dialog box of the **Workspace** command has been modified and now allows you to select 2D and 3D elements (sketches, frames, planes, axes and points). These elements are presented in a tree. They can also be selected from the graphics area or the Entities tree.

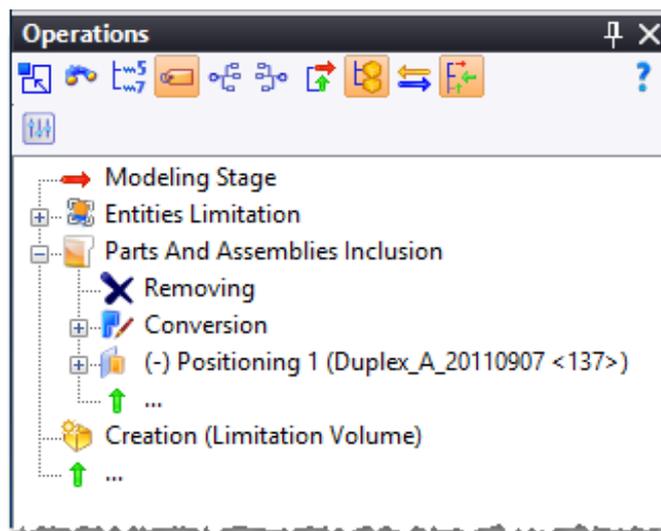
### Context

The new **Context** command allows you to create contexts. Its operating principle is the same as when you create a set.



### Context management

In the workspace background documents generated by the Workspace Manager, the operations are now clearly identified.



## Visualization

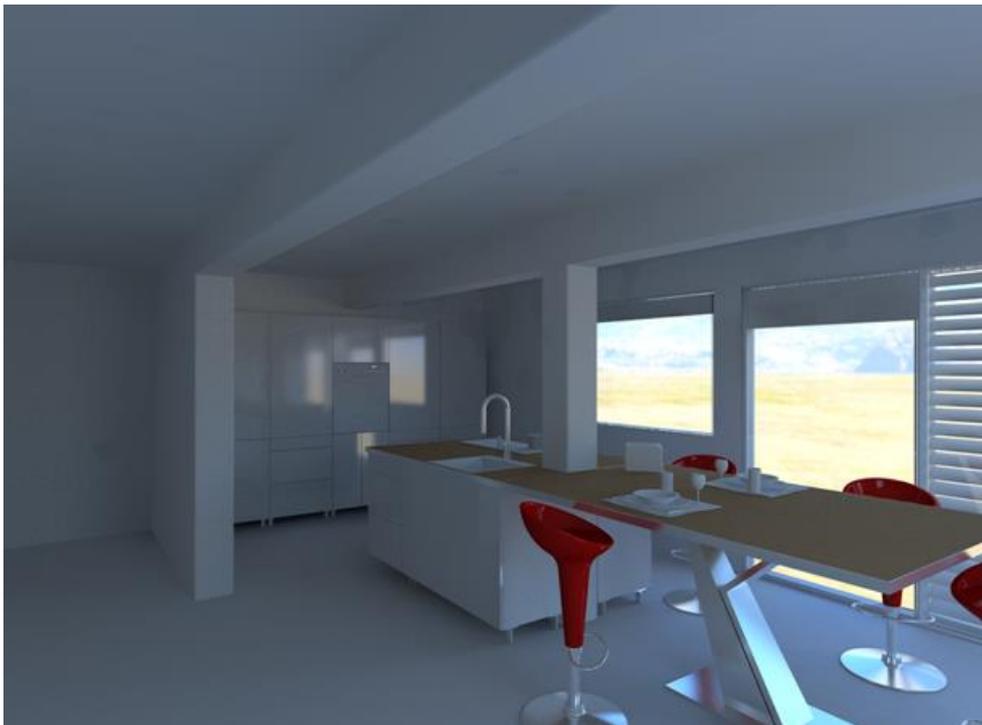
### Vertical locking

The new **Lock Vertical** mode locks the Z axis of the absolute frame when the view is dynamically rotated. The Z axis then remains collinear with the Z axis of the screen.



### Emissive environment

An environment can now be defined as emissive, i.e. it will behave like a light source. It can then replace a sky light source.



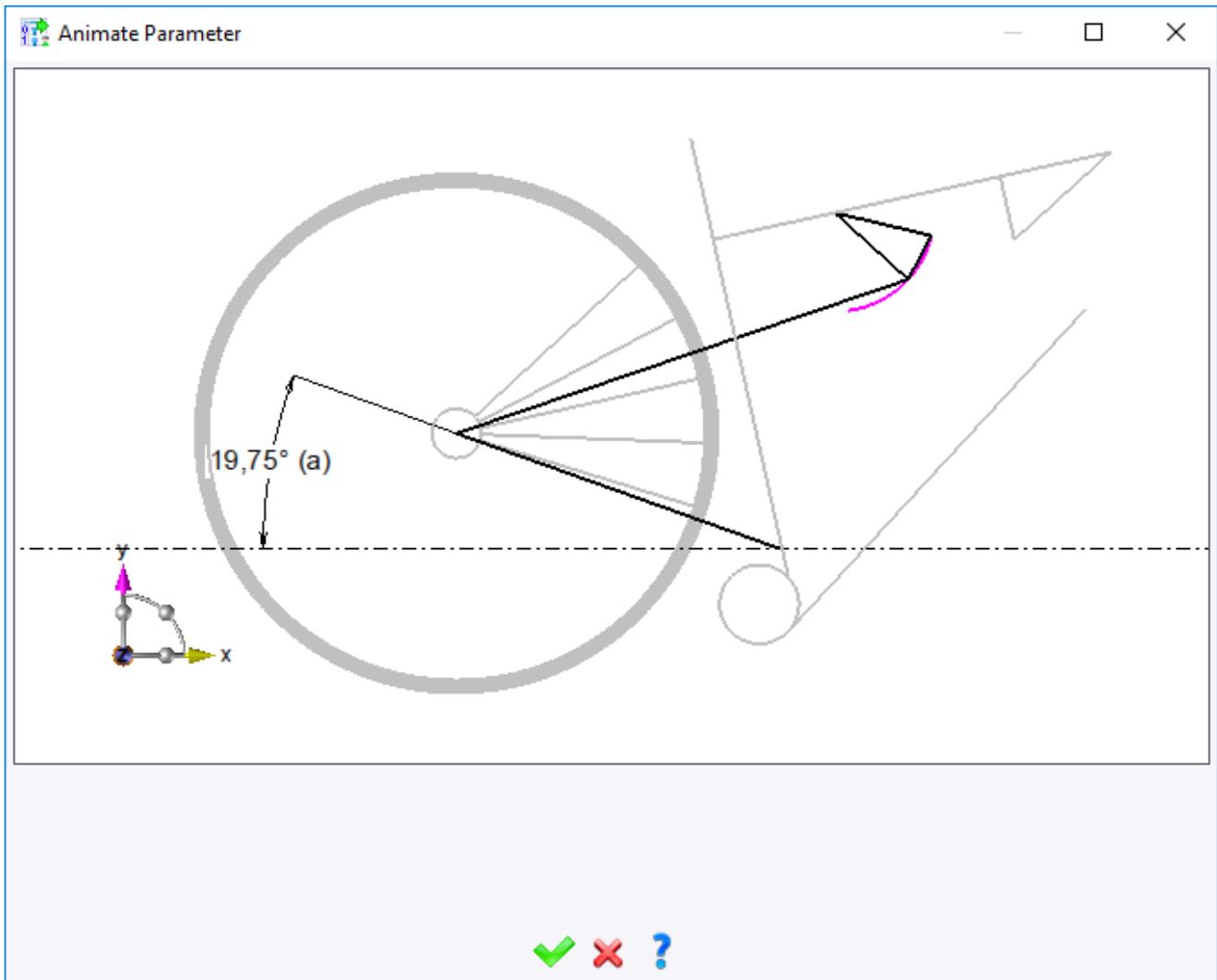
*Example of an image calculated with only an emissive environment.*

## Construction

### Animate parameter

The new **Construction > Parameters > Animate Parameter** command allows you to vary a parameter between two values and to view the animation in a separate window.

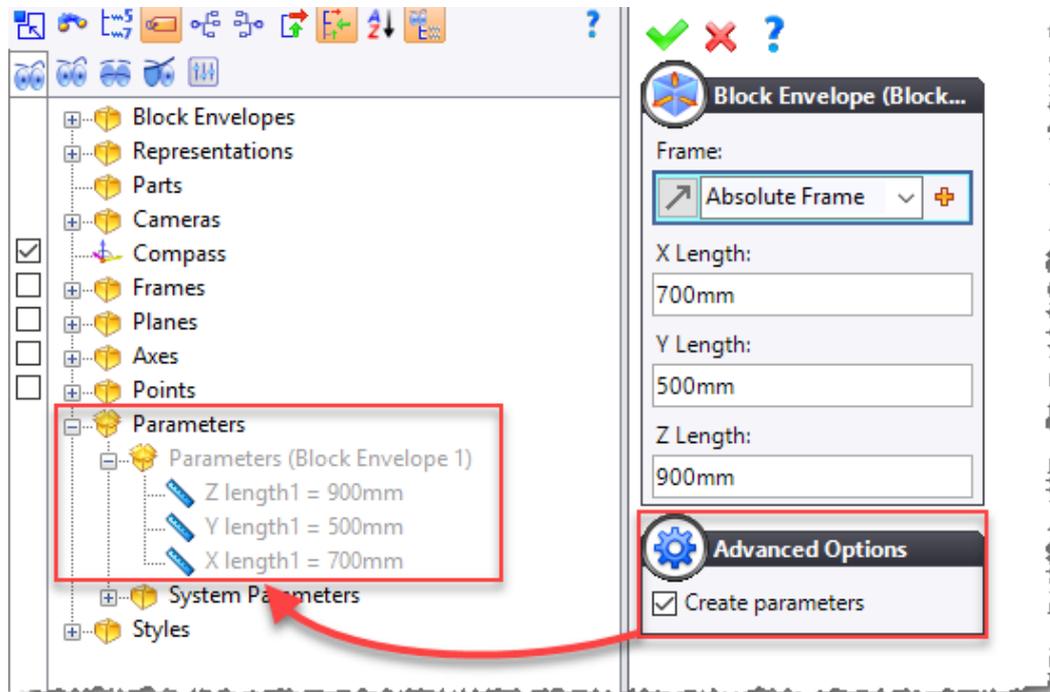
The **Point to follow** and **Create the curve in the document** boxes respectively allow you to view the trajectory of a particular point and to retrieve it as a basic sketch in the document.



Example of a parameter animation with curve creation (in magenta).

### Envelope block

The **Advanced Options** section containing the **Create parameters** box has been added to the **Construction > Block Envelopes > Block Envelope** command's dialog box. It allows you to create a length parameter for each dimension of the envelope block.



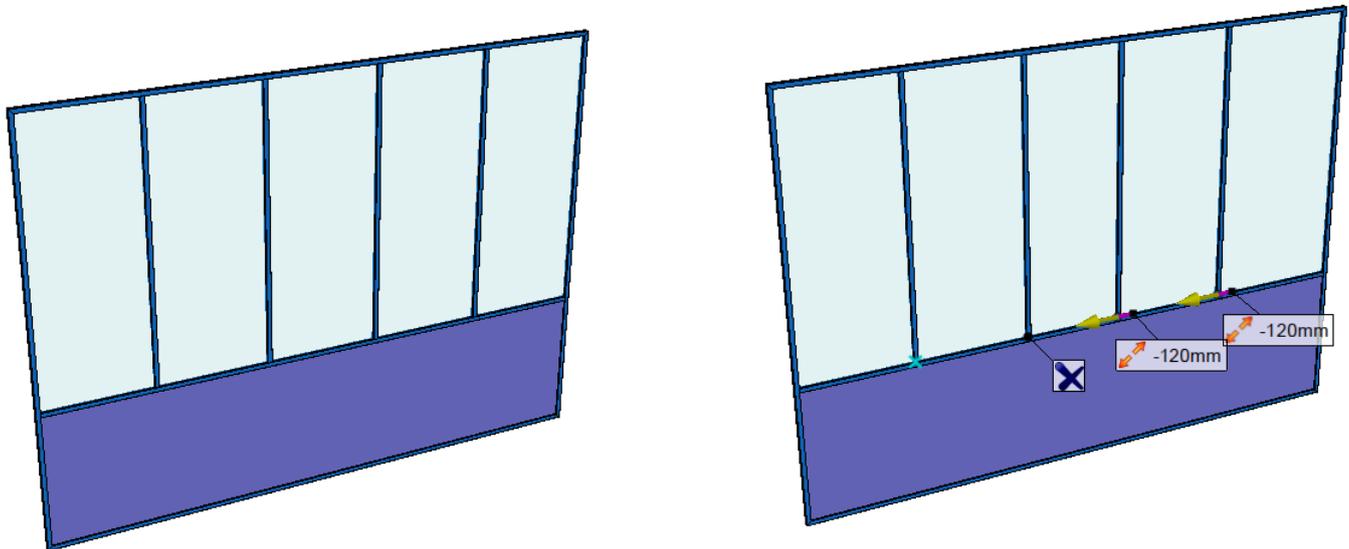
### Constrained linear pattern

The **Create parameters** box has been added to the **Advanced Options** section of the **Constrained Linear Pattern** command. When this box is checked, all calculated values are available as parameters in the Entities tree. Two new **Edge to edge** and **Thickness** boxes are available in the **Fixed count** and **Step** modes.

### Pattern modifier

The new **Construction > Patterns > Pattern Modifier** allows you to define a pattern as drivable so that you can delete or shift one of its instances in the assembly.

The drivable pattern must be defined in the generic document of the family and must be defined as an optional driver in the family document. Once the family has been included in an assembly document, the contextual menu called on the inclusion operation provides the **Edit Pattern Modifier** command which allows you to remove or shift an instance of the controllable pattern.

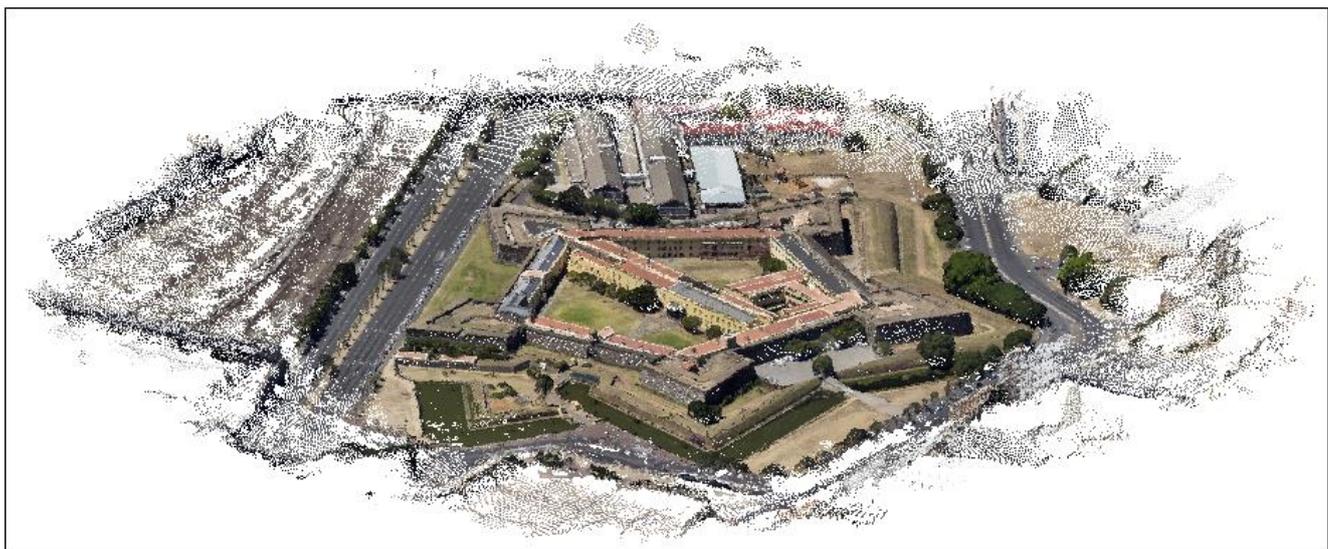


Example of a pattern modifier.

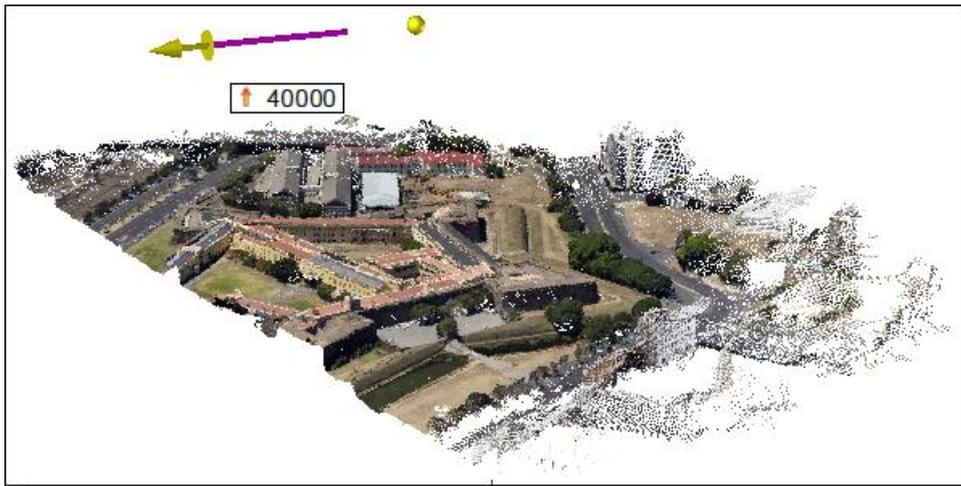
### Point cloud

The new **Construction > Point Cloud** command allows you to quickly import a point file (.pts or .xyz) into a **Part** or **Assembly** document. The created points are stored in the **Point Cloud** folder of the Entities tree. When the **Visualization > Faceted Shape Vertex Selection** mode is enabled, you can select points to take measurements.

In addition, the **point cloud** entities are supported by the **Visualization > Cut** command.



Example of recovering a point file as a point cloud.

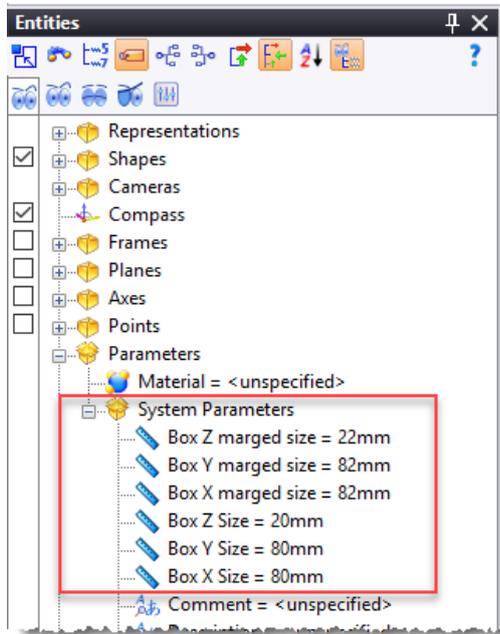


*Example of a cut on a point cloud.*

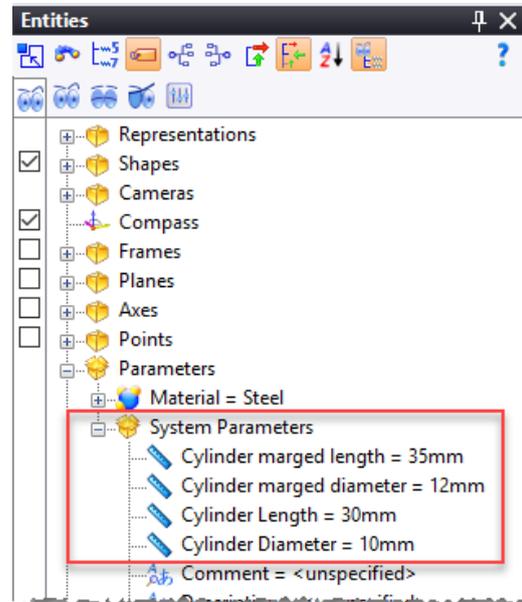
## Tools

### Enclosing block and cylinder

The **Margins** and **Round** sections have been added to the **Tools > Enclosing Box** and **Tools > Enclosing Cylinder** command's dialog boxes. For each operation, parameters are created in the Entities tree, which allows them to be retrieved in a bill of materials using the **Tools > Stock** command (see *Stock* section).



Parameters of an enclosing box.



Parameters of an enclosing cylinder.

In addition, these margins can also be defined according to a material using the new **Box Margin Rules** and **Cylinder Margin Rules** document types available in the **Special** tab of the document creation dialog box. These rules are then used by the machining modules.

### Stock

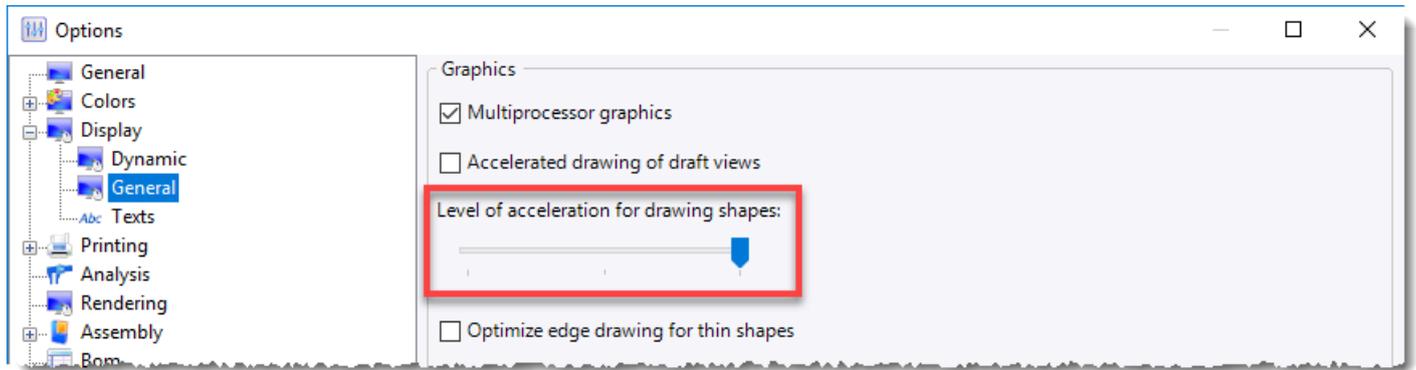
In **Manual** mode, the **Stock** command's dialog box now allows you to associate the parameters of the enclosing box with the general **Length**, **Width** and **Thickness** parameters.

### Spare video memory

The **Spare video memory** option avoids you to allocate too large video memory blocks in order to better support the graphics cards with less than 2 GB of memory.

## Graphic optimization

The **Level of acceleration for drawing shapes** option uses an acceleration feature available in the recent graphics cards. This feature can be disabled if the graphics card does not support it.



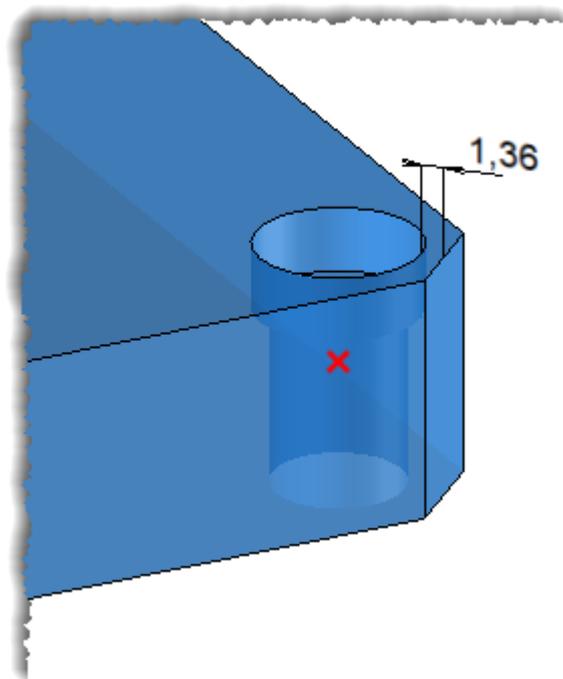
	TopSolid 7.14	TopSolid 7.15
Shading + Edges	16 img/s	27 img/s
Shading only	19 img/s	31 img/s
GPU occupancy (according to the Task Manager)	25%	70%

## Drilling collisions

The **Tools > Drillings Collisions** command has been improved and now detects the collisions between the holes and the edges of the part.

The dialog box has been simplified and the **Drillings** and **Threadings** modes have been removed.

For the threading operations, the analysis is now systematically performed on the face corresponding to the thread root.



*Example of a drilling collision analysis with a 2mm safety margin.  
A collision is detected between the spot faced hole and the edge of the part.*

## Nesting (Optional Module)

### ***Automatic nesting: maximum filling***

The new **Maximum** filling calculation mode is available.

For each part and in each available support, **TopSolid** calculates the nesting giving the lowest loss rate, regardless of the number of nested parts. Then it continues with the other parts by testing on the remaining supports, taking into account the priority of the parts and the priority of the supports.

Only the best nesting is displayed with the resulting part quantity.

### ***Manual nesting***

#### **Deleting parts and deleting support**

Two new commands are available:

- **Delete**: Allows you to remove one or more parts from the nesting.
- **Delete support**: Allows you to delete the selected support(s), as well as the parts nested on the support(s).

#### **Disable security area of parts already nested**

Thanks to the new **Disable security area of parts already nested** advanced option, only the security gap of the moved or copied part is taken into account. The gaps of the already nested parts and the support are no longer displayed.

#### **Disable collisions**

The new **Disable collisions** advanced option disables the collisions with the other parts and the security gaps of the support. A part can be moved or copied anywhere on the support. If the box is checked, the security gaps are no longer displayed.

#### **Precise positioning**

You can now choose a precise positioning to move or copy one or more parts.

The positioning is defined by an **origin point** and a **destination point** with an X offset and a Y offset.

#### **Linear pattern**

The **Part rotation** box in the **Linear Pattern** section now allows you to rotate the selected part before applying the pattern.

#### **Rotations**

You can now apply a rotation to a set of parts. In the previous version, rotation could be applied to only one part.

In addition, when the **Linear Pattern** box is checked, you are able, from the **Rotations** section, to rotate all parts of the pattern with the center of the enclosing box of the reference parts as the center.

#### **Modification**

The **Manual Nesting** contextual command is now available in the documents generated by the nesting document (assembly, machined part setup and machining documents). It is also available in the **Tools** menu.

You can therefore modify the nesting from these documents. You must save the nesting document once before you can modify the nesting.

## TopSolid'Fea

### ***Modifying the shells***

The new **Analysis > Modify Shells** command allows you to quickly modify the material and thickness of one or more shells. The shells can be selected manually or by criteria (material and/or thickness).

### ***Modifying the beams***

The new **Analysis > Modify Beams** command allows you to quickly modify the material, the profile type and its orientation for one or more beams. The beams can be selected manually or by criteria (material and/or family, code).

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## Drafting

### View alignment

The alignment of the views has been improved:

- The dynamic movement of an aligned view respects the alignment constraints.
- When moving a view, all dependent views move as well.
- The alignment constraints are displayed when hovering over a view.

### Interrupted view

The dialog box of the **View > Interrupted View** command has been modified. The **Standard** mode has been broken down into two modes:

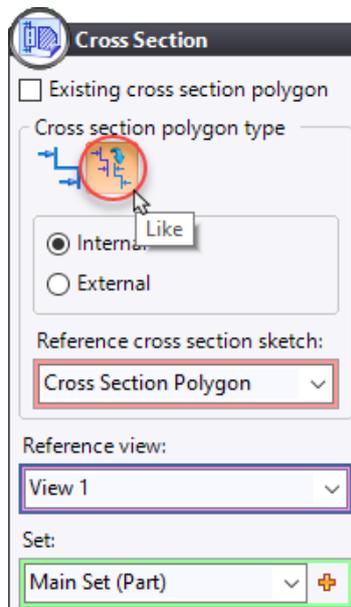
- **Manual:** The interruption profiles are drawn manually.
- **Manual Standard:** The interruption profiles are selected from a list of predefined types (line, wave, etc.).

In addition, the new **Automatic Standard** mode allows you to automatically interrupt a view, taking into account the topology of the part.

### Cross section polygon

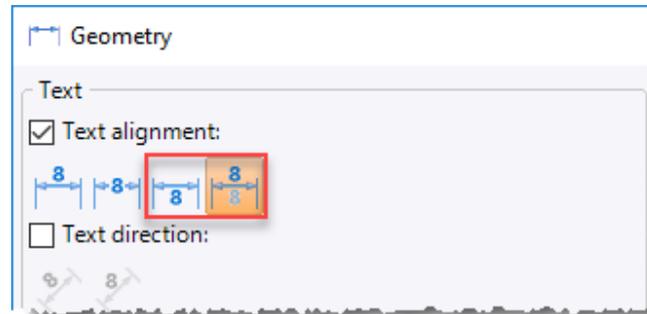
When creating a cross section polygon, the new **Like** option allows you to create the same cross section polygon on another view as long as the directions of the two views are parallel.

Once the polygon has been created, the **Make Manual** contextual command allows you to make it independent.



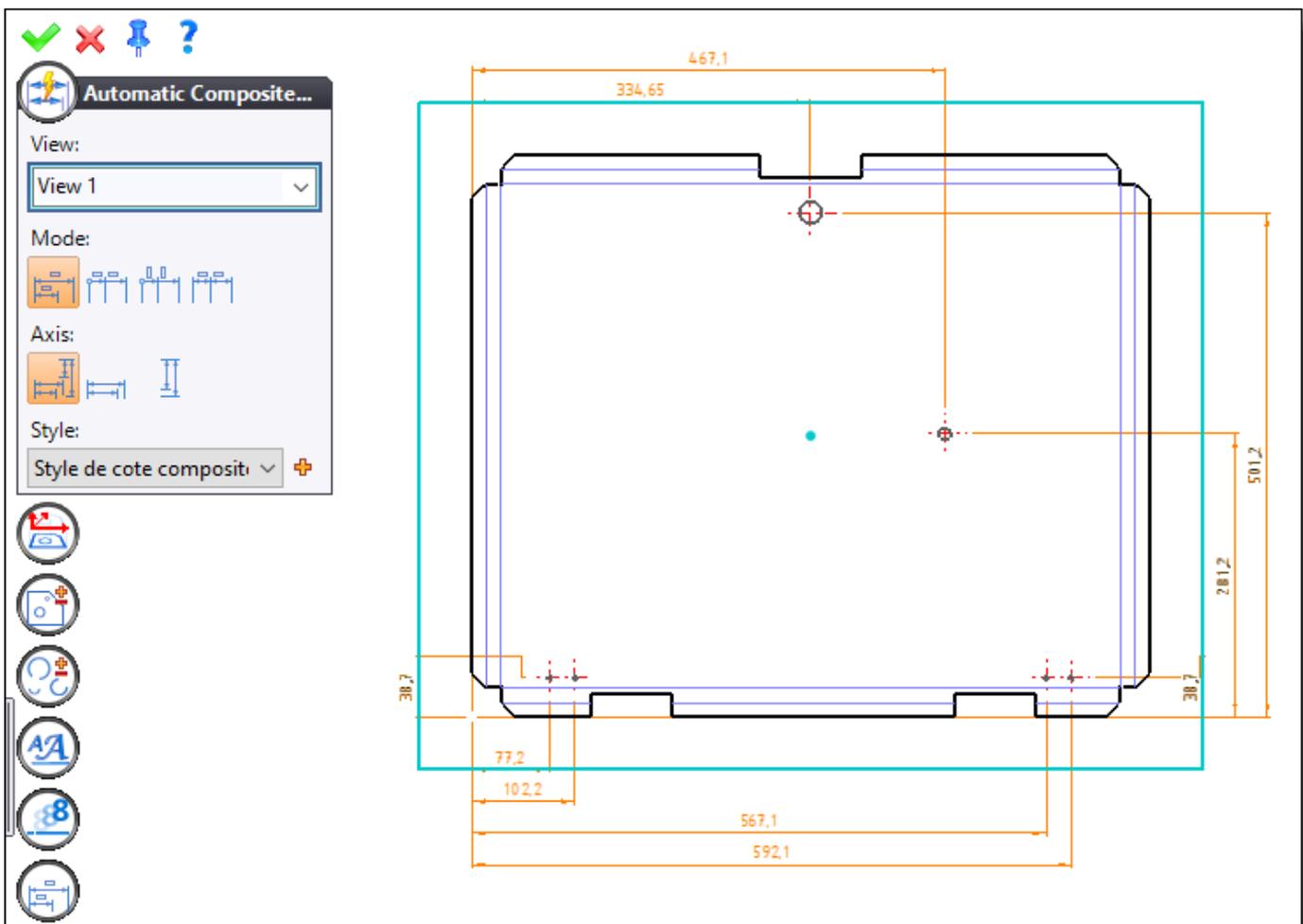
### Dimensions

The **Below** and **Opposite to geometry** text alignment modes have been added and the offset value for the vertical dimensions can now be adjusted.



### Automatic composite dimensions

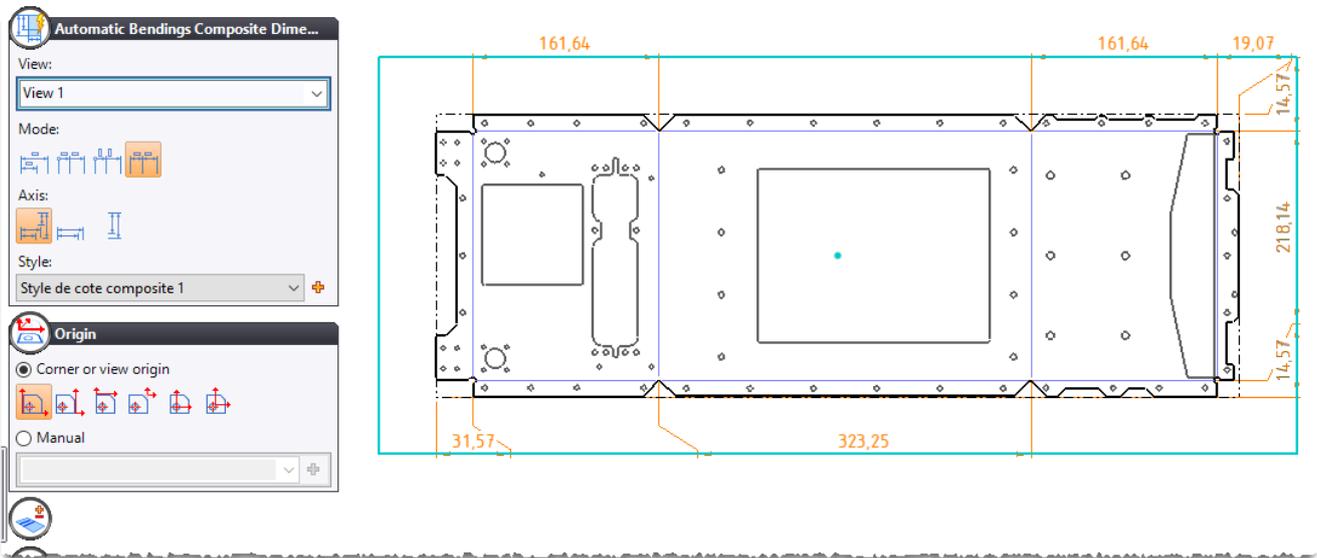
The **Detailing > Automatic Composite Dimensions** command now supports the views containing unfoldings.



Example of automatic composite dimensions on a view containing an unfolding.

### Automatic bending composite dimensions

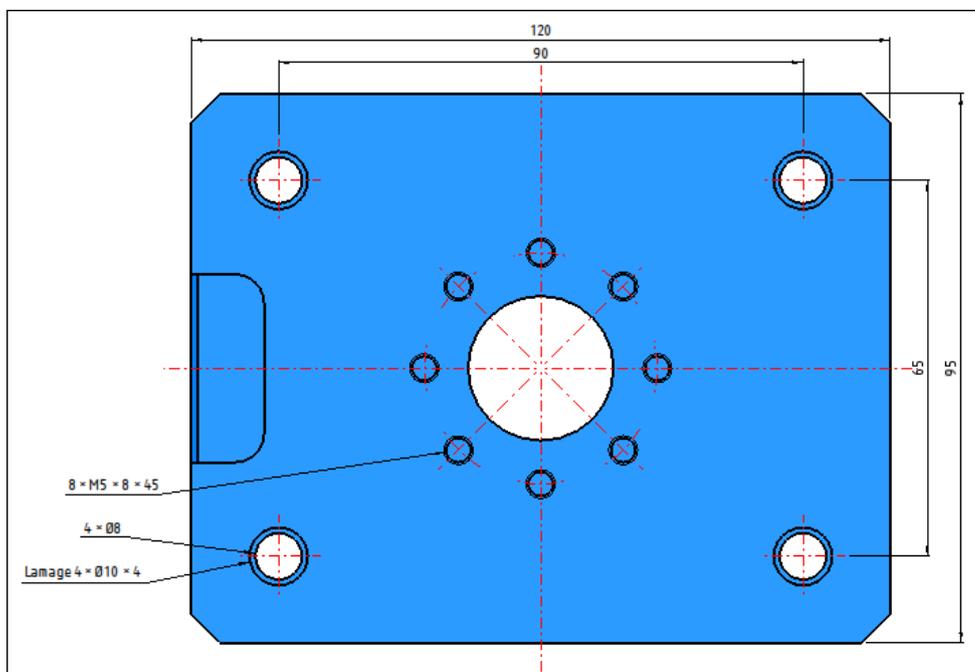
The new **Detailing > Automatic Bendings Composite Dimensions** command allows you to automatically dimension all bends of a view containing an unfolding. As with the automatic composite dimensions, the **Base line**, **Cumulated**, **Ordinate** and **Continue** modes are available and you can also exclude certain bends and add certain geometries.



Example of automatic bending composite dimensions.

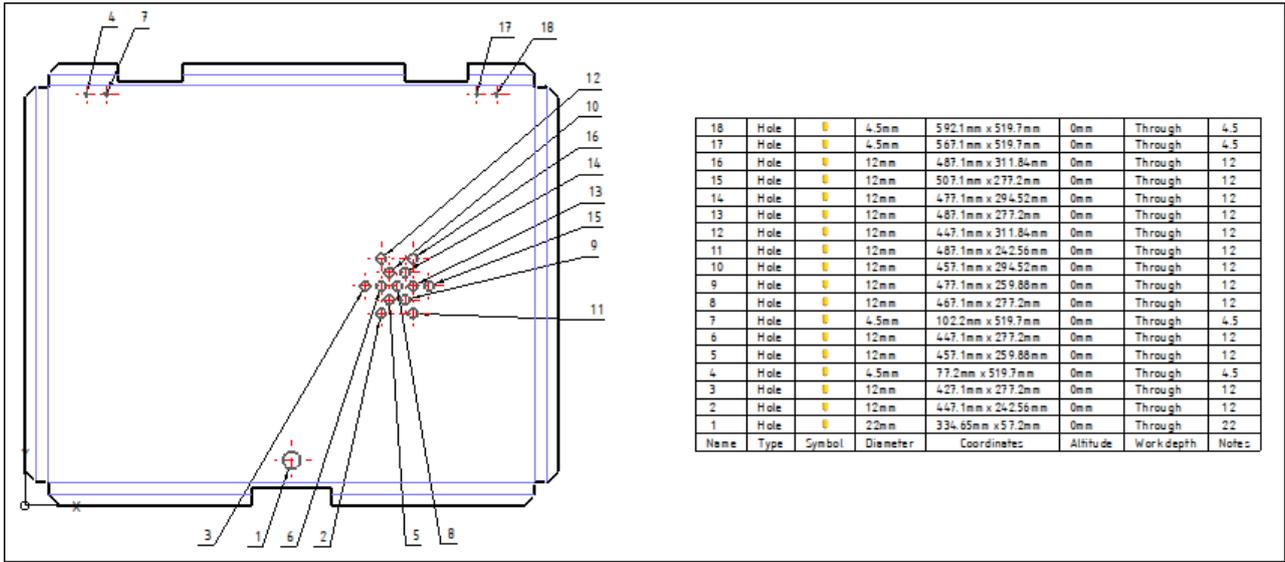
### Drilling table

When creating a drilling note using the **Detailing > Drillings Table** command and the **As table** box is unchecked, the new **Index all the occurrences** box allows you to create a dimension on each drilling (checked box) or to create only one dimension by indicating the number of identical drillings (unchecked box).



Example of drilling dimensions created using the **Drillings Table** command. The **As table** box is checked and the **Index all the occurrences** box is unchecked.

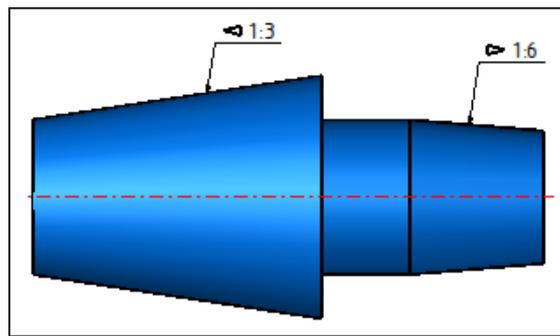
In addition, the command now supports the views containing unfoldings.



Example of a drilling table on a view containing an unfolding.

### Cone note

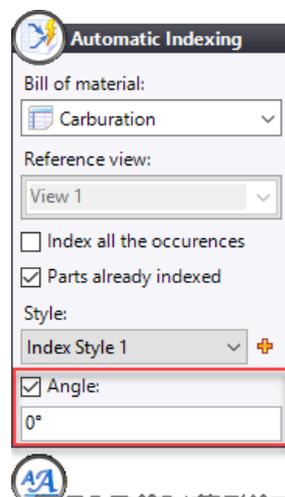
The new **Detailing > Cone Note** command allows you to create standard cone dimensions.



Example of a cone note.

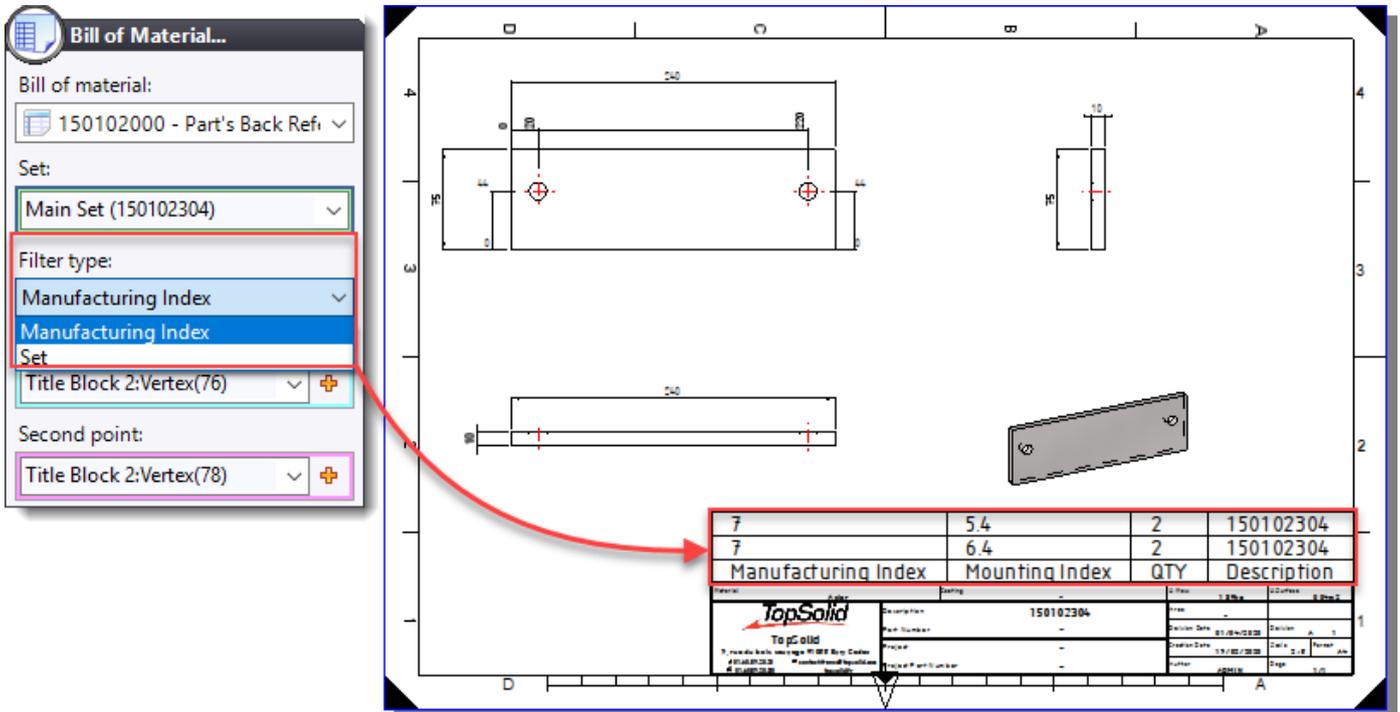
### BOM index

When creating a manual or automatic BOM index, you can adjust its inclination by checking the new **Angle** box.



### BOM table

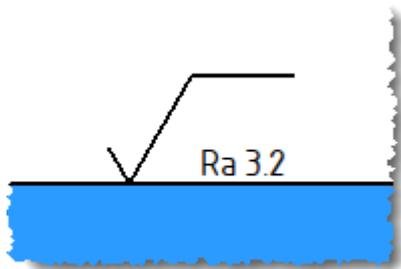
The **Filter type** option has been added to the **Detailing > Bill of Material Table** command. It allows you to display either the set (value with the same name) or the back references of the part (**Manufacturing Index** value).



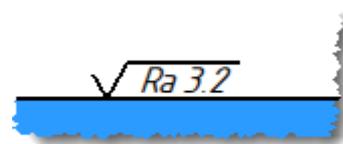
### Surface finish symbol

You can now place a surface finish symbol anywhere on the drawing, as long as there is a view. You can also hang the symbol on the arrows or the frame of an annotation.

When the document is the GOST standard (Russian standard), if the symbol contains only the bottom text, it automatically adapts to the height of the text.



ISO standard



GOST standard

### Drilling notes and indexes

The position of the drilling notes and indexes has been improved. The notes and indexes are now automatically positioned on the red frame of the view when optimizing the scale factor.

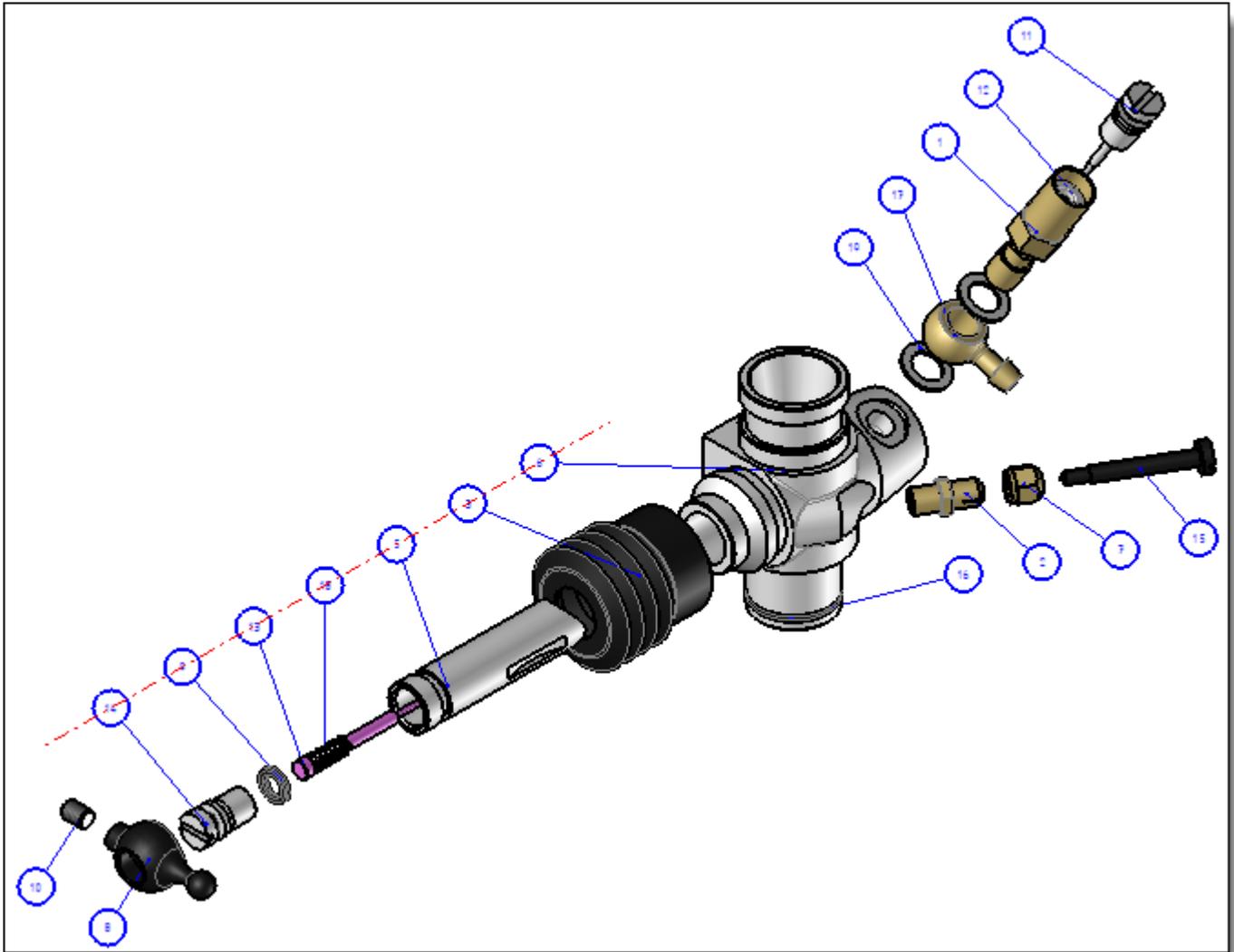
### BOM index

The new **Detailing > Magnetic Line** command allows you to create a sketch on which the BOM indexes can be positioned by magnetization. This command is also available in contextual mode when selecting multiple BOM indexes.

When moving a frame, these magnetic lines appear in yellow. You can magnetize the indexes by hovering over them.

When hovering over a magnetic line, it is shown as a red axis line. It can be moved or enlarged by selecting one of its ends.

A magnetic line can also be edited, like any sketch, so that it can be moved or transformed. The magnetic indexes will then follow the changes.

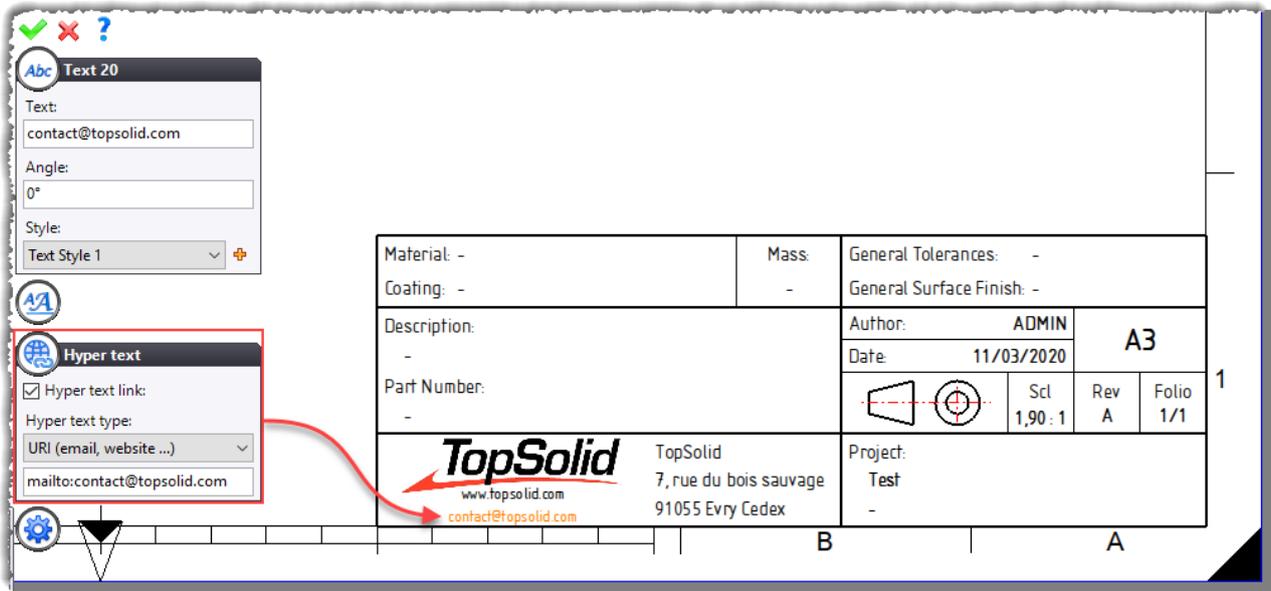


*Example of magnetizing indexes on magnetic lines.*

## Hypertext link

The **Hypertext** section has been added in the text, note and rich text creation commands, as well as in the different text styles. It creates a link between a text of the document and either a PDM document, an e-mail address, a website, a file or a folder on the computer or network.

You can then follow the link using the **Ctrl** key + **double-click** with the mouse (the **Ctrl** + **click** combination used in other applications is already used to duplicate an element).



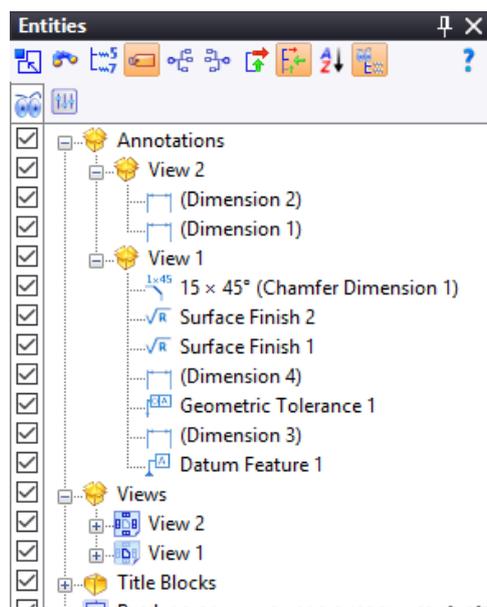
Example of a hyperlink with an e-mail address.

## Symbols

When creating a symbol in a sketch of the 3D document using the **Sketch > Other Curves > Symbol** command, you no longer have to check the **Profiles** box to retrieve the symbol in the drafting document.

## Storage

The dimensions and annotations are now arranged view by view in the **Annotations** folder of the Entities tree.



### Cutting the links

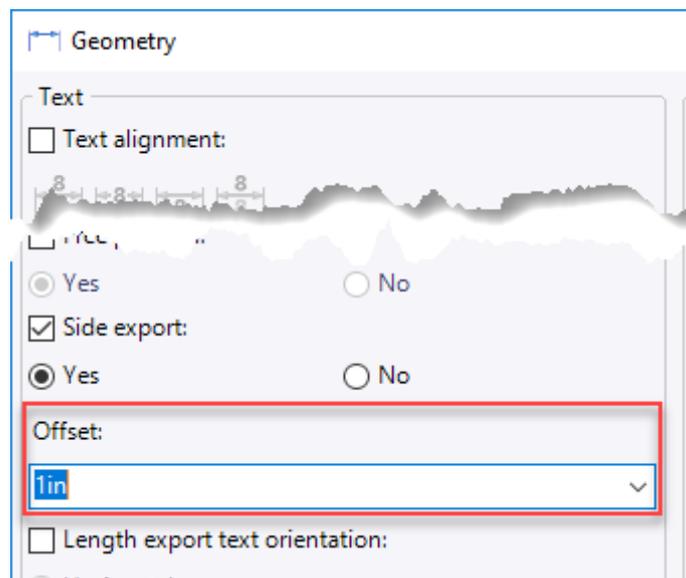
The new **Edit > Isolate Drafting** command allows you to cut the links between the drafting document and its referenced documents.

### Format

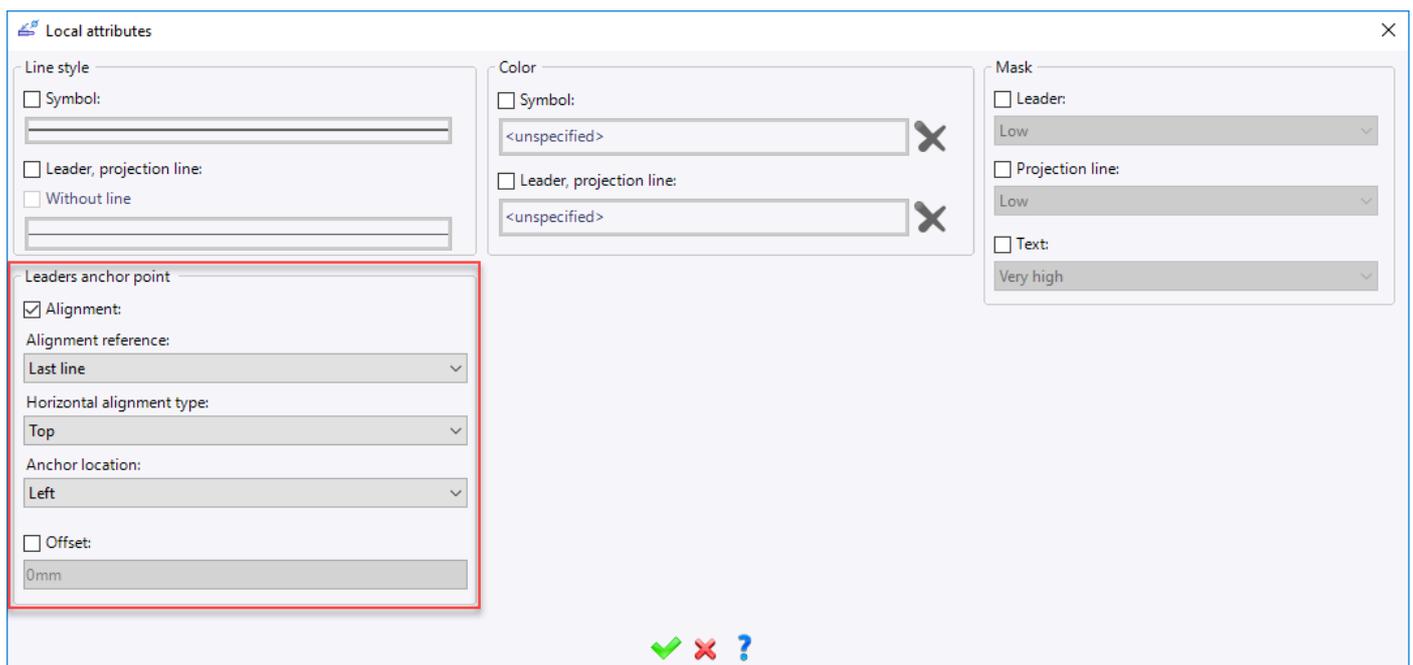
In the GOST standard, the multiple formats (A4x3, A4x4, A3x3, etc.) have been added to the list of predefined formats.

### ANSI standard

The dimensions have been modified to comply with the ANSI standard. You can now adjust the offset value for the vertical dimensions.



For the drilling notes or any command using text (simple or rich text), you can now adjust the leader anchor point and the offset value from the **Local attributes** button.



## Unfolding

### Parameter from a bill of materials

In an unfolding document, the new **Construction > Parameters > Parameter from Bill of Material** command allows you to retrieve information from a bill of materials such as the quantity of parts or the manufacturing index.

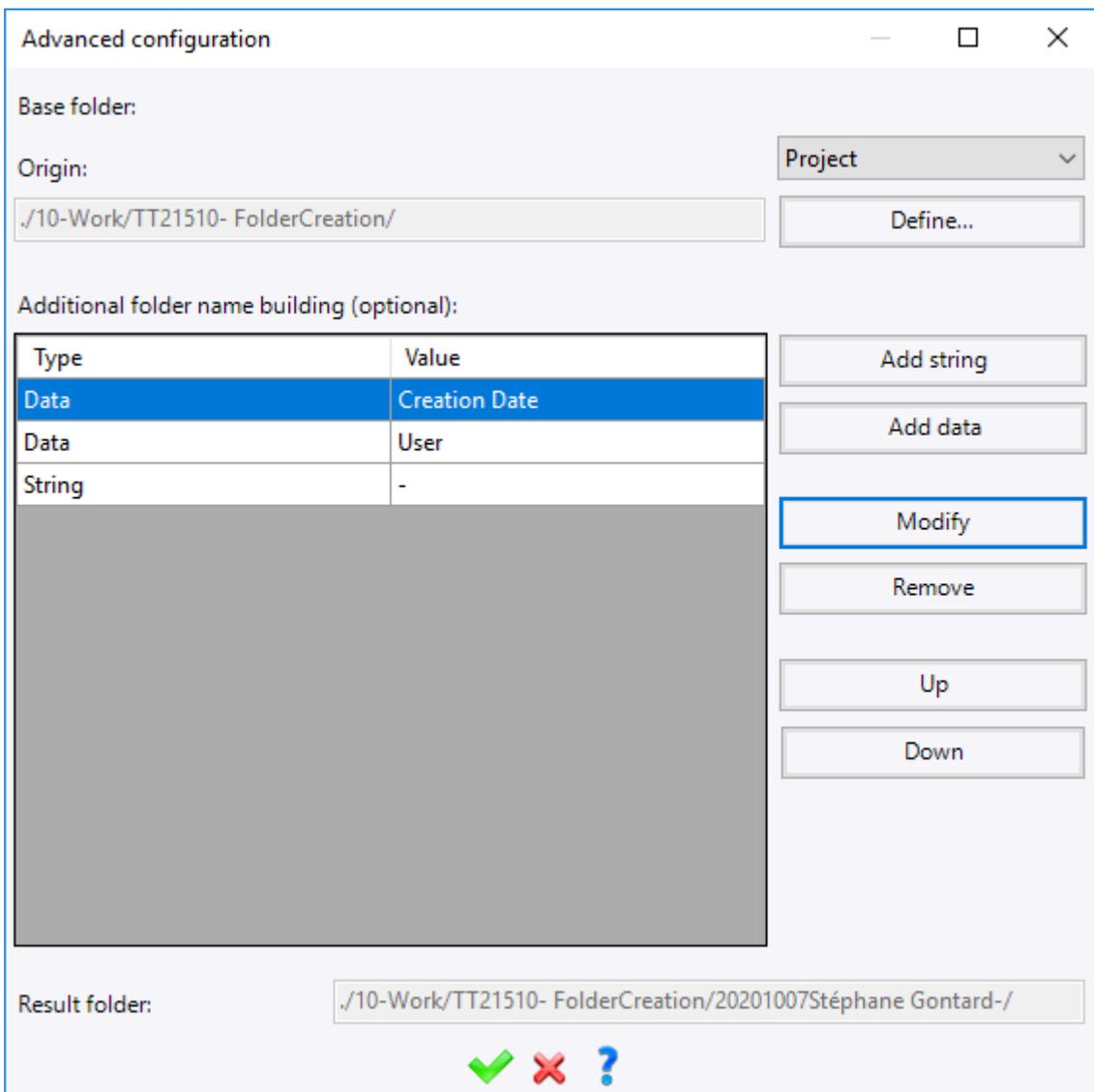
## Work

### Document status

In a **Work** document, the used documents no longer need to be checked into the vault. It is enough that they have been saved at least once.

### Folder creation process

A folder creation process with a customizable name has been added. It allows you to define the name of the folder with the user's name and/or the creation date, as well as characters.



## Contextual menu on assembly

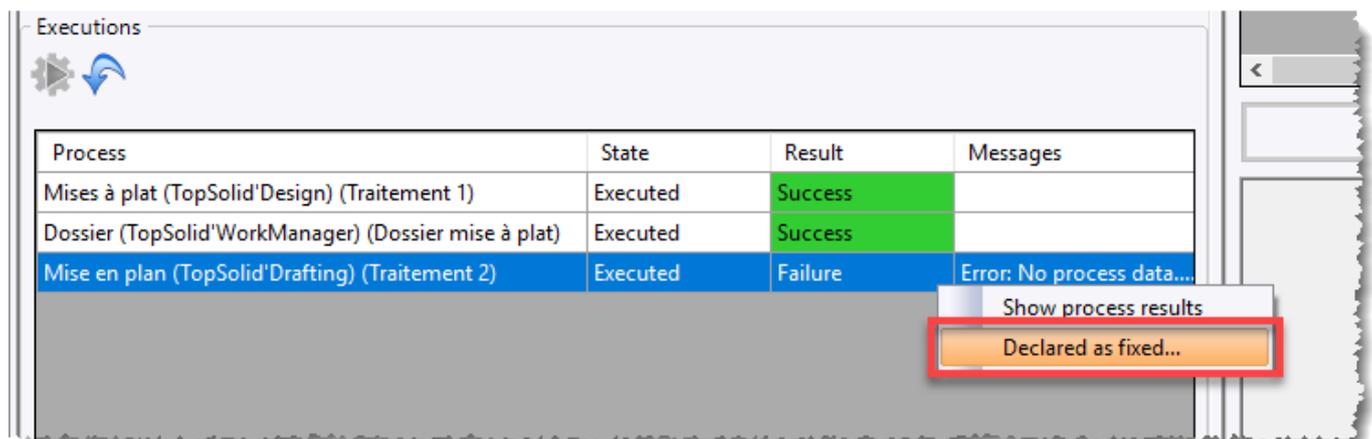
You can now create a **Work** document by selecting one or more documents in the Project tree or via the tab of an open document. For example, this allows you to select a **Work** document template containing processes and to execute them automatically if the **Tools > Automatic execution** box is checked in this template.

## Adding tasks by bill of materials

In the **Tasks** contextual command available from a BOM document, you can now create a new **Work** document on the fly. For example, this allows you to select a **Work** document template containing processes and to execute them automatically if the **Tools > Automatic execution** box is checked in this template.

## Fixing a process

When a process has produced invalid documents and failed, you can now declare it as fixed by the user thanks to the **Declared as fixed** contextual command available on the failed process.



## New processes

The following processes have been added to the list of available processes:

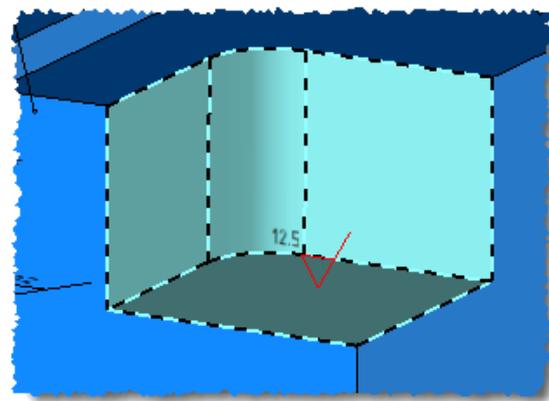
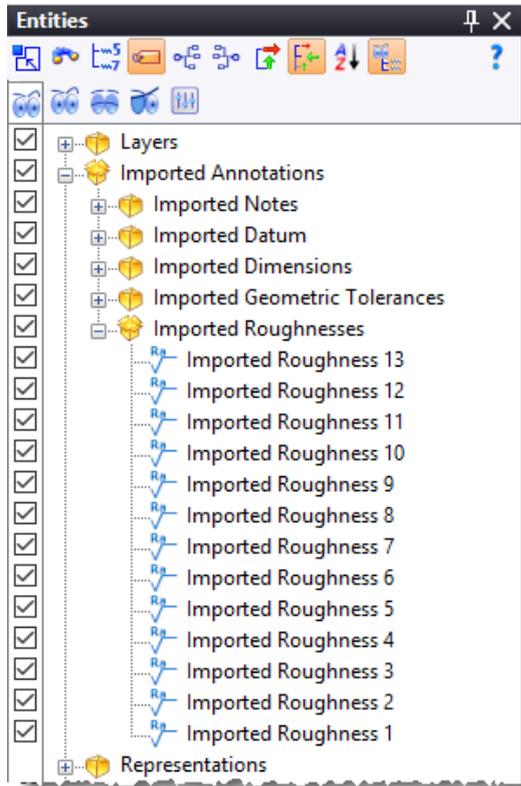
- Conversion into unfoldings
- Creation of folders
- Exports (AutoCAD, PDF, Step, Parasolid, etc.)
- Creation of drafting bundles

# Translators

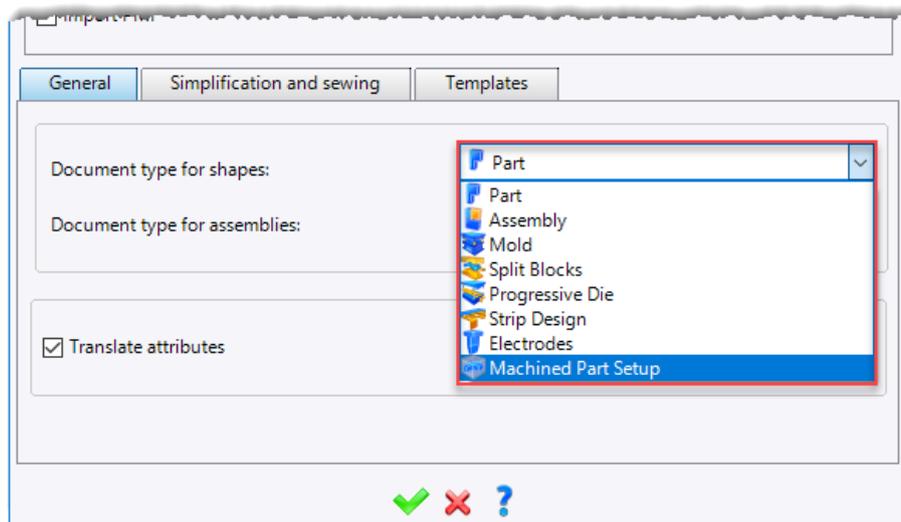
## Spatial import

Several enhancements have been made to the import of PMIs:

- The **Hole Feature** PMIs are now converted into inoperative holes so that **MFs (Manufacturing Features)** can be produced for the machining. In addition, the drilling tolerances are automatically applied to the drilling operations.
- The imported annotations are stored in sub-folders of the Entities tree. In the graphics area, hovering over an imported annotation highlights the referenced faces.



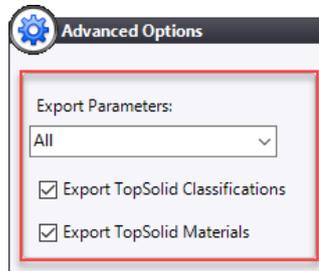
When importing a document via a Spatial translator (Step, Acis, etc.), you can now choose to create a **machined part setup** document.



## IFC translator

### Export

The **Advanced Options** section has been added to the IFC export command. It allows you to choose the parameters to be exported (**None**, **System Only** or **All**) and to export the different classifications defined with the **Tools > Classifying** command.



The single and multi-layer materials are now exported as **Material Layer Set** entities.

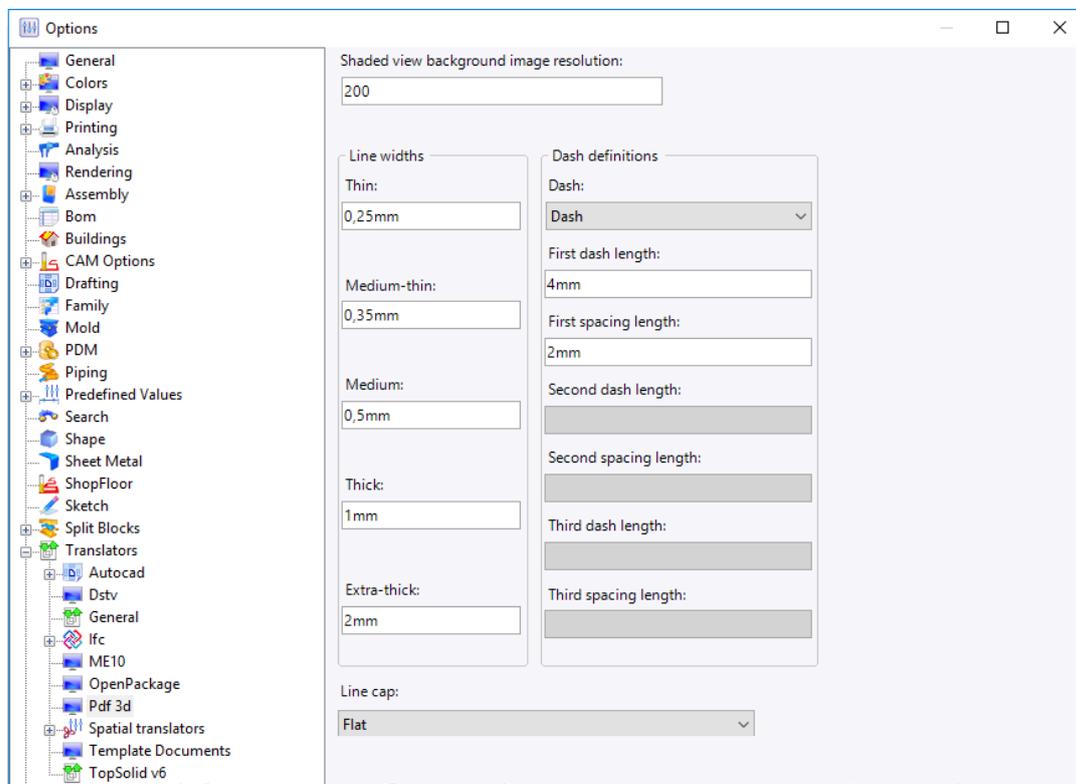
### XYZ import

A color attribute defined with three values (RGB) is now supported for each point.

In addition, the new **Point cloud** type allows you to import a point file very quickly (see *Point cloud* section in the *Construction* chapter).

### PDF3D export

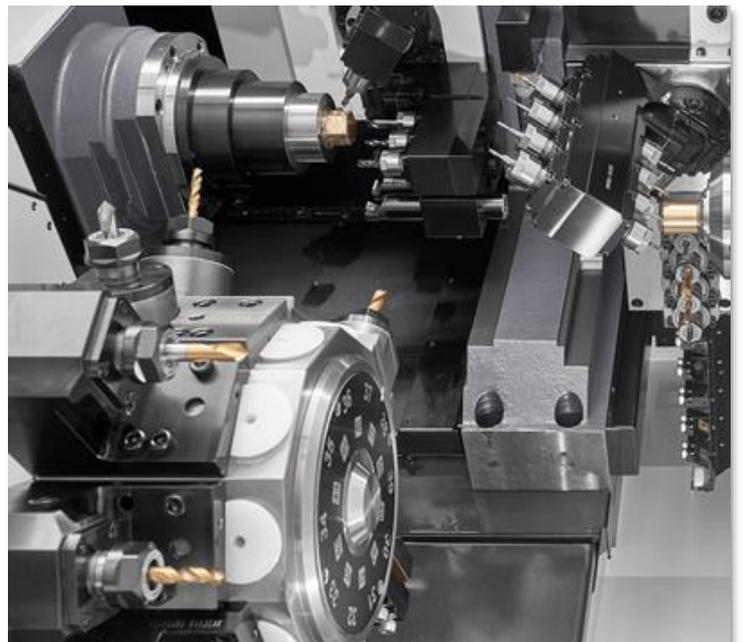
Line thickness management options have been added to the **Tools > Options > Translators > Pdf 3d** command. They allow you to generate a PDF document with different thicknesses for each type of line without having to perform a print operation (via PDFCreator for example).



*Adjustment of the line thicknesses during a PDF3D export.*



## What's New in TopSolid'Cam 7.15

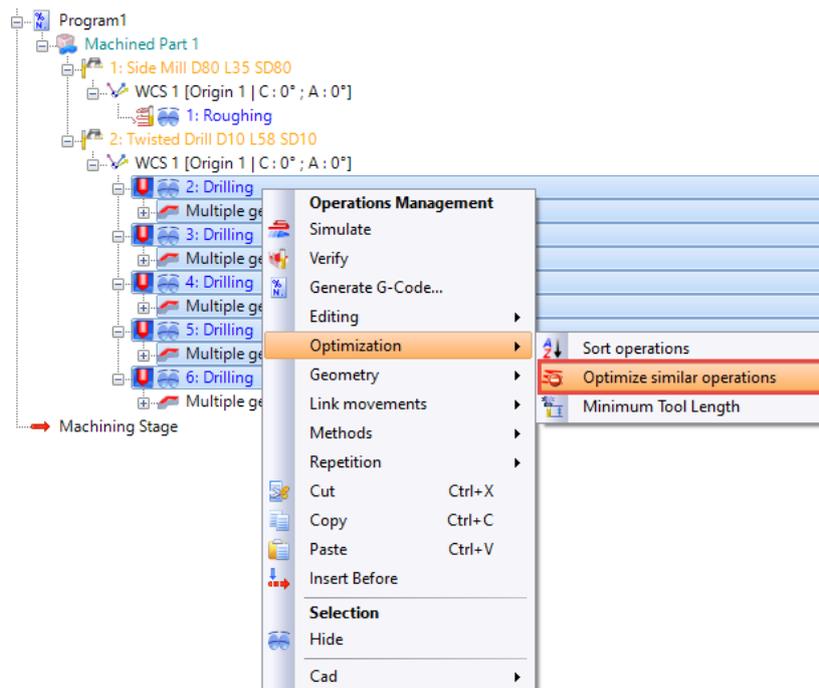


This section describes the enhancements made to the **Machining** application in version **7.15** of **TopSolid 7**.

## Drilling

### Tool path optimization

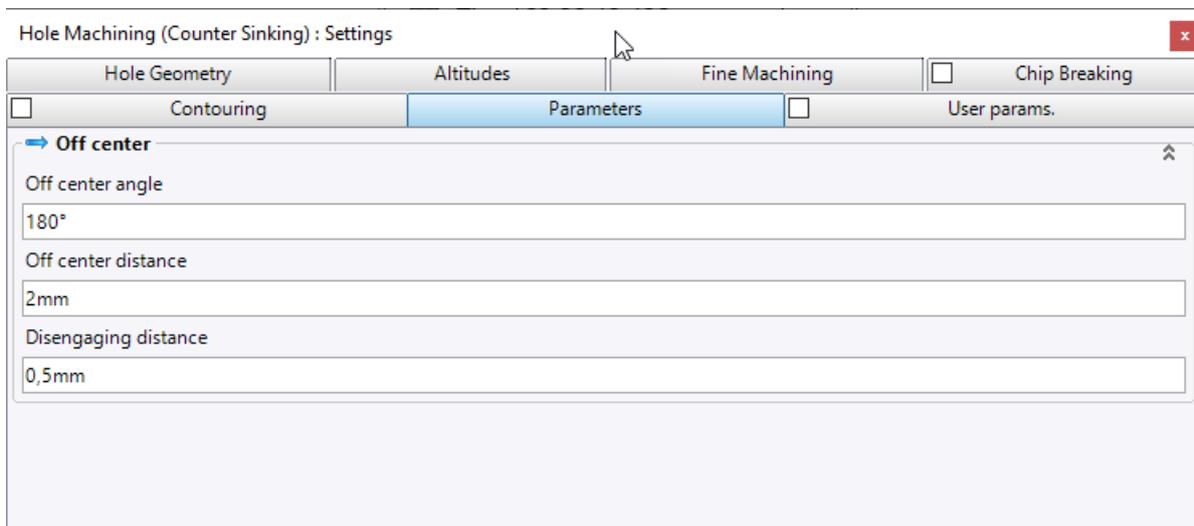
The new **Optimize similar operations** command allows you to optimize the tool paths of several identical (same primitive, same tool, same work coordinate system and same part) and consecutive hole machining operations by improving the paths between holes. A new master operation of the same type grouping the identical operations together is created under a new name.



**Restriction:** Only the hole machinings are processed.

### Back countersinking using a reverse chamfer mill with a rigid insert

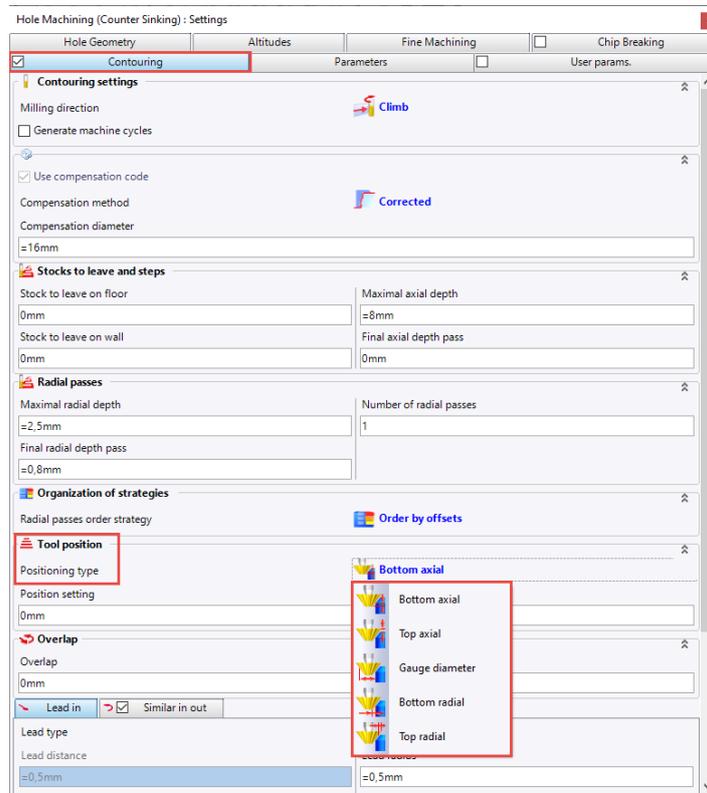
You can now perform a back countersinking operation using a reverse chamfer mill with a rigid insert. The positioning parameters of the tool can be adjusted before plunging into the hole and retracting.



**Restriction:** The double chamfer mill is not managed.

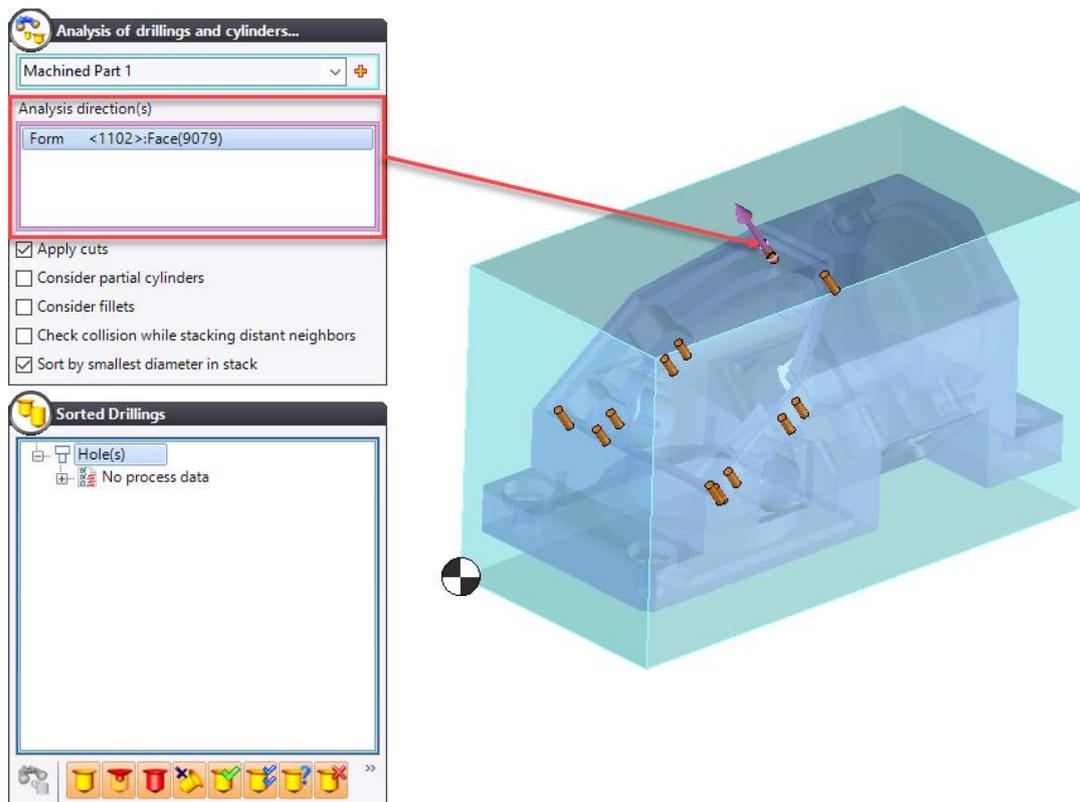
### Counter sinking: Tool position management

When performing a countersinking by contouring operation, you can manage the position of the tool, as is the case with the chamfer milling and edge breaking milling operations.



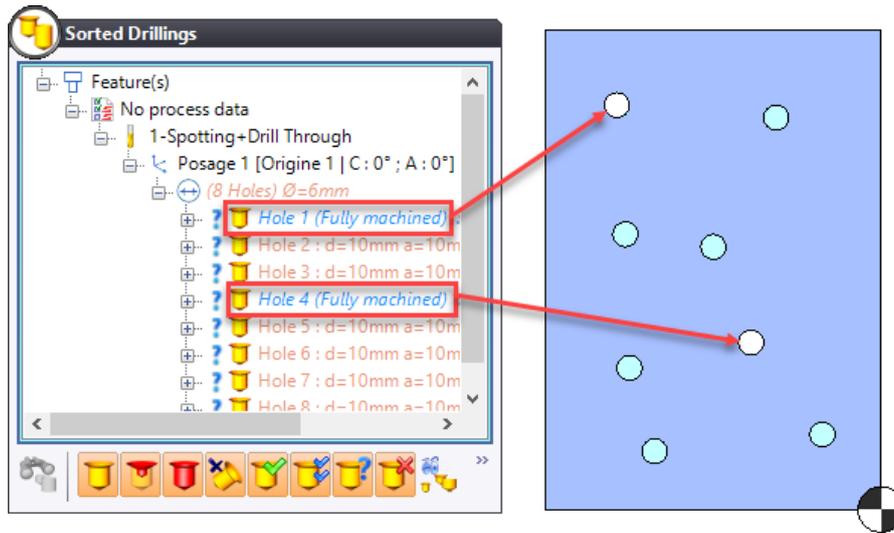
### Analysis of drillings and cylinders according to directions

The analysis of drillings and cylinders now allows you to add a list of directions to restrict the analysis to certain faces in order to save analysis time and to filter the list of results.



### Analysis of drillings and cylinders: Repositioning

You can now store the already machined holes after repositioning the part.

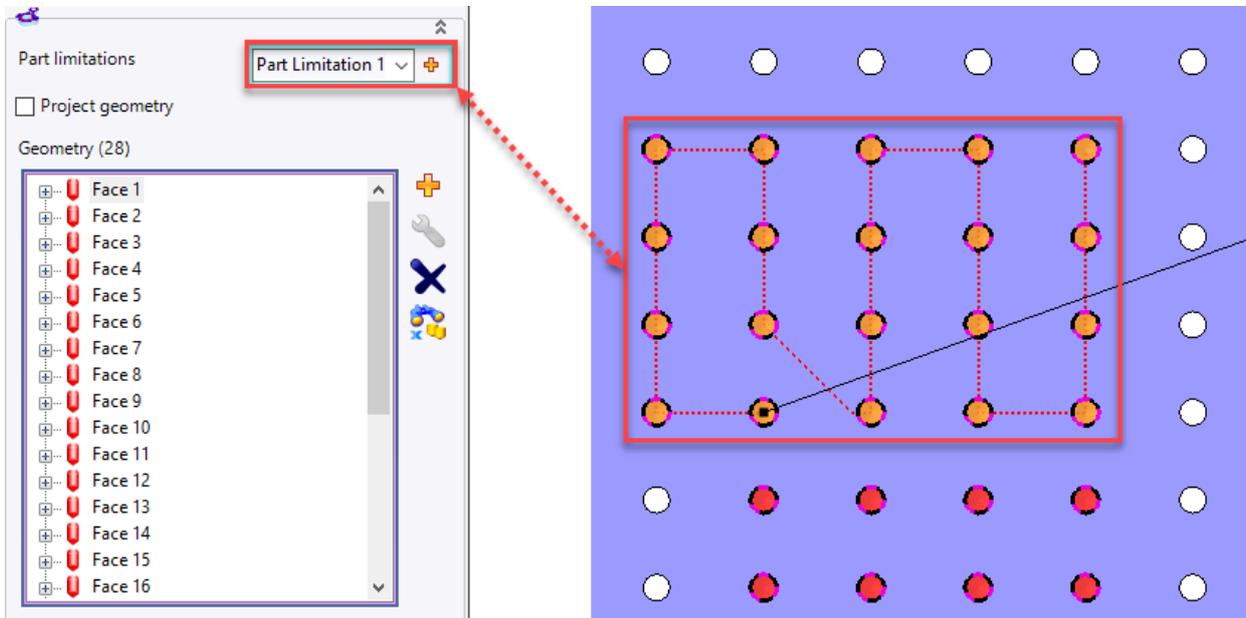


### Limiting the analysis area: Associativity

When modifying an analysis limitation area, the already created relevant drilling operations are replayed according to the new limitation area.

The effect of this change only applies to the geometries available in the operation. Under no circumstances will new geometries be automatically added or deleted.

Thus, these hole machining operations are able to exclude selected geometries (holes) that were previously included in the analysis limitation area, but which are no longer reachable after modification.



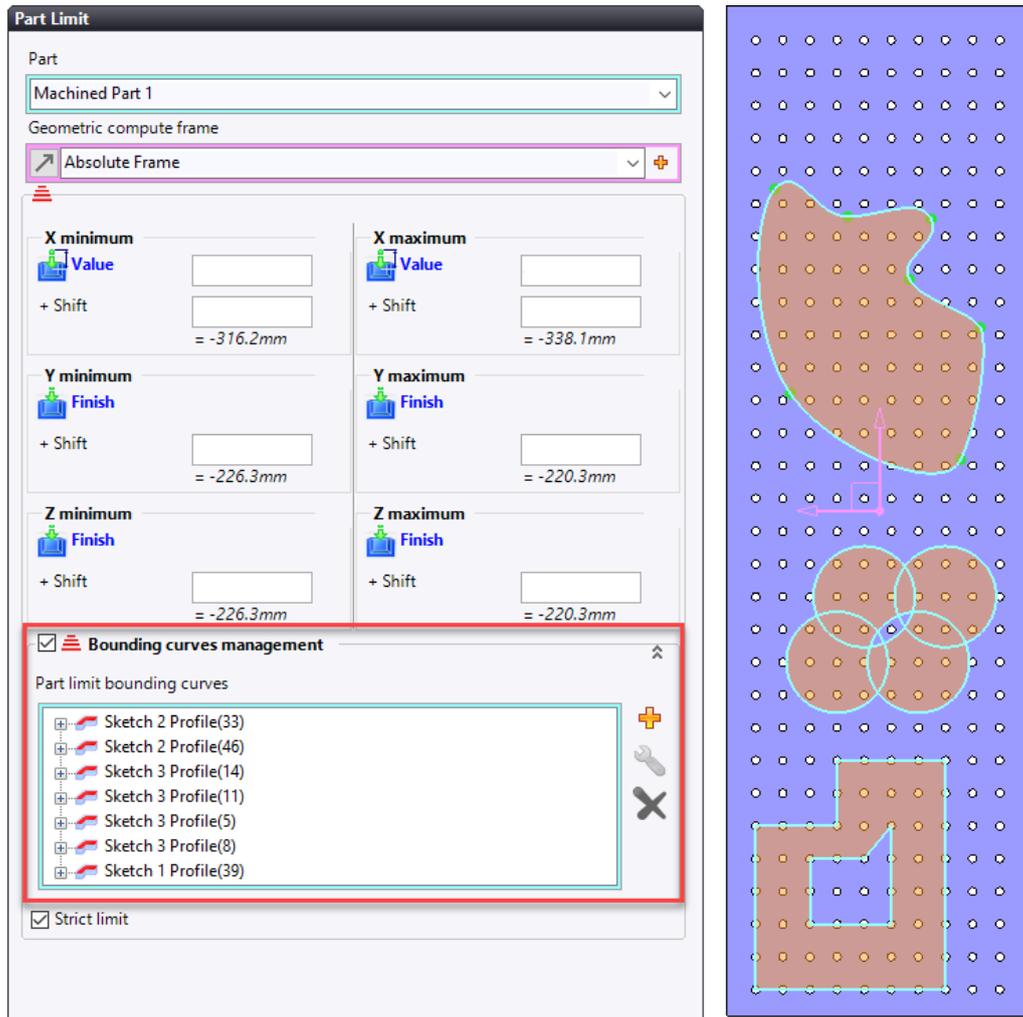
Conversely, the selected geometries that were previously outside of the limitation area and that re-enter the area after modification are therefore included and machined.

**Restrictions:** The changes in the limitations of the analysis areas do not automatically restart the analysis in the operations. For example, if the limitation area is enlarged to add additional holes, you will have to edit the operation and restart the analysis from the **Geometry** section.

For holes that are outside of the area (shown in red because they are unreachable) and that you wish to remove from the list, you have to delete them manually.

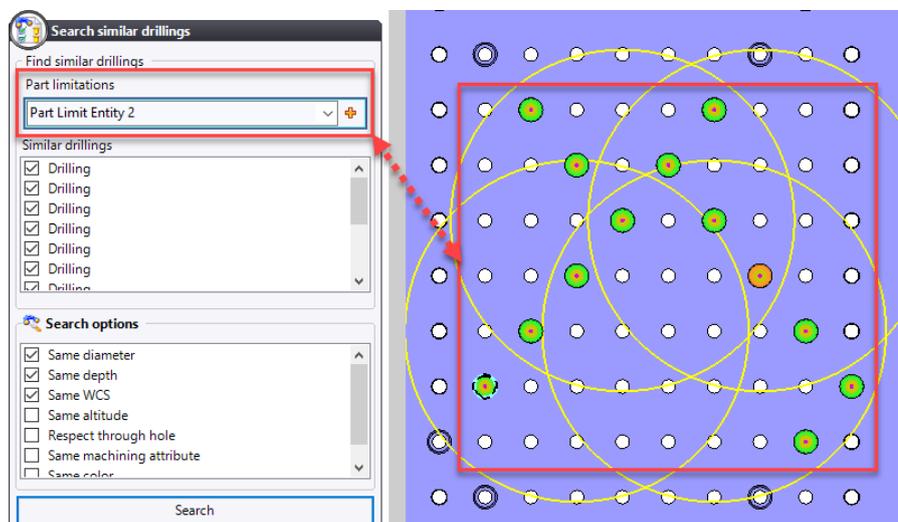
### Limiting the analysis area: Curves

You can now define an analysis limitation area using bounding curves, in addition to the XYZ bounds.



### Limiting the analysis area: Applying a method

Similar to what already exists in the drilling and cylinder analysis, you can limit the analysis area using the **Apply Method** command.



**Restriction:** The selected reference face must be included in the limitation area to be able to perform the analysis in the referenced area.

### ***Searching for identical MFs in the hole machining operation***

You can now search for the MFs (manufacturing features) of the identical holes in the **Geometry** option of the hole machining operation.

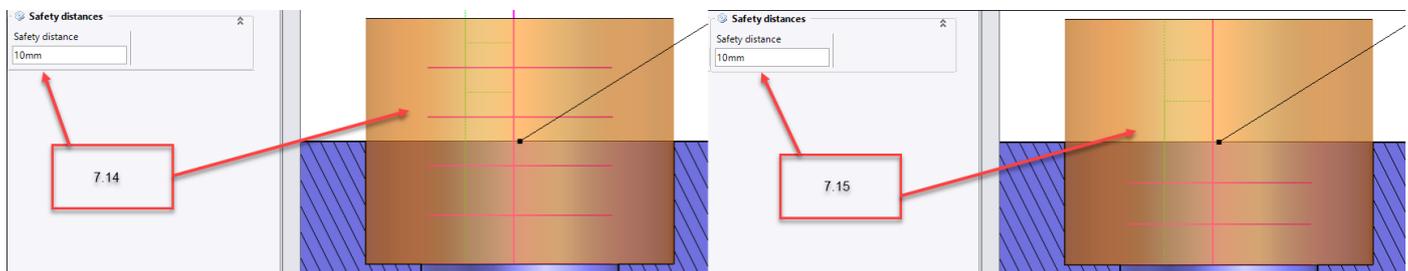
To do this, you only have to click on the  **Find geometry similar to the current one** icon in the **Geometry** section. The icon is only accessible if only one element is selected in the list.

**Restriction:** The stock of the part is not taken into account in the selection. The depth is that of the hole defined in the CAD environment.

### ***Management of the starting altitude in drilling/spot facing by contouring and pocketing***

In version 7.14, the safety distance affected the starting altitude of the drilling/spot facing by contouring and by pocketing operations, which could produce empty passes.

In version 7.15, the safety distance no longer has any influence on the starting altitude of the drilling/spot facing by contouring and by pocketing operations, but it is still active for the drill cycles.

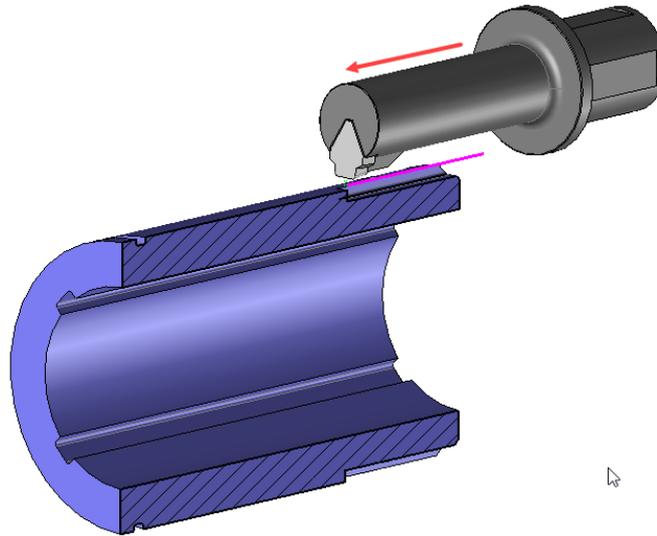


## 2D Milling

### Broaching

**Broaching** is a machining process based on the use of a broaching tool (shape tool).

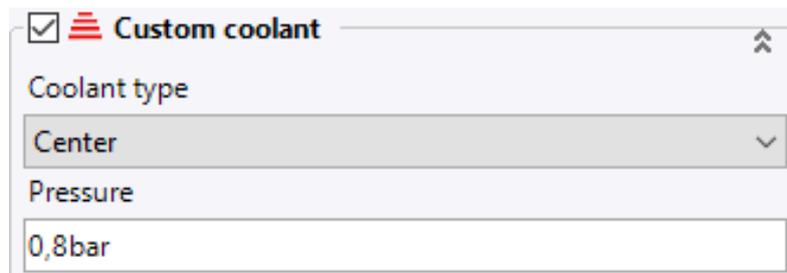
The principle consists in removing the material along a slot in a strict direction, between the finished part and the stock, by extending it to the limits of the geometry.



### End milling: BoostMilling

#### Custom coolant for plunge

You can enable the custom coolant feature for the plunge in the operations using the **BoostMilling** strategy (end milling and 3D roughing).



### Integrated contouring

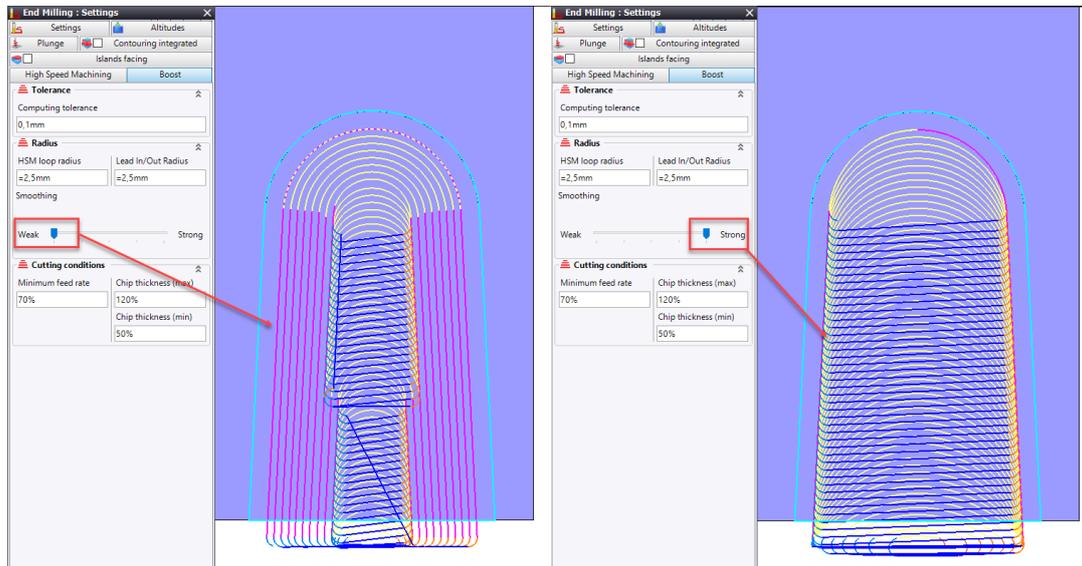
Integrated contouring is available in **Boost** mode but, unlike a classic strategy, the feed rate is automatic and reduced on corner arcs to preserve the rate of material to be removed.

## Path smoothing

The path smoothing tolerates not being as close as possible to the finished shape with paths parallel to the contour. This reduces the abrupt changes in direction by opening the material with larger loops and reduces the variations in the feed rate.

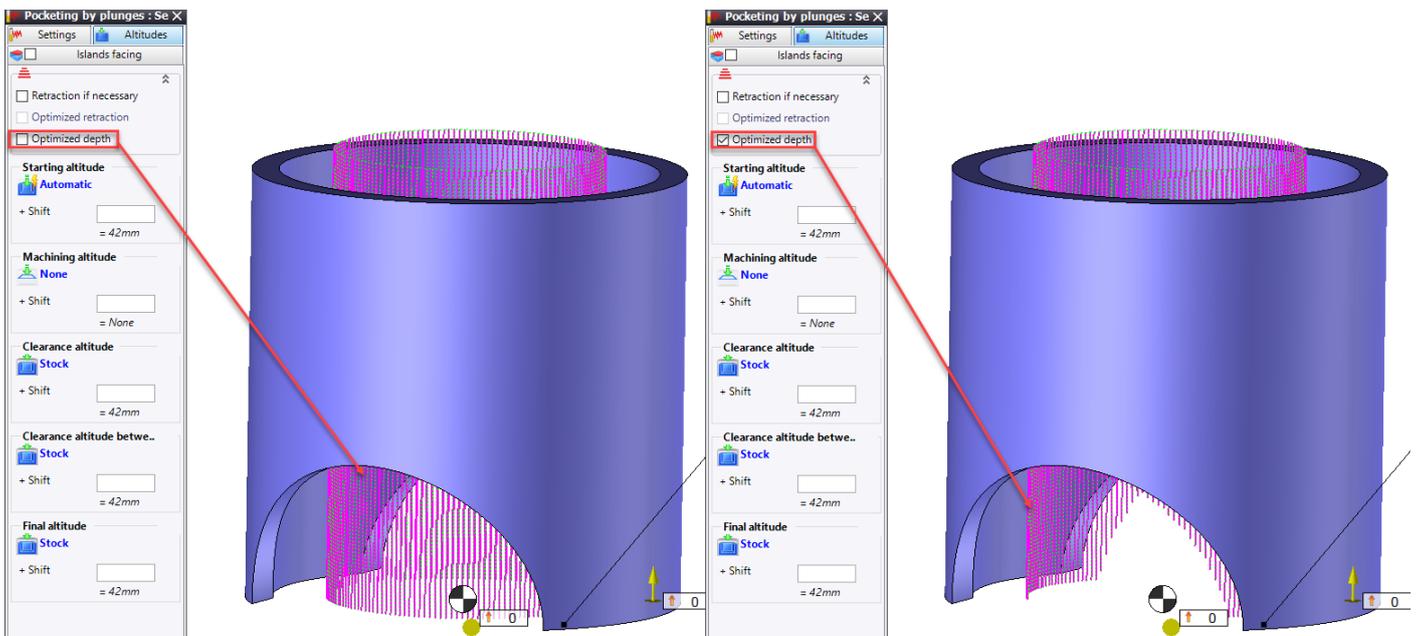
Thus, by smoothing the path, the variation of the feed rates is less important and the machine is used much less. However, it should be noted that for a machine that would be very fast, smoothing could have the opposite effect and increase the machining times.

It is then now possible to influence this smoothing to match the characteristics of the machine to be controlled.



## Management of hollow stocks in vertical milling

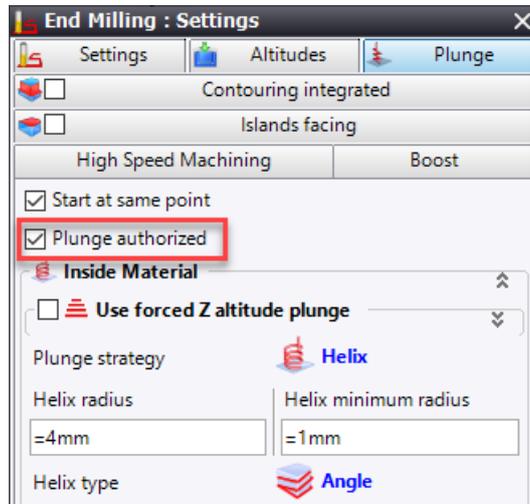
In the 2D contouring by plunge, pocketing by plunge and punching pocketing operations, you can now manage the hollow stocks, i.e. calculate plunge altitudes that are above the machining altitude as soon as the tool passes through the stock.



**Restriction:** This setting is not available if the compensation plunge strategy is selected.

### End milling: Plunge into material allowed

The **Plunge authorized** option is now available in an end milling operation, as it was in the side milling operation. The box is checked by default for center-cutting tools and unchecked by default for non-center-cutting tools.



### End milling: Straight lead out in open pocket

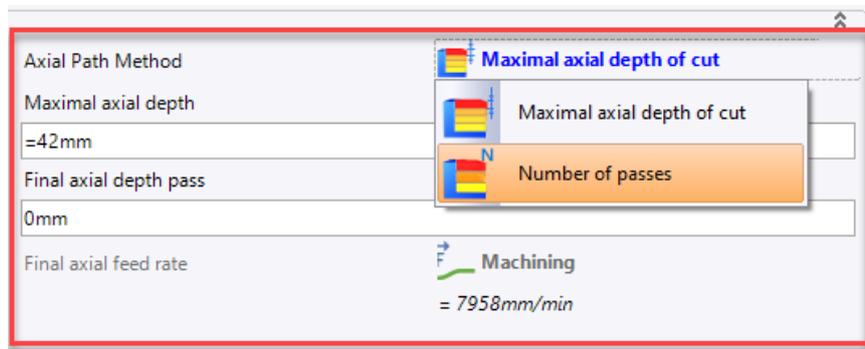
In order to optimize the machining paths, the end milling algorithm no longer machines around the sharp edges at the exit of an open pocket, where possible.



### Number of passes for Ap

In end milling operations, you can now select the axial path method either by number of passes or by maximum axial depth. This option is available in the settings or in the label of the operation.

The number of passes is applied between the Z maximum and Z minimum of the limitation area.



This option is also available for 3D roughing, swarf machining and 5-axis contouring operations.

### Breaking edge milling: Three positions

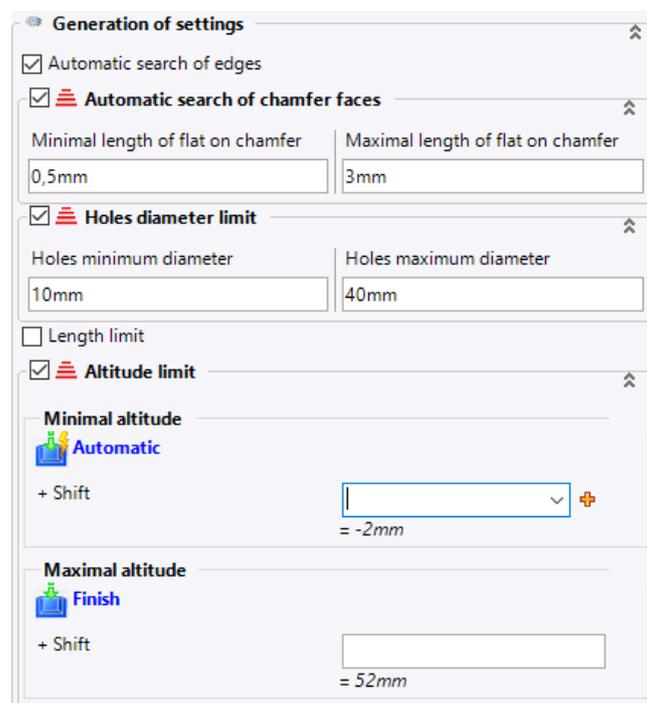
The breaking edge milling operation provides a new **Three positions** method to calculate the position of the tool. The calculation is performed on three positions (low, normal and high) and takes the one that maximizes the machined length of the initial profile.

The operation of the other two modes remains the same for the old files. Only the activation of the **Three positions** mode causes the changes.

### Breaking edge milling: Minimum/maximum values

When performing a breaking edge milling operation, you can now:

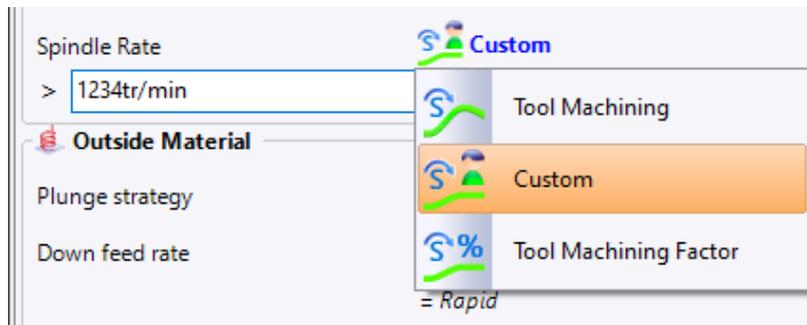
- enter the minimum and maximum values for the flat length (automatic chamfer selection);
- enter the minimum and maximum diameters for the search of full circles (automatic edge selection).



### Plunge spindle rate

You can now define the type of spindle rate during the plunge operation.

This option is also available for hole machining (**Settings** option > **Drilling intersections** tab) and roughing (**Settings** option > **Altitudes** tab > **Scallop optimization** box).



### Manufacturing Features: Pocket with islands - Island contouring

The islands can now be managed when machining on a MF (Manufacturing Feature). For example, when islands are present when creating the pocket operation in the CAD environment, the corresponding MF will be able to indicate their presence in the CAM environment.

### Marking Manufacturing Feature

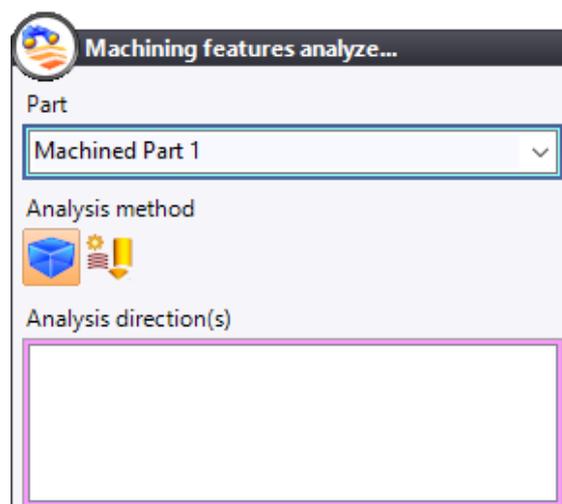
The marking Manufacturing Feature is now included in the engraving operation. It can be selected directly by right-clicking on the element in the graphics area.

The marking Manufacturing Feature is also available in:

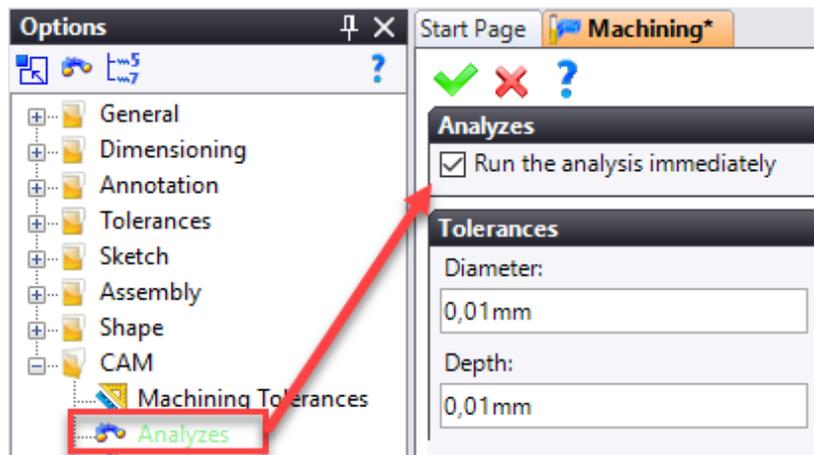
- the methods (element selection);
- the association documents;
- the machining feature analysis.

### Machining feature analysis in topology mode

The **Machining features analysis** command now allows you to add a list of directions to restrict the analysis to certain faces in order to save analysis time and to filter the list of results. Defining preferential directions allows a finer analysis and the detection of through pockets (bottomless pockets).



It is of course possible to not automatically start the analysis in order to be able to select these directions by unchecking the **Run the analysis immediately** box in the options of the machining document.



**Restrictions:**

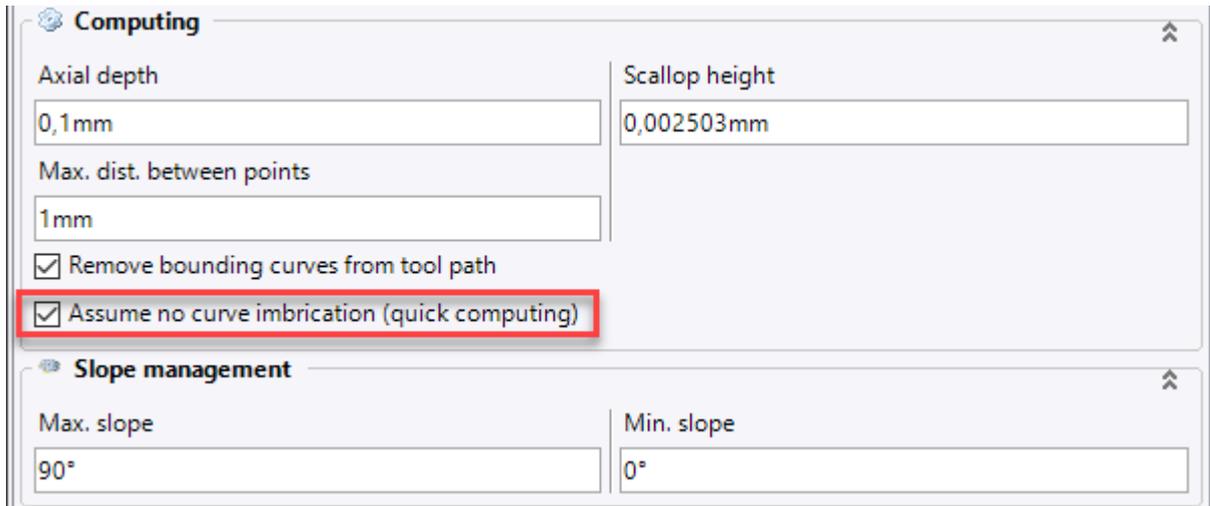
- The analysis of through pockets is specific to the topological analysis. In **MF** mode, this additional analysis is not performed because it makes no sense: the MF fully describes each machinable shape.
- A through cylindrical shape (through hole) is not filtered by default and is recognized as a through pocket.
- The outer contour of the part is not recognized as a machining feature to possibly apply a contouring operation depending on the analysis direction.

## 3D Milling

### Constant Z performance improvement

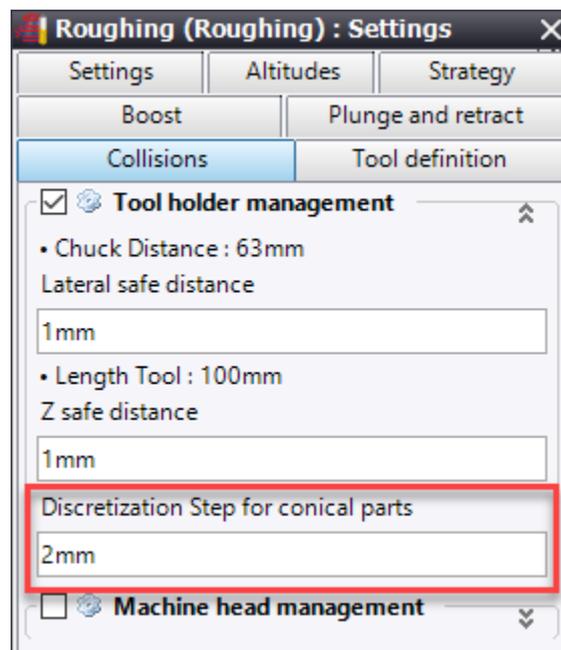
In a constant Z operation containing many bounding curves\*, checking that these curves do not intersect can take a long time. Accordingly, in the case of molds for which you are sure that these curves do not intersect, you can now ask **TopSolid** not to perform any check in order to save time thanks to the new **Assume no curve imbrication (quick computing)** box.

(\*) Below 75 curves, this option is not visible because the time saving would not be significant.



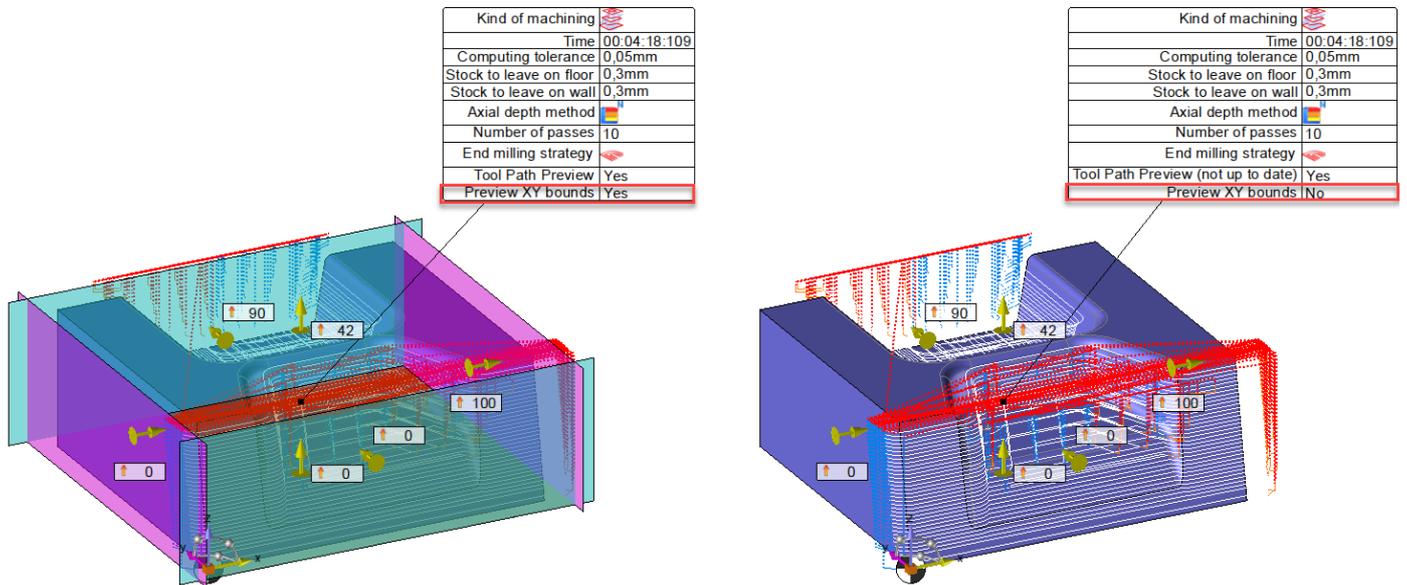
### Avoiding collision with the conical tool shank

Prior to version 7.15, when the collision management was enabled, the tool shank and its conical shape were completely ignored. Thanks to version 7.15, this conical shape is now supported. In order to correctly take into account the conical part, you can enter a **discretization step** according to the tool axis.

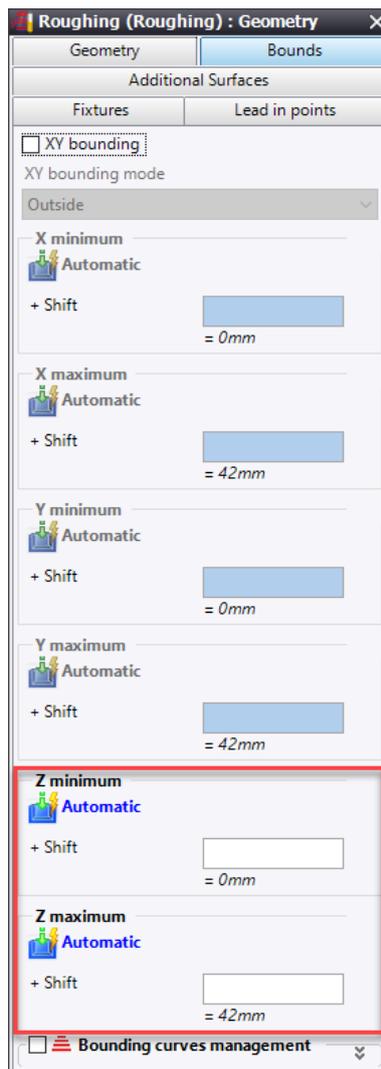


### XY bounding

In the 3D milling operations, you can now hide the preview of the XY bounds from the operation's label in the graphics area.



You can also choose not to have XY bounds, but only Z bounds.



### **Constant Z: Compensation in the XY plane in tangent mode**

When performing a constant Z machining, when the lead in and the lead out are in **Tangent Arc** mode, you can now allow the compensation code to be sent.

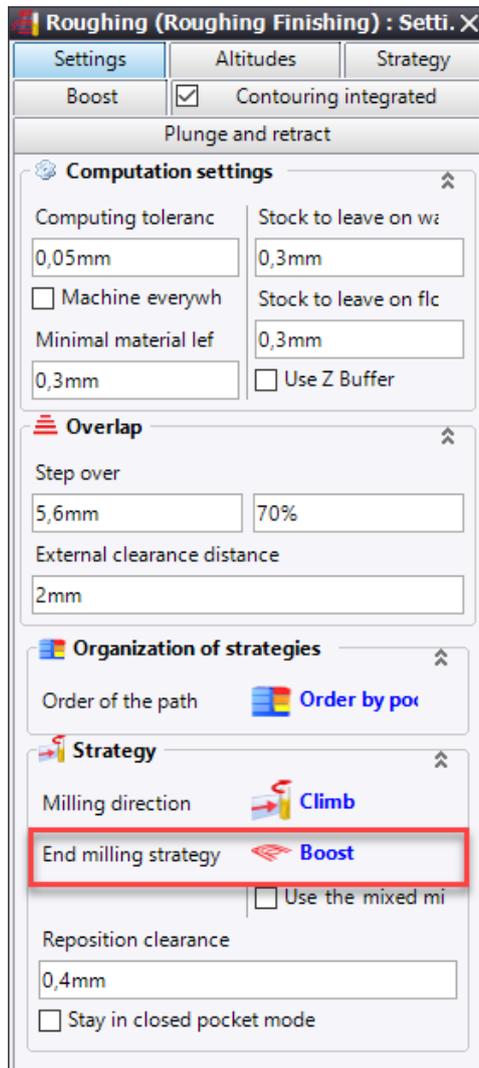
This was already possible when orthogonal lead in and lead out were performed.

### **Automatic machining of electrodes or parts**

A different machining method can now be selected for each electrode when executing the automatic machining process in the work document.

### **BoostMilling roughing finishing**

The **BoostMilling** strategy is now available in the roughing finishing operation.



## 4D Milling

### 4-axis radial engraving

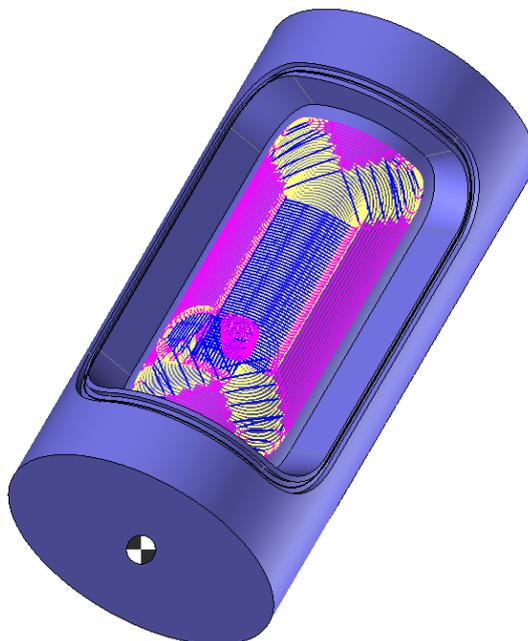
As is already the case in the side milling and slot milling operations, you can now perform 4-axis radial engraving operations.

Spotting Drill D6 SD6  
Engraving  
1: Engraving (Radial)



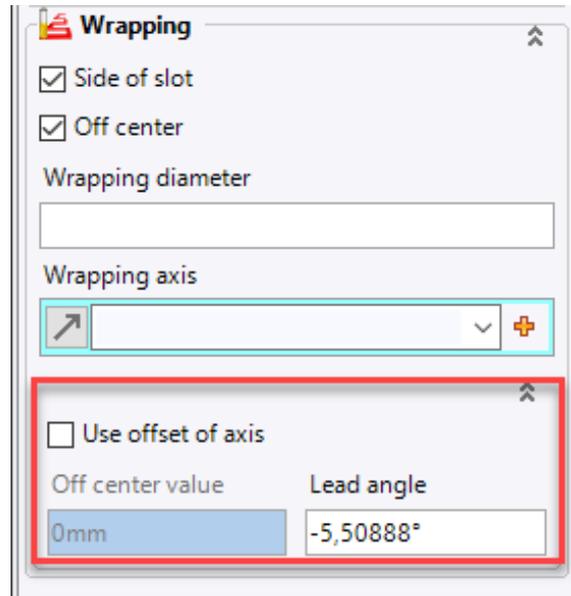
### 4-axis radial BoostMilling

The **BoostMilling** strategy is now available in the 4-axis radial roughing and 4-axis radial end machining operations.



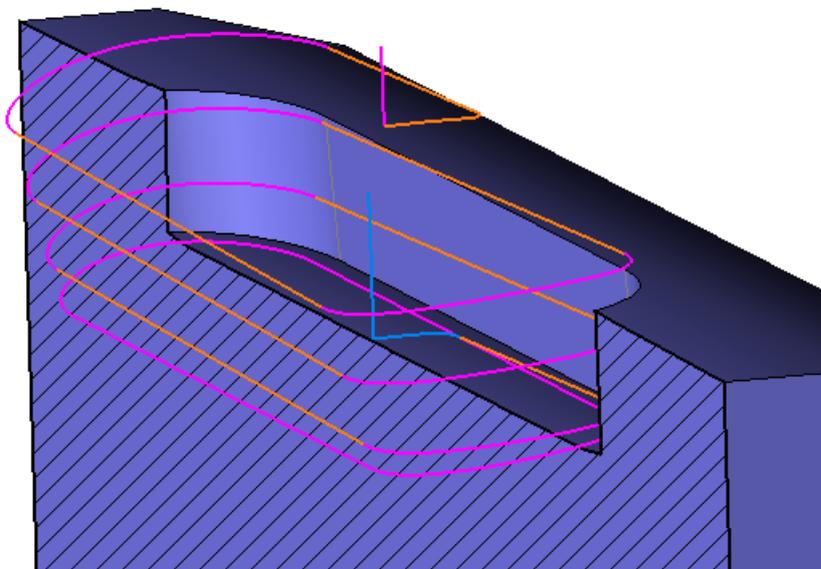
### Y-axis offset side milling

When performing a 4-axis radial operation, you can now machine sides that are offset in relation to the rotation axis of the part. You can enter either the **lead angle** or the **off center value**.



### 4-axis radial side milling: Helical mode

The **Helical mode** option is now available in the 4-axis radial side machining operations.

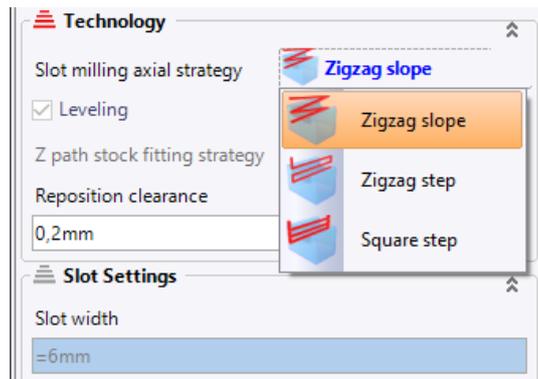


This mode works in the following cases:

- with a climb milling or conventional milling direction;
- with and without off-center;
- with one or more passes;
- when the profile consists of segments or arcs and segments;
- in corrected and shifted compensation mode.

### 4-axis radial slot milling: Slope machining

You can now perform a 4-axis radial slot milling operation using the **Zigzag slope** axial strategy.



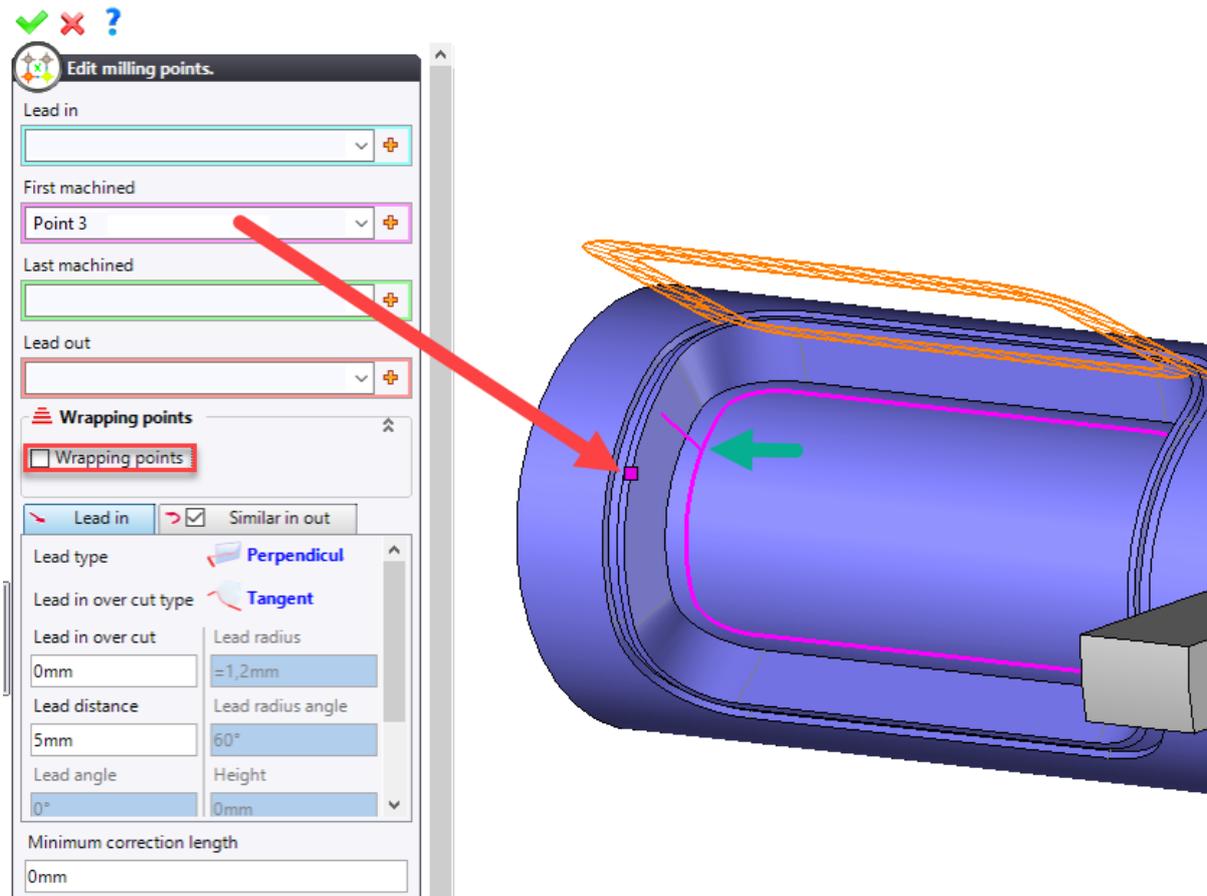
### 4-axis radial end milling: Plunge outside material

When performing a 4-axis radial end milling operation, you can now select the **Outside material** plunge strategy.

### 4-axis radial end milling: Wrapping lead in/lead out point

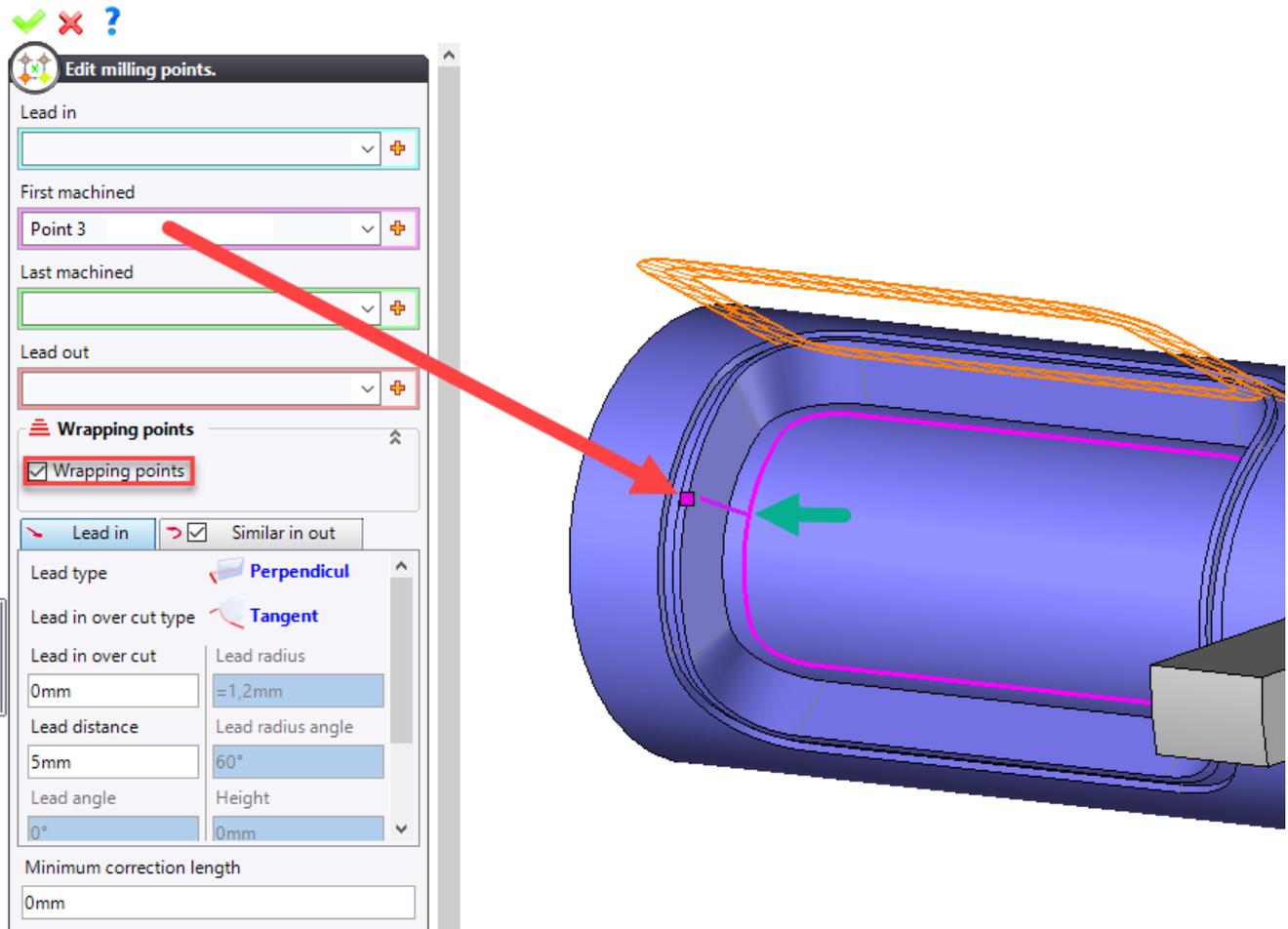
The new **Wrapping points** option allows you to indicate whether the lead in/lead out point to be selected is already on the part (wrapped) or not.

If the box is unchecked (default behavior), the point is projected in the WCS direction, before being wrapped.



The red arrow indicates the selected point and the green arrow indicates the starting point for the machining.

If the box is checked, **TopSolid** considers that the point is already wrapped.

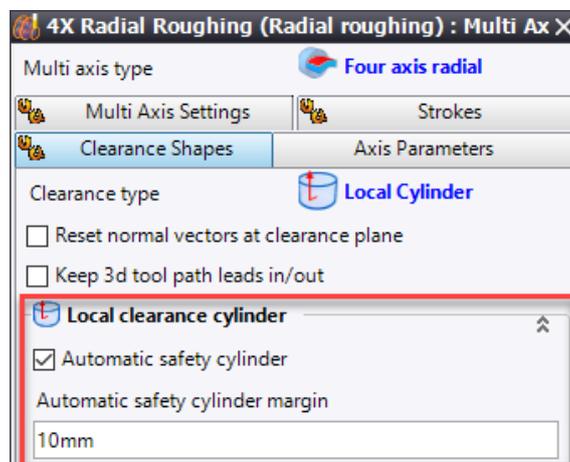


The red arrow indicates the selected point and the green arrow indicates the starting point for the machining.

The **Wrapping points** option is available in the end machining and contouring operations.

### 4-axis machining using a head-head machine Automatic clearance shape

Version **7.15** integrates a new mode to automatically calculate the clearance shapes which automatically builds the local clearance shape selected according to the margin defined around the part to be machined.



The automatic mode is enabled by default for all new documents. For old documents, this mode is disabled and the already entered values are kept.

## 5D Milling

### Impeller/blisk machining

Numerous commands have been added for the machining of impellers and blisks (blade disk parts).



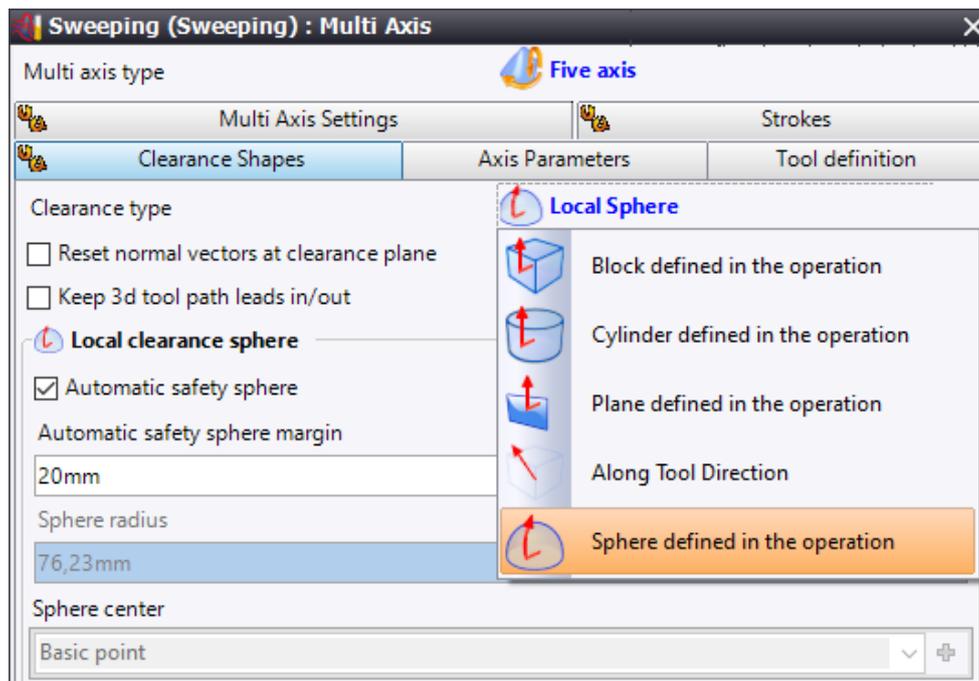
### 5-axis pocket machining

You can now machine a pocket in 5-axis continuous mode.



### 4-axis and 5-axis clearance shape: Clearance sphere

The sphere shape has been added to the list of clearance shapes for 5-axis machining operations.

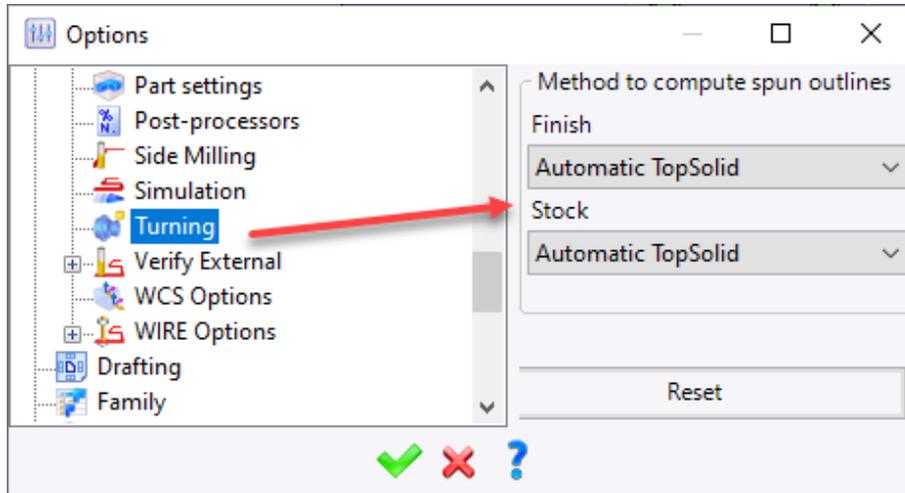


**Restriction:** If the sphere is contained in the part, the tool paths will still be performed towards the sphere, which creates a risk of collision. It is therefore essential that you properly adjust the size of this sphere to avoid collisions.

# Turning

## Configuring the method to calculate spun outlines

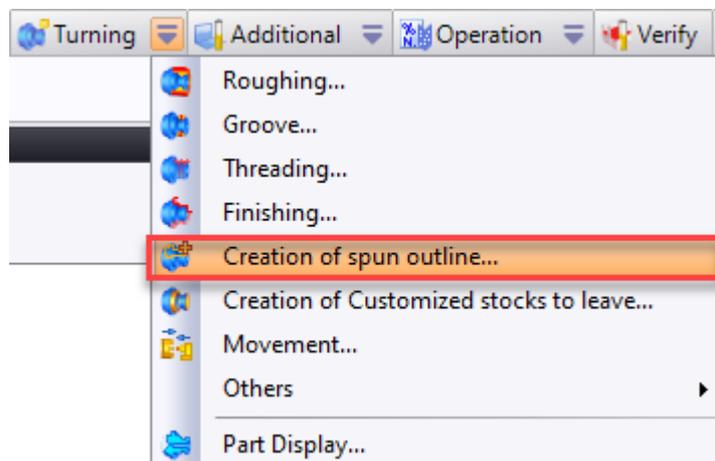
A **Turning** section has been added to the **TopSolid** options, allowing you to configure the method to calculate the spun outlines for the finished part and the stock.



A new **Turning settings** section is also available in the Options tree of the machining document. By default, the calculation method is the one defined in the application options.

## Preparing the revolved silhouettes

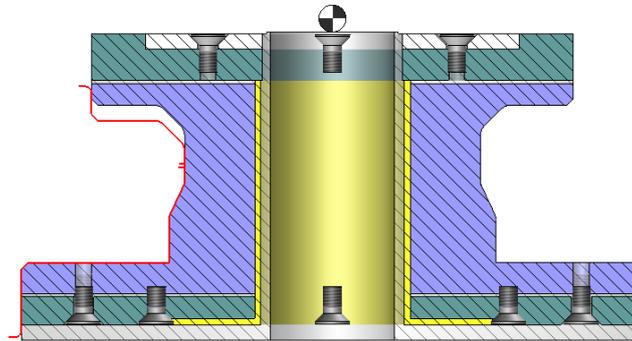
You can now prepare several revolved silhouettes before performing a turning operation. To do this, you only have to select the **Creation of spun outline** command as many times as needed.



**Restriction:** If a spun outline with an identical revolution axis already exists, you cannot confirm the dialog box and a message warns you of the problem.

### **Assembly machining**

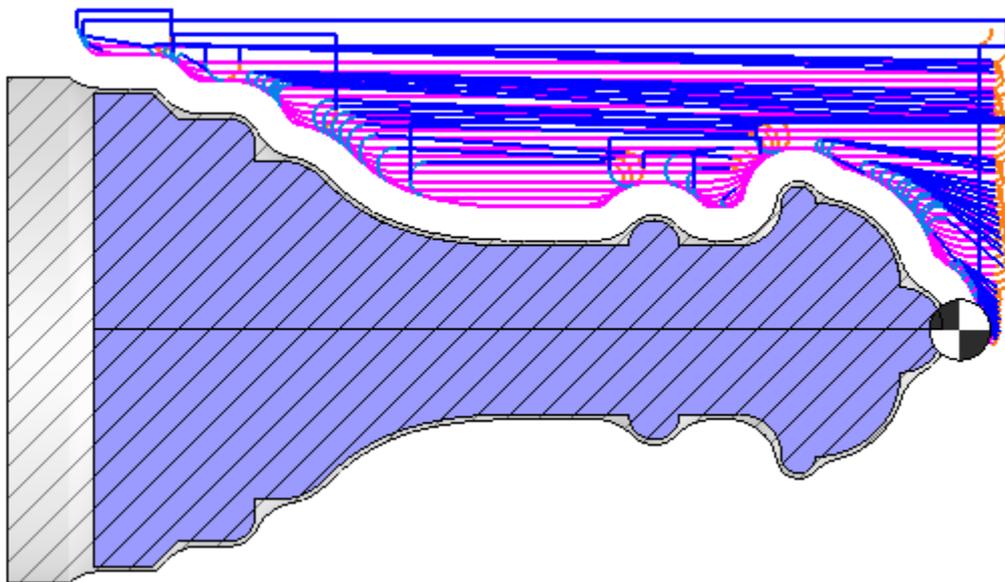
The new version **7.15** allows the machining of assembled parts in turning. The way in which start/end faces can be selected has been improved so that you can select faces that do not belong to the same finish, but belong to the same machining part.



**Restriction:** TopSolid does not support the case of X parts separated by a gap greater than 0.01mm in the same bar.

### **BoostTurn**

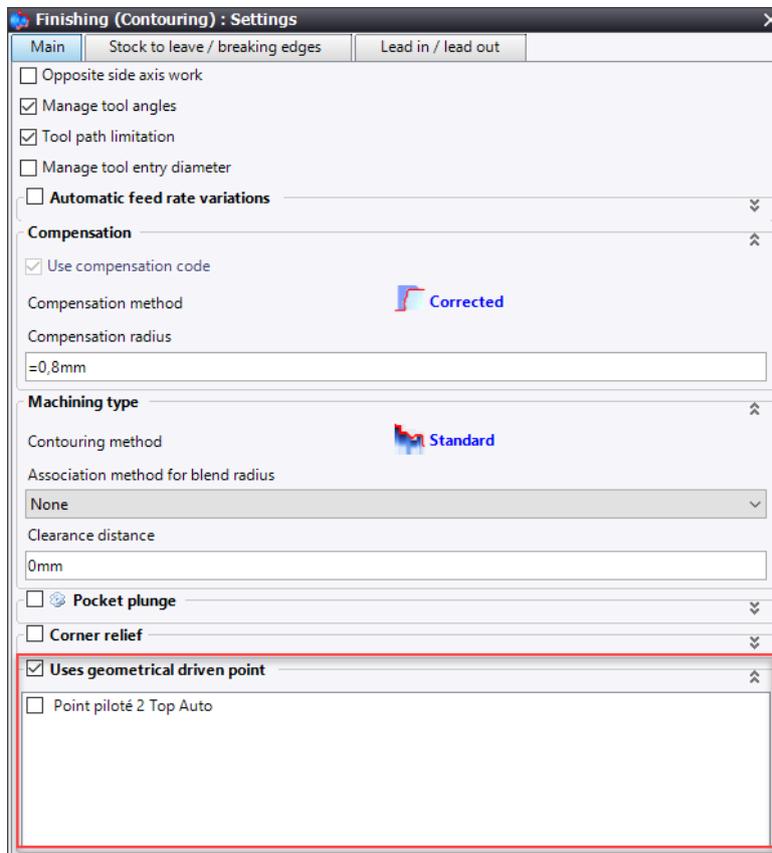
The new **BoostTurn** strategy allows you to perform dynamic machining paths in turning operations by using a monodirectional or bidirectional mode.



**Restriction:** Only tools with circular inserts are supported.

### Using a geometrical driven point in contouring

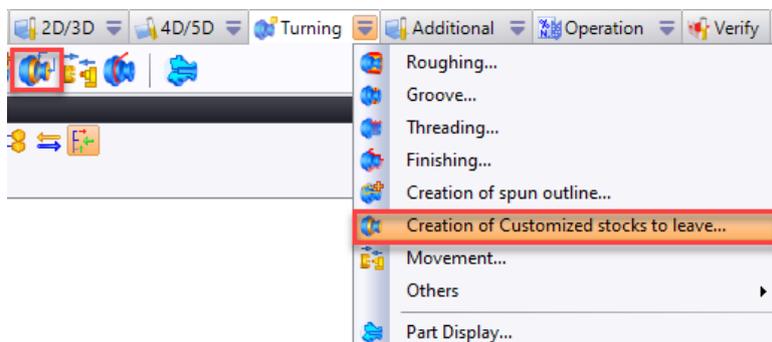
The objective is to allow tool path calculation with a driven point, but to drive the tool using another point. The new **Use geometrical driven point** box has been added to the finishing operation settings, allowing you to select a geometrical driven point.



**Restriction:** Only tools with more than one driven point are managed.

### Customized stocks to leave

You can now define customized stocks to leave on the faces/diameters of the finished part. The new **Creation of customized stocks to leave** command allows you to generate a custom profile from the revolved spun outline of the finished part.



### Groove roughing: First plunge retraction

When performing a groove roughing operation, a clearance can now be automatically applied to the retraction of the first plunge, if possible.

## Virtual Jog

### Improved description of virtual jog items

Improvements have been made to the description of some virtual jog items in order to make user-defined information visible before editing.

- **Approach to hold part:** You can adjust the approach type (stock, finish or point), as well as the shift value. In the case of a point, its coordinates indicated in the operation WCS are specified in brackets.

Type	Value
Approach to hold part	Approach type: Stock Shift: 10mm

*Stock approach type.*

Type	Value
Approach to hold part	Approach type: Finish Shift: 10mm

*Finish approach type.*

Type	Value
Approach to hold part	Approach type: Point (0,03;0,11;0,055) Shift: 10mm

*Point approach type.*

- **Part transfer:** The name of the item is mentioned in the description.

Type	Value
Part transfer	Part transfer

- **Part pulling:** You can indicate if you are performing a shift equal to the part length and/or enter the value of the simple shift.

Type	Value
Part pulling	Shift: part length + 10mm

*Part length shift and value.*

### Enhanced ergonomics and information

Many enhancements have been made to make sure that the information sent to the user is clearer and more understandable:

- **Invalidity messages:** Improvement and translation of the invalidity messages displayed on the **Settings** icon of the virtual jog.
- **Turret indexing:** This item no longer appears in the list of jog items when the machining document uses a machine that does not have a turret.
- **Rotating element:** In the virtual jog editing context, selecting the rotating element triggers the automatic selection of the part if there are less than two parts in the document.
- **Asterisk on the Cutting conditions icon:** An asterisk symbol now appears on the **Cutting conditions** icon to indicate that cutting conditions are required (when a jog item performs a feed rate movement).
- **Part selection:** When editing a virtual jog operation, if the machining document contains only one part, the **Part** field of the **Geometry** option is automatically filled in.
- **Tool selection:** When editing a virtual jog operation, if the **Tool** icon box is checked, the last tool used (if a tool is used) is selected by default.
- **Automatic WCS selection:** When selecting and confirming an **Approach to hold part**, **Part pulling** or **Part transfer** jog item, the WCS is automatically created.

**Warning:** The automatic WCS selection is not possible in the following cases:

- o More than one part holder that is not a "drivable part holder".
- o More than one part on the part holder.

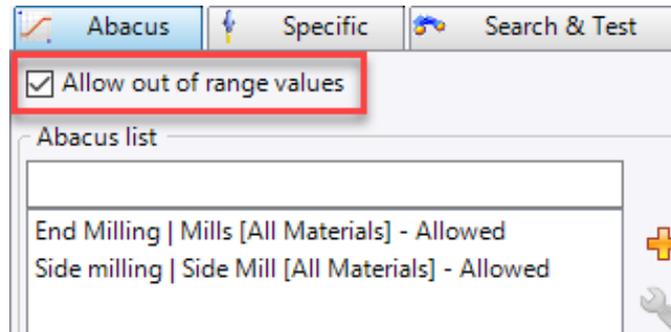
## Tools and Cutting Conditions

### Exceeding cutting condition ranges

When using an abacus, you can now allow cutting condition ranges to be exceeded.

If the **Allow out of range values** box is checked, you can use the abacus even if the Ae/AP values are outside the defined ranges. The highest cutting speed value is taken or a cross-multiplication is applied with the next value.

If the **Allow out of range values** box is unchecked, the NaN ("Not a Number" or "Not a Numeric") value is returned and you cannot confirm the operation.

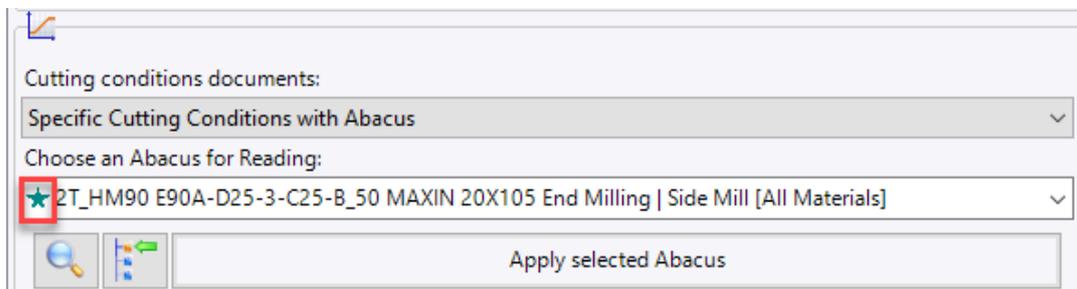


**Restriction:** This option applies to all abacuses in a cutting condition document.

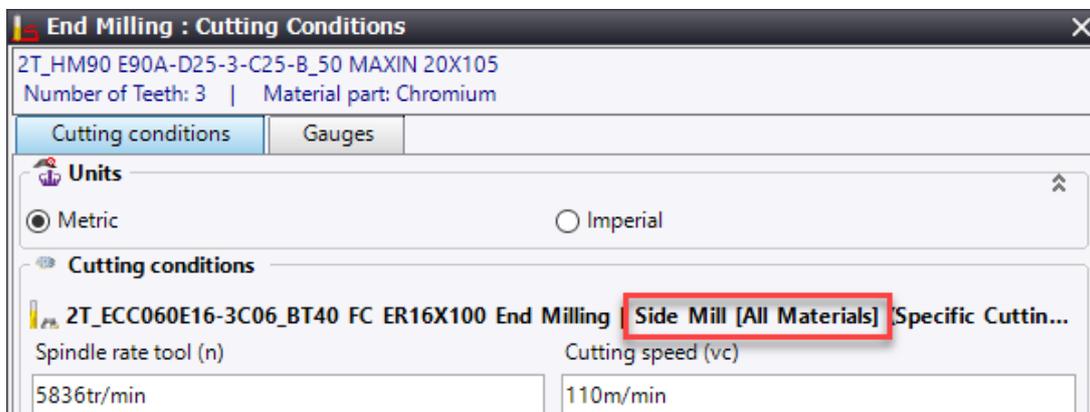
### Specific cutting conditions of a tool linked to an abacus

The abacus/tool readability has been improved through two options.

In the **Cutting conditions** tab of the operation, a star symbol is now visible in front of the tool abacus and the general abacus.

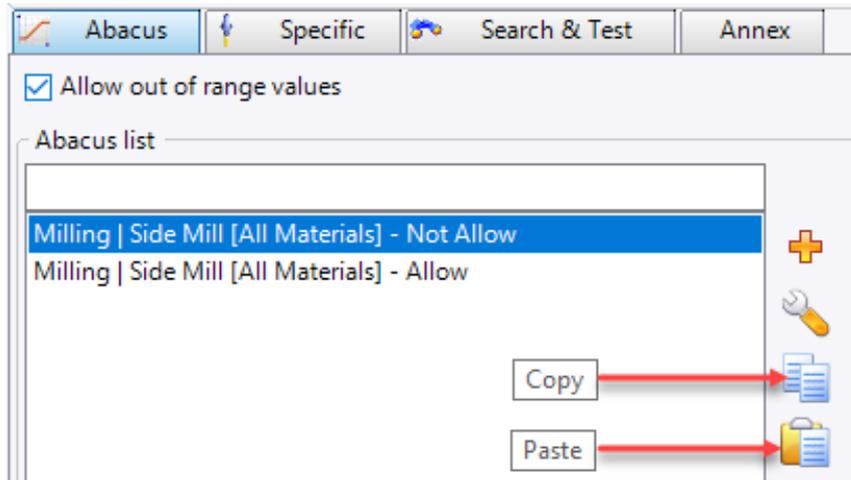


In addition, the name of the abacus linked to the tool is recalled when the abacus is selected.



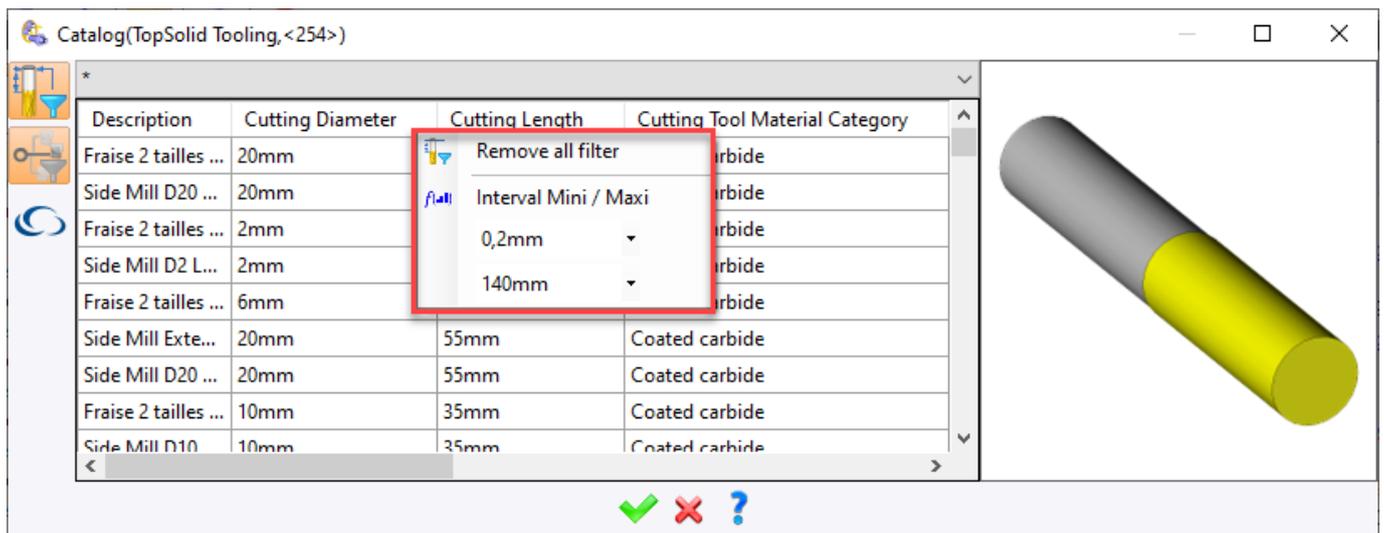
### Copying an abacus between two cutting condition documents

You can now copy an abacus from a cutting condition document and paste it into another document of the same type.



### Tool catalog column filter

As in the Tool Manager, a filter to indicate a minimum/maximum interval has been added to the tool catalog column headers.

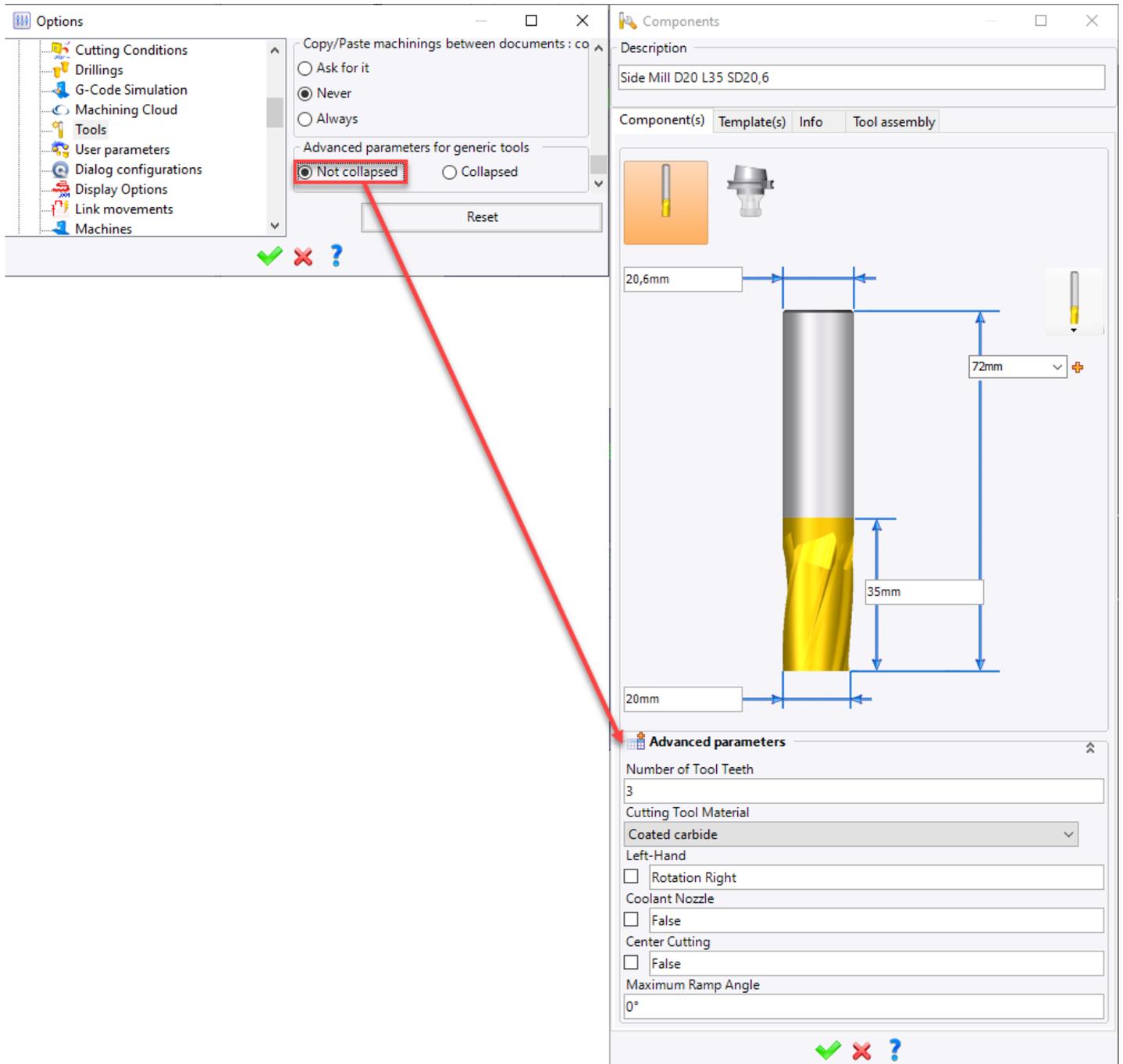


**Restriction:** The filters are not saved when closing the dialog box.

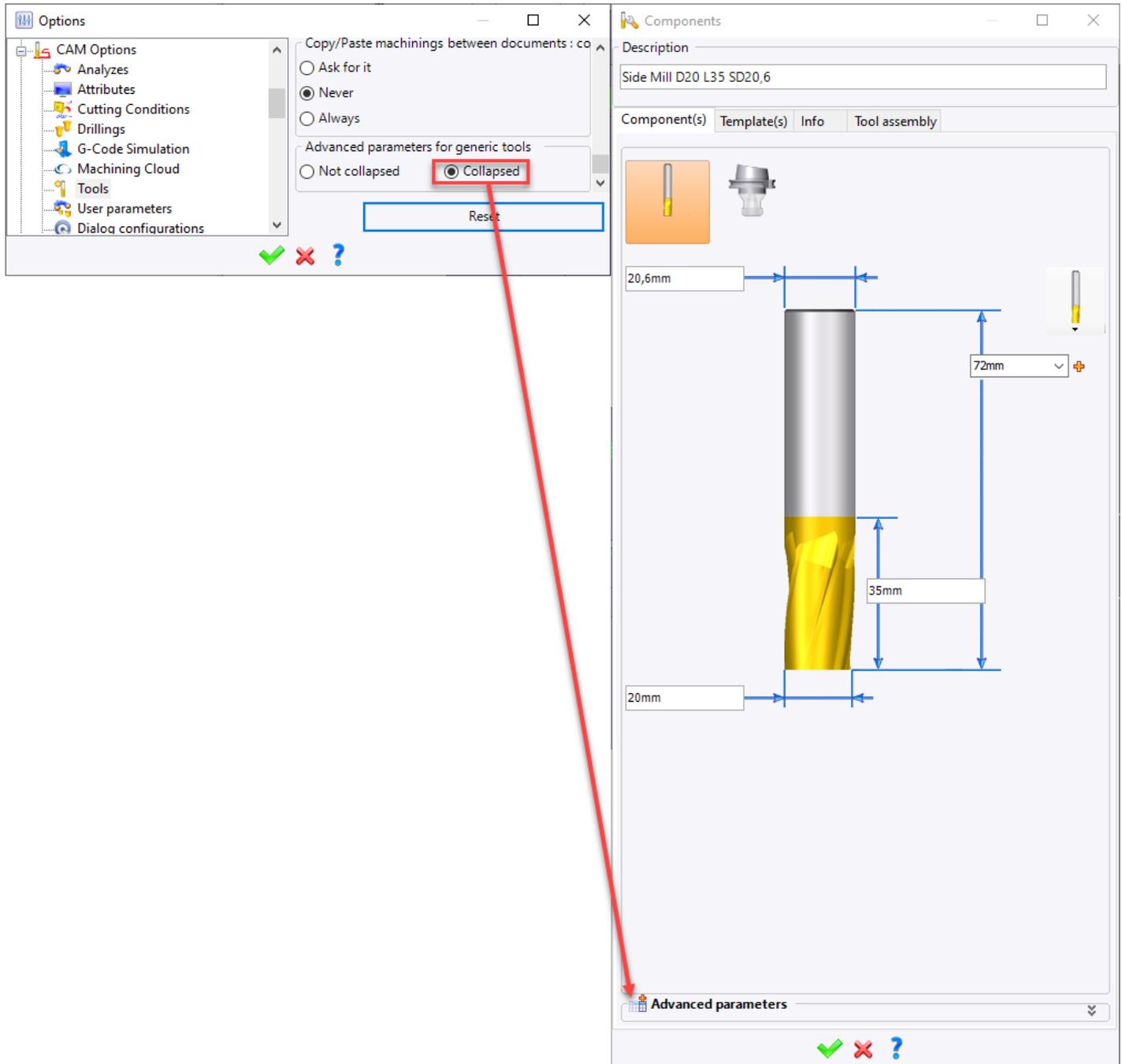
### Advanced parameters for generic tools

You can now choose whether or not to display the advanced parameters of the generic tools by default. This option can be adjusted in the **TopSolid** options.

### Not collapsed mode

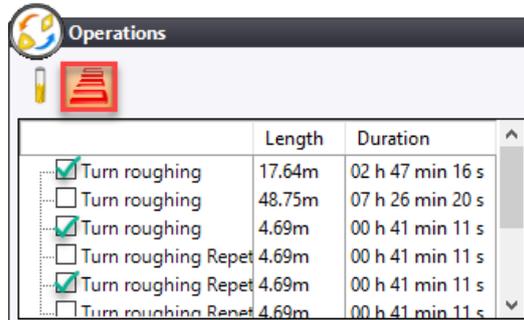


### Collapsed mode

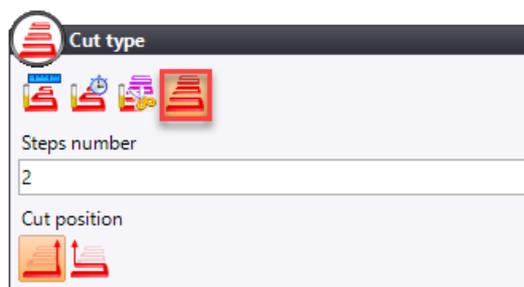


### Tool/Insert lifetime

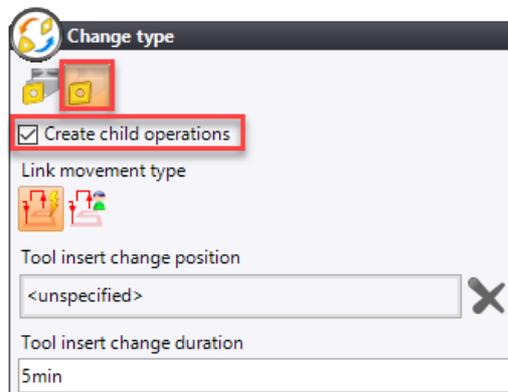
For both milling and turning operations, a new mode has been added to the **Lifetime management** command, allowing you to adjust the lifetime for certain operations rather than for the entire machining range.



In addition, a new **Steps number** cutting type is available. It allows you to enter the number of steps allowed before considering the tool or insert to be worn out.

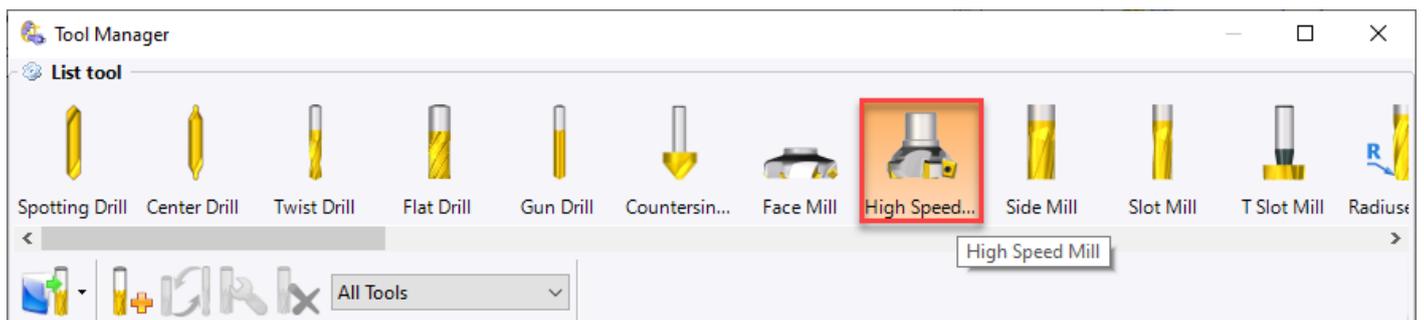


In order not to multiply the operation lines in the NC Operations tree each time the insert is changed, the new **Create child operations** box allows you to choose whether or not to create a child operation in the machining range. This option is only available in insert change mode.



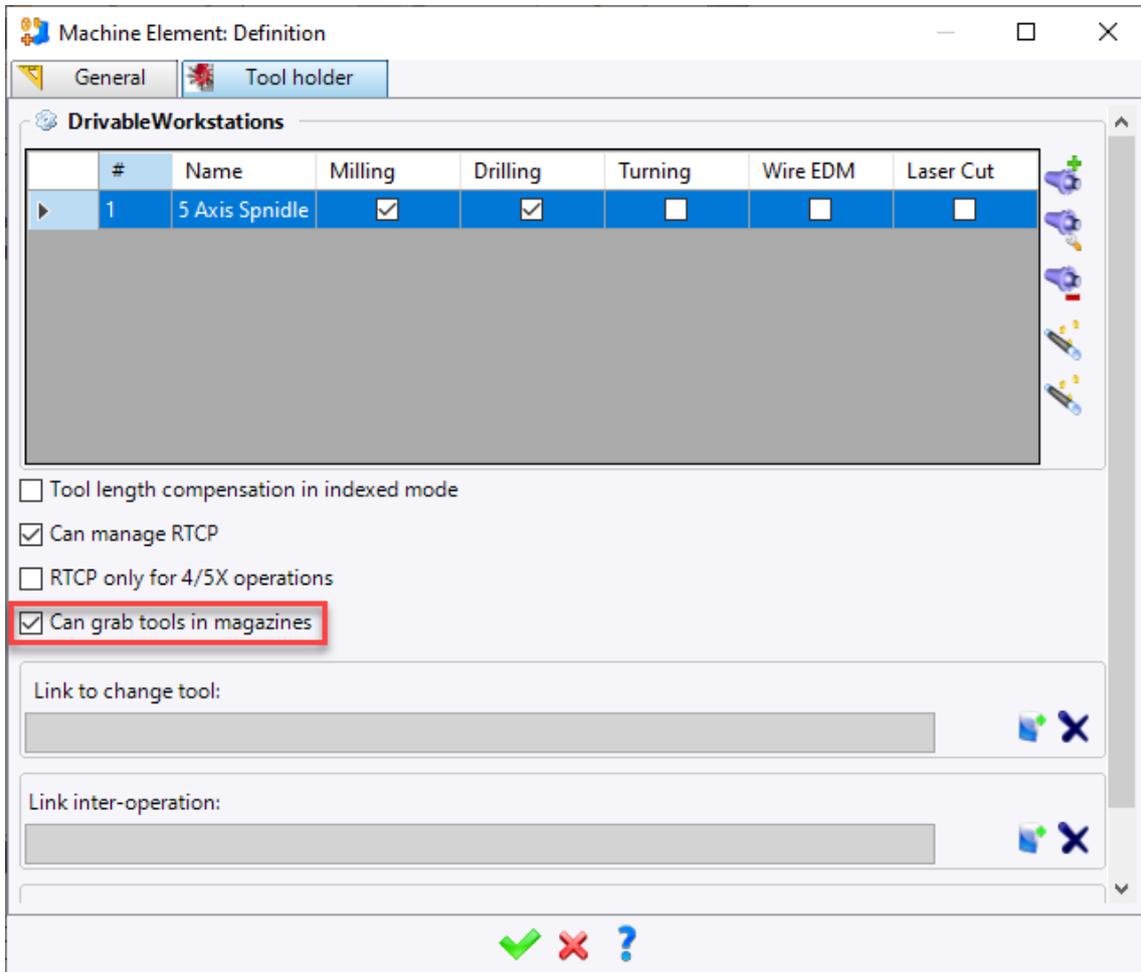
### New family of tools

A new family of tools named **High Speed Mill** is available in the Tool Manager.



### ***Tool magazine with a tool holder having several mounting faces (turret)***

In a machine document, the new **Can grab tools in magazines** box allows you to search for tools in a magazine even if the tool holder does not have a power axis.

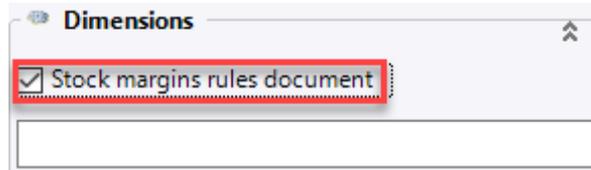


**Restriction:** The sides of the tool holder receiving the tools from a magazine must not have any tools mounted.

## Machined Part Setup

### Margin rules for block and cylinder stocks

In the machined part setup document, when defining the NC part, you can now determine the margins to be used for the block and cylinder stocks. The margin rule documents work with the dimensions of the finished part to be enclosed and the material used. The dimensions are automatically extracted from the selected finished part and the material is set in the **Materials** option.



**Restriction:** You must first have created and adjusted a margin rule document (**Special** tab > **Cylinder Margin Rules** or **Box Margin Rules**).

### Show/Hide finish

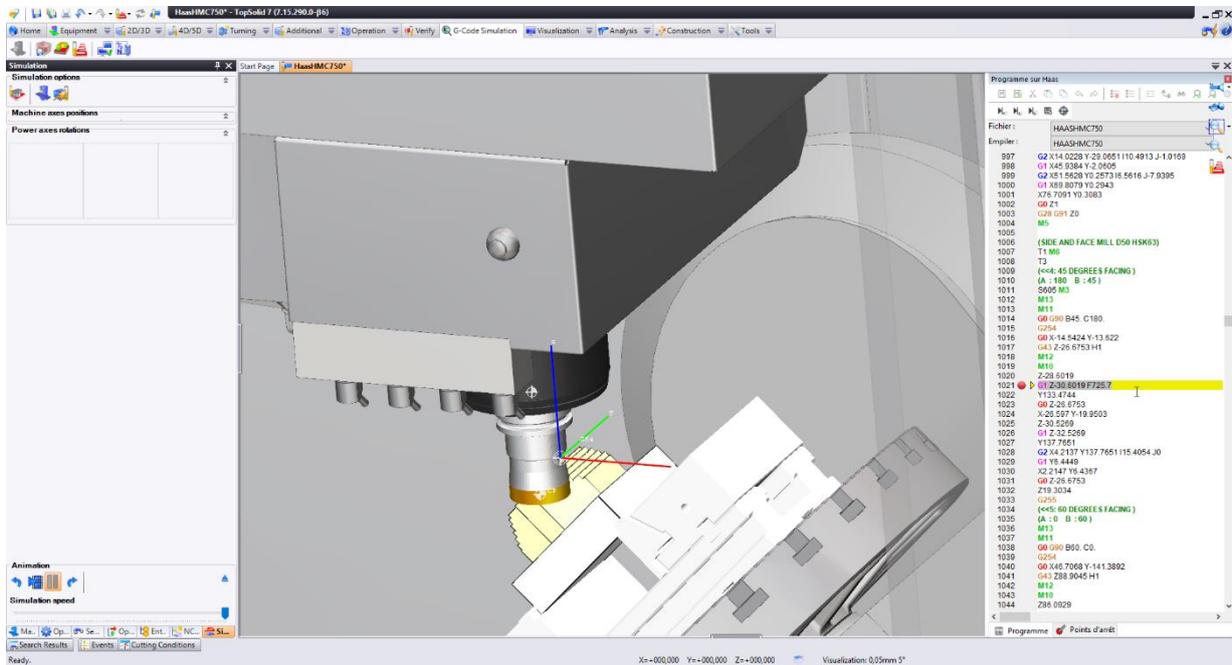
You can now show or hide the stock and the finished part directly from the graphics area.



# Simulation and Verification

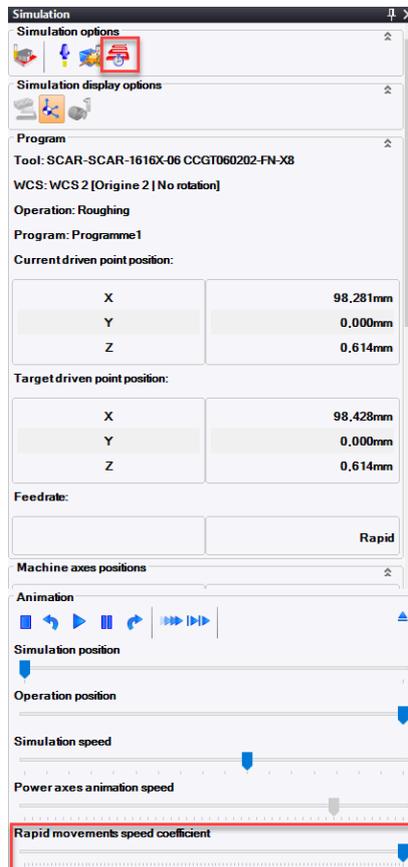
## G-code simulation

You can now simulate the G-code generated by the post-processor in the verification phase.



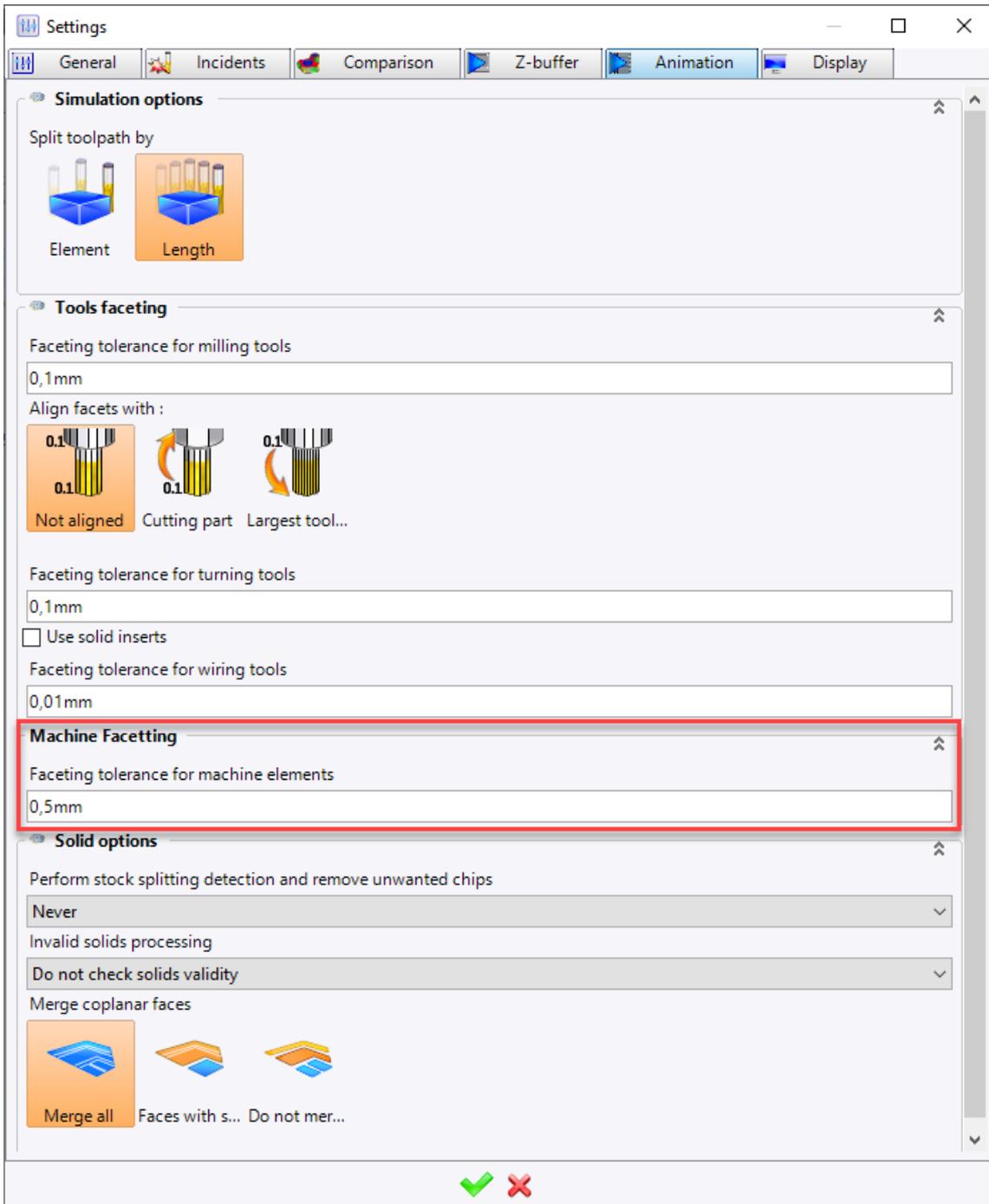
## Speed coefficient for rapid movements

In the simulation window, a new slider allows you to adjust a speed coefficient to be applied when simulating rapid movements. It is only available in simulation with time increment mode.



### Faceting tolerance for machine elements

In the verification configuration, you can now apply a **faceting tolerance** to the machine elements and environments. In machine mode only, the tolerance change forces you to recreate the machine, which repositions the simulation to the beginning (a confirmation is requested from you).

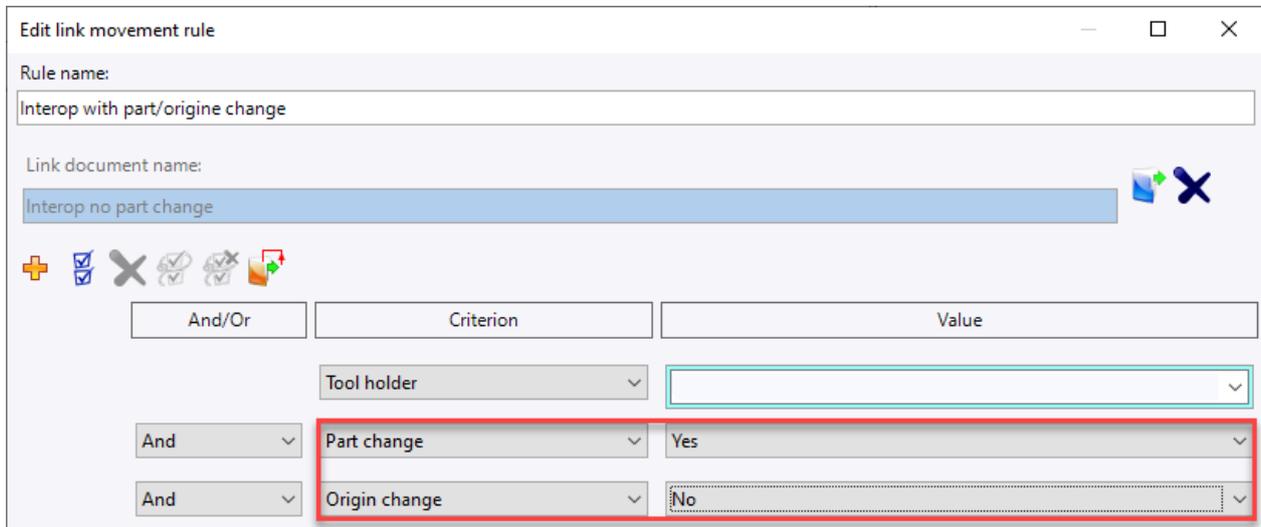


## Link Movements

### Link movement rules

#### Part/origin change

Two new **Part change** and **Origin change** criteria are available for the link movement rules in the interoperation movements. The selectable values for these two criteria are **Yes** or **No**.



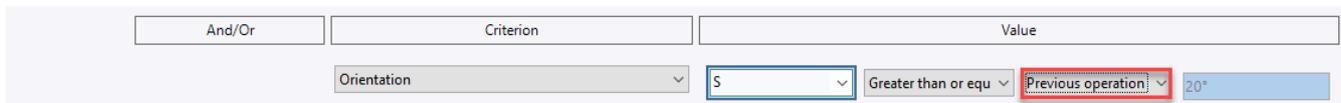
#### Comparison of orientation in relation to the previous operation

You can now compare the orientation of an axis with that of the previous operation.

- **With a value:** It is always possible to compare the orientation with a fixed value. To do this, you have to select the **Value** entry from the drop-down list.



- **With the value of the previous operation:** To be able to compare with the orientation value of the previous operation, you have to select the **Previous operation** entry from the drop-down list.



### Program start/end

Two new **Program start** and **Program end** criteria allow you to assign a particular link movement to start or end a program. The selectable values for these two criteria are **Yes** or **No**.

Criterion	Value
Program start	Yes
Criterion	Value
Program end	No

### Forcing the interoperation loading

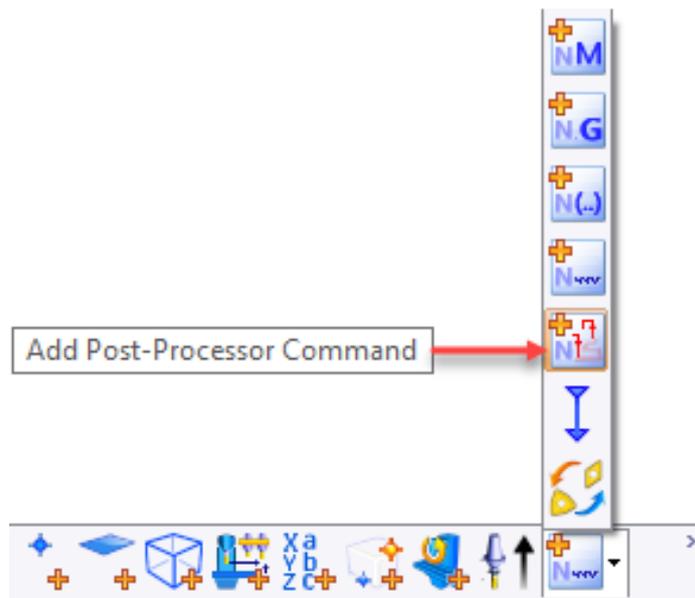
Thanks to the new **Force loading** option, you can now force the loading of the interoperation, even if there is no head and/or table rotation between the two operations. This option is available in the icon bar above the criteria.



**Restriction:** This option is only available for interoperation type rules.

### PP word for link movement

The PP word is now available in the link movements.

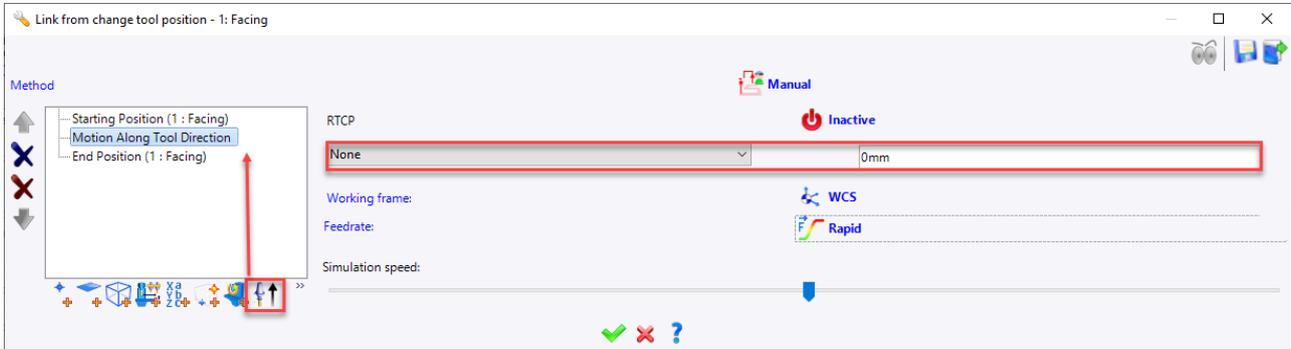


**Restriction:** This menu is only accessible in **Manual** mode.

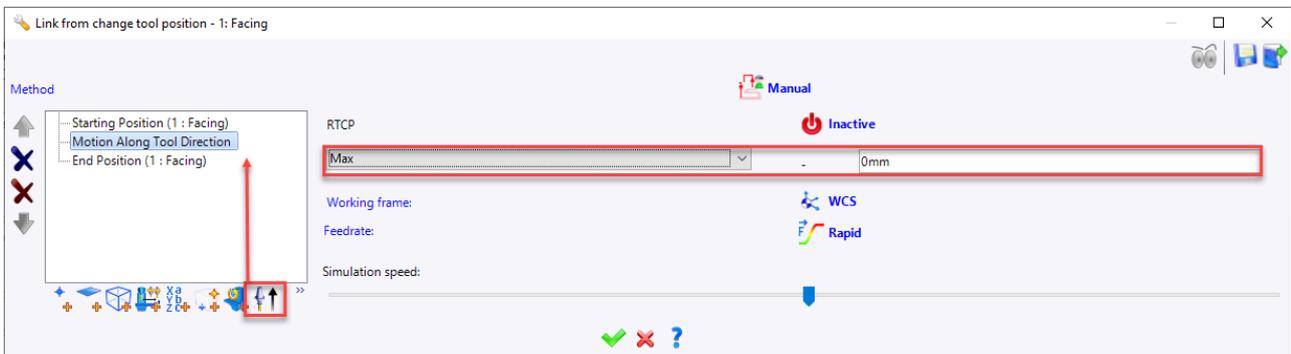
## Movement along the tool direction

A new movement named **Motion along tool direction** is available in the link movements, as well as in the virtual jog operation. It can be used in two different ways:

- **None mode:** In this case, there is no automatic movement, but only a retract movement in the tool direction by the indicated length.

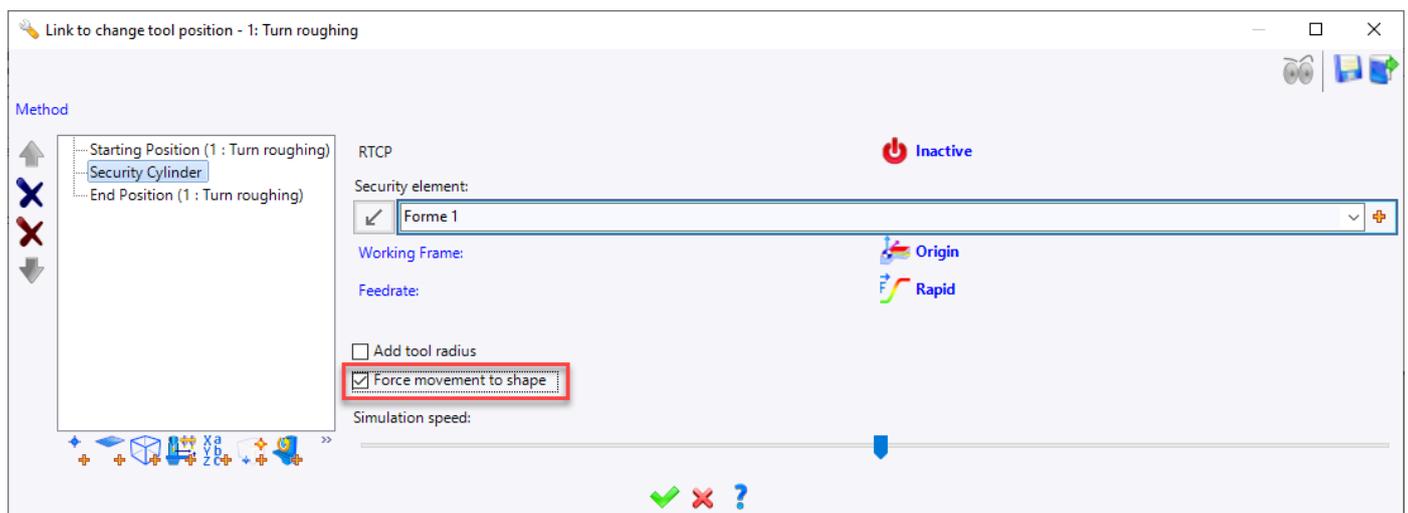


- **Max mode:** In this case, the largest possible movement is performed along the tool direction according to the machine strokes.



## Using security shapes in the approaches

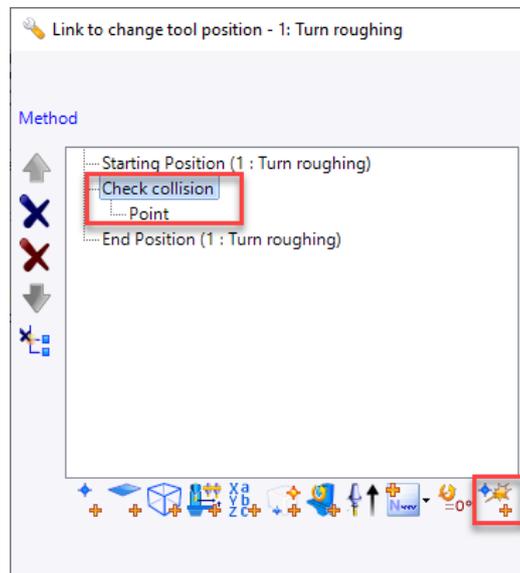
When adding a movement onto a security element, you can choose to go to the security shape by checking the **Force movement to shape** box and thus use this shape to perform an approach.



**Restriction:** If the tool is in the security shape, the behavior will be the same whether the box is checked or not. A movement is performed only if there is an intersection between the security shape and the tool axis.

## Movement on point(s) with collision management

In a **turning** link movement, you can now add a new **Check collision** element on point when the **Manual** mode is enabled.



**Restriction:** We have the same limitations as the collision check in automatic mode. If collisions already exist at the start or end point, no avoidance point can be found.

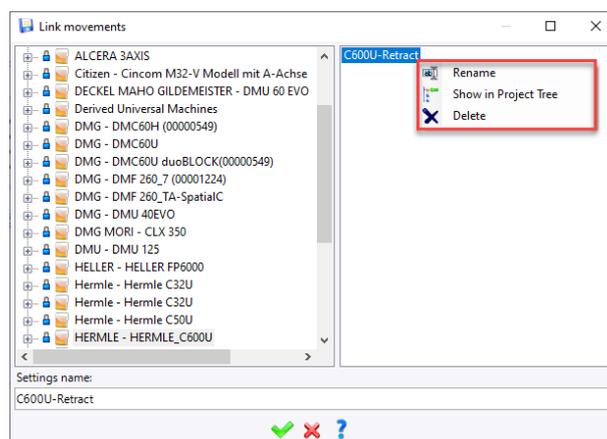
## Editing link movements

### Saving the link movement document

When saving a link movement document, a message tells you whether the document already exists and asks whether or not you want to replace it.

### Contextual menu

A right-click contextual menu is now available on the elements in the list on the right. In particular, it allows you to delete, rename or show the location of an element in the Project tree.



**Restriction:** You cannot rename or delete a link movement document that is already in use in another document (link movement rule document, machine document, methods, etc.).

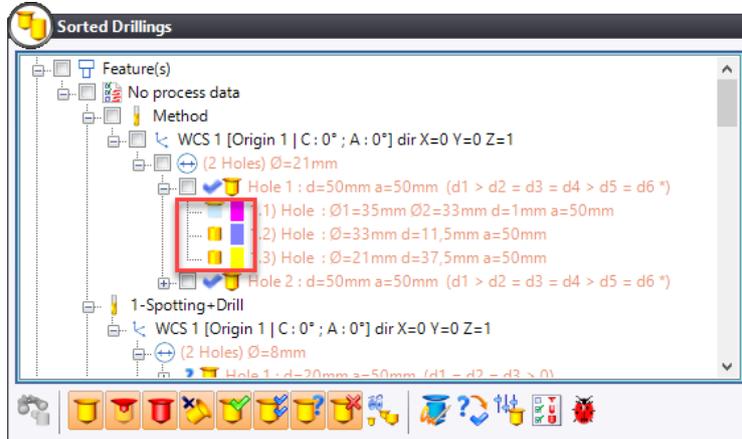
In this case, an error message indicates that the action is impossible and lists the documents in which the document is being used before canceling the current action.

You can then make the movement obsolete so that it is no longer available in a new document.

## Methods

### Colored hole creation in the drilling and cylinder analysis

You can create a general method with a **Hole creation** action respecting the colors of the hole stack directly from the drilling and cylinder analysis.



In the drilling and cylinder analysis, when selecting the **Create hole stack method** command, the colors of each hole of the selected stack are included in the corresponding sub-action.

### Associating methods with colored holes of a basic part

You can now associate a method with a hole stack in the drilling and cylinder analysis if the hole colors match. In the **Hole creation** action of a general method, a new **Visualization color** parameter is available for each sub-action.



**Restriction:** If a method/process association document is used, it takes precedence.

### Selecting a tool from the magazine when executing the method

In the general method, two new parameters are available in the **Select a tool** function:

- **Question:** Text to be displayed when the tool choice dialog box is displayed.
- **Choose tool only in document:** If the selected value is **Always**, the function will search in the machining document only and/or open the tool choice dialog box.



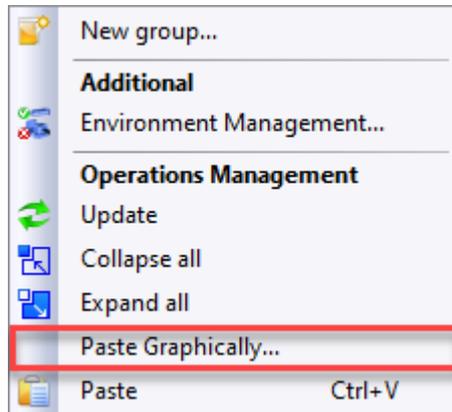
In case of manual tool selection, it will also be possible to filter the results via the Tool Manager.

**Restriction:** The **Pocket** parameter is ignored.

### ***Copying and pasting operations in the same machining document***

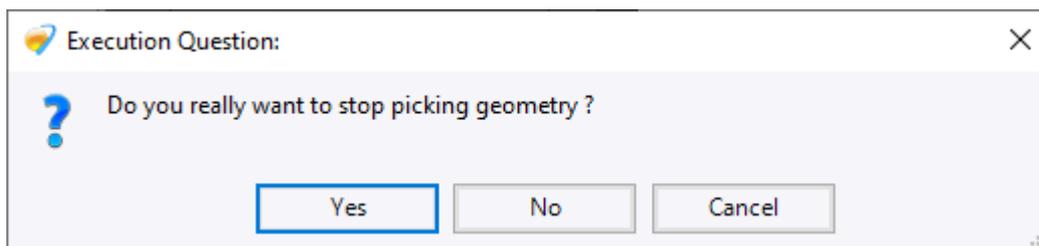
To copy operations in the same machining document, you should always use the **Copy** operation that copies everything, including the machined geometry.

The new **Paste graphically** command allows you to paste the operations by selecting one or more new geometries.



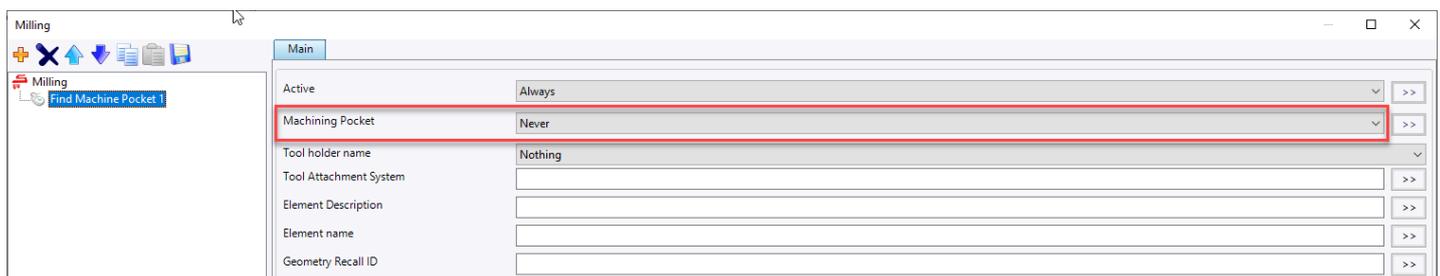
### ***Copying and pasting operations without geometry selection***

After performing a classic copy and paste operation of a machining range between two documents, you no longer have to select a geometry when a question relating to the method is asked. You can proceed to the next step even if no geometry is selected. The following question is then displayed.



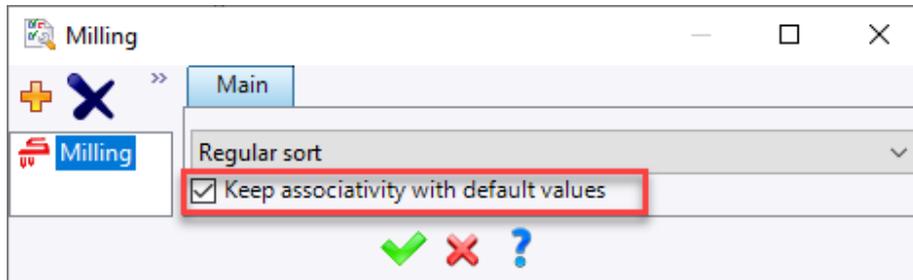
### ***Adjusting the tool's machining pocket***

In the machining pocket search action, the new **Machining pocket** parameter allows you to search for the tool only in machining pockets.



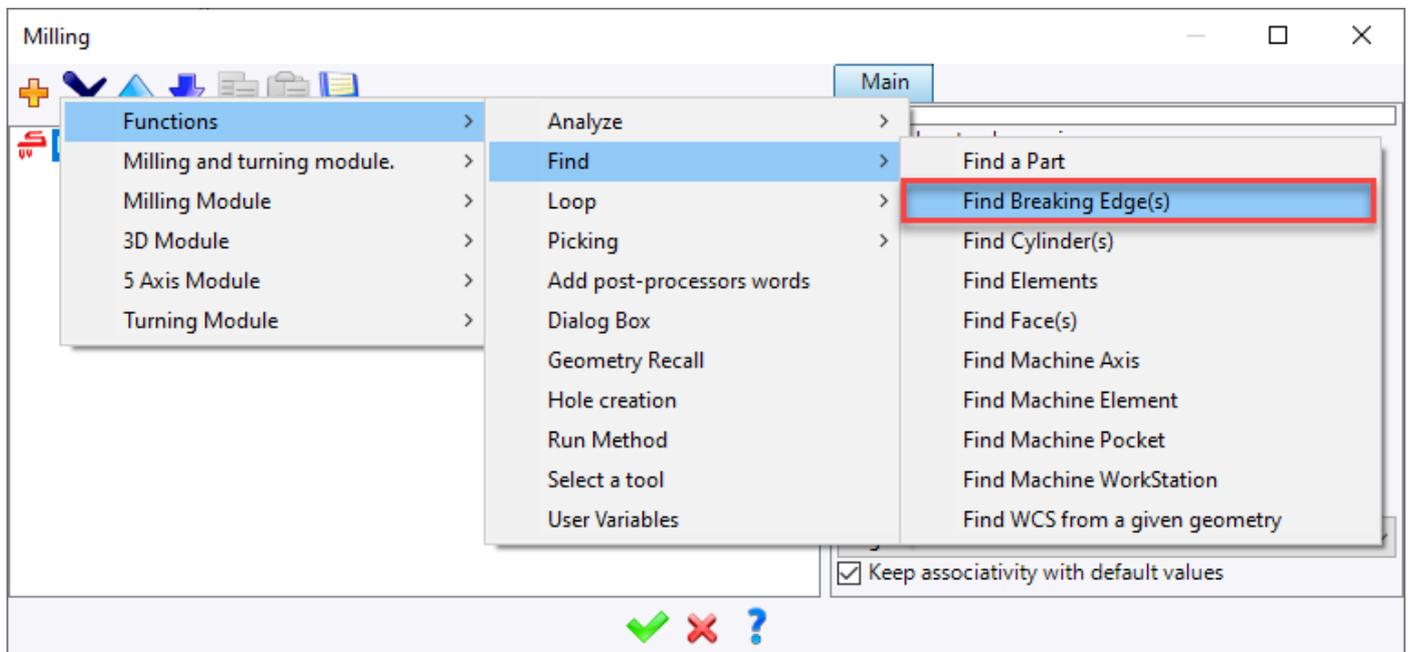
### Using the default value set

When defining a method, you can choose whether or not to use the default set of values for machining operations when creating the machining action or when executing the method. The method is then associative with the default values.



### Search for edges and chamfers

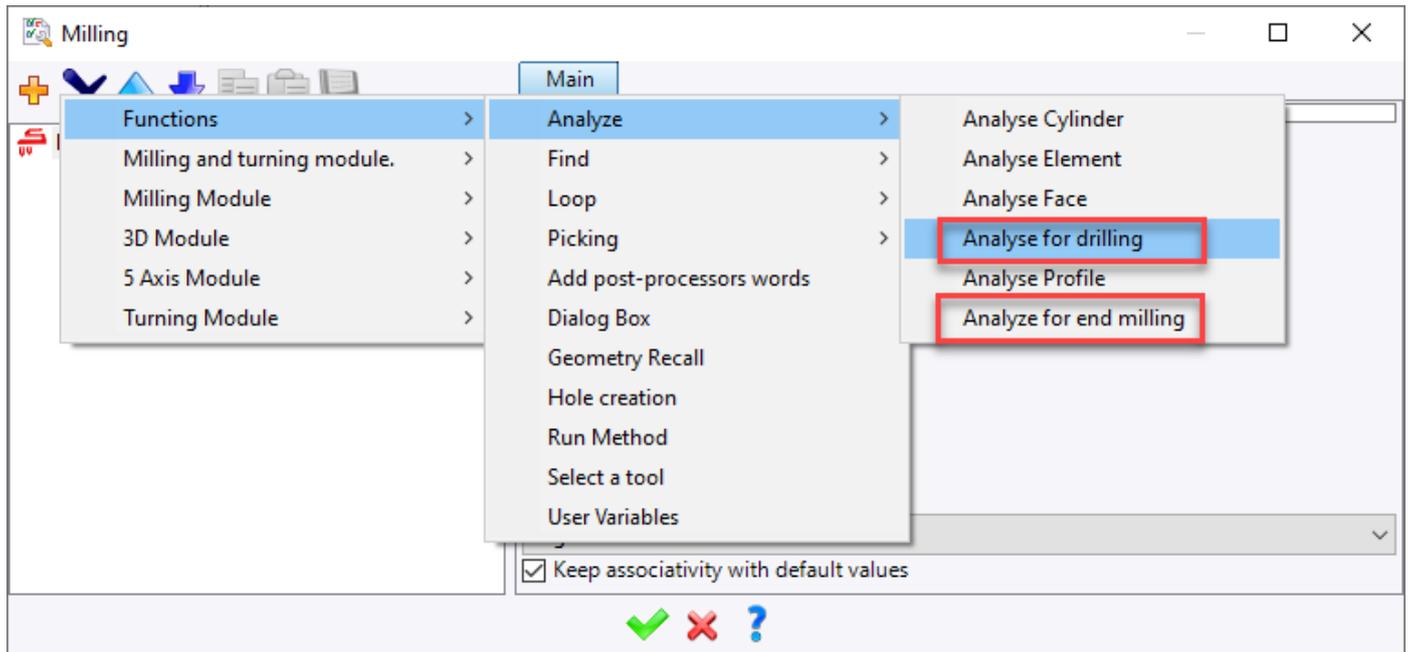
A new **Find breaking edge(s)** function is available in the **Find** category. It allows you to provide geometries for the breaking edge milling operation (equivalent to the **Generation of geometry** option in this operation).



**Restriction:** The search result can only be used by the breaking edge milling operation.

### ***New analysis functions in the methods***

In the general method, two new **Analyze for drilling** and **Analyze for end milling** functions are available.



These functions are capable of analyzing all types of geometries that can be machined by drilling and end milling operations.

### ***Analysis of an element length***

In the general method, the **Length** variable has been added to the **Analyze element** action. It returns the length of a 2D or 3D profile, segment or edge given by the **Find elements** or **Picking elements** actions.

**Restriction:** This is only available for profile, segment and edge elements. If the element is of another type, the length will be 0.

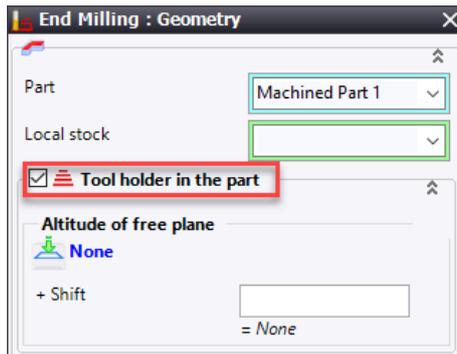
## Miscellaneous

### ***Machining inside a part: Managed Z bounded plane for the stock and finished part***

To machine inside a part, you can define a cutting plane, i.e. a plane resulting from the defined work coordinate system with an altitude. This plane allows you to cut the shapes (stock, finish, etc.) and to avoid finding collisions that forbid the machining or altitudes that are too high.

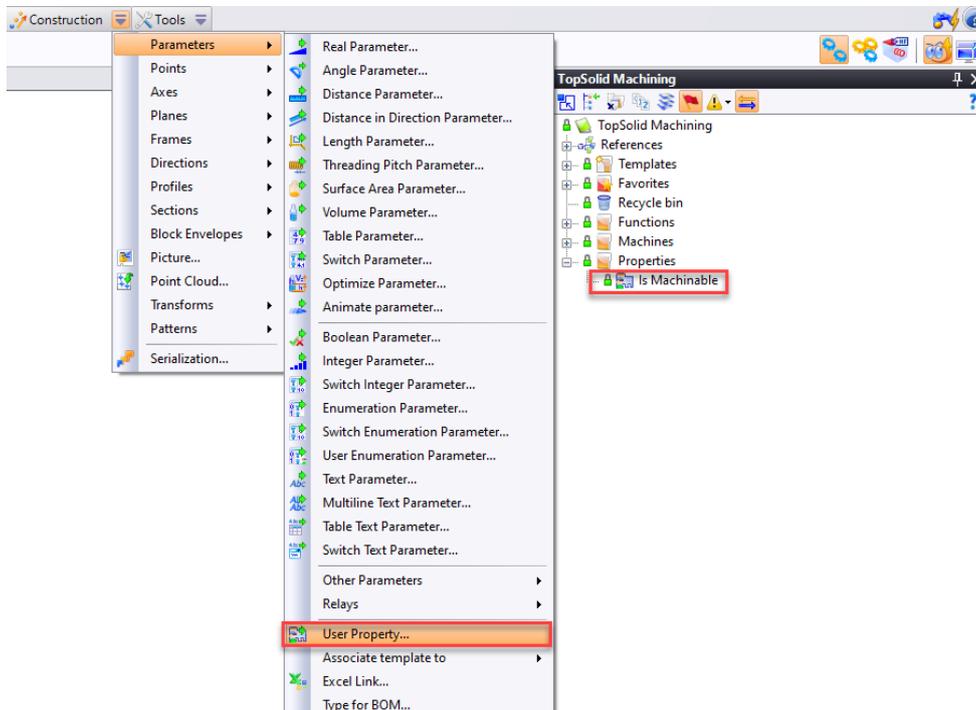
Accordingly, two new parameters have been added:

- **Tool holder in the part** (which allows machining inside the part)
- **Altitude of free plane**



### ***Inclusion property***

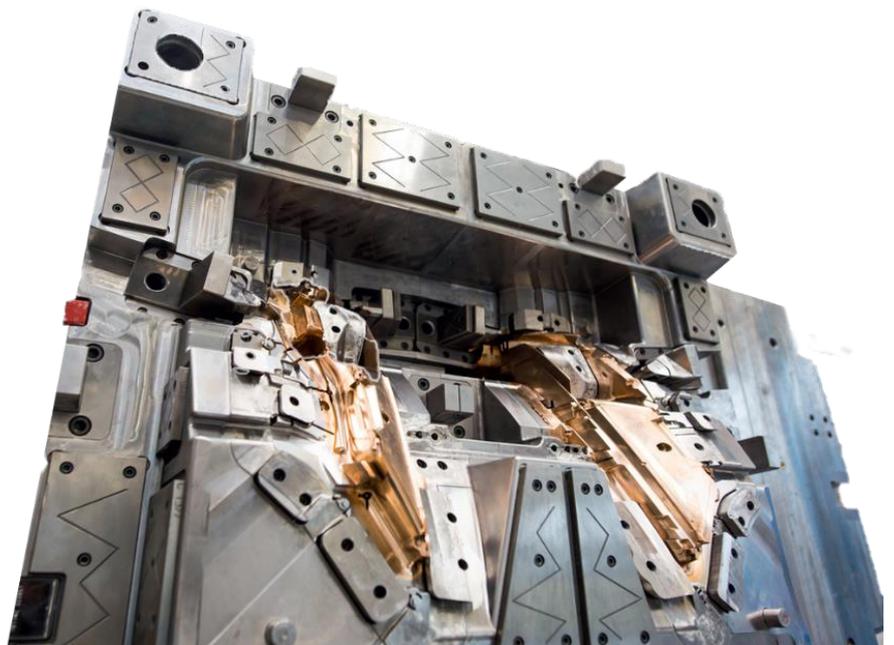
A new **Machinable** user property is available in the **TopSolid Machining** library. It avoids displaying the dialog box asking whether the included part is to be used for an NC part or a normal inclusion.



### ***Exiting the "Show machine preview in position" mode***

When creating or editing a machining operation, when calling up a command to create an entity (point, profile, sketch, etc.), the preview of the machine in position pauses and the machine is displayed in the rest position to facilitate the selection.

## What's New in TopSolid'Tooling 7.15



This section describes the enhancements made to the **Split**, **Mold**, **Electrode**, **Strip** and **Progress** applications in version **7.15** of **TopSolid 7**.

## TopSolid'Split

### Performance improvement

Version **7.15** offers significant improvements in terms of software performance, especially in the **TopSolid'Split** module as it relates to the processing of very large and geometrically complex parts:

- Launching the **Parting Edges** command: **90%** time saving.
- Creating the **shells**: **40%** time saving.
- Improving calculation time of the **core cavity blocks** when the dialog box opens: **30%** time saving.
- Improving calculation time of the **insert surfaces** created using the **Faces** mode: **75%** time saving.

### Pre-study mode

The core cavity blocks created in pre-study mode now include the shapes to split. You can still choose to split the stock using the opening plane in order to create the two core cavity blocks via the **options** in the **Split Blocks** document.

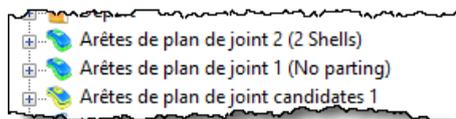
### Improved candidate edge detection

The way in which candidate edges can be detected has been improved, especially for the parts with grids.

### Parting edges

#### Number of shells

The **number of shells** obtained after each parting edge operation is now displayed in the Operations tree. It is also indicated that the operation creates **no parting** in the case where you have selected edges that does not allow the part to be split into at least two shells.

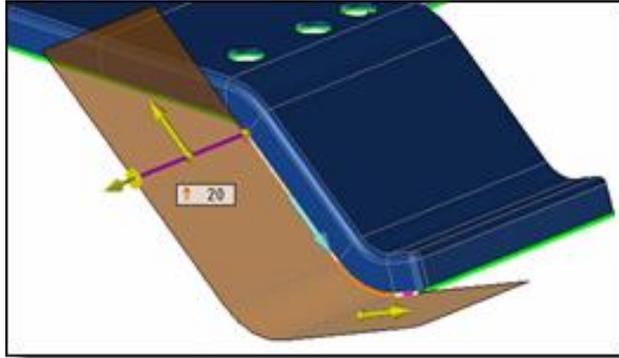


### Edge path between vertices

The new **Edges Path between Vertices** command helps you to create parting edges and is available via the  icon in the **Parting Edges** command. It identifies the **shortest** parting edge path between two vertices on the part. You can calculate this path by forcing it to pass over the selected edges.

## ***Extension parting surfaces***

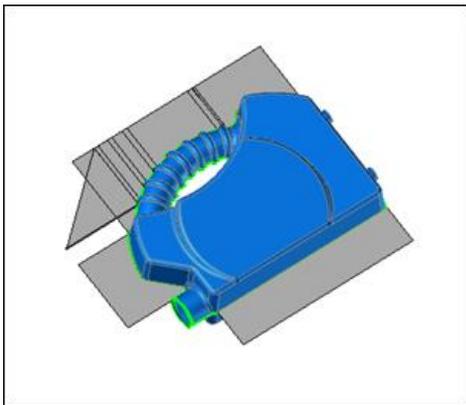
A new **Tangent** mode is available to create **lateral extensions** on the **Extension** parting surfaces.



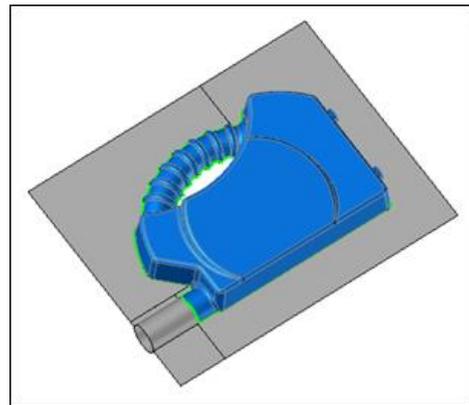
## ***Automatic parting surfaces***

The creation of **automatic parting surfaces** has been greatly improved:

- The calculation time has been optimized (from 50% to 80% time saving according to the type of part).
- The results are much better.



*Automatic surfaces produced in version 7.14.*



*Automatic surfaces produced in version 7.15.*

# TopSolid'Mold

## Pin preview

Positioning many pins simultaneously in sketch mode in a large mold file can be quite time-consuming. In order to avoid calculating the preview of each pin, you can now enable or disable the preview in this positioning mode.



## Cooling

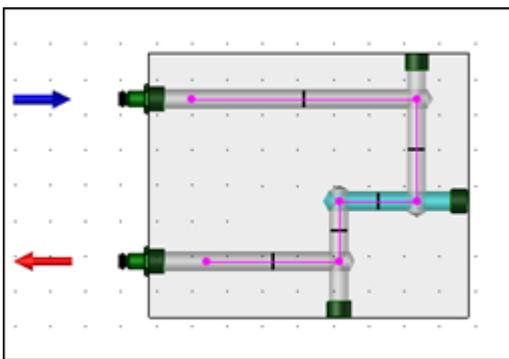
### Creation on existing 3D sketch

You can now create cooling circuits on 3D sketches, which offers many new options:

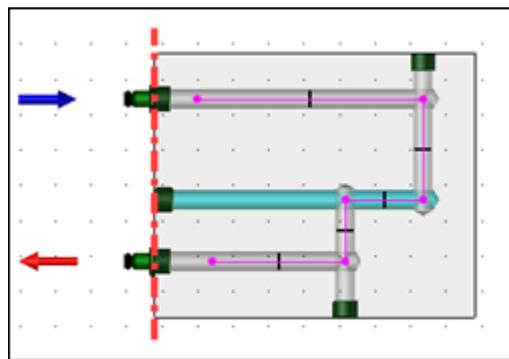
- standard cooling circuits based on sketch components;
- symmetrical repetition of a circuit by repeating the sketch;
- repetitions of segments within the sketch;
- associativity between the sketch and the cooling circuit.

### Preferred entry faces

You can now consider certain faces of the part as the entry faces of the drillings created by the cooling sketch.



*It might have been interesting to create the blue drilling in the other direction.*



*The drilling is automatically oriented by defining an entry face to the circuit.*

### Baffle line point

The new **Baffle Line Point** command, which is only available in a cooling sketch, simplifies the creation of baffle lines. A baffle line is automatically performed for each created point, as you defined it in the cooling style.

### Offset

The new **Offset** option has been added to the **Modify components** command in a cooling sketch. It allows you to modify the offset of the selected components.

### Angle pin

You can now create a parameter corresponding to the real stroke of the angle pin when including each angle pin. The parameter is named **Stroke - Ange pin inclusion operation name**.

## Drilling collisions

The **Drillings Collisions** command has been completely redesigned to be able to test for collisions between all the drillings available in all the mold's parts, whether the drillings are performed by an operation in the mold or whether they were created locally in the parts.

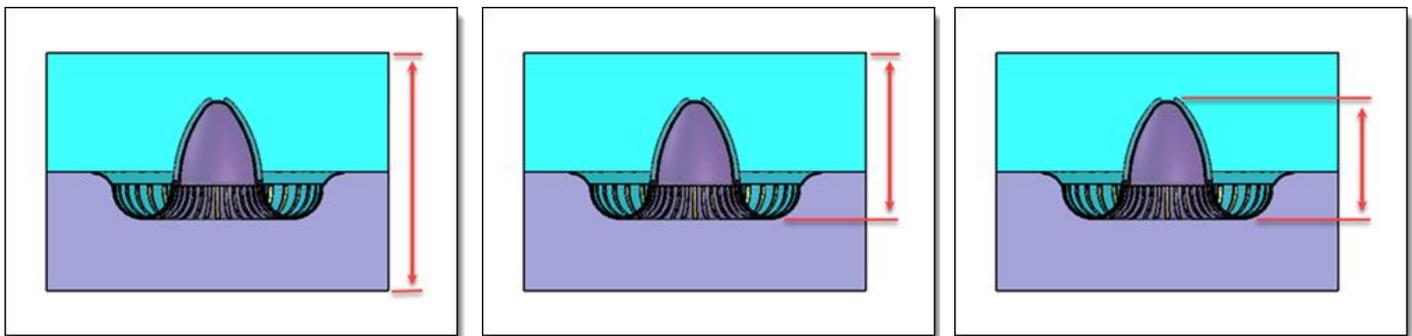
This command automates the launch of the **Drillings Collisions** command available in any part document:

- You do not need to open every part.
- This allows the different collisions identified in each part of the mold to be grouped in the **Drillings Collisions** folder of the Entities tree.
- The collisions between the different cooling drillings are automatically excluded.

## Mold simulation

### Ejection stroke parameter

The new **Ejection Stroke Parameter** command makes it easy to create an ejection stroke parameter based on the parts and blocks included in the mold document. Several calculation modes are available.



*Inferior and superior blocks extremities mode.*

*Part and superior block extremities mode.*

*Part extremities mode.*

This stroke can then be selected to calculate the mold **simulation**.

### Open tooling configuration

The new **Open tooling configuration** option allows you to create and maintain a **configuration** of the tooling in **open** position. This option is useful for controlling the mold design.

In particular, you will be able to easily test the collisions in open position thanks to the **Configuration Collisions** command, as you will be able to copy this configuration in order to modify the joint driving values to test the collisions in different positions of the tooling.

### Decorrelation

You can now decorrelate a mold. The **Decorrelate** command allows you to basify the links between the mold and one or more of its synchronized documents. All entities referencing an entity in another document are replaced by a basic entity.

At the end of the mold design and before editing it and put it into production, this operation allows you to manage independent life cycles on each part and also allows you to modify a mold very locally without impacting all the other parts of this mold.

## TopSolid'Electrode

### Drilling detection

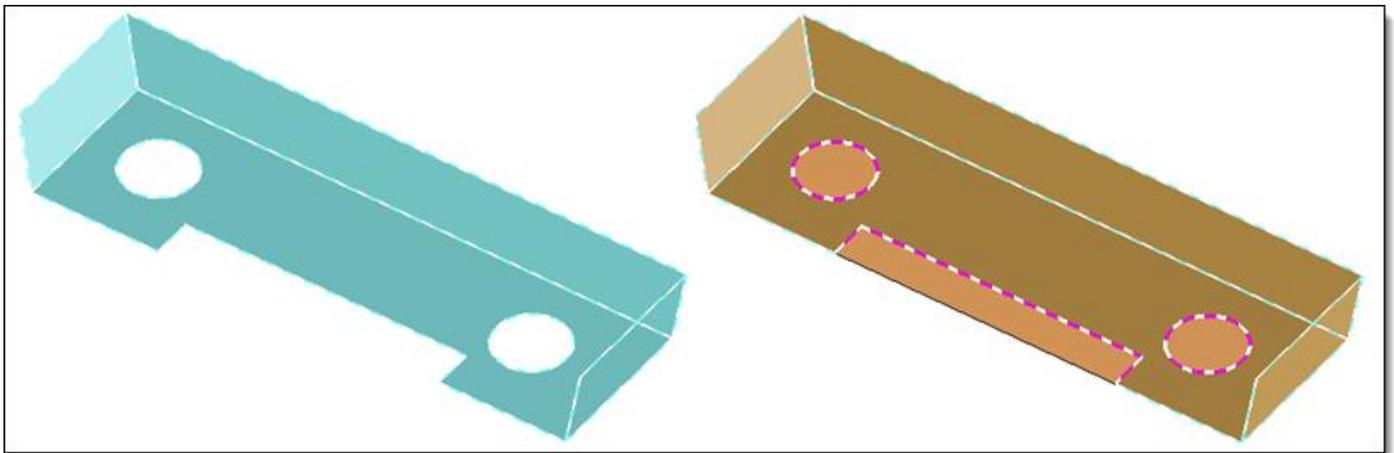
The **Drillings Detection** command has been significantly improved in terms of performance, with time savings of up to 75% compared to the previous version of **TopSolid'Electrode**.

### Removing cooling circuits and drillings

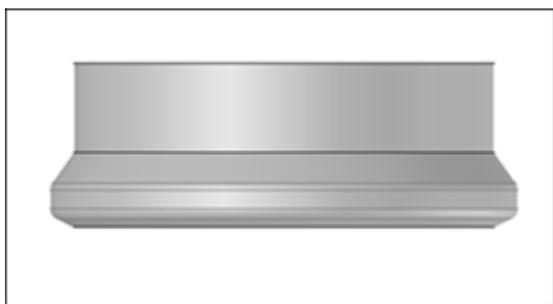
The **Removing Cooling Circuits and Drillings** command has been significantly improved in terms of performance in the case of hole removal. Time savings of up to 50% compared to the previous version of **TopSolid'Electrode** can be achieved.

### Shells

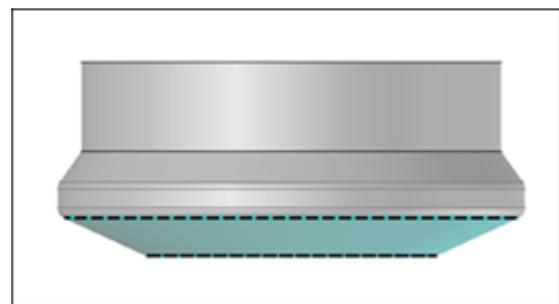
You can now select several holes to be filled simultaneously. The unclosed openings can also be automatically filled in the simple cases.



When the faces connected to the edges of the holes to be filled are not in the plane of the hole, you can shift the created face to fill the hole. The connected faces will be extended.



*Filled hole.*



*Filled hole with shift.*

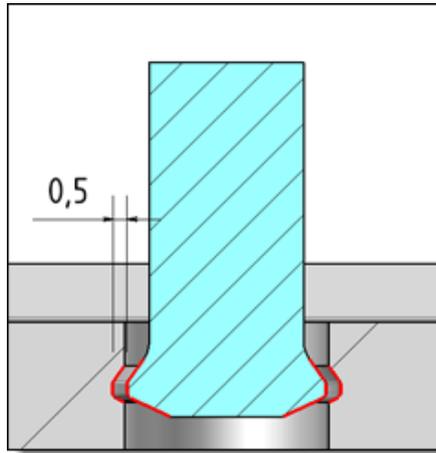
### Eroding shape

In the **Eroding Shape** command, the new **Total height** information indicates the height of the eroding shape that will be generated when confirming the command. The height is calculated according to the given extension direction.

In addition, the new **Blend edges** option allows you to automatically add fillets onto the blend edges between the first and second extension.

### Orbital

A new **Translated** creation mode is available and can only be applied to orbital electrodes with a **revolved** shape. In this mode, the undersizing is applied by translating the **silhouette** of the eroding shape.



### Electrode

#### Dimensioning origin of the EDM frame

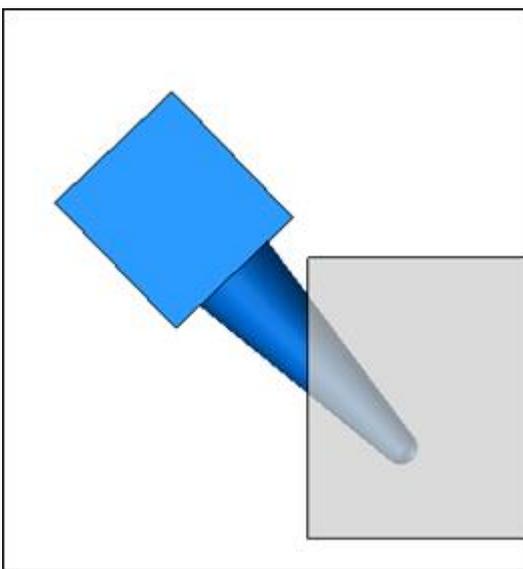
When creating an electrode or including an electrode in the **Electrodes** document, you can now select the dimensioning origin frame.

Therefore, regardless of the electrode's diving direction, you can determine its position according to a frame you have selected.

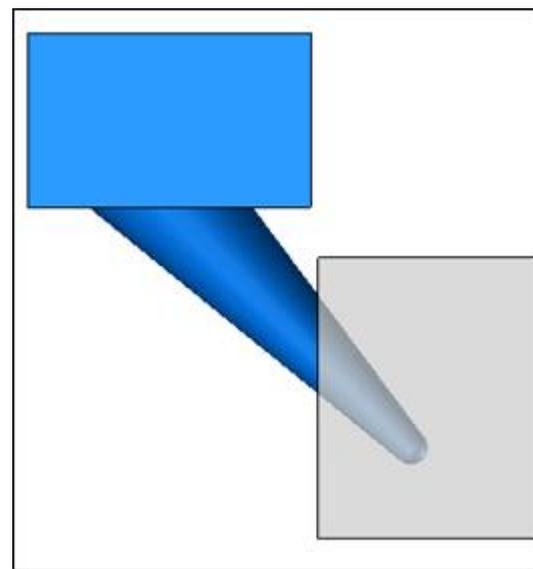
#### Base orientation

The base orientation can now be defined in two ways:

- **Eroding shape direction:** The Z direction of the base corresponds to the direction of the last extension of the eroding shapes.
- **Positioning origin direction:** The Z direction of the base corresponds to the Z direction of the positioning origin.



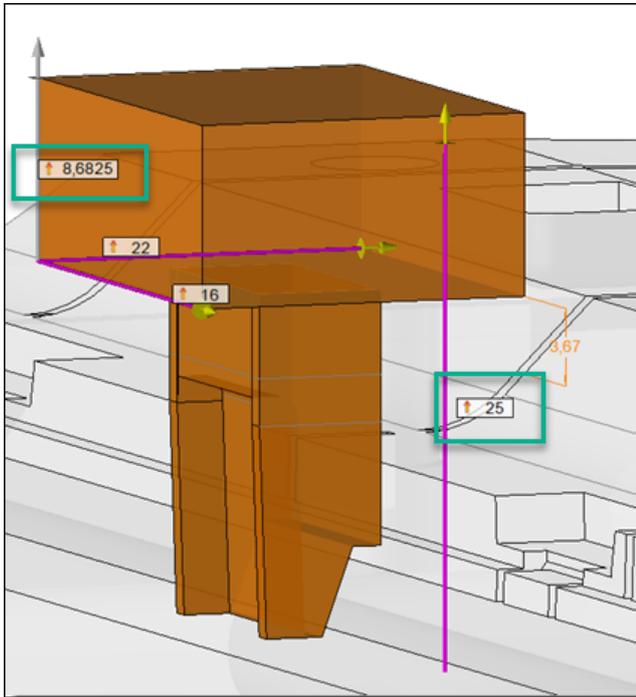
*Eroding shape direction.*



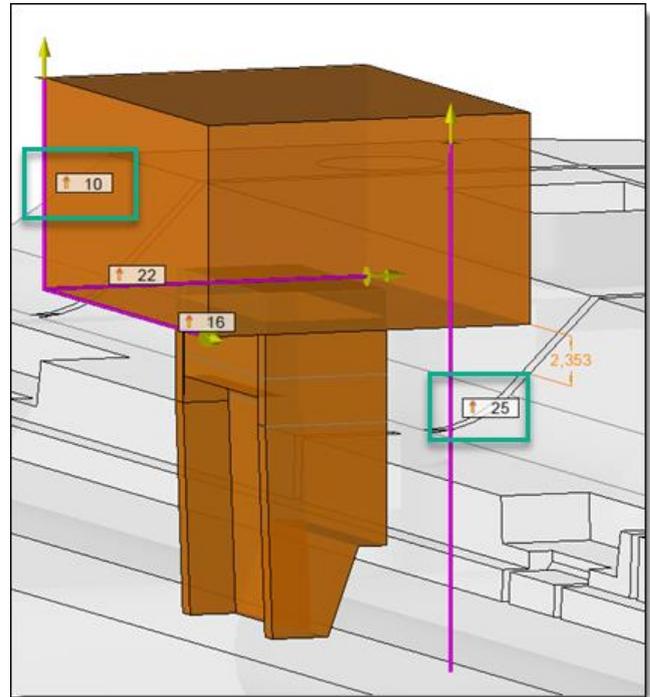
*Positioning origin direction.*

## Total height

For the electrodes created from version 7.15 onwards, you can create them by indicating their **total height**. Depending on the given total height, two adjustment modes are available.



*The height of the head is adjusted to create an electrode of the desired total height.*



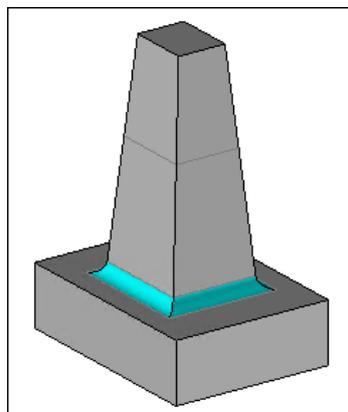
*The total height and the head height are given. The height of the eroding shape is automatically adjusted.*

## Height of the base in relation to the shape to be eroded

A new dimension is visible in the electrode creation preview. This informal dimension is displayed in the color of the preview. It allows you to easily view the minimum height between the electrode base and the shape to be eroded.

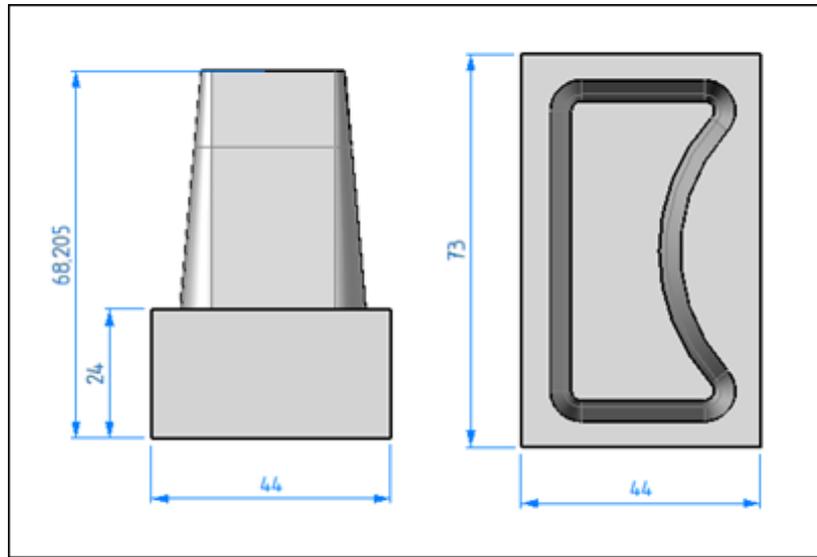
## Blend edges

You can now automatically add **fillets** or **chamfers** to the blend edges between the base and the eroding shape.



## Automatic detailing

When creating the electrode, the new **Detailing** advanced option allows you to automatically create the detailing dimensions of the electrodes in the drafting documents. You also have to check this box in the views of the drafting templates to benefit from this automation.



## Creating frames on the fly

Thanks to the  icon, you can now create the following elements on the fly:

- An **additional origin frame** as the dimensioning origin in the **Electrode** and **Include Electrode** commands.
- A **frame** as the positioning origin of the electrode base in the **Electrode** command.

## Removing eroded faces

Detecting eroded faces by the theoretical electrodes did not take into account the possible modifications performed on the area to be eroded when creating the shell of the original electrode.

The **Removing Eroded Faces** command has been enhanced and now allows you to better identify the eroded faces to be removed and to automatically perform the removal operation as it was defined by the original electrode shell.

## Electrode occurrence properties

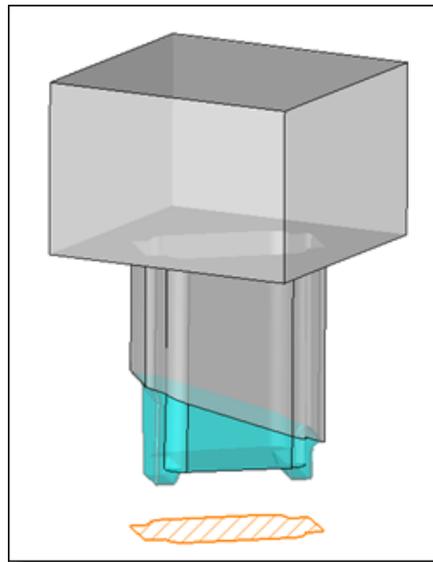
The occurrences of an electrode are its theoretical positions. The new **Electrode Occurrence Properties** allows you to assign different properties to these occurrences, e.g. a number of electrodes property for each occurrence.

These properties can then be displayed in a drafting document and/or an electrode table.

### Eroding area properties

The new **Eroding Areas Properties** command is used to calculate the **projected area of the silhouette** and the **sum of the face areas** of the eroding shapes defined for each electrode. The calculation is performed in the **Analysis** stage of the document. The properties will only be updated after switching the document to this stage or after saving the document.

Created in the electrode's part document in the **Analysis** stage, the properties can be used in the **electrode tables** and **electrode notes**.



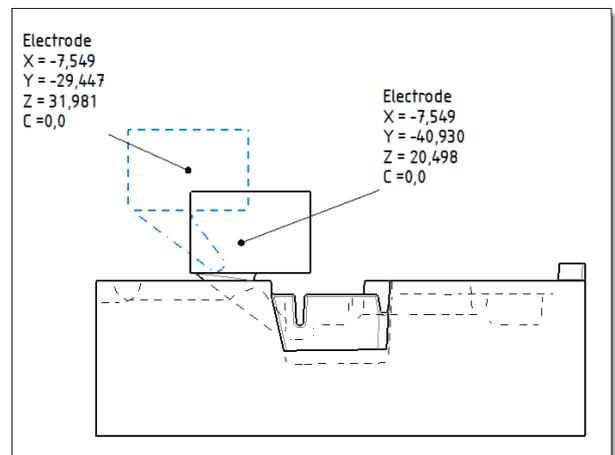
### Electrode drafting

#### Projection set

You can now add the **initial positions** of the electrodes to an **Electrodes** projection set. The initial positions can therefore be projected in the drafting documents.

#### Initial positions

The new **Initial positions** option allows you to add in the views the occurrences of the electrodes in **initial position**. If the initial positions are projected, they are annotated by the **Automatic Electrode Note** command.



### Electrode table

In the **Electrodes Table** command, the new **Sort by electrode name** advanced option allows you to sort the electrodes alphabetically in the table. If the box is unchecked, the electrodes are sorted in the order in which they were created.

The following two sorting modes are available: **Ascending sort** and **Descending sort**.

## TopSolid'Strip (Strip Design)

### *Unstamping Materials 7.14 library*

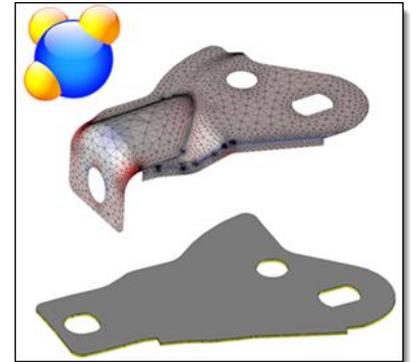
A new material library named **Unstamping Materials 7.14** is available on **TopSolid'Store**.

[www.store.topsolid.com](http://www.store.topsolid.com)

This library contains examples of materials for FTI simulation.

In an **Unfolding** or **Strip Design** document, if the unstamping operation is performed on a part with a partially completed material, **TopSolid** completes the mechanical characteristics of the material with values corresponding to the steel values.

The decision was made to share this library via **TopSolid'Store** and not via the software installation since we cannot guarantee the accuracy of the data and simulation results.



### *Station shape management*

Two new **Basify station shapes** and **Basify cutting operations sketches** options are available in the **Advanced** tab of the **Station Shapes Management** command.

In order to lighten the strip file and simplify its design, you can now **basify** all the station shapes and punching sketches.

The operations performed on the shape to strip are not deleted. To further simplify and lighten the file, they can be deleted in the station preparation stage.

Following the basifying operations, any changes to the strip design must be made directly on the station shapes that are grouped together in the **Stations Preparation Shapes** set of the Entities tree.

### *Derived strip*

In a strip design document, the **Tools > Derivations > Derived Part** command now generates a **Strip Design** document. Using this command can be interesting if the calculation of the merged strip and the strip properties is quite time-consuming.

In the "work strip", you can delete the two **Merged Strip** and **Strip Properties Management** operations, and then create the derived part from the strip. The **Merged Strip** and **Strip Properties Management** operations must then be recreated in this document. It will then be possible to update the derived document in a desynchronized way when it is opened.

The original document should be used to create the progressive die and the derived strip should be used for the strip drafting.

## TopSolid'Progress (Progressive Die)

### Pilots

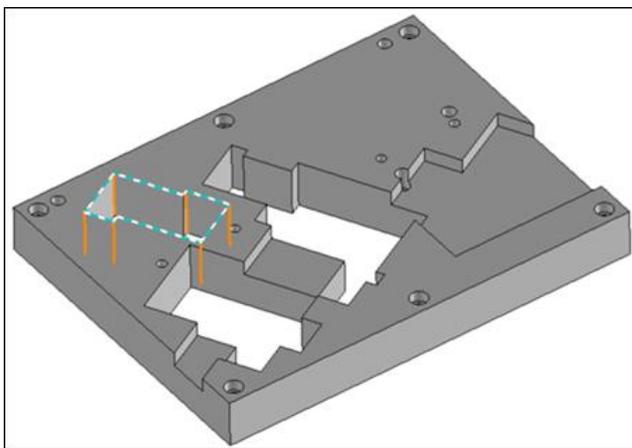
The pilot symbols included in the **Strip Design** document are transferred when including the strip in the progressive die document.

### Angle clearance

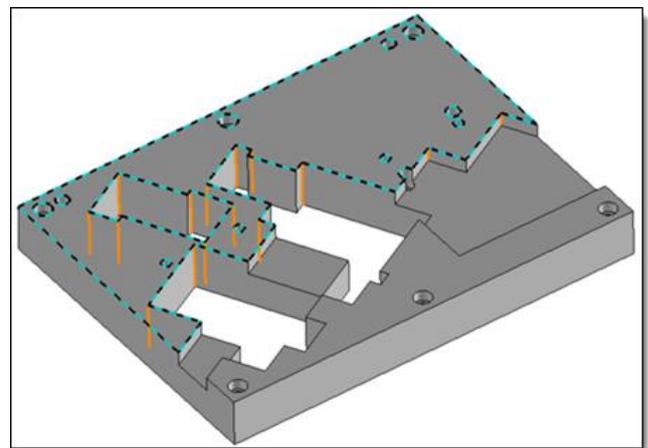
The **Angles Clearance** command is now available in the **Shape** menu of the part documents and no longer in the **Progressive Die** menu.

Two new selection modes available via the  icon make it easier to select edges:

- **Pocket Edges from Loop**
- **Pocket Edges from Face**



*Pocket edges from loop.*



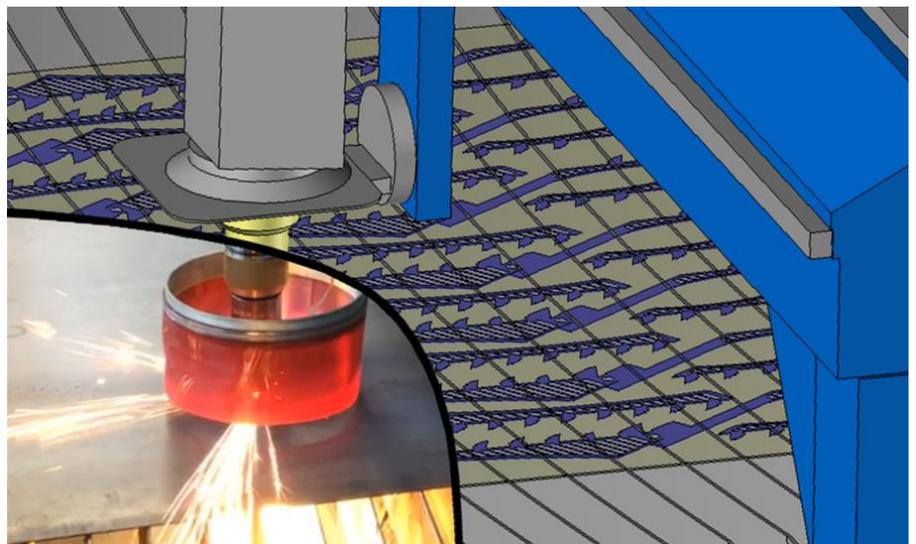
*Pocket edges from face.*

### Progressive die simulation

In the **Progressive Die Simulation** command, the new **Create an open tooling configuration** box allows you to create and maintain a **configuration** of the tooling in **open** position. This option is useful for controlling the progressive die design.

In particular, you will be able to easily test the collisions in open position thanks to the **Configuration Collisions** command, as you will be able to copy this configuration in order to modify the joint driving values to test the collisions in different positions of the tooling.

## What's New in TopSolid'Cut 7.15



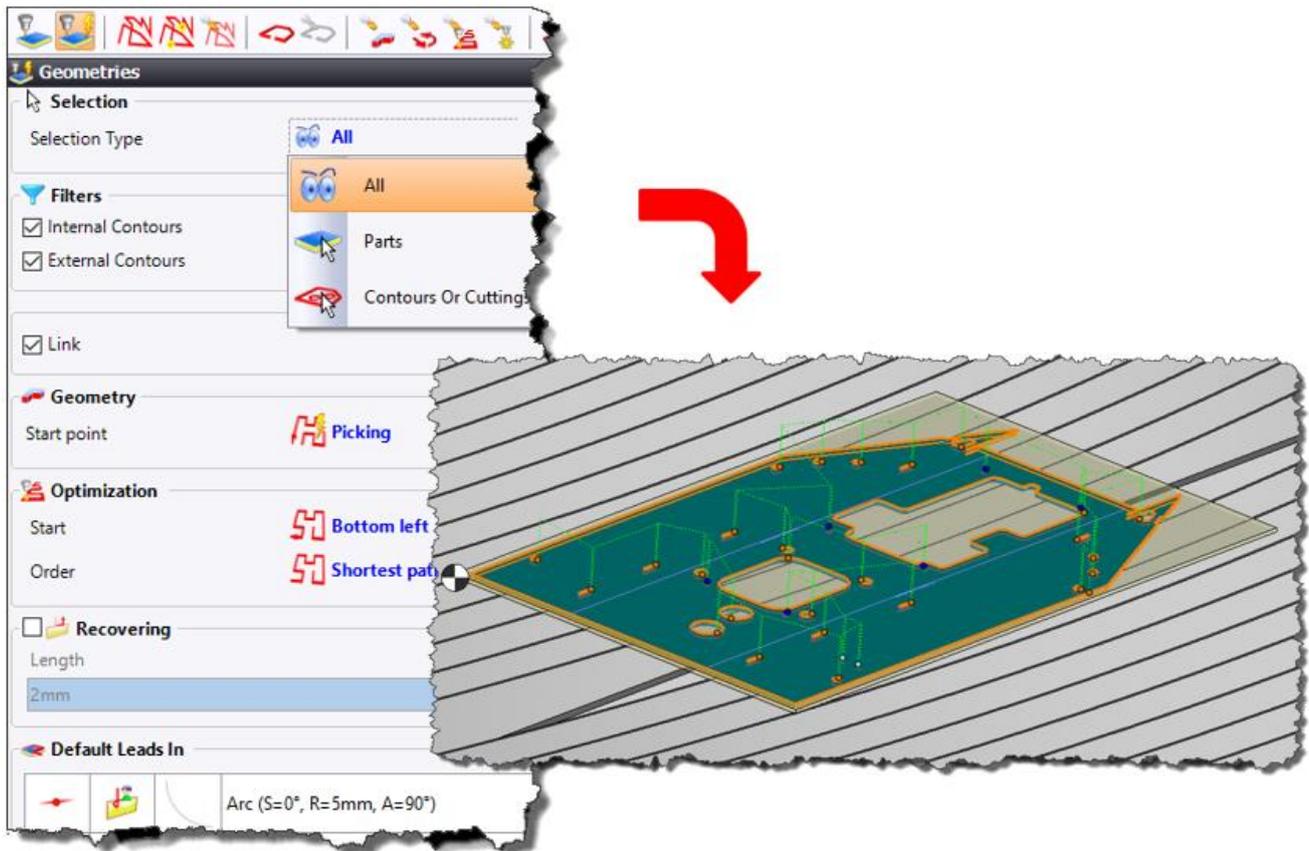
This section describes the enhancements made to the **Sheet Metal CAM** application in version **7.15** of **TopSolid 7**.

## 2D cutting

### Automatic cutting

The new **Automatic Cutting** command allows you to automatically assign cutting operations to several elements or parts simultaneously based on the parameters adjusted in the management operations (leads, micro-tabs, and cuttings). You can also modify these parameters directly in the dialog box of the operation.

You can select different types of elements: parts, contours or cuttings, etc.



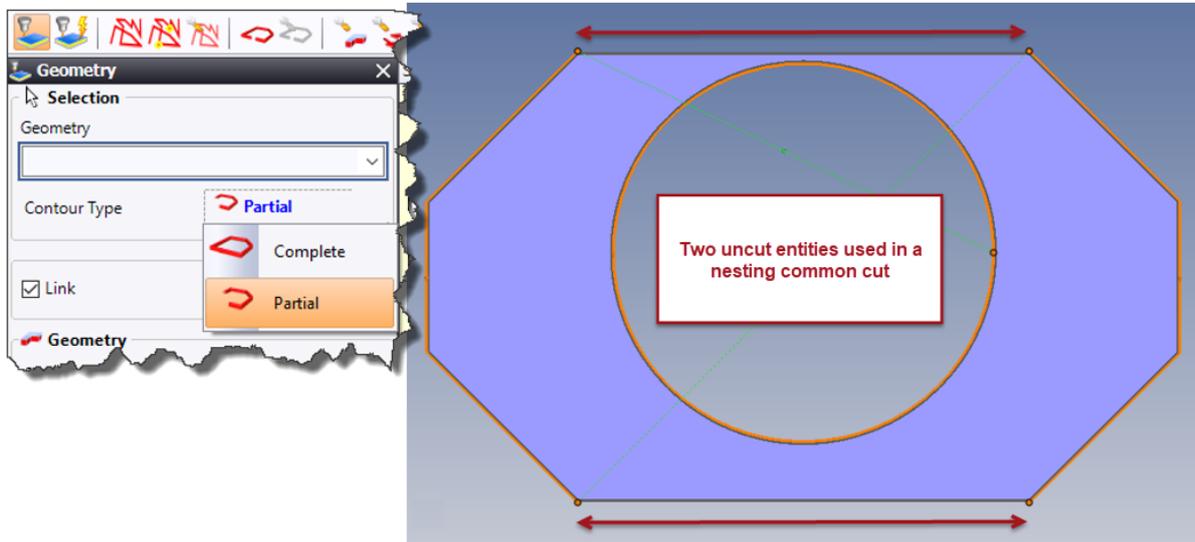
*Automatic cutting.*

For the operation to run smoothly, the management operations must obviously be correctly pre-configured.

You can edit the automatic cutting operation after confirming it without deleting it completely by using the various editing commands available in the **Cutting** menu.

### Partial profile cutting

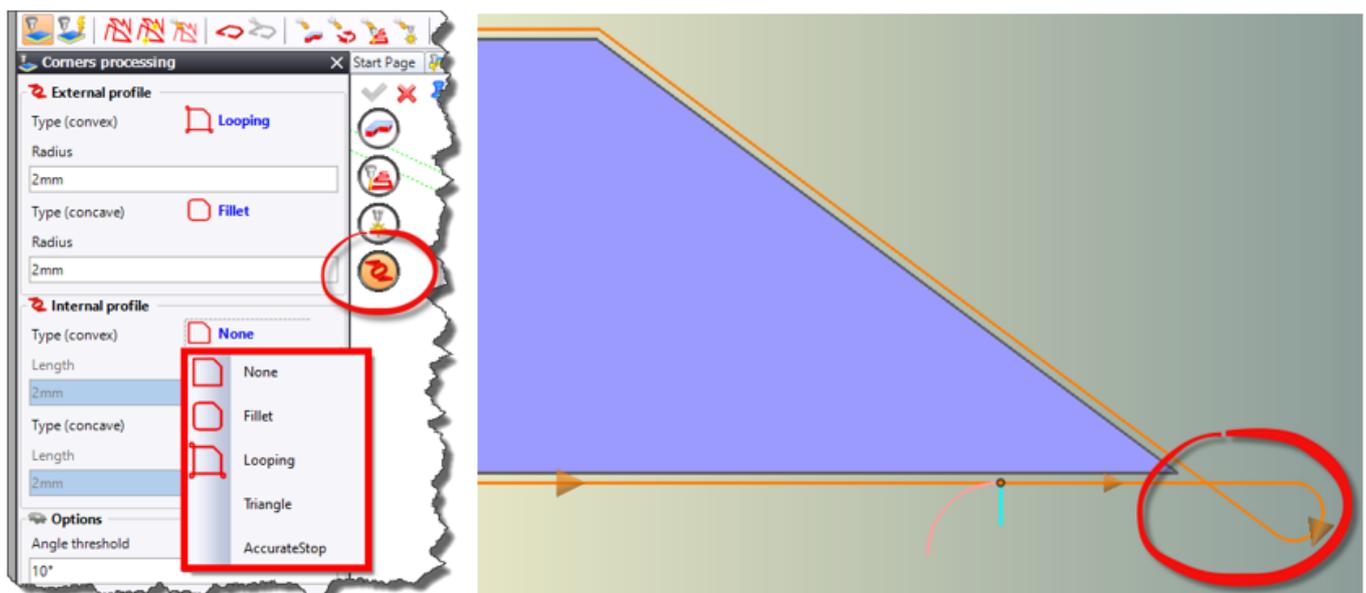
The partial cutting feature is now available in the 2D cutting operations. This allows you to freely cut out a section of the contour. Indeed, either the starting points are anchored on specific sections of each entity (corners, middle, etc.), or you can position the points dynamically, as needed, by moving the mouse cursor. The advantage is to be able to optimize the tool path, especially in the case of a common cutting operation occurring later in the cutting cycle.



Partial profile cutting.

### Corner processing

The new **Corners Processing** option allows you to place geometric entities at specific locations on the tool path. These entities are geometrically parameterized to increase the curvatures and decrease the breaks in the tool path. More specifically, the interest of this new feature is to optimize the dynamic behavior of the machine by reducing the accelerations and decelerations which can generate important mechanical stresses on the axes. This improves the quality of the part cutting.

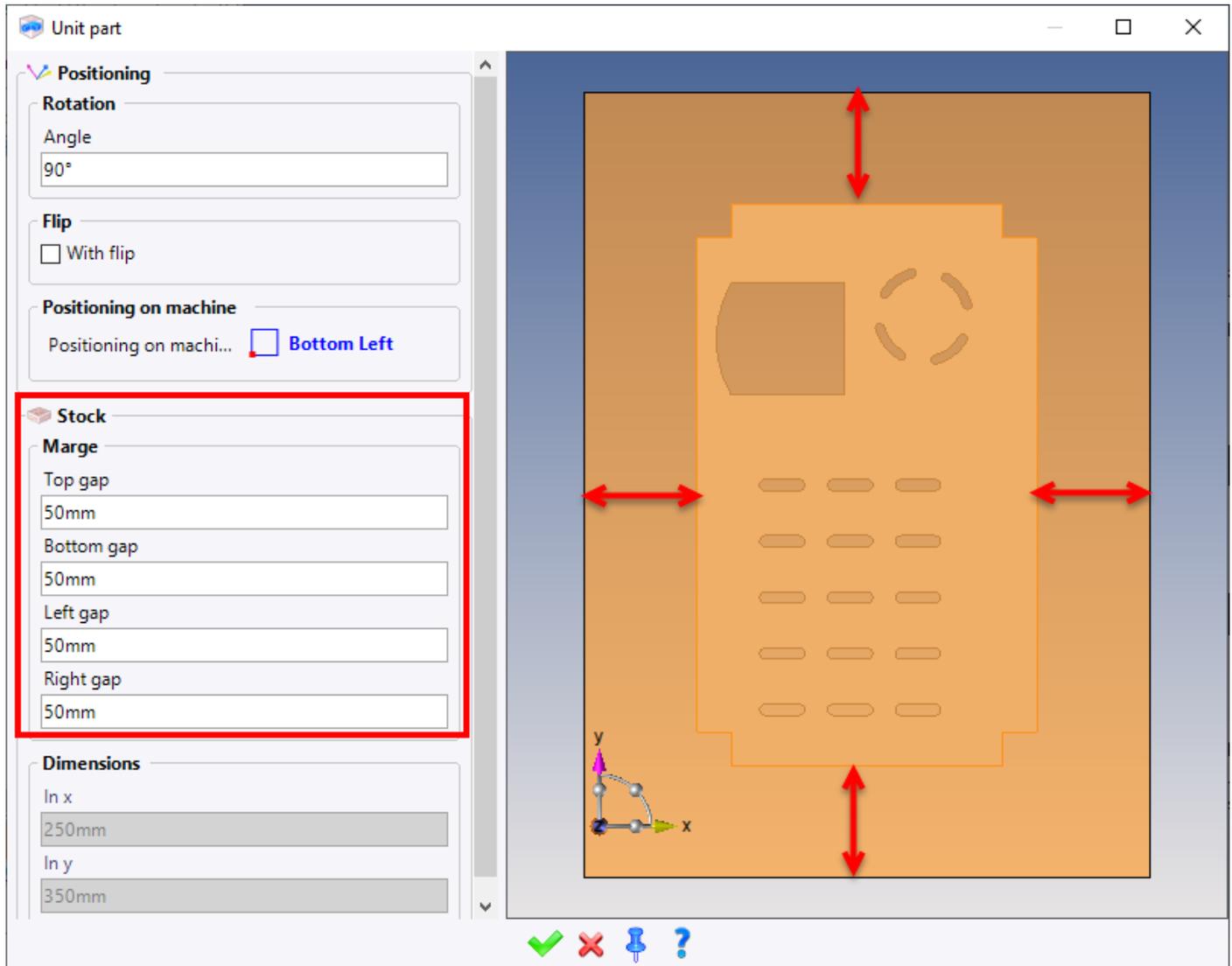


Corner processing in looping mode.

# Nesting

## Unit part

The new **Unit Part** command, which is available in the **Nesting** menu, has been developed to be able to generate safe machining in an extremely fast and efficient way. This command can be very useful in the case of a test part, before starting to cut a complete series of parts.



Dialog box of the **Unit Part** command.

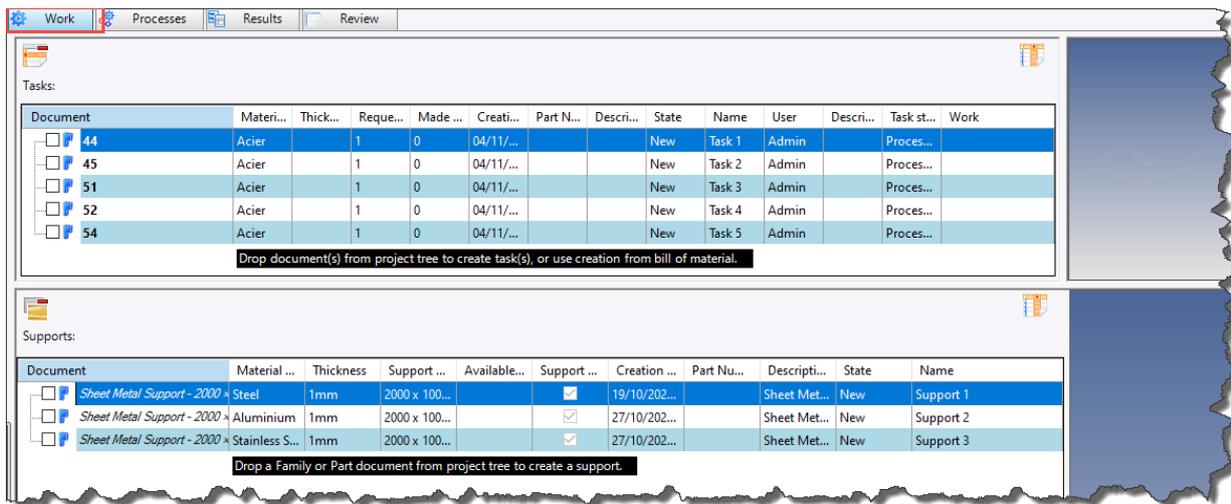
# Process automation

## Work document

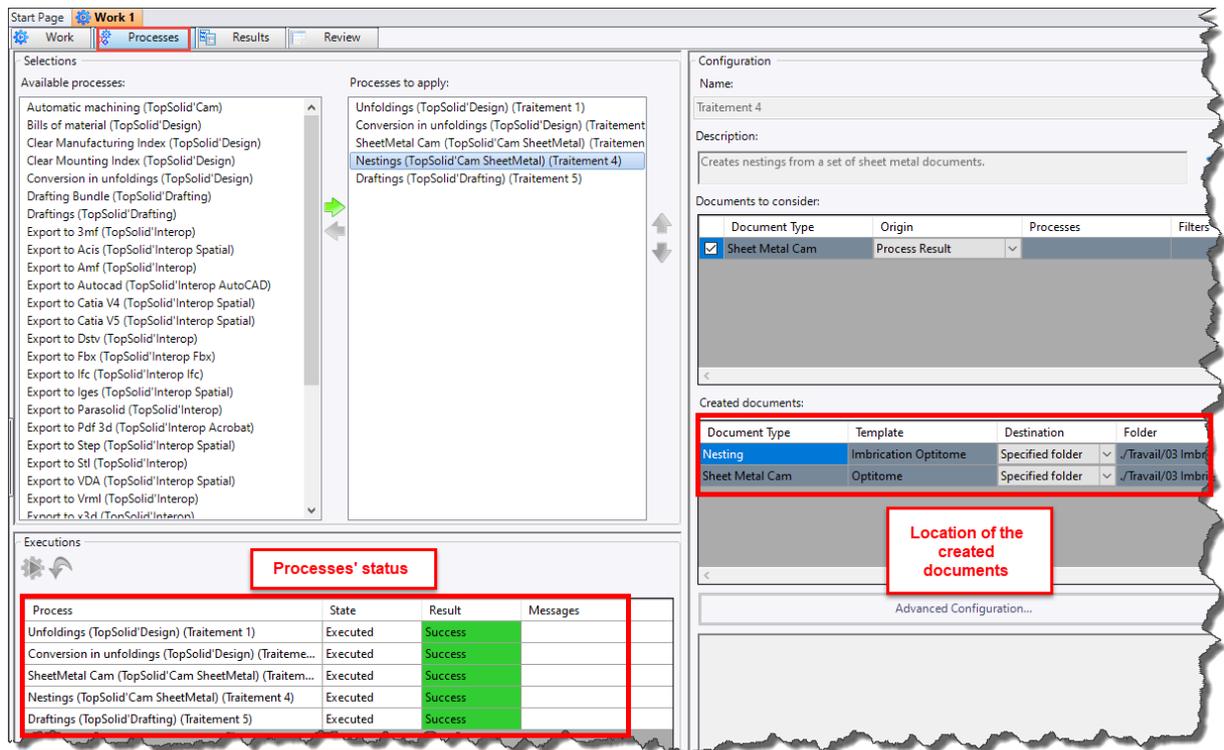
The **Work** document allows you to generate several processes automatically. It is based on template documents that are parameterized according to the machines, materials and production types. There are many processes available: unfolding, machining, nesting, drafting, etc.

Once all the templates have been adjusted, you only have to insert the parts to be processed and the available supports as well as select the processes according to your needs.

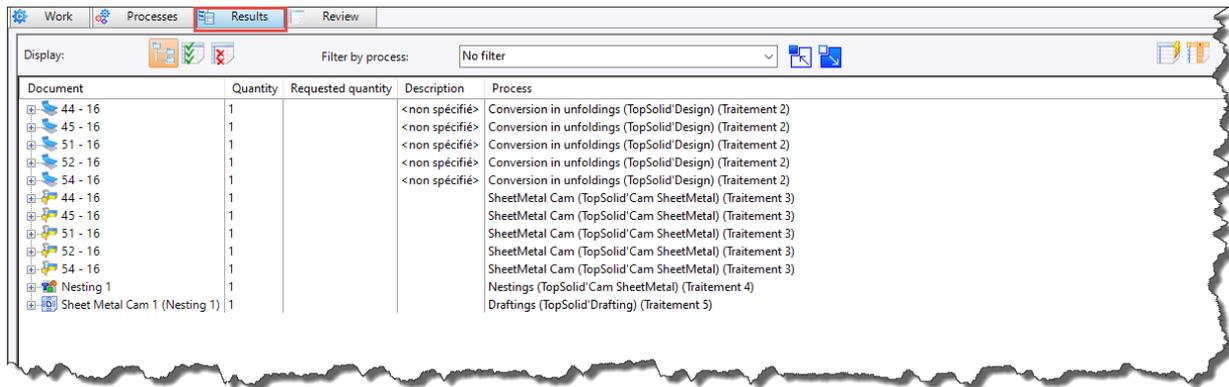
This document is very useful if you want to improve your efficiency. It meets the requirements of high-volume production by allowing the automatic generation of documents in hidden time to be optimized, and also by structuring the location of these documents in the current project.



Work tab



Processes tab



Process result

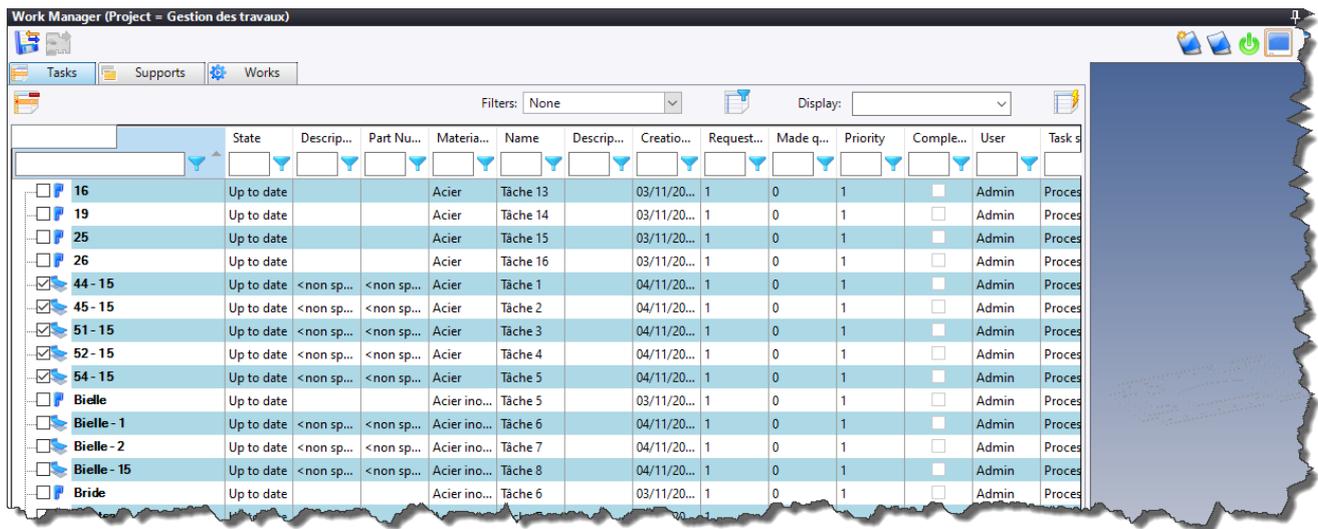
## Work Manager

The **Work Manager** is a scheduling tool that allows you to manage a workflow in a simple, efficient and orderly manner. It can be adapted to documents of different types: CAD (drafting, bill of materials, etc.), CAM (machining, nesting).

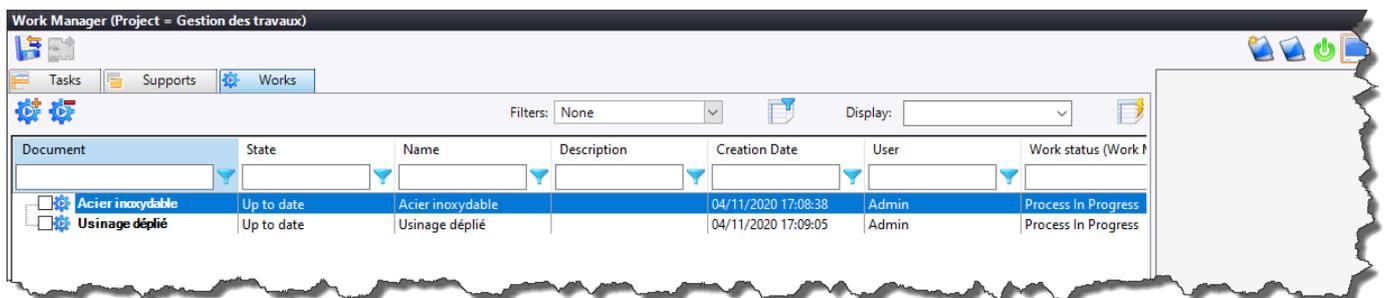
In addition, a number of settings and filters allow tasks to be scheduled according to the company's habits. The customer can therefore choose his scheduling order by material, thickness, part family, etc.

In PDM server mode, you can also assign tasks to several users and thus distribute the workflow within a team.

This tool is an additional complementary step to the work document by combining automation and management.



The **Tasks** tab listing the tasks to be distributed.



The **Works** tab grouping the works to be managed.