TopSolid 2022

TopSolid 7.16

What's New

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Welcome to TopSolid 2022

Throughout this document, you will discover the main enhancements and learn about the new features and benefits of the latest **2022** 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 2022** features, please contact your local reseller.

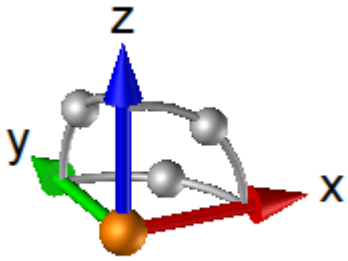
What's New in TopSolid'Design 7.16

This section describes the enhancements made to the **PDM**, **Design**, **Sheet Metal**, **Nesting**, **FEA**, **Drafting** and **Unfolding** applications in version **7.16** of **TopSolid 7**.

User Interface

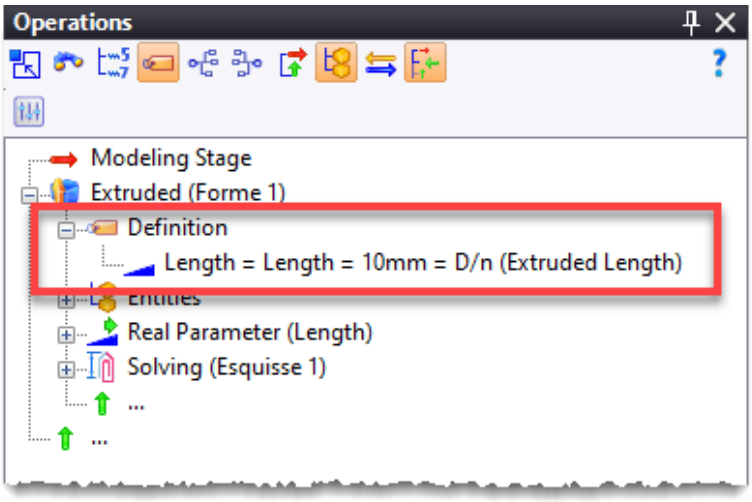
Compass

The color of the compass axes has been changed to be consistent with other software on the market. Now, the X axis is displayed in red, the Y axis in green and the Z axis in blue.
However, these colors can be modified in the **Tools** > **Options** > **Colors** > **System colors** section.



Displaying parameters

In the Operations tree, when a parameter is named, the type, name, value, expression and designation of the parameter are now displayed in the **Definition** node (as in the Entities tree).

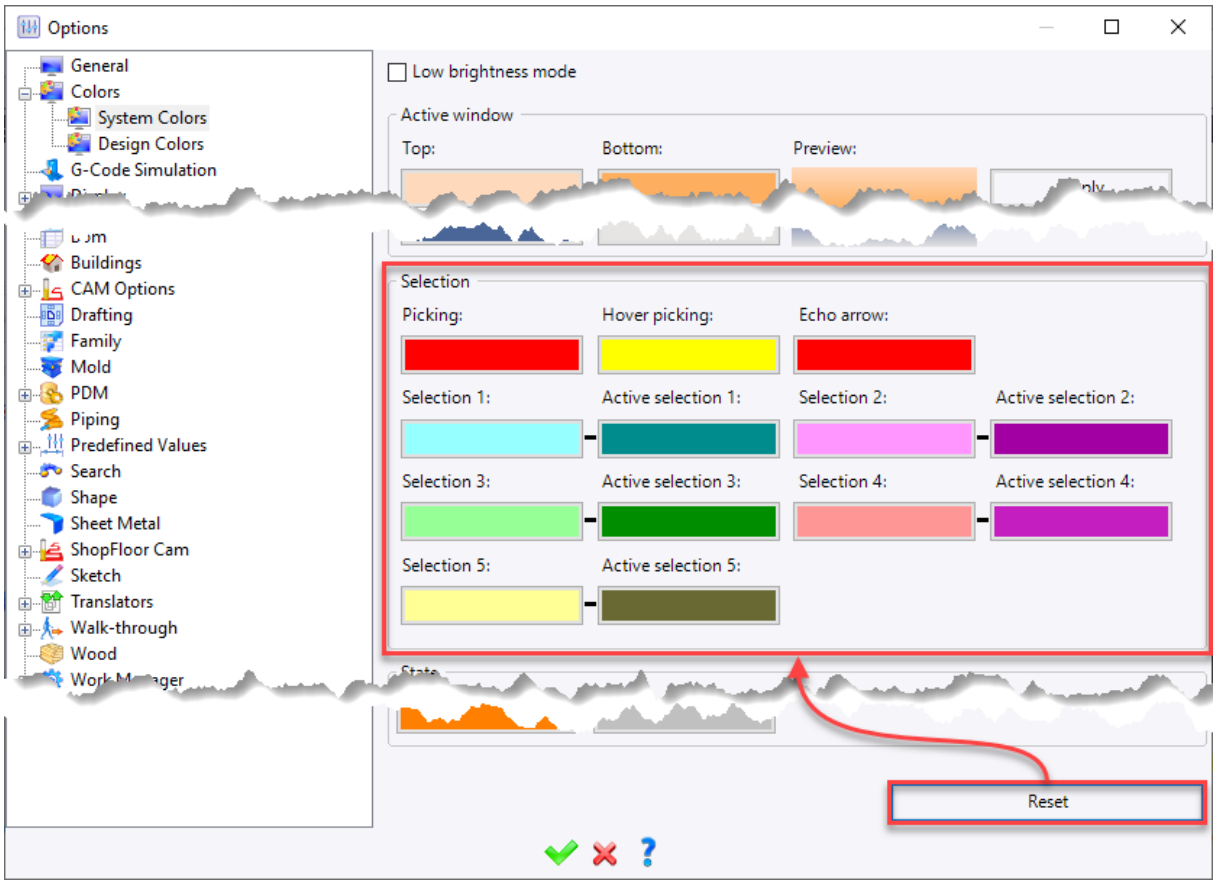


Manufacturing Features

In order not to overload the display, the arrows have been removed.

Selection colors

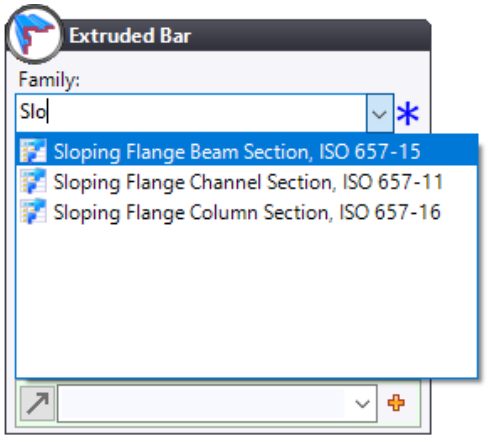
The selection colors accessible from **Tools** > **Options** > **Colors** > **System colors** have been slightly modified to be more contrasted. To apply this new setting, you have to click on the **Reset** button.



Selection filter

A dynamic selection filter that allows you to limit the drop-down list using a few entered characters has been added to the drop-down lists of the following commands:

- **Modeling** > **Extruded Bar** (**Assembly** document)
- **Tools** > **Functions** > **Provide Function** (**Part** and **Assembly** documents)
- **Tools** > **Material and coating** (document **Part**)
- **Tools** > **Parts material and coating** (**Assembly** document)



Example of a filter when including an extruded bar.

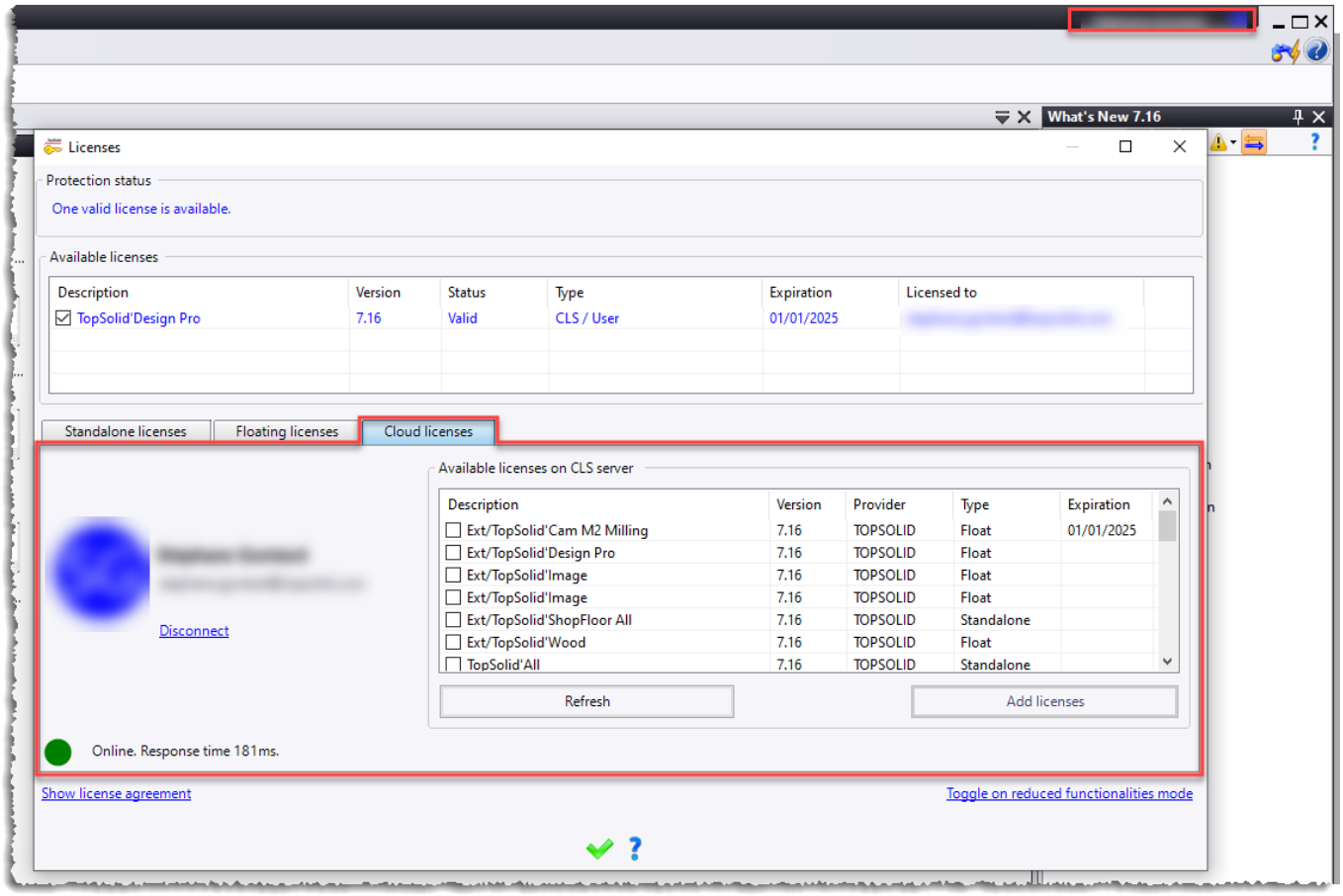
Upgrade

The **Upgrade** command is now available from a multi-selection from the Operations tree. This command allows you to replay an operation in the current version provided that the selected operations accept this command.

Licenses management

The new protection mode named **CLS** (**Cloud Licensing System**) allows you to simplify the license management. Users log in with an account (email address) and three types of licenses are available:

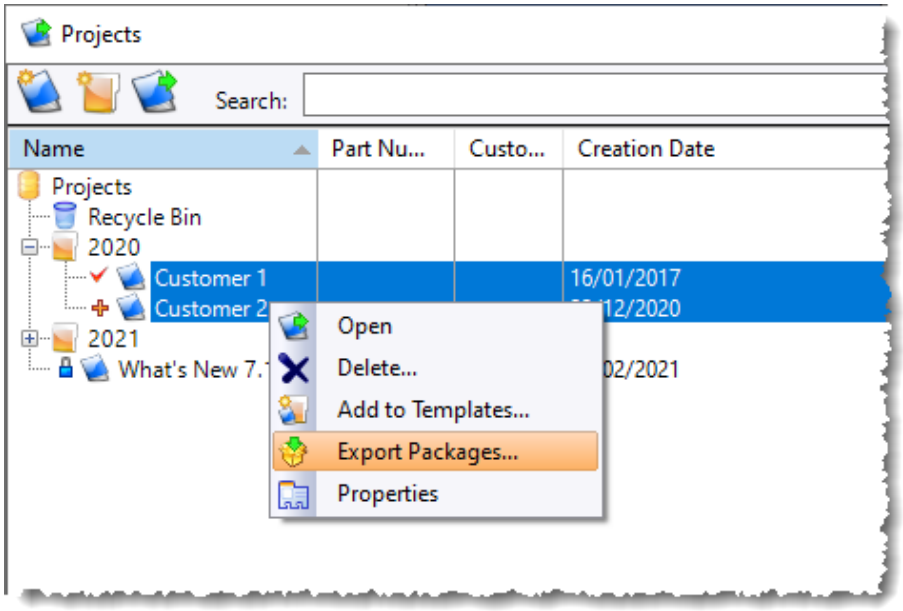
- **Standalone**: The license is assigned on a protection key or unique identifier.
- **User**: The license is assigned to a user identified by his email address.
- **Float**: A number of licenses is allocated per company. Licenses can be taken from any location, making it easy to manage licenses from home, for example.



PDM

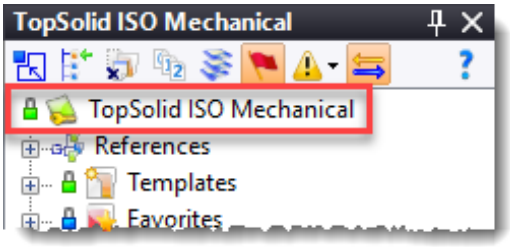
Multiple exports

From the **Project Manager** dialog box, the new **Export Packages** command allows you to export a selection of projects. This new command creates as many packages as selected projects.



Libraries

Protected (read-only) libraries are now identified by a specific icon with a key.



Package import

When importing a package, the new **Forced Copy** option allows you to import the documents by duplicating them. This new option will be particularly useful to solve index and/or version problems.

Export for subcontracting

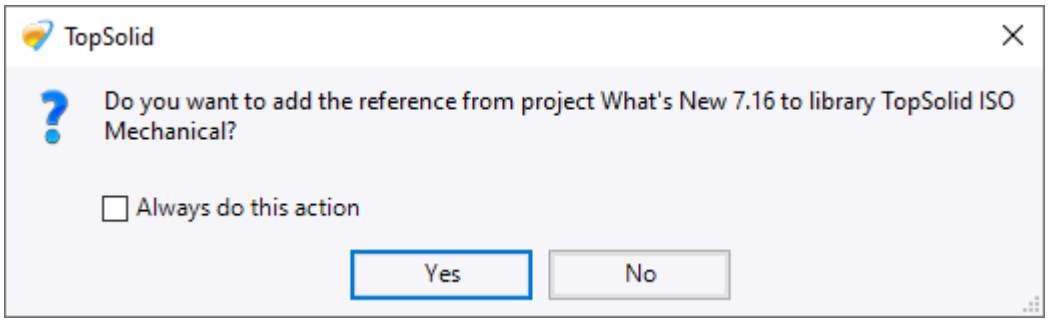
When exporting a package, the new **Export for delivery** option allows you to generate a package whose elements will not be modifiable after import on another site (subcontractor site).
To use these documents, you have to copy or derive them, which allows:

- on the subcontractor's side, not to modify the originals;
- on the customer side, to continue the modifications.

This same mechanism must be used by the subcontractor to return his project to his customer.

On-the-fly referencing

When a component is included from a library (or a project) that is not referenced by the project, **TopSolid** now offers to automatically reference this library.

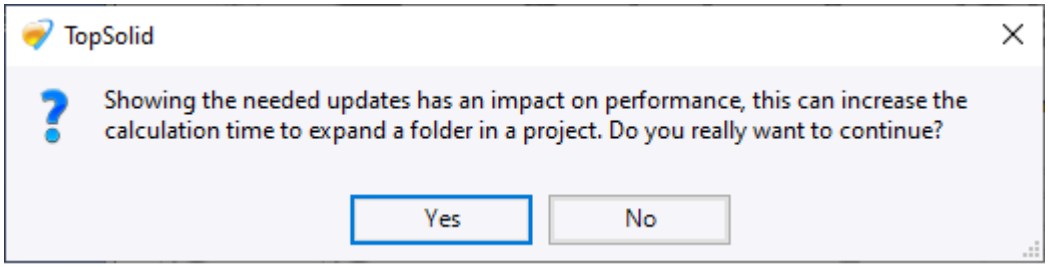


Connection

The new **PDM** > **Connection as** command allows an administrator to login with another user's account without entering the password. This allows you, for example, to check the documents into the vault for a person who is absent (user having left the company, employee on leave, etc.).

Displaying updates

The update display mode has been identified as a factor that decreases performance: as a result, a warning message is now displayed when this mode is enabled.
When the update display mode has been disabled, after modifying a document, the **Update** command should be used to display the list of documents to be updated.
During a new installation, this mode is now disabled by default.

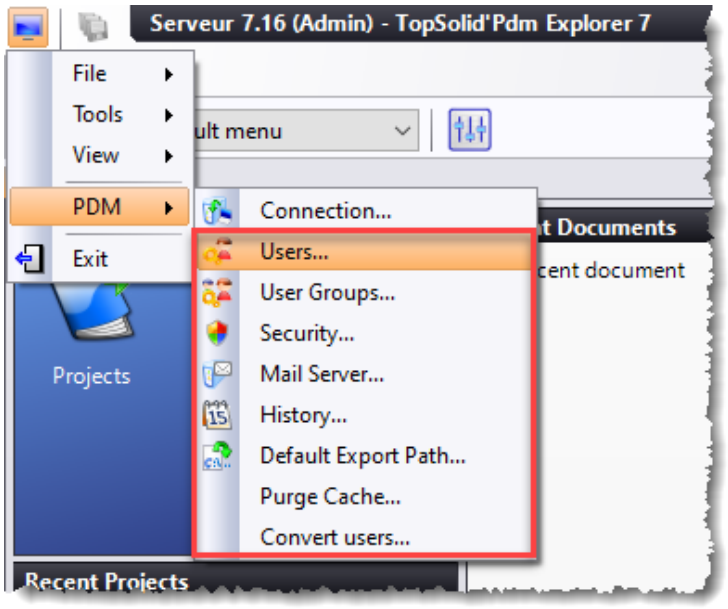


Managing external documents

When an external document (.docx, .xlsx, .txt, etc.) managed by the PDM is opened for modification and then saved in its application (Word, Excel, Notepad, etc.), it is now automatically saved in the PDM.

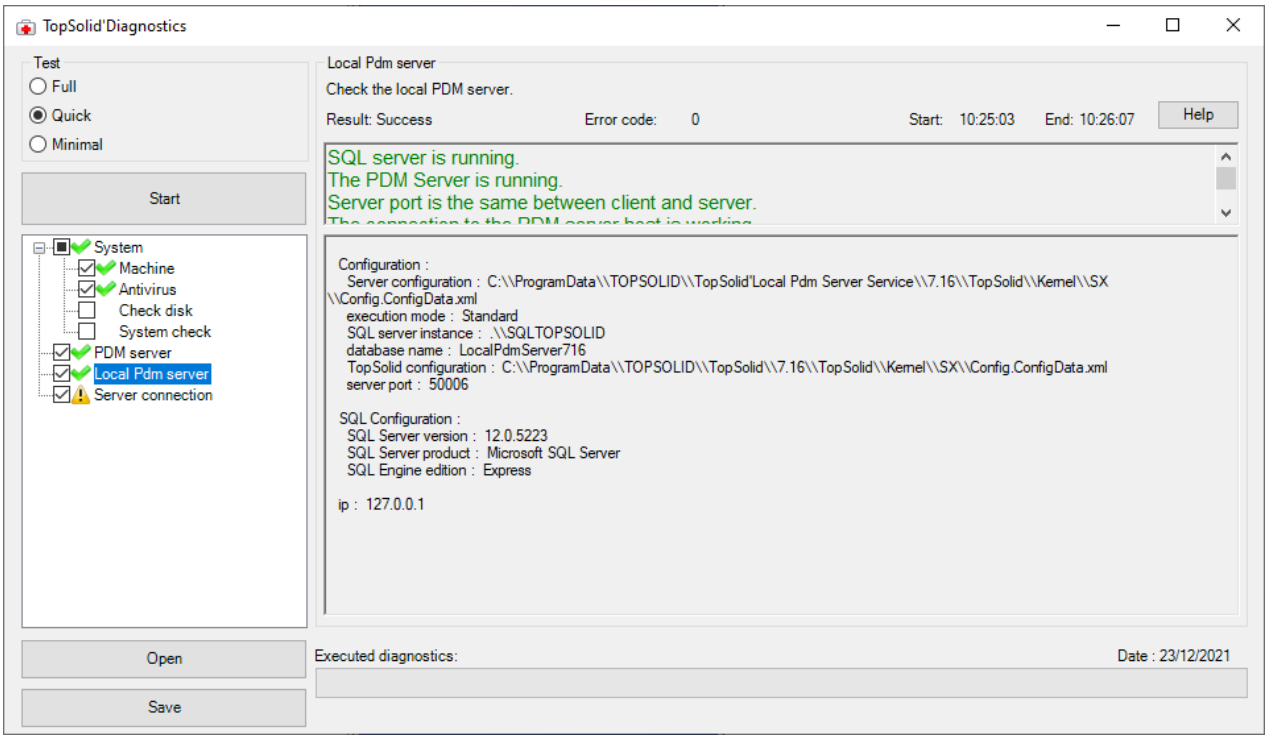
TopSolid'Pdm Explorer

The administration **Users**, **User groups**, **Security**, **Mail server**, **History**, **Default export path**, **Purge cache** and **Convert users** commands have been added. They allow **TopSolid** to be administered using the **TopSolid'Pdm Explorer** thin client.



TopSolid'Diagnostics

A new **TopSolid'Diagnostics** application allows you to make a diagnosis (memory, disk, network, antivirus, etc.) of the machines (clients and servers) and to offer solutions when problems are detected. It is available in the program installation folder (by default in C:\Programs\TOPSOLID\TopSolid Diagnostics 7.16\bin\TopSolid**.Diagnostics.exe**).



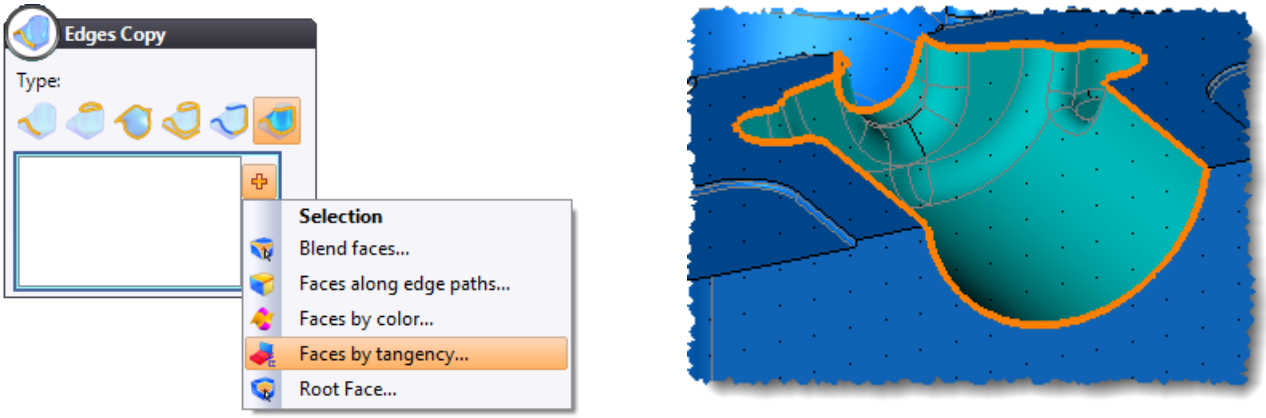
Sketch

Parabola and hyperbola

Two new commands have been added to the **2D Sketch** > **Other Curves** menu. They allow you to create parabola and hyperbolic arcs.

Edge copy

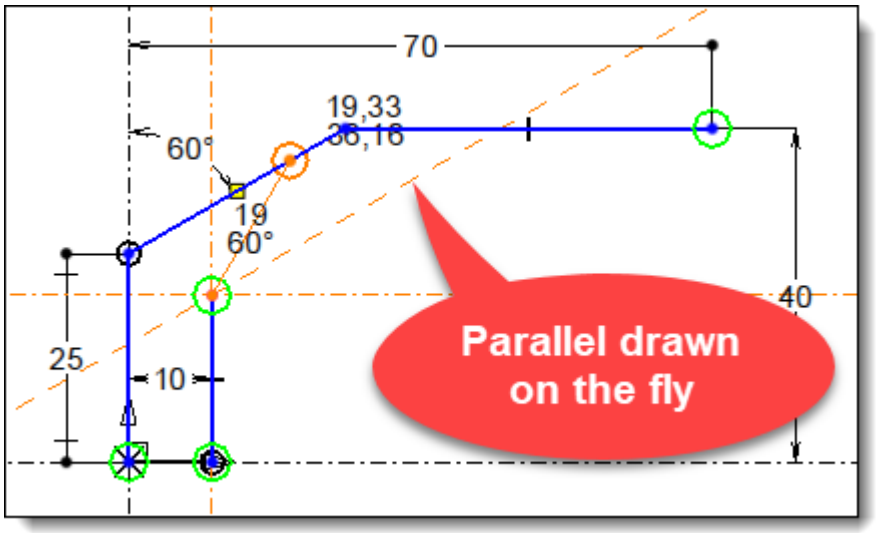
In a 3D sketch, the **Edges Copy** command with the **Paths around faces** option now has a **Special inputs** icon that allows you to select the edges of a part according to a root face, blend faces, faces by color, tangent faces, faces along edge paths.



Example of edges copy with selection of tangent faces.

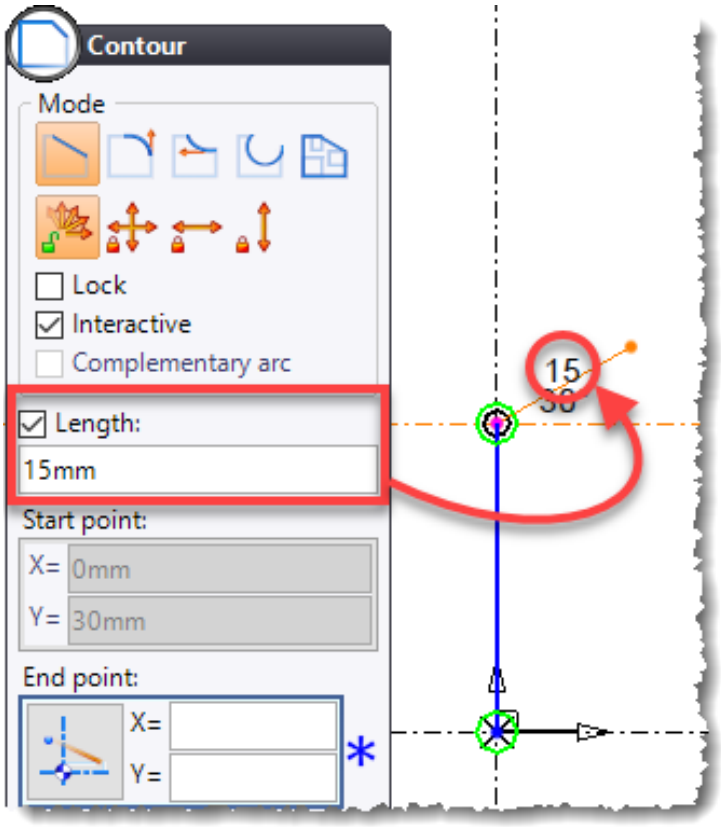
Parallel drawn on-the-fly

When creating a contour and/or a line, if the cursor is placed on another linear segment, **TopSolid** draws a magnetic axis parallel to the linear segment, passing through the last point of the contour being created. A dimension is created automatically if you base on this magnetic axis.



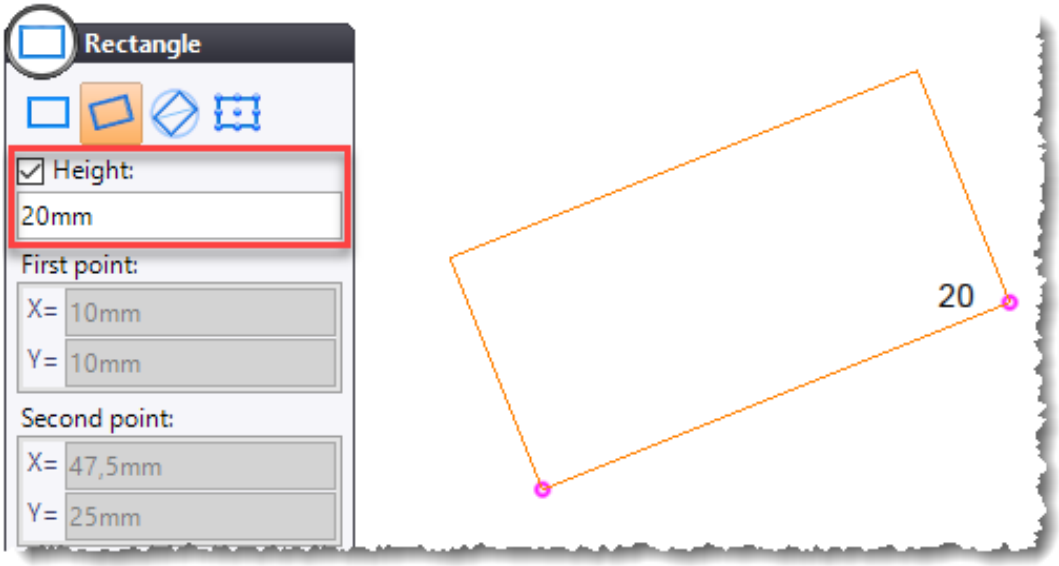
Contour

A new **Length** field allows you to enter a value that will be assigned to the length of the created segment and the corresponding dimension will be automatically added.



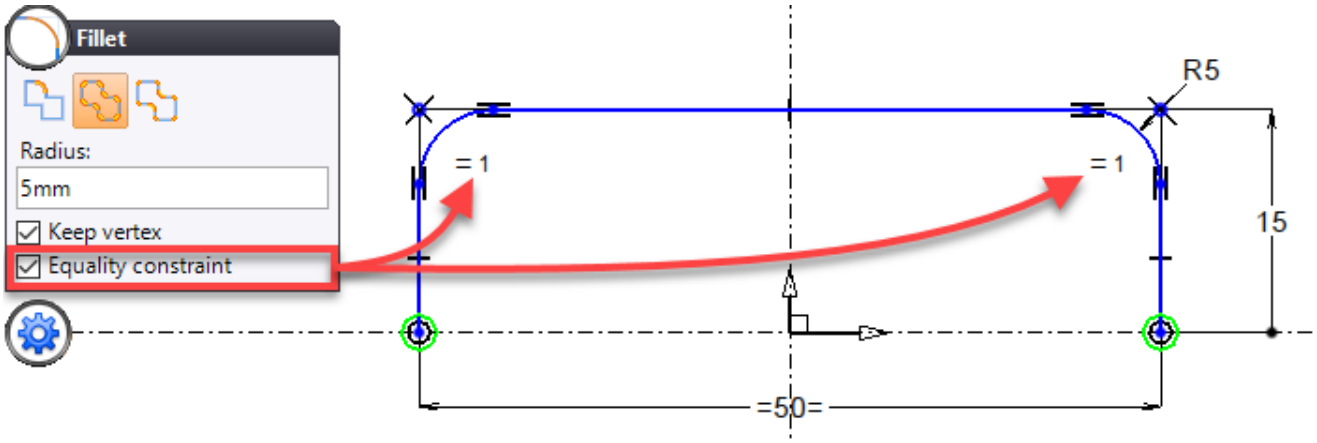
Rectangle

With the **Three points, by base** mode, the new **Height** option now allows you to indicate a value (thickness) and to only give two construction points.



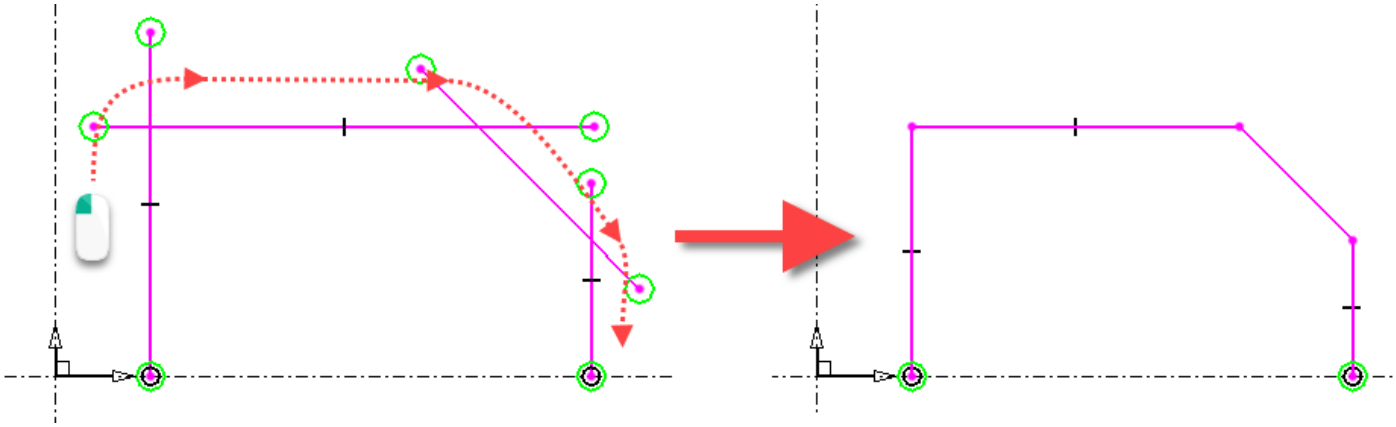
Fillet

The **2D Sketch** > **Fillet** command has a new option called **Equality constraint** for the **Global** and **Internal external** modes. This option allows you to create only one dimension and to add an equality constraint on each fillet.



Trimming by dragging

In the **2D Sketch** > **Trim** command, you simply have to now click in the graphics area and to move the cursor (keeping the left button pressed) on the parts to be deleted in order to trim them.



Cell

The **Explode** contextual command allows you to split a cell type entity into four sketch segments in order to make modifications (fillets, chamfers, etc.).

Half-part dimension

The **2D Sketch** > **Revolution dimension** command has been renamed **2D Sketch** > **Half Part Dimension**.
This new type of entity has three types of displays (half-part, large diameter, or whole part) that can be adjusted using the **Display** contextual command. This display mode will be used by the **Detailing** > **Projected annotations** command in a drafting document.

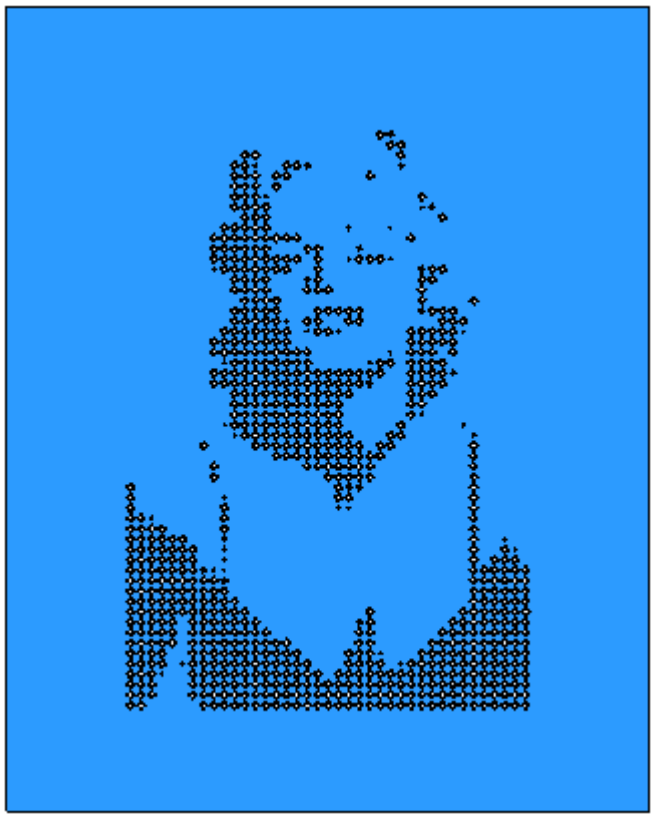
Symbol copy

Symbols, except for symbols with geometric drivers, can now be copied using the **Ctrl** + **Drag** technique**.**

Pixelating

The new **2D Sketch** > **Operations** > **Profiles Batch** > **Pixelating** command allows you to create a sketch and patterns from an image. These entities can be used, for example, to repeat drillings in order to reproduce the image on a sheet metal.

Base image



Perforated sheet metal plate made from the sketch generated by the pixelating operation.

Options

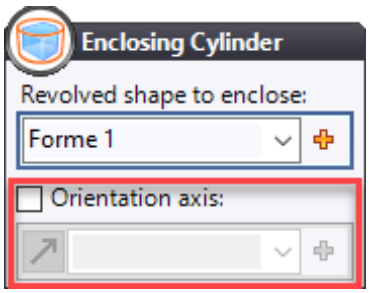
In the document options, several changes have been made to the **Planar sketch** and **Spatial sketch** options:

- The **Grid** section has been renamed by **Snap**.
- A new **Grid** section has been added in order to modify the default settings of the grid (**Type**, **Coordinates**, **Step**).

Shape

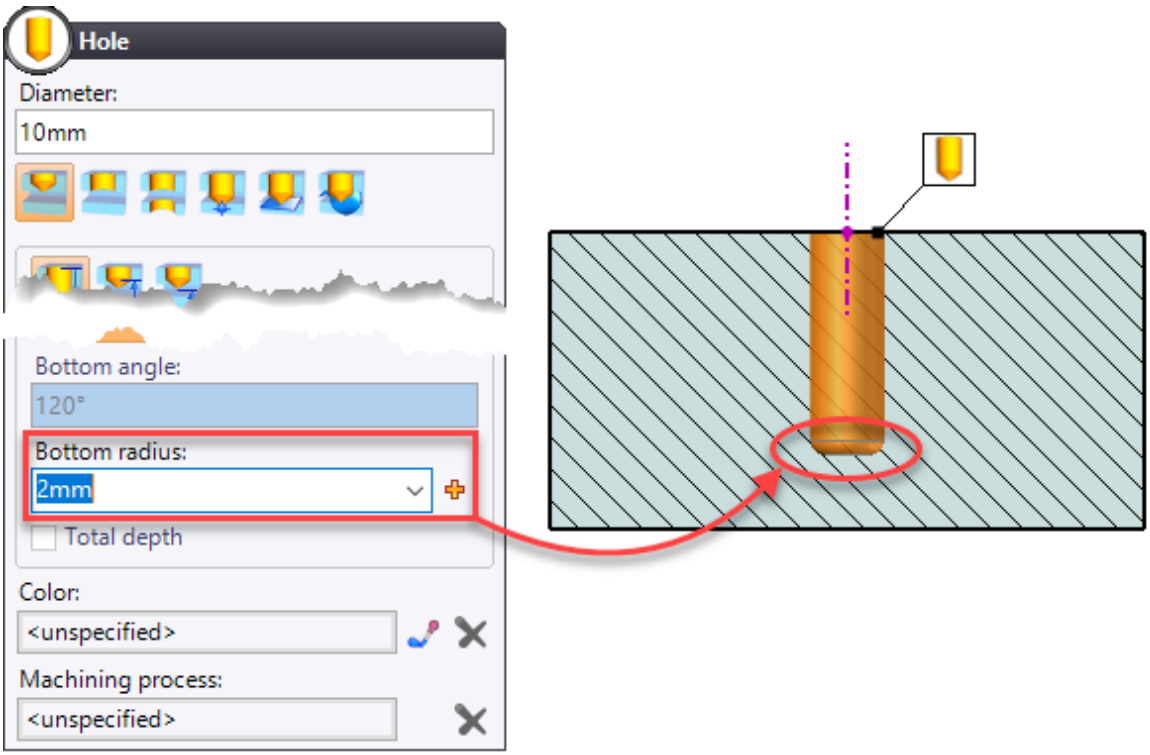
Enclosing cylinder

The calculation of the enclosing cylinder axis is now automatic. However, you can define the axis manually by checking the **Orientation axis** box and then selecting an axis.



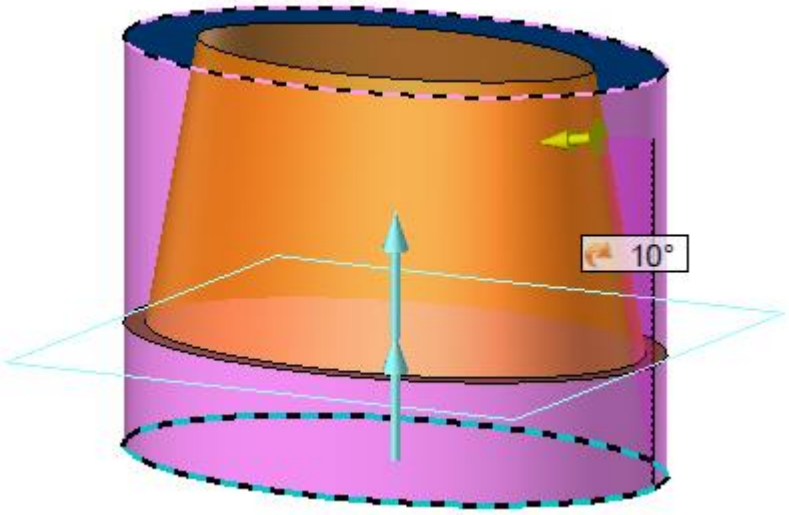
Drillings

For blind drillings with a flat drill (smooth hole and counterbore), you can now add a bottom radius. This information can be retrieved from a table or a drilling dimension.



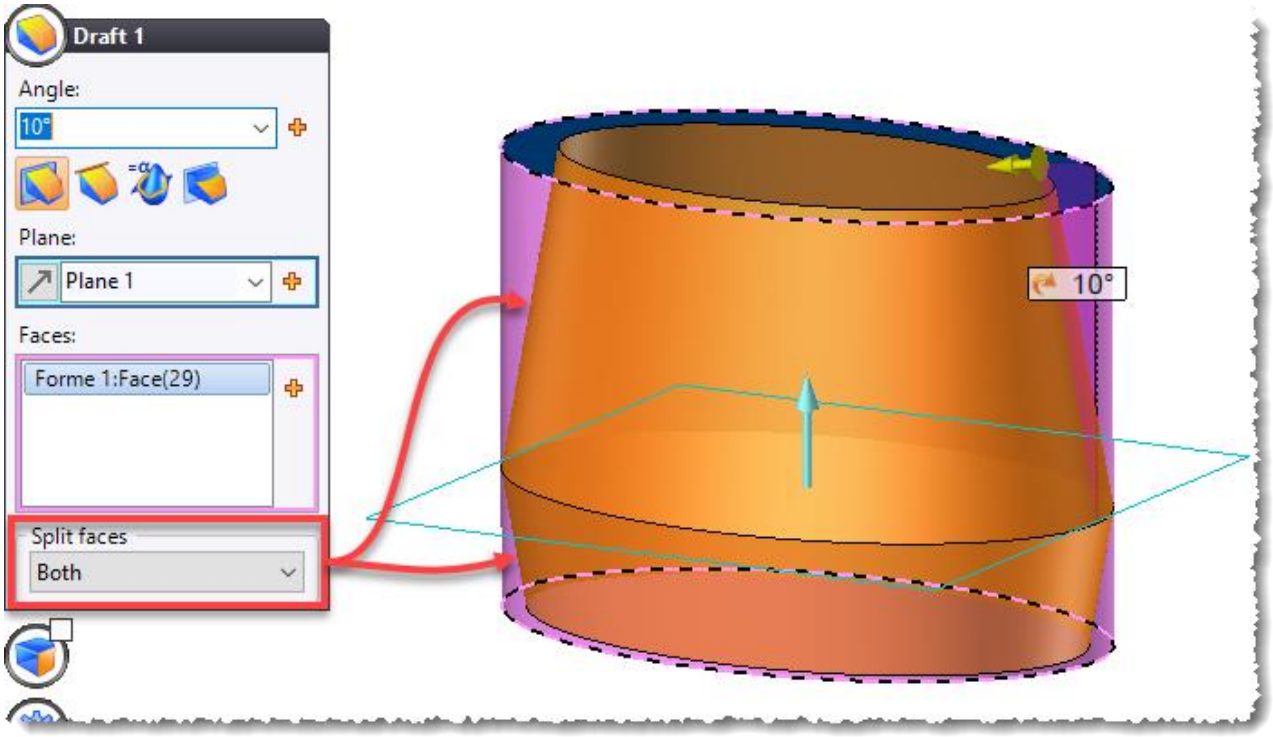
Draft

The **Step with plane** option has been added to the **Draft with step** mode. It allows you to select a plane, which avoids imprinting an edge on the shape to be drafted.



Example of a draft with step and plane selection.

In addition, the **Split faces** option available in the **Draft with reference plane** and **Draft with step** modes allows you to either separate the faces located on each side of the reference plane or not.



Example of a draft with split faces.

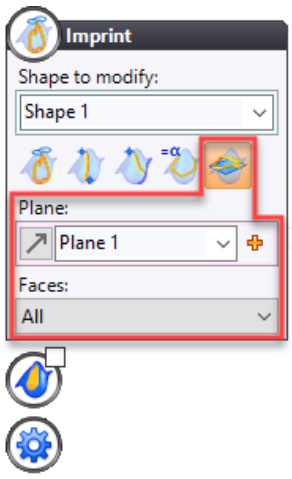
Enclosing block

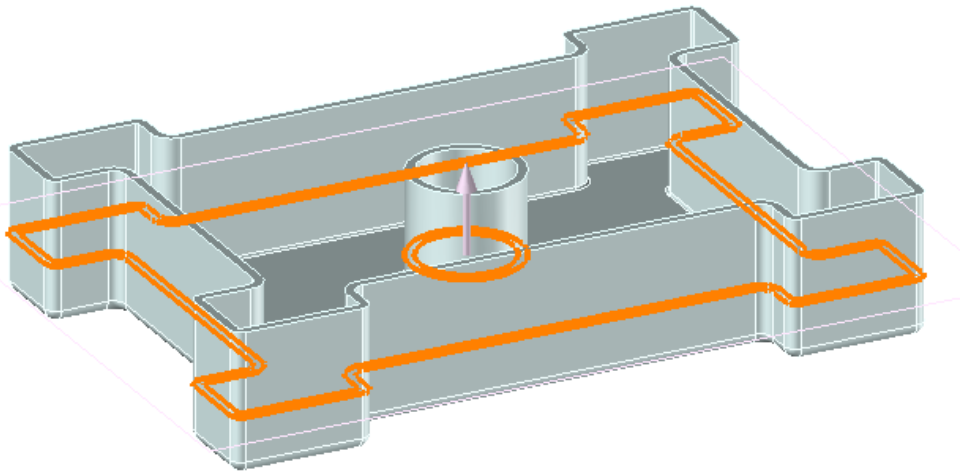
Polyhedral shapes from an import (STL, FBX, etc.) are now compatible.

Surface

Imprint by plane

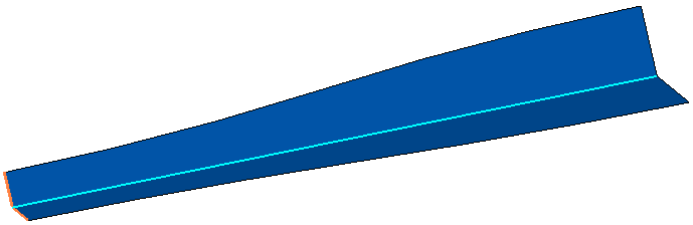
The **By plane** mode has been added, it allows you to imprint the intersection profiles between the part and the selected plane. In addition, the **Faces** option allows you to choose the faces on which the profiles will be imprinted.



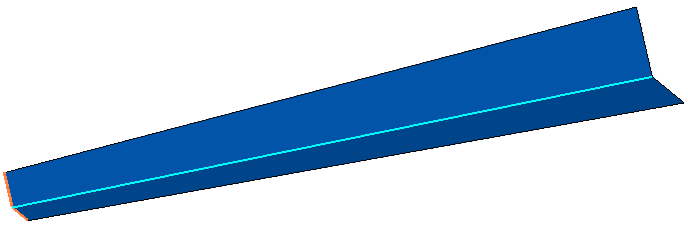


Pipe

The **Linear Variation** option has been added to the **Rotation** and **Scale** dialog boxes. It allows the section to evolve either linearly or not along the path.



*Example of a pipe with scale and the **Linear Variation** box unchecked.*

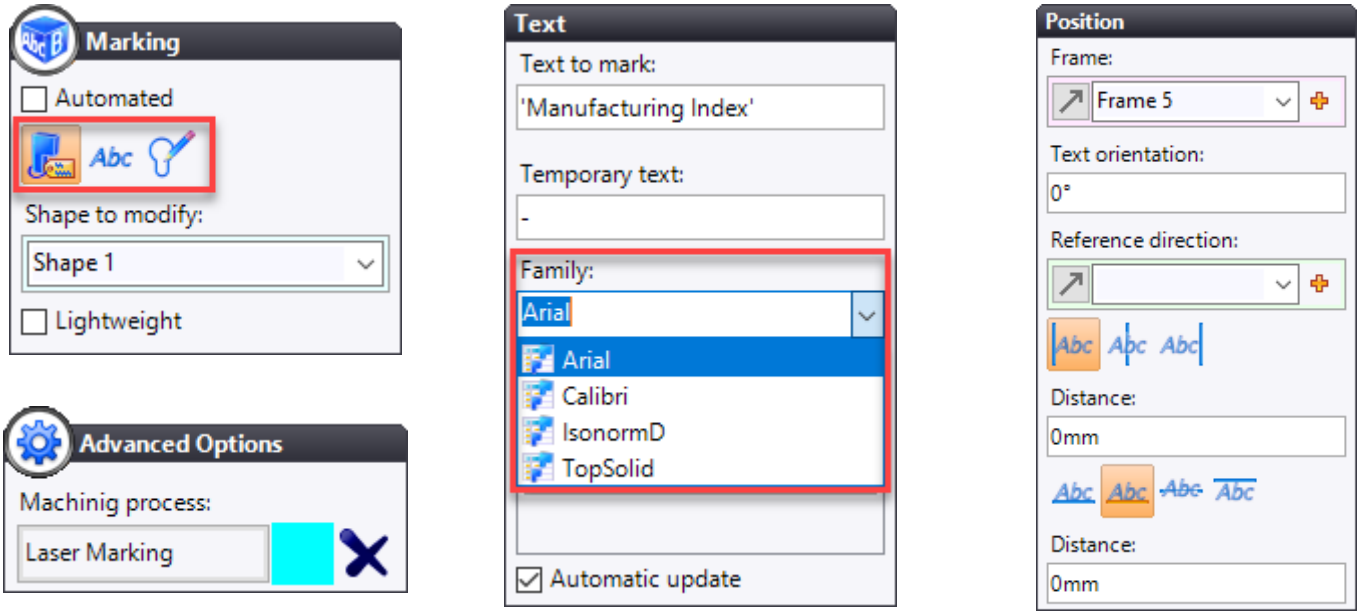


*Example of a pipe with scale and the **Linear Variation** box checked.*

Marking

The **Marking** dialog box has been modified:

- Three icons allow you to choose the type of information to mark.
- **Arial**, **Calibri** and **IsonormD** fonts have been added.
- A section allowing you to orientate and position the text has been added.
- You can define a specific machining process.

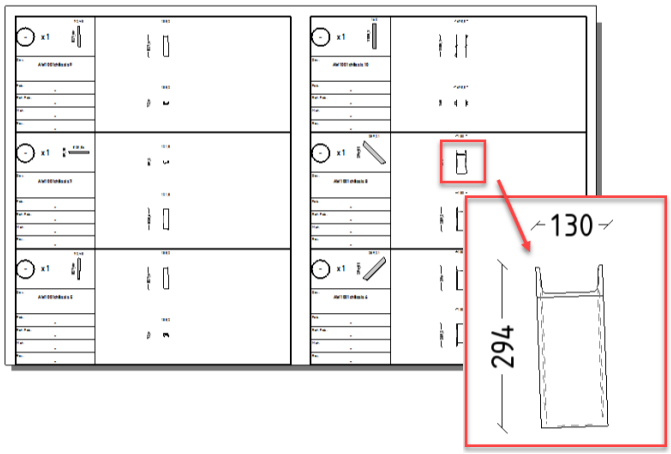


The new dialog boxes of the marking command.

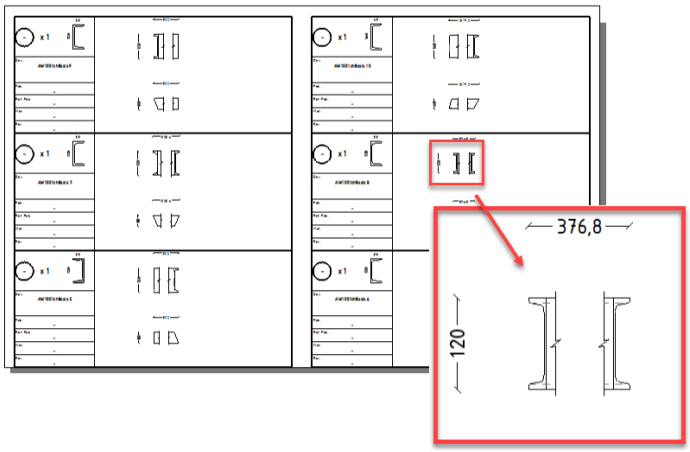
Sheet Metal

Sheet metals recognition

When recognizing **Profile** and **Tube** parts, the camera orientation is automatically adjusted to the orientation of the profile in the document. This improvement now allows you to create views in multiple draftings that are automatically oriented along the axes of the profiles.



Recognition and conversion of parts into profiles, without modification of the camera.



*With the new version of **TopSolid**, the views are automatically oriented along the axes of the profiles.*

In addition, in the case of recognition of standardized profiles, the section type and its code are associated with it. And the corresponding parameters are created allowing you to format the designations of the profiles, for example.

Sheet metal recognition processes

In the **Work** document, a new **Sheet metal recognition (TopSolid'Design)** process is available.
This process automates the creation of sheet metal parts, from part or assembly documents.
The options for this process are similar to those available for the classic assembly document command.
In the **Results** tab, only the documents modified by the conversion are listed.

Performance improvement: Sheet metal properties computation

A very important improvement regarding calculation time has been added when generating the sheet metal properties. This improvement is very significant in the case of parts with many bends and/or many holes.

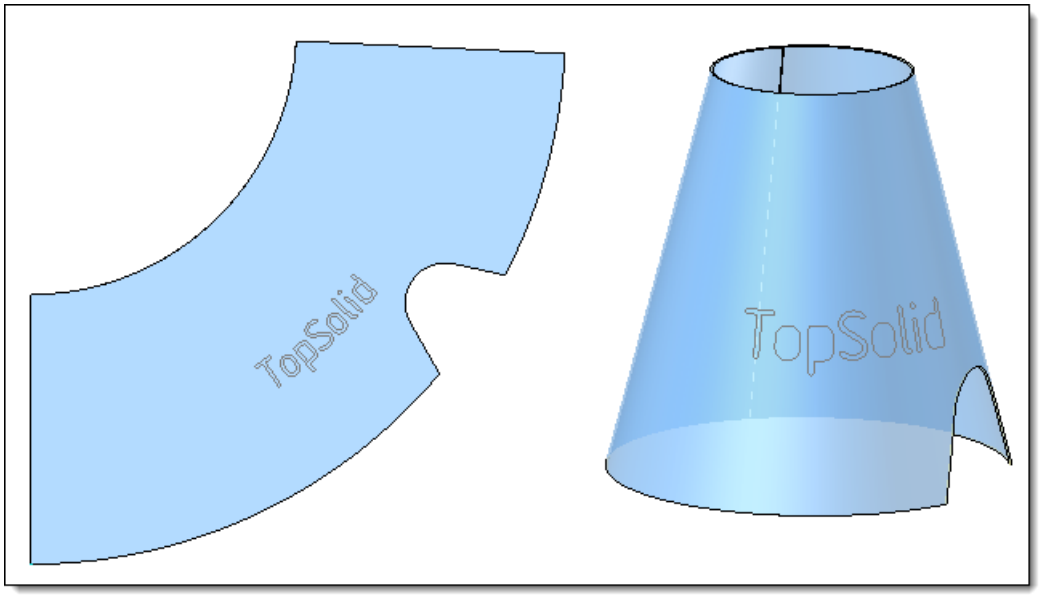
Conversion into unfoldings

This contextual command, launched from the Project tree, allows you to manage Autocad imports containing multiple unfoldings in order to create unfolding documents for each identified part.
You can now create the part document of the sheet metal for each unfolding. Therefore, the information only available on 3D parts, such as the MFs from the analysis operation, can be used for costing, for example.

Unfolding and rebending of bend

The unfolding and rebending of bend commands have been enhanced with a new **Developable bend** mode. They now allow you to unfold and rebend strictly developable surfaces, in particular the option can be applied to straight cones or inclined cones.

You can impose an unbending method, which is suitable for unrolling surfaces and therefore has the **Approximation tolerance** and **k factor** parameters (as in the **Unbending Process** command in **Rolling** mode).
As for the classic unbending bends, you can perform Boolean operations (pockets, drillings, trimmings, etc.) or imprint operations on the unbended faces. The dimensions of the removals and imprints carried out before the rebending operation are kept, as well as their geometry. The drillings are identified as drillings in the drafting document (and therefore listed in the drilling tables).



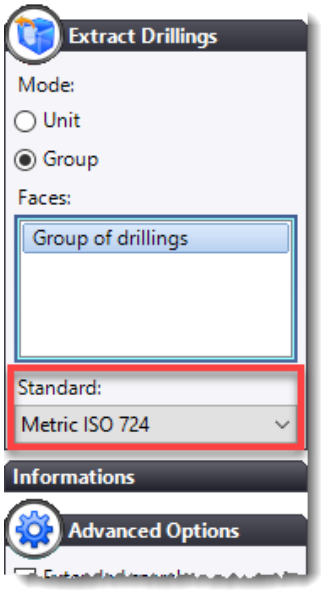
FreeShape

Progress bar

When extracting one or more drilling groups, a progress bar is now displayed at the bottom of the screen.

Tapping standard

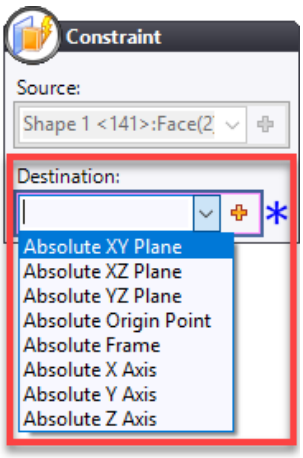
In the **FreeShape** > **Extract holes** command, a new drop-down list allows you to choose the standard of the tapped hole.



Assembly

Automatic positioning

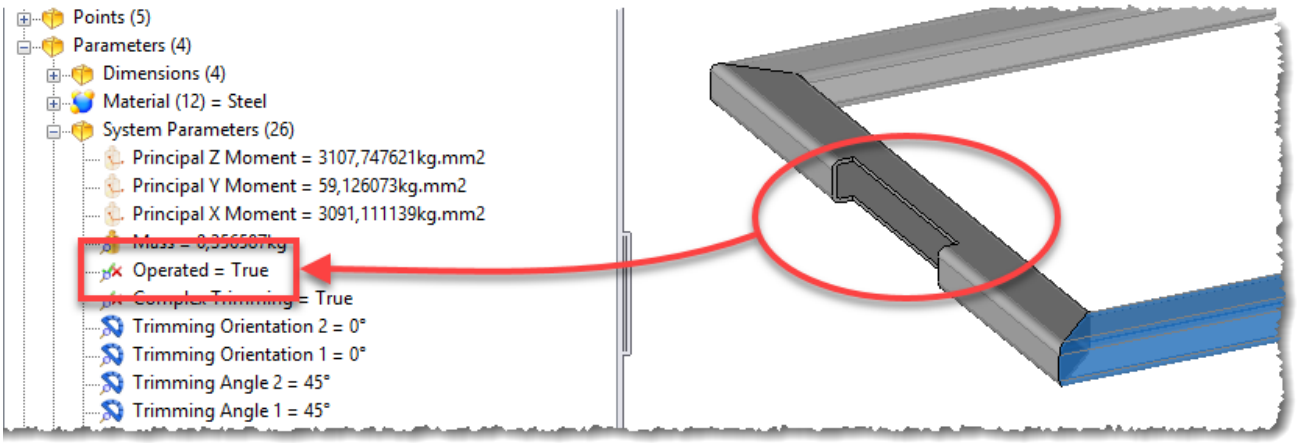
In the **Constraint** command, the absolute origin point, axes (X, Y and Z absolute), planes (XY, XZ and YZ absolute) and the absolute frame are now offered in the **Destination** field.



Extruded bars

Operations

When a profile has operations other than cutting operations, it is now identified by a specific management operation in the **Analysis** step that produces an **Operated = True** parameter in the **Parameters** > **System parameters** folder located in the Entities tree. This parameter can then be retrieved in a bill of materials, allowing the user to create a more detailed drafting document of these profiles.



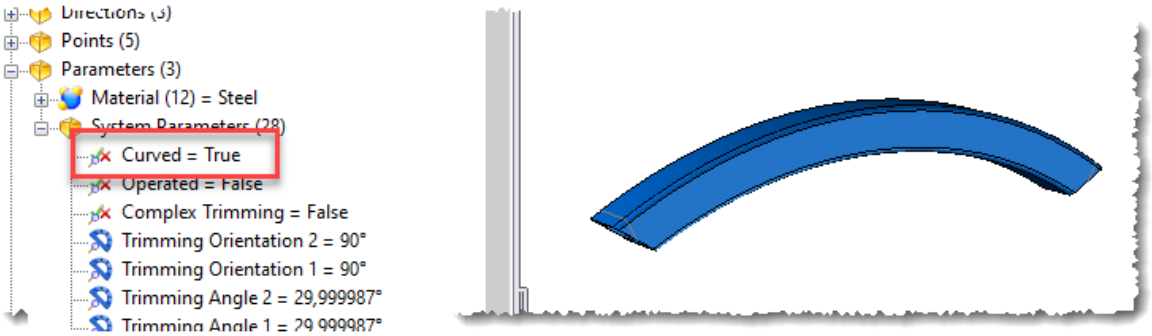
In addition, the profiles derived for modification are now compatible with cutting operations (mitre trim, main trim, etc.).

Possible lengths

This improvement avoids you to create copies with lengths of 100m. **TopSolid** then uses the first possible value just above the actual length as the length. For example, when installing a profile on a 2.5m profile, if the possible values of 1m, 3m and 10m have been declared in the generic document, then the copy created will use the length 3m. This is very interesting when the profile generic contains a multitude of operations because the calculation time during the inclusion is considerably optimized.

Curved extruded bars

As for the operated extruded bars, the curved extruded bars are now identified by a specific management operation in the **Analysis** stage which produces a **Curved = True** parameter in the **Parameters** > **System parameters** folder located in the Entities tree. This parameter can then be retrieved in a bill of materials, allowing the user to create a more detailed drafting document of these profiles.

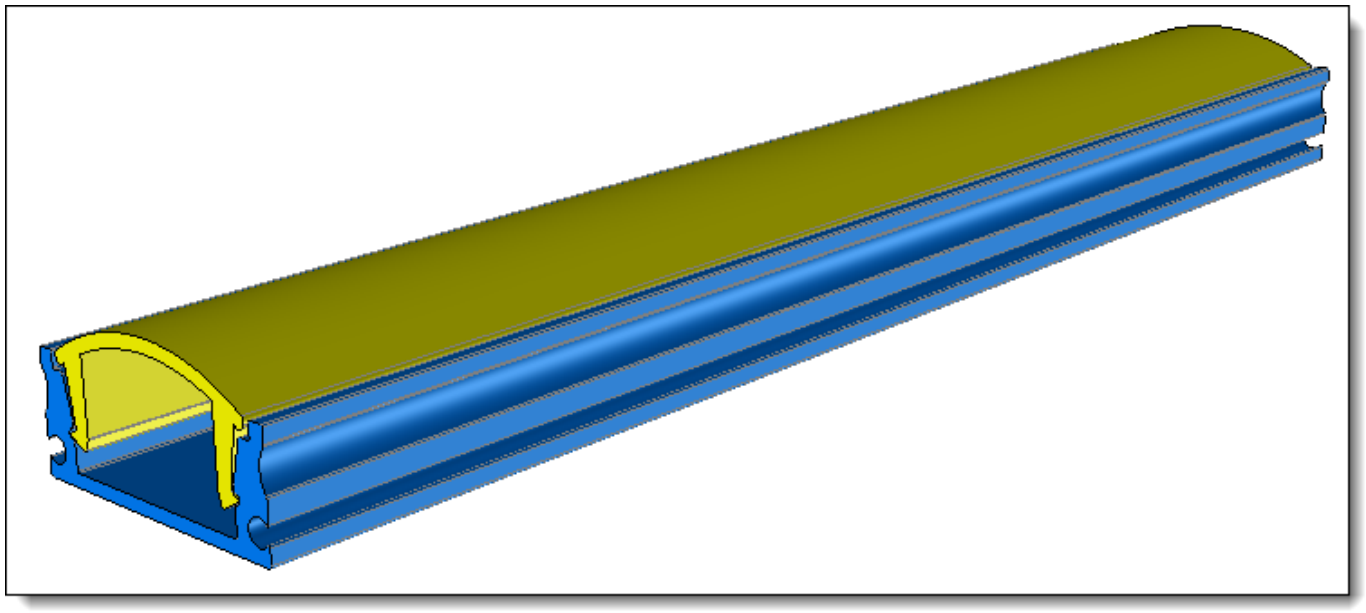


In addition, two improvements have been added:

- Curved extruded bars are now compatible with the **Mitre Trim** command.
- The curved extruded bars are divided into simple shapes (straight part, curved part), which allows them to be dimensioned in the drafting document.

Profile of extruded bar

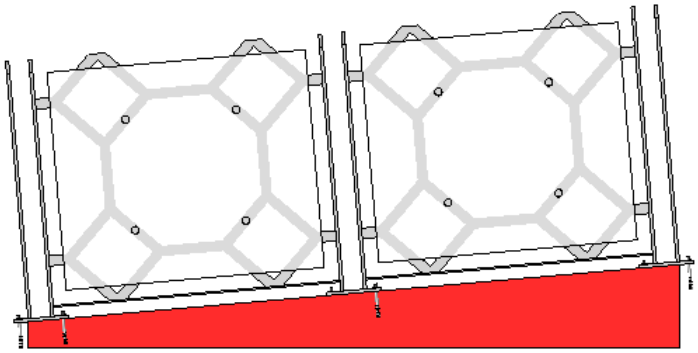
In an assembly document, you can now include several extruded bars and provide the **Extruded Bar** function to define a profile of the extruded bars. This allows you, for example, to create a component consisting of an extruded bar and its joint. This profile of extruded bars behaves like any simple extruded bar and supports cutting operations (mitre trim, main trim, curved extruded bar, etc.).



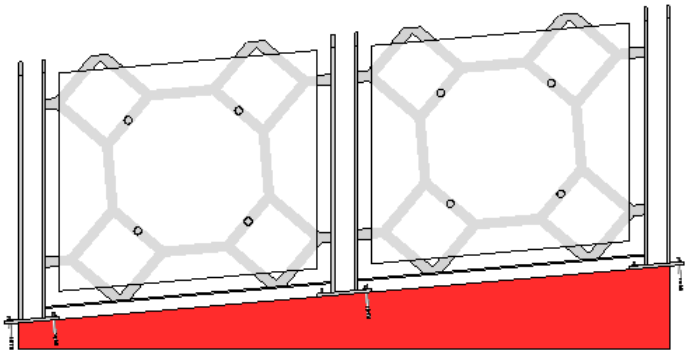
Distribution

In a family document, the **Distributable Family** driving now offers an **Angle** parameter to manage the orientation of the distributed component.
Therefore, in the distribution command, two modes are now available:

- **Normal orientation** mode: the orientation of the component is perpendicular to the slope.
- **Vertical orientation** mode: the orientation of the component is parallel to the vertical direction (Z axis).



***Normal** mode*



***Vertical** mode*

In addition, when distributing a component such as a railing on a staircase, the new **First component centered** option allows you to reposition the first component on the origin of the distribution.

Local assembly

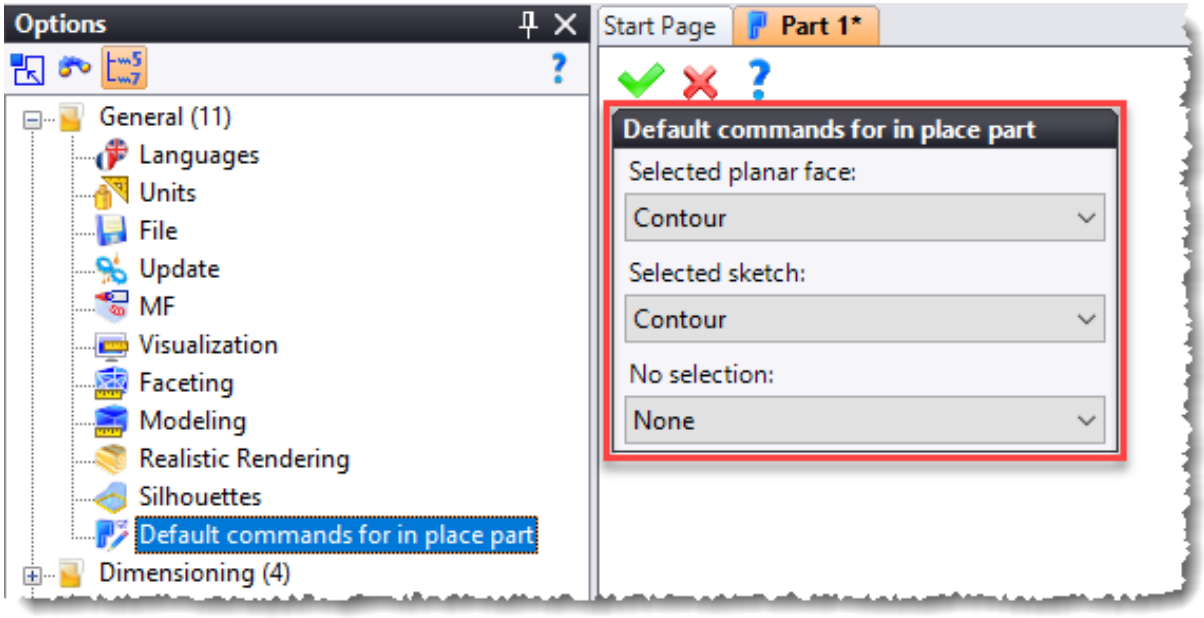
When converting a part from a sub-assembly to a local part, the sub-assembly is converted to a local assembly and now a warning message is displayed to warn the user of this conversion.

Locally modifiable parts

The **Modeling** > **Local shapes** > **Locally modifiable parts** command has been added: when a part has been defined as locally modifiable, if an assembly operation (drilling, trimming or pocketing) is applied, it is generated only in the assembly. As a result, there is no synchronization between the documents. This operation is automatically launched when the parts are connected (see also the **Tools** > **Joined parts** paragraph).

In-place part

In the options of a **Part** document, the **General** > **Default commands for part in place** option allows you to choose the commands to be launched when creating an in-place part. You can choose a command to select a planar face, a sketch or when no selection is active.



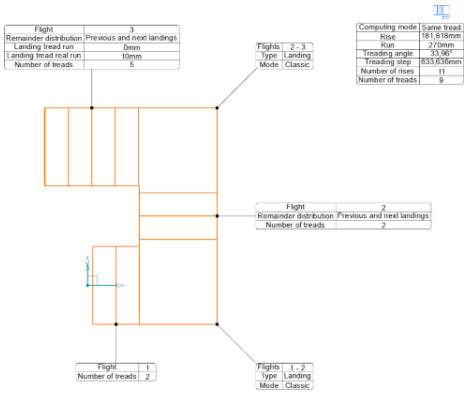
Building

Multi-flight staircase

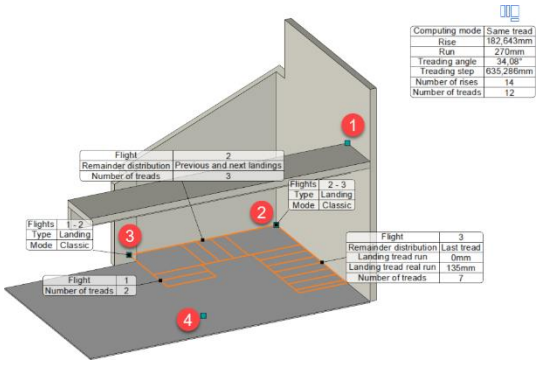
TopSolid'Building now integrates the **Multi-flight staircase** command. This command allows you to calculate a staircase distribution according to many criteria.

Key features:

- two laying methods:

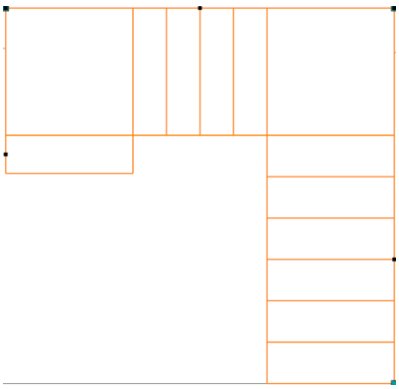


By table
(entry of the values of the different flights)

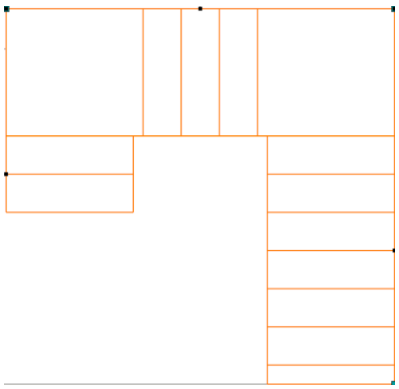


By points
(direct installation on environment)

- two calculation methods:

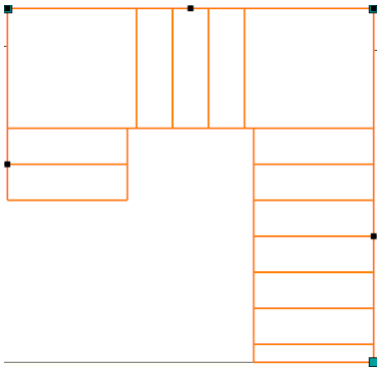


Same rise
Straight steps have the same rise but different runs

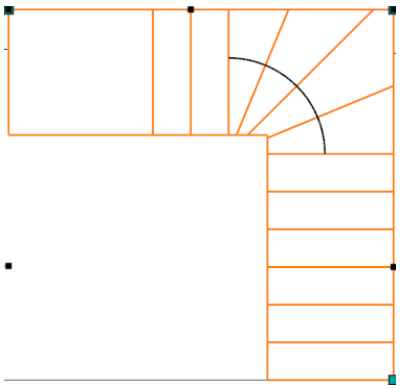


Same step
Straight steps are completely identical

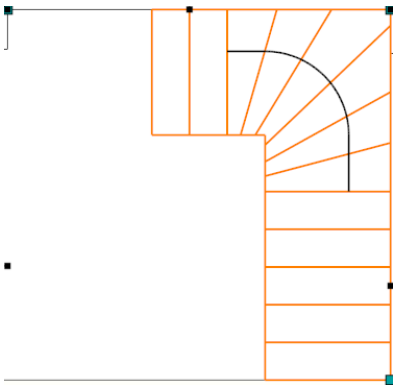
- three methods of angle management:



Landing

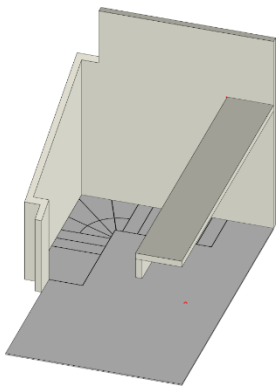


Spiral treads

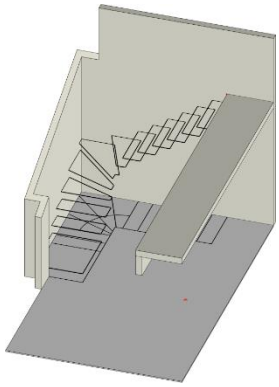


Winder treads

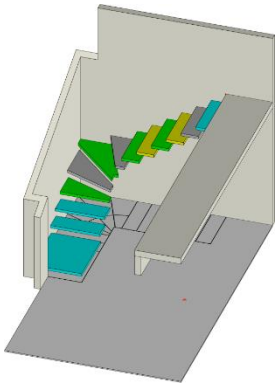
- four possible results:



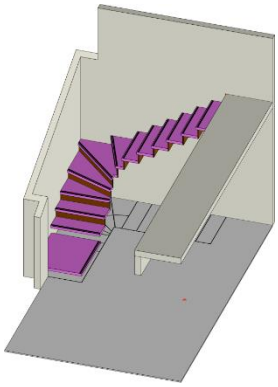
Sketch on the ground



Profile on each step



3D Shapes



Component (family)

- five applicable constraints (min, max, objective):
  - **Rise**;
  - **Run**;
  - **Treading angle**;
  - **Treading step**;
  - **Number of treads**.

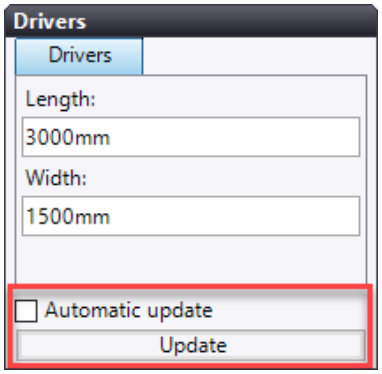
Family

Parameter

The tree structure of the parameters folder is now retrieved in the **Generics** folder of the **Family** document.

Driver update

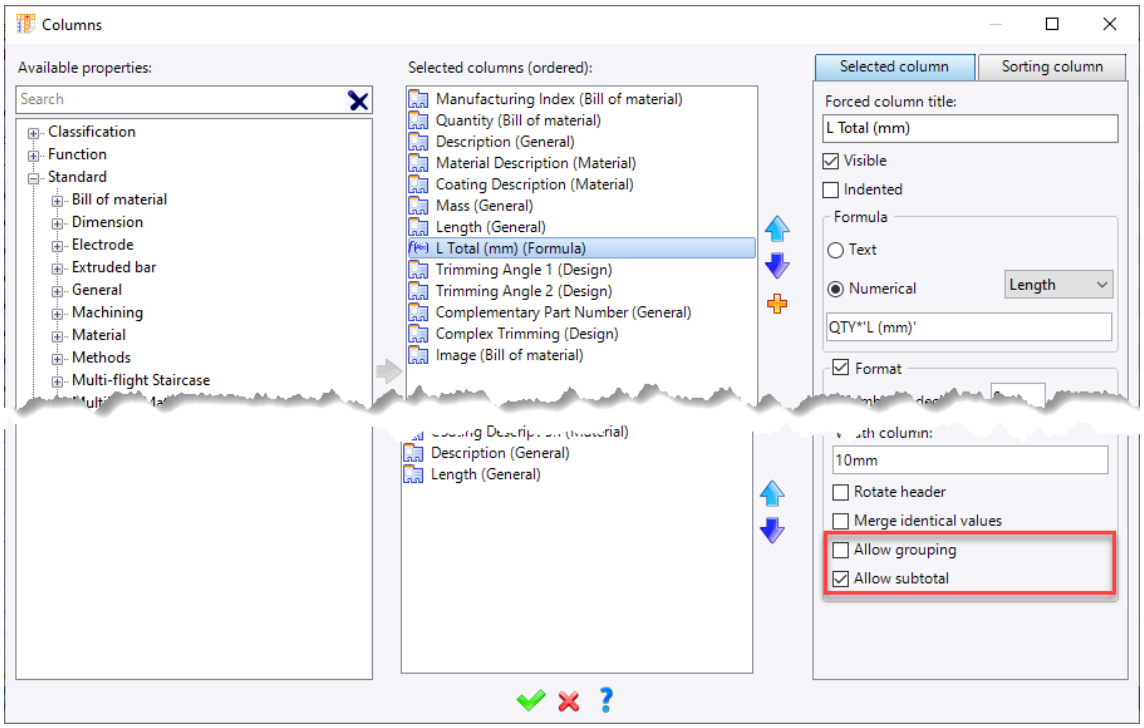
When including a family, the drivers can now be updated manually. For families containing many drivers, this allows the component to be **updated** only when all drivers have been completed.



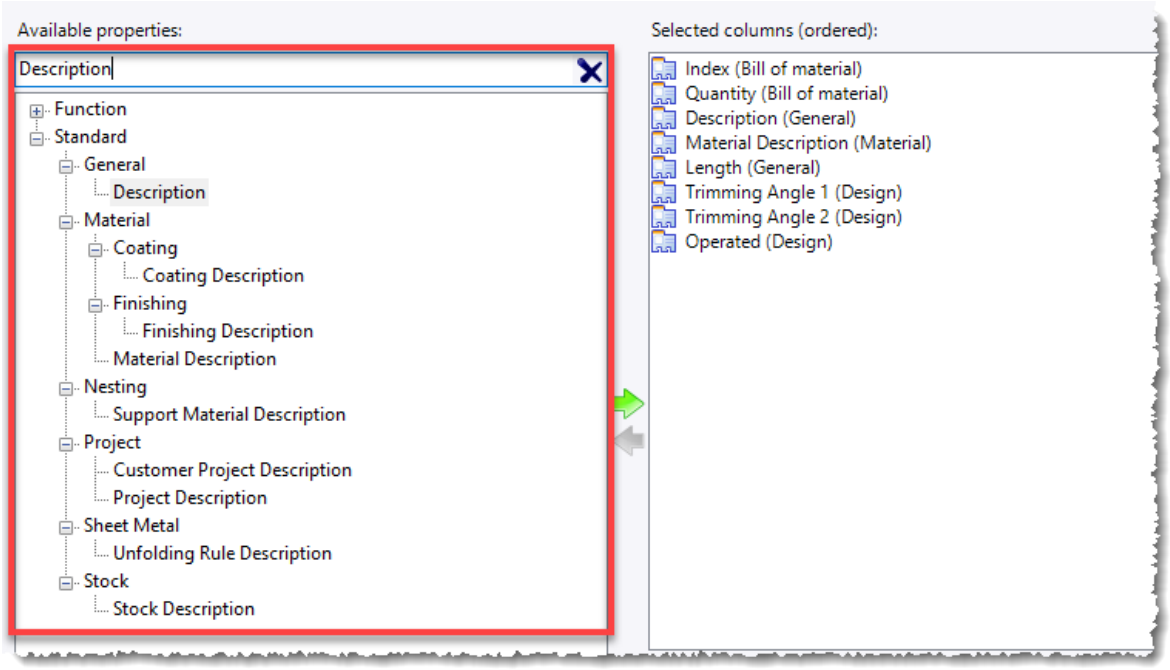
Bill of Materials

Grouping

When defining a column, the **Allow grouping** and **Allow subtotal** options allow you to group the parts with the same property and to count them.
In the drafting document, these groupings and subtotals can be displayed by checking the **Visible** options of the new **Bill of material** section available in the **Detailing** > **Bill of Material Table** command.



In the **Columns** command, the properties are now offered in the tree form and a search field makes it easier to choose the properties to be displayed.



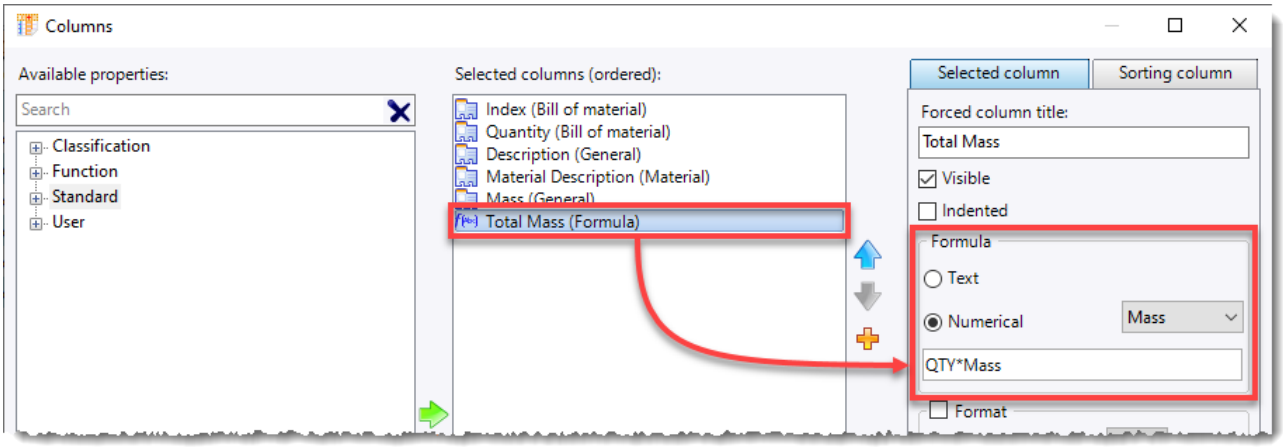
This feature is also available in a filter and search document.

Calculated column

When defining a column, the **Add** option allows you to perform an operation (simple or complex) from the contents of the existing columns. Two types of formula are available:

- **Text**: allows you to "add" the content of several columns (example: **Material** + **Coating**).
- **Numerical**: allows you to make a calculation between several columns (example: **Length** x **Quantity**).

The principle of the calculation formulas is identical to that of creating a parameter.



Example of a column calculating the total mass (unit mass multiplied by the number of parts).

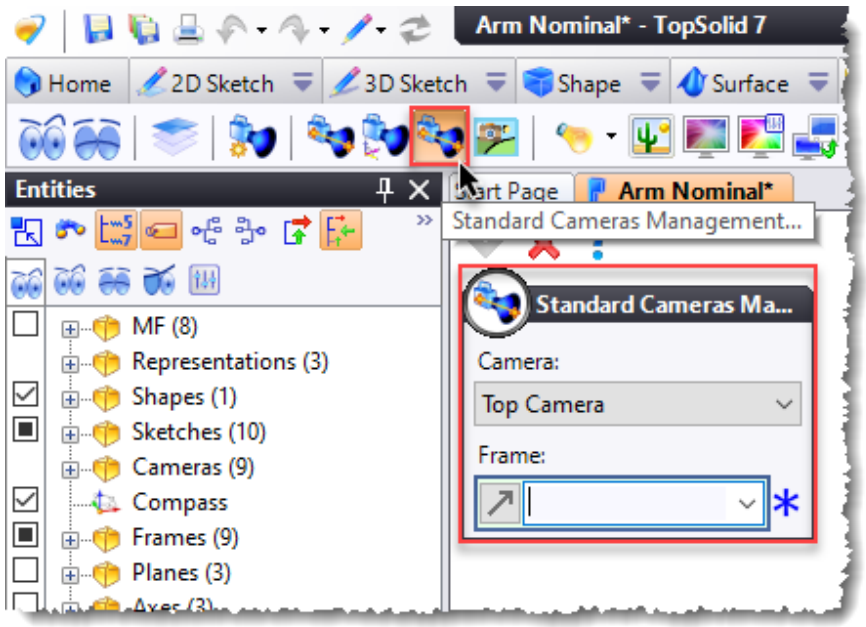
Parameter from project

All the project parameters can now be retrieved in a BOM column.
To do this, you simply have to create a relay parameter (for example, a text relay parameter) in each part or assembly document that retrieves the desired parameter.

Visualization

Camera from a frame

The new **Visualization** > **Cameras** > **Standard cameras management** command allows you to redefine the orientation of standard cameras from a frame.

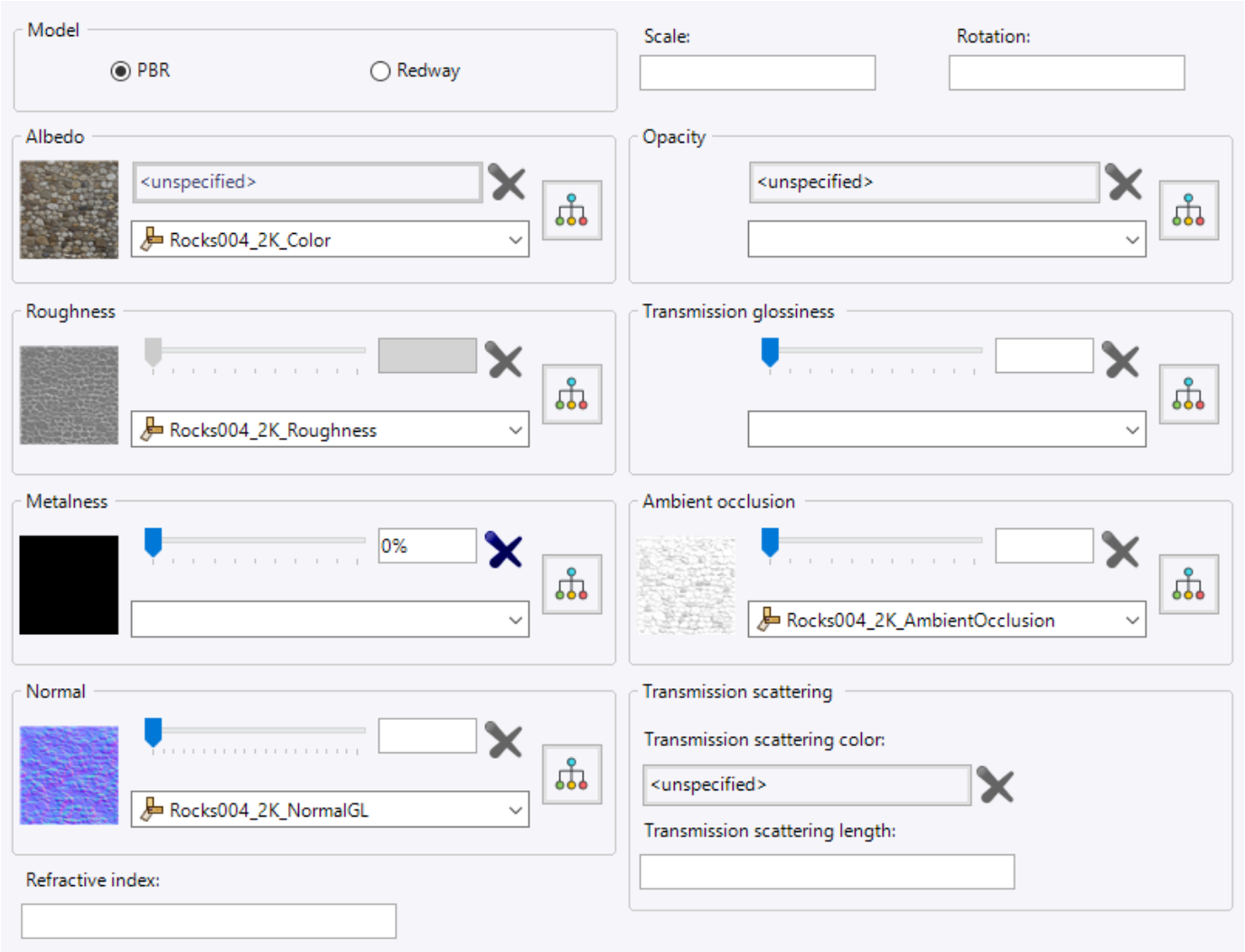


Studio document

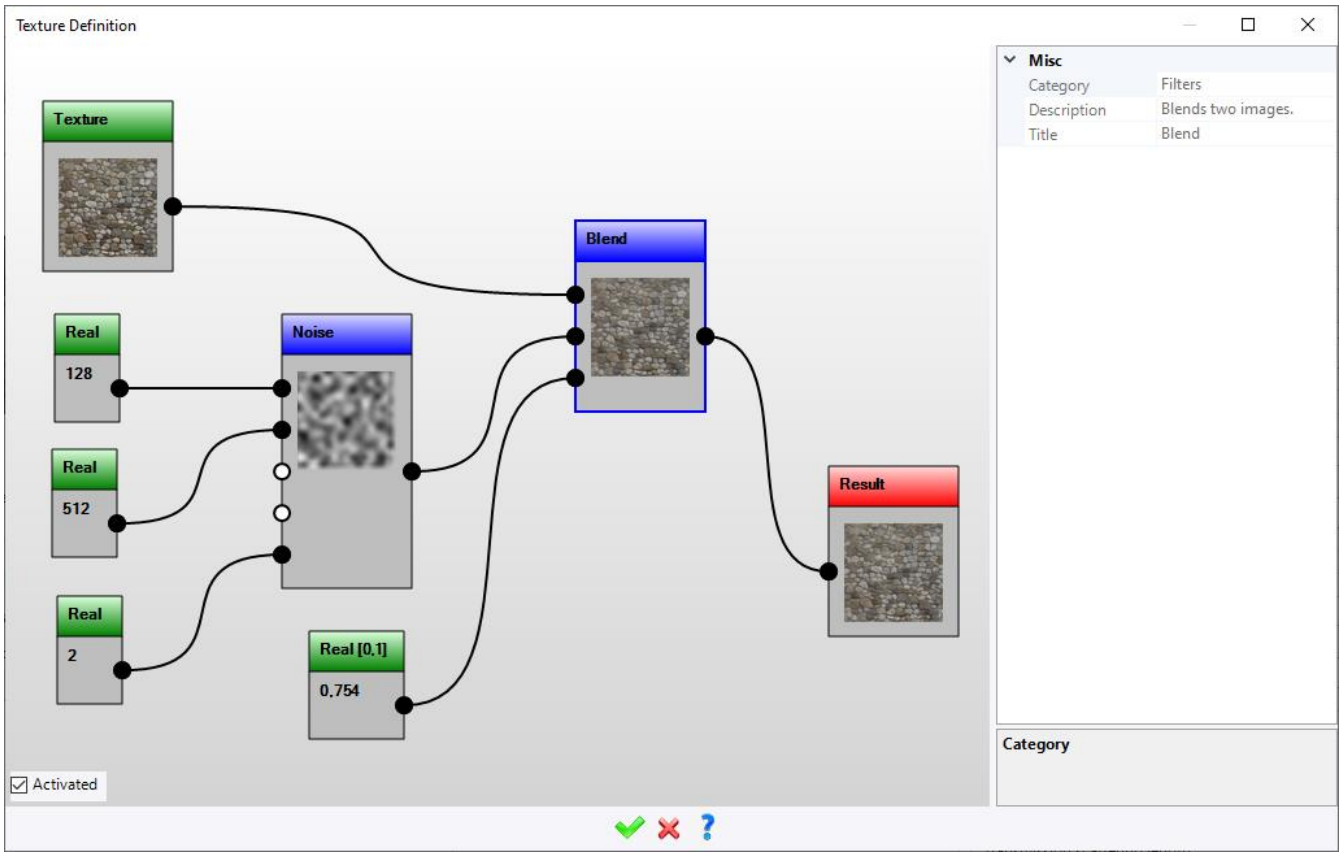
The new **Studio** document has been added to the **Advanced** tab of the document creation dialog box. This type of document has a **Ground** entity by default and allows you to position a part or an assembly document, to define an environment, light sources and cameras in order to calculate an image.
The main advantage of this type of document is that you can change the material and coating of the parts without checking them out of the vault. For this purpose, the **Parts Material and Coating** and **Face Coating Management** commands are available in the **Tools** menu.

Physically Based Rendering (PBR) material and coating

You can now define **PBR** materials or coatings. They are defined by their **albedo**, **roughness**, **metallicity**, **normal**, **opacity**, **transmission glossiness**, **ambient occlusion** and **transmission scattering**. Each feature (available as a category) is defined by a **Texture** document.



Moreover, a graph editor allows you to adjust some parameters of the different features in order to add more granularity to a material for example.



Example of adding noise to a texture.

Basic texture

Result

Rendering camera

The new **Visualization** > **Rendering Camera** command allows you to create a conical camera by indicating the position of the eye, the point of view, the type of lens and the features of the image to be calculated (size, rendering scale). This type of camera also allows you to use a 2D background image to superimpose a 3D design.

Rendering style and post-processing style

The **Rendering Style** and **Post-Processing Style** commands have been added to the **Visualization** menu. They allow you to predefine styles that can then be used when calculating an image in realistic rendering.

Realistic rendering

The **Realistic Rendering** dialogue box has been modified:

- The preview dialog box is displayed as soon as the command is launched.
- The command can be launched using the **R** shortcut key.
- The **Styles** tab has been added, it allows you to choose the rendering camera, the rendering style and the post-processing style.
- In the **Post-Processing** tab, the **Denoise** option has been added. It allows to remove the noise caused by the lighting and to perform a kind of "smoothing".

*Example of an image without the **Denoise** option.*

*Example of an image with the **Denoise** option.*

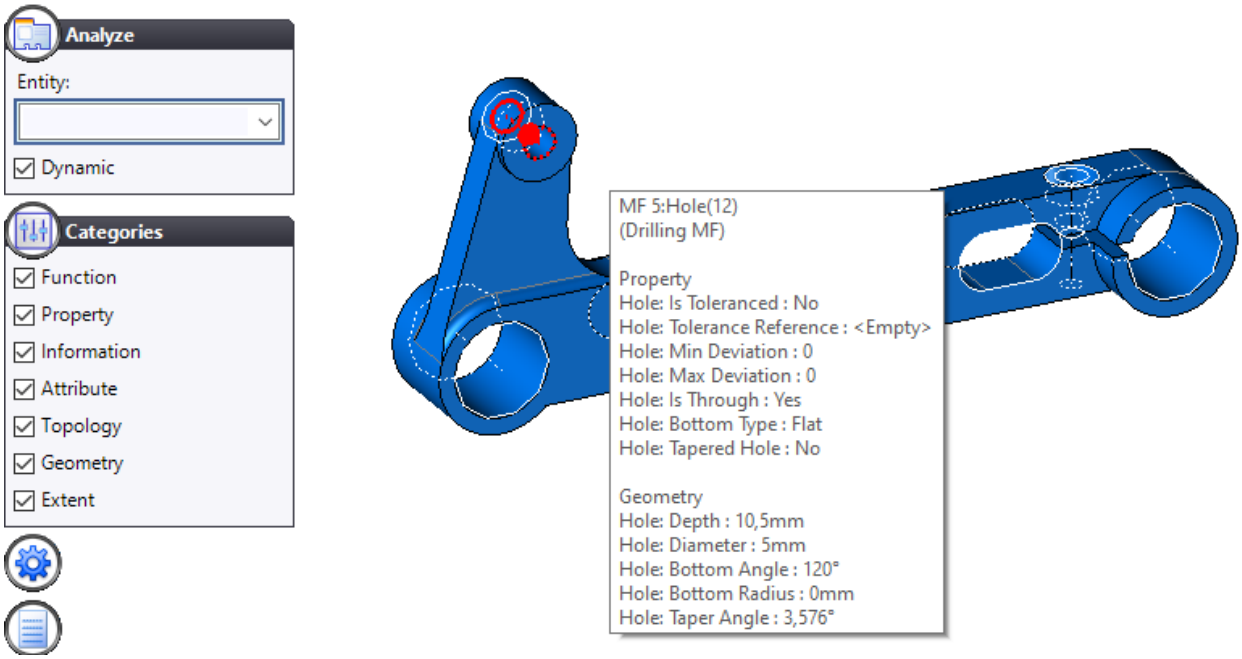
Analysis

Geometry analysis

In non-automatic mode, the **Point**, **Axis** and **Plane** modes now allow you to create entities on-the-fly.

Analysis

The details of the MFs as well as the annotations (tolerancing, roughness, etc.) are now displayed on the label.



Construction

Point clouds

When selecting a plane cloud, the **Search planes** contextual command has been renamed **Search geometries**. It can now be used to search for planes, cylinders, axes or frames.
In addition, the **Edit the point cloud** contextual command allows you to delete, separate or add a batch of points from the cloud.

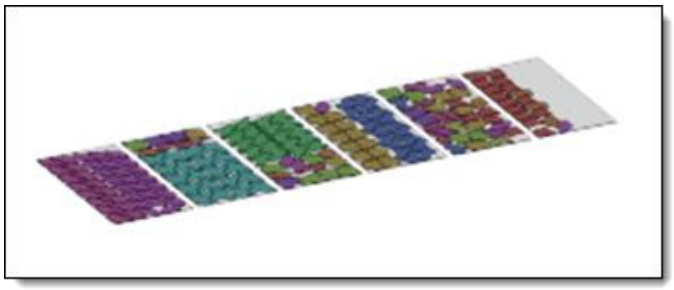
Copying and pasting parameters

You can now copy and paste the parameters from one document to another, whether they are in the same project or not.
The **Copy** command is available either by right-clicking on the selected parameters or from the **Parameters** folder (in this case, all parameters are copied).

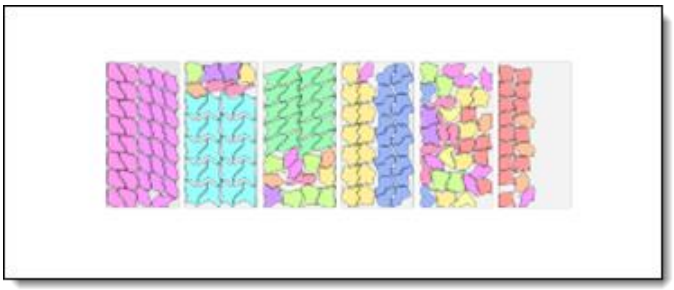
Nesting

Document preview

In order to make the preview of a nesting document more readable, especially in case of many supports, the preview is now automatically created in **top view** instead of perspective view.
In addition, you can configure the appearance of this preview in a nesting template document via the new **File** > **Document Preview** command.



*Preview of a nesting document in **7.15**.*



*Preview of the same document in **7.16**.*

Performance improvement

An important improvement regarding calculation time has been added to the nesting document. This improvement is very visible in the case of parts with many faces and/or many holes.
The improvement deals with the opening time of a nesting document and the nesting calculation.

Automatic nesting

Increasing check options

Many checkings have been added, blocking the creation of an automatic nesting in case of problems.

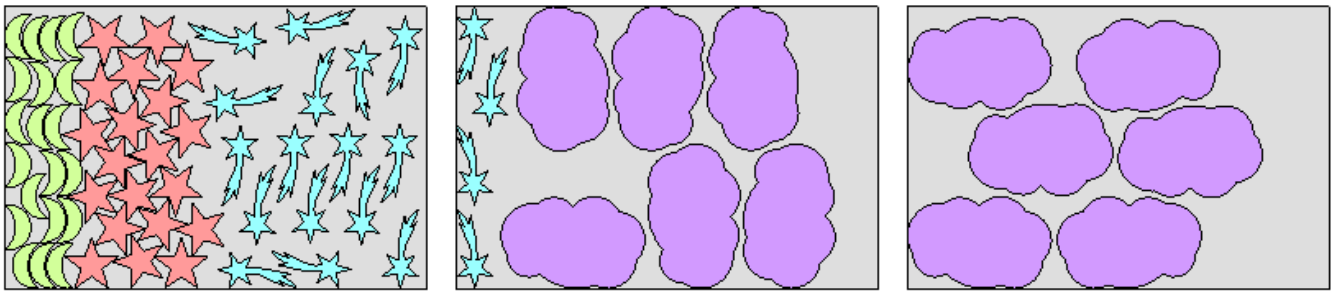
- Verification of the **documents** to be nested: the thickness and material compatibility between parts and supports is verified.
- Verification of the documents generated by the nesting: for example, for the **Sheet Metal Cam** documents, we check if there is a template document and if it has a machine.
- In **Sheet Metal Cam**, flip is not possible. An error message is displayed on the part line if the flip is selected.

Manufacturing index

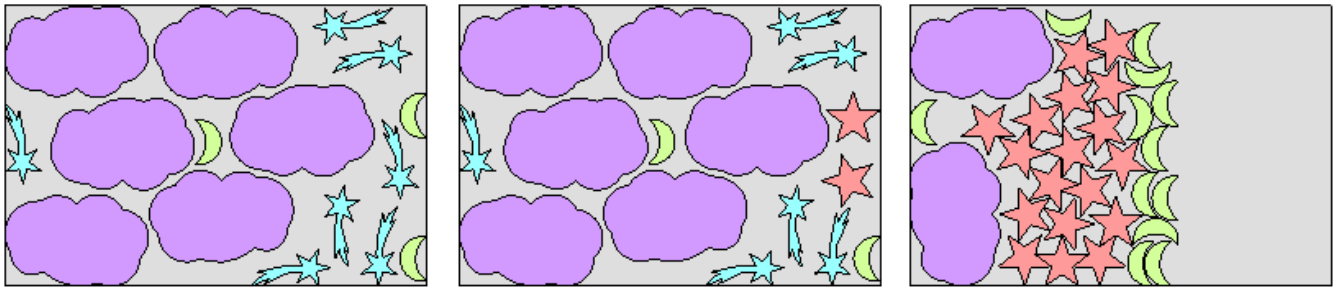
If the nesting is created from a bill of materials exposing the **Manufacturing index** column, these are now correctly associated with the parts in the nesting document, as in the parts of the assembly documents produced by the nesting.

Large priority parts

A new **Large priority parts** option in automatic nesting allows you to calculate the priority of the parts to be positioned from the largest to the smallest enclosing box in the **Parts** tab.



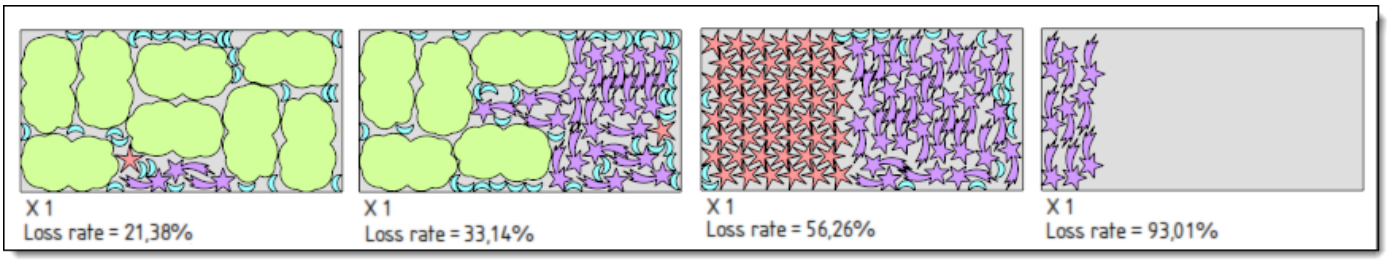
In this example, the priorities are added manually: Yellow 1, pink 2, blue 3, purple 4 part.
Not all the purple parts could be positioned.



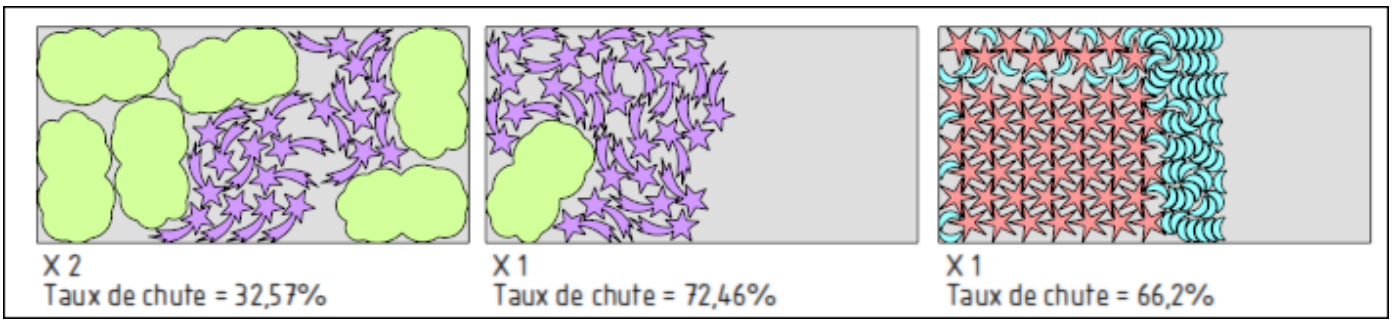
In this second example, the priorities are added automatically in order of size. All the purple parts are first positioned, then the blue ones in the remaining space, then the pink ones, and finally the yellow ones.

Maximum number of part references by support

This new option available in the **Strategy** tab consists of imposing a maximum number of part references per support to help and therefore save time in unloading the batches of parts corresponding to a command, for example.



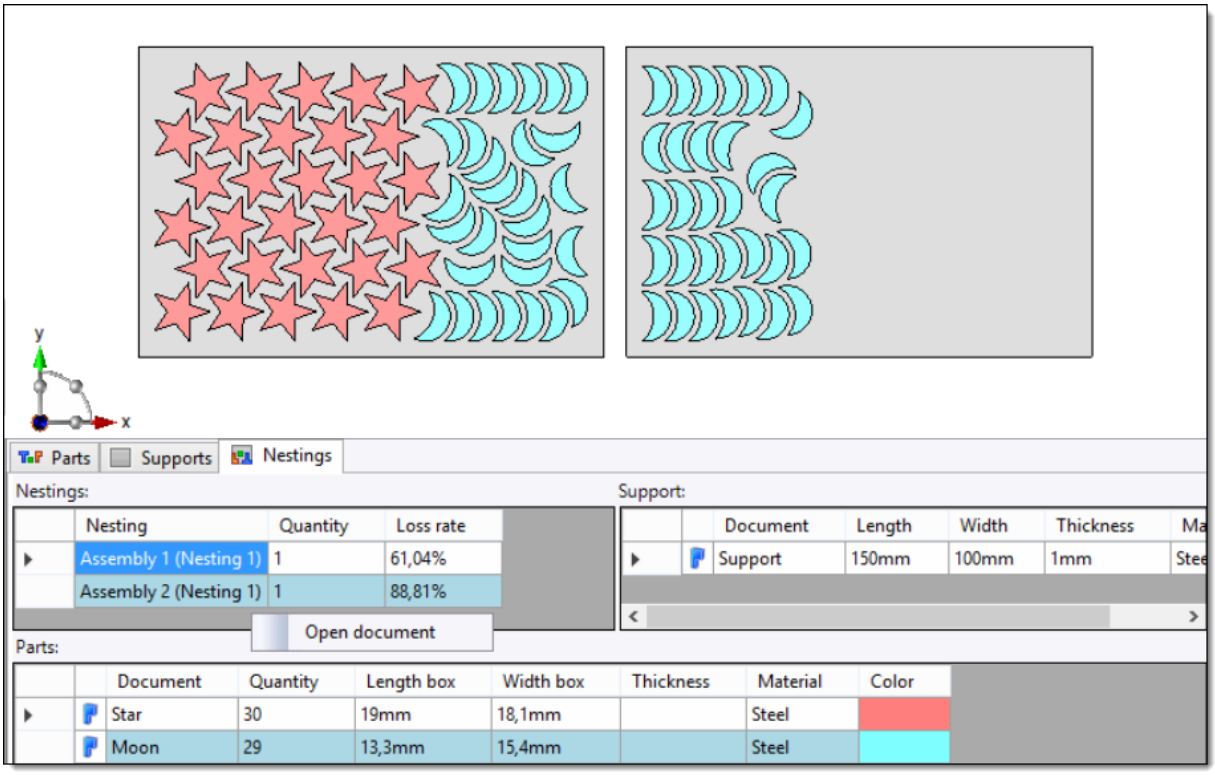
In this example, no limitation of part references is added.



In this second example, a maximum of two part references are allowed per support.

Nesting tab in the nesting results display panel

The **Nesting** tab, only available before via automatic nesting editing, is now displayed in the results tab.
From this tab, you can open each of the documents produced by the nesting.



Empty supports

You can now create a manual nesting part, in empty supports. This command allows you to choose the desired support, as well as the desired nesting verification options.

Manual nesting: Dynamic control of the position

A new **Dynamic control of the position** option allows you to choose to check for collisions with parts already positioned in the support either:

- **Dynamically**: the part preview is only visible if you can position the part.
- **Non-dynamically**: the part preview is always visible at the "end" of the mouse. The verification is only done on click.

In the latter case:

- If you can position the part, the move or copy modifications are made.
- If you are outside of the support or if the position is not valid (no place available on the support or too far from a valid position), nothing happens.
- If you are close to a valid position, the parts are positioned at the nearest valid position.

Nesting process

In the **Work** document, a general **Nesting (TopSolid'Design)** process for the parts is available. The process is available with the **Imbrication** license.
This process automates the creation of nesting documents, by creating **automatic nestings** from part, unfolding or bill of materials documents.
The process groups the **parts/unfolding** by compatible **materials/thickness** and will produce as many nesting documents as groupings found.

Tube Nesting

Automatic tube nesting: Partial common cut

In the **Strategy** tab of the tube nesting, you can now define the matching criteria for partial common cuts thanks to the two added parameters:

- **Minimum length proportion per part**: each portion in common cut must cover at least x% of the perimeter
- **Minimum length proportion**: the total length in common cut must cover at least x% of the perimeter

These values are given in percentage.

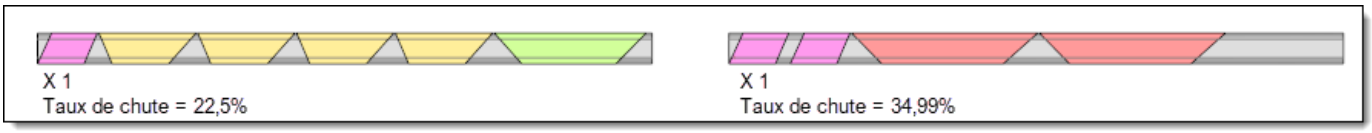
Nesting from a bill of materials

You can now add the parts list with the relative quantities of parts.
You can also change the quantities of the parts with an overall multiplicity of the assembly.

Bar Nesting

Automatic bar nesting

This new command allows you to nest profiles in a support bar of the same section type. After defining the **profile nesting characteristics**, for both profiles and support bars, the parts will be automatically positioned on the bars to optimize the overall loss rate of the bar required to nest all the parts.



In this first version of the command, it is the profile enclosing boxes that are nested, without management of the possible flips and/or cutting angles.

Profile nesting characteristics: part as a support

A new **Usable as support** option allows you to define a profile family as nesting part and as support.
Thanks to this new option, you no longer have to duplicate the family and its generic to identify the supports. In addition, the type of profile to be nested is now defined by its possible degrees of freedom in rotation, and no longer by a type of section.

Bar nesting process

In the **Work** document, a **Nesting (TopSolid'Design)** process for profiles is available.
This process automates the creation of nesting documents, by creating **automatic bar nesting** from part or bill of material documents.

Tools

Mirror from a document

The new **Tools** > **Mirror Definition** command allows you to define a document as a mirror of another document. This command will be mostly used when importing a right-hand part and its left-hand part.

Enclosing block

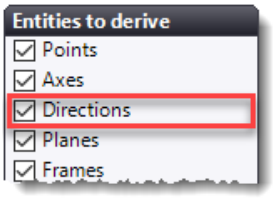
The **Enclosing block** command has been added to the **Tools** menu. As in a part document, it allows you to calculate the enclosing box of a representation of the assembly.

Joined parts

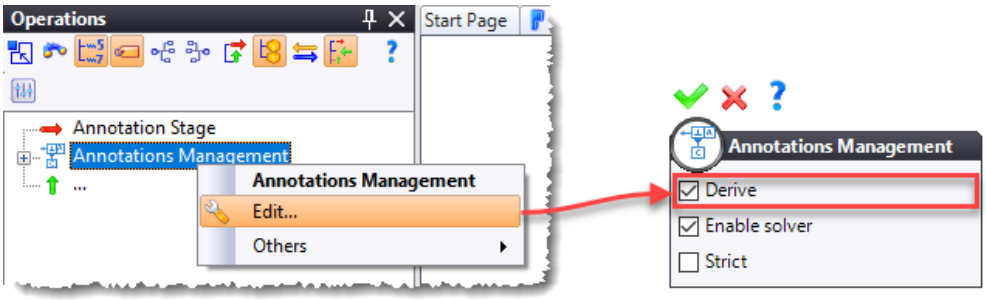
The new **Tools** > **Joined Parts** command is used to declare parts as glued or welded. When including a sub-assembly containing joined parts, if an assembly operation (drilling, trimming or pocketing) is applied, the shapes are made modifiable automatically and the operations on shapes are only generated in the assembly. As a result, there is no synchronization between the documents.

Derivation

When deriving a document, you can now derive **Directions** entities.



The annotations created in the document as well as the imported annotations can be derived by editing the **Annotations Management** operation (or the **Imported Annotations Management** operation) and checking the **Derive** box.



Publishing

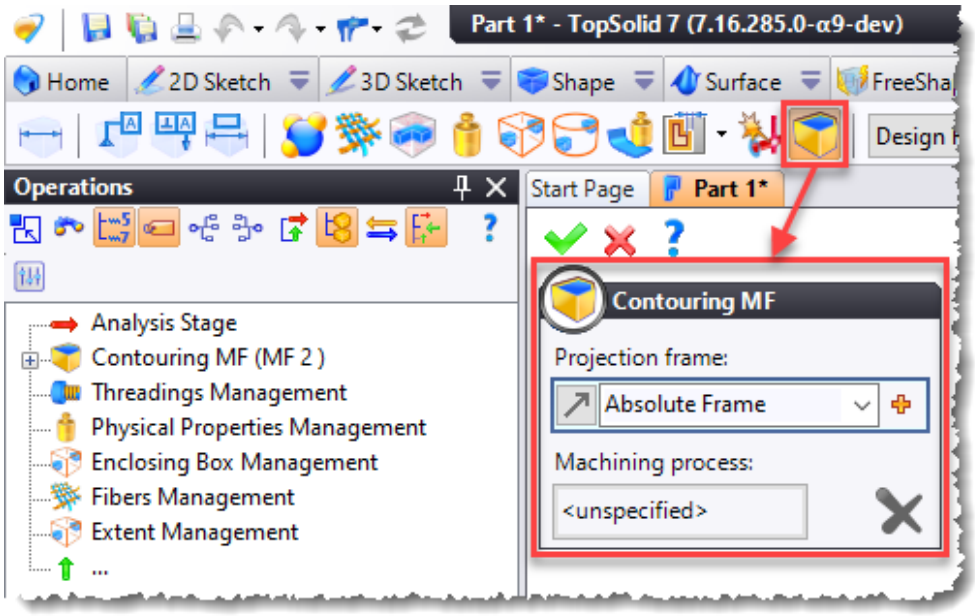
The **Publish Block Envelope** command has been added to the **Tools** > **Publishings** menu. This avoids, for example, publishing the six planes of the block envelope.

Level reference

The new **Tools** > **Annotations** > **Level References Management** command allows you to define the reference of heights (**NGF** and **Project**) of a part or an assembly from a frame or an IFC file. This reference will be used when creating a level note (see also the **Drafting** > **Level note** paragraph).

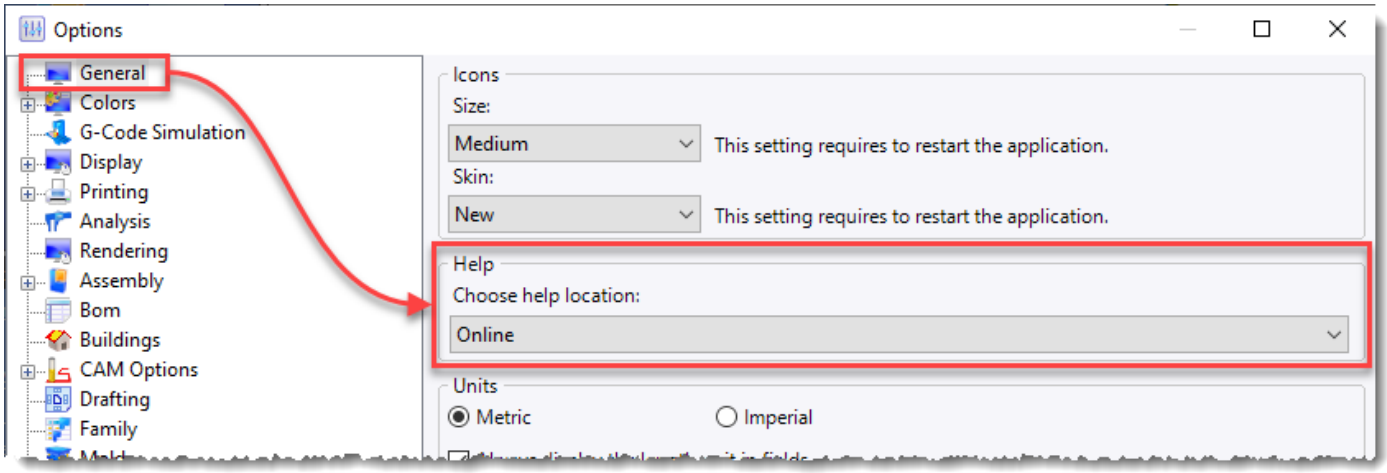
Contouring MF

The **Tools** > **Contouring MF** command has been added. It allows the machining to be prepared for **TopSolid'Cam** by indicating a frame and possibly a machining process.



Online help

From now on, the online help is also published on the Internet, which makes it possible to update without waiting for the release of a service pack. For users who do not have access to the Internet, the local version is always available in the **TopSolid** installation folder and the **Tools** > **Options** > **General** section allows you to choose to display either the local or the online version.



TopSolid'Fea

Selection

The root face selection option has been added to the **Uniform Pressure**, **Surface Load** and **Hydrostatic Pressure** commands.

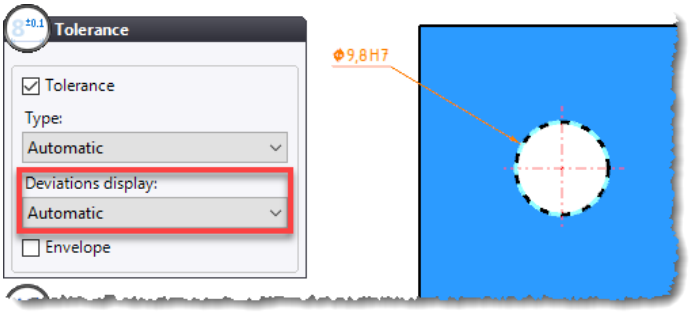
Drafting

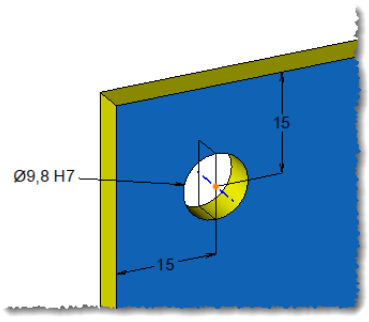
Draft bundle

In a draft bundle, when performing automatic nesting, the **Nest one drafting by page** option allows you to automatically create a new page for each plane of the nesting.

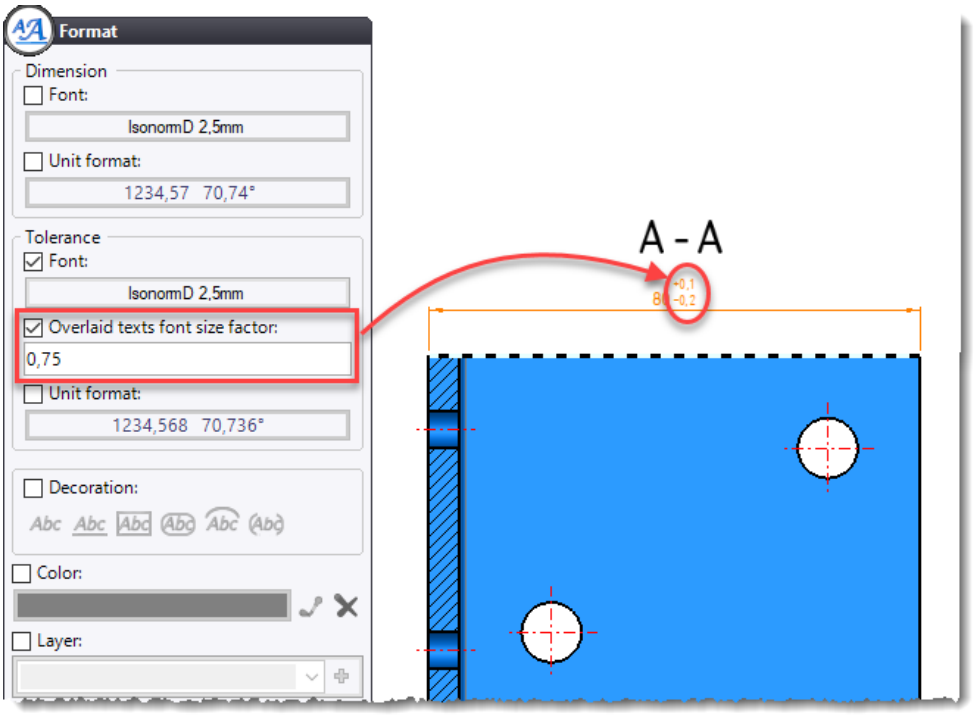
Dimensioning

When selecting a toleranced hole, the **Tolerances** section now offers an option to automatically display this tolerance as it appears in the 3D document.

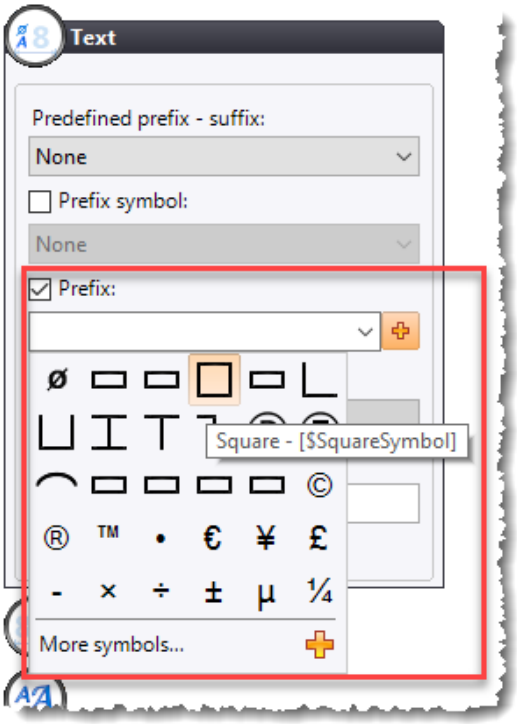




In the **Format** section of the dimensioning dialogue box, the new **Overlaid texts font size factor** option allows you to assign a scale factor to the height defined in the tolerance font.

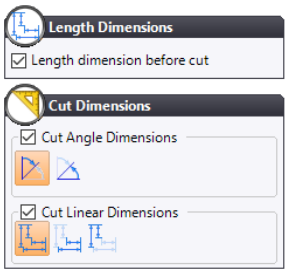


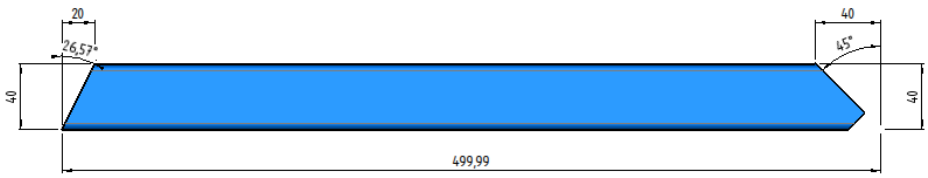
In all the fields allowing you enter a text in a dimension or a dimension style, the " **+** " button now displays a selection panel of the usual symbols. For other symbols, this panel also offers access to the symbols in the Windows special characters table.



Automatic dimensions

The **Detailing** > **Automatic Dimensions** command has been enriched to better manage the dimensioning of the profiles.
The **Length Dimensions** section allows you to add only one length dimension corresponding to the length of the profile before cut.
The **Cut dimensions** section allows you to either enable the length and/or angle dimensions or not.

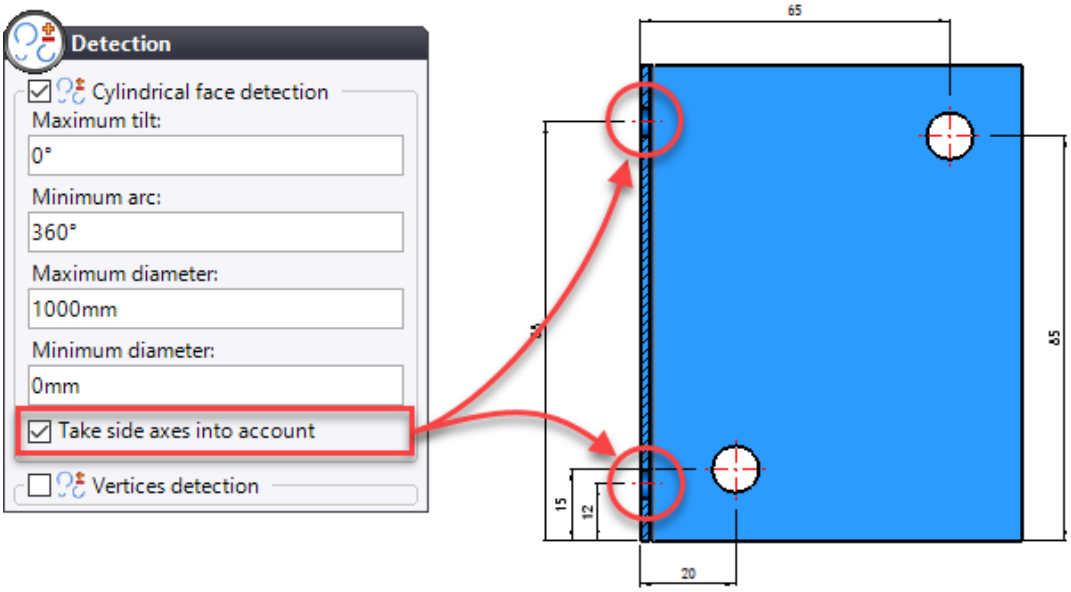




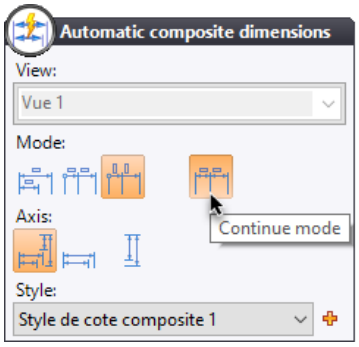
In addition, imported profiles (using the recognition option) as well as derived profiles are now compatible with the automatic dimensioning.

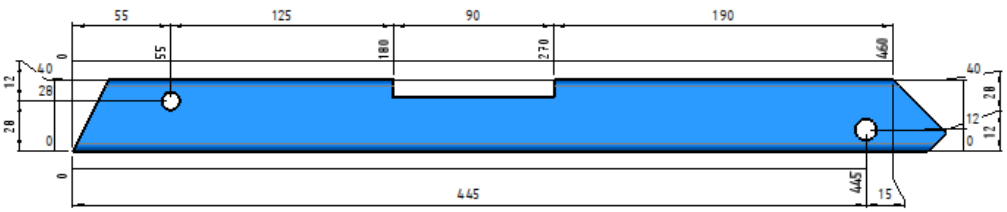
Automatic composite dimension

The new **Detection** > **Take side axes into account** option available in the **Detailing** > **Automatic composite dimensions** command now allows you to detect drillings not represented by a circle. However, you have to create the automatic axes on the view beforehand.



In addition, the **Continue Mode** has been separated from the other modes, it can now be combined with any one of them. When two modes are combined, the **Distance in multiple mode** advanced option allows you to manage the distance between the dimension lines. If the distance is zero, the two dimension lines are superimposed.





For more complex parts, the **Continue per side** option has been added in the **Continue Mode** section, it allows you to create the dimensions in relation to the sides of the part.
In addition, the **Detection** and **Advanced Options** sections have been enhanced with new options:

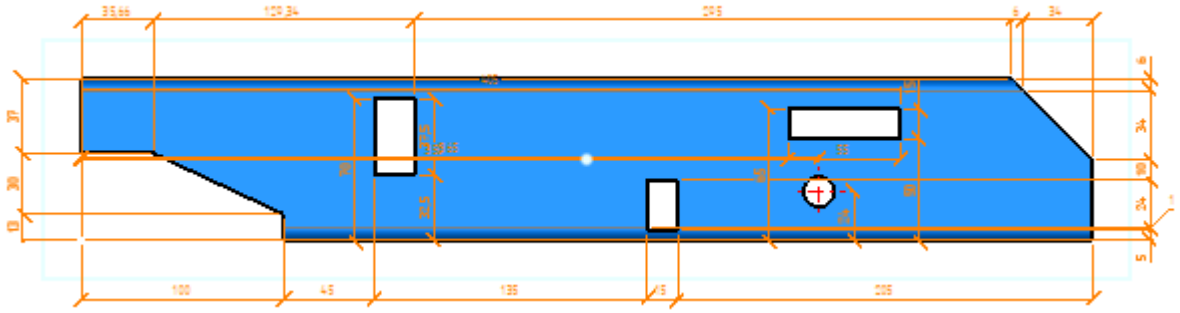
Detection:

- **Cylindrical face detection**: allows you to take into account drillings, circular pockets, etc.
- **Vertices detection**: allows you to take into account the vertices of the segments of the view.

Advanced options:

- **Use auxiliary axes for isolated geometries**: This option creates axes near isolated geometries (closed geometries included in the external contour of the part) on which the dimensions of these geometries will be hooked.

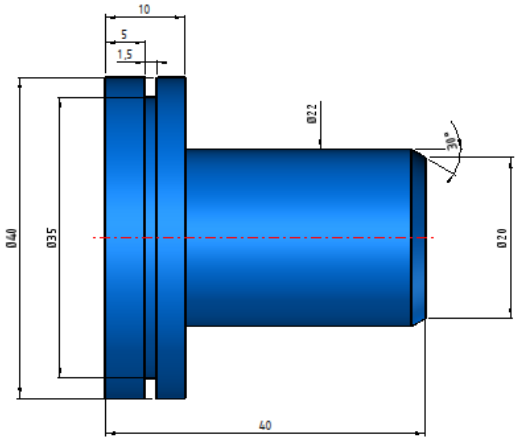
  This option has two sub-options:
  - **Group isolated geometries**: This option allows you to offset the dimensions on the main axis when there are no overlapping dimensions.
  - **Use two axes for isolated geometries**: This option allows you to distribute the dimensions on each side of the isolated geometry.



Example of automatic dimensioning on a profile.

Projected annotations

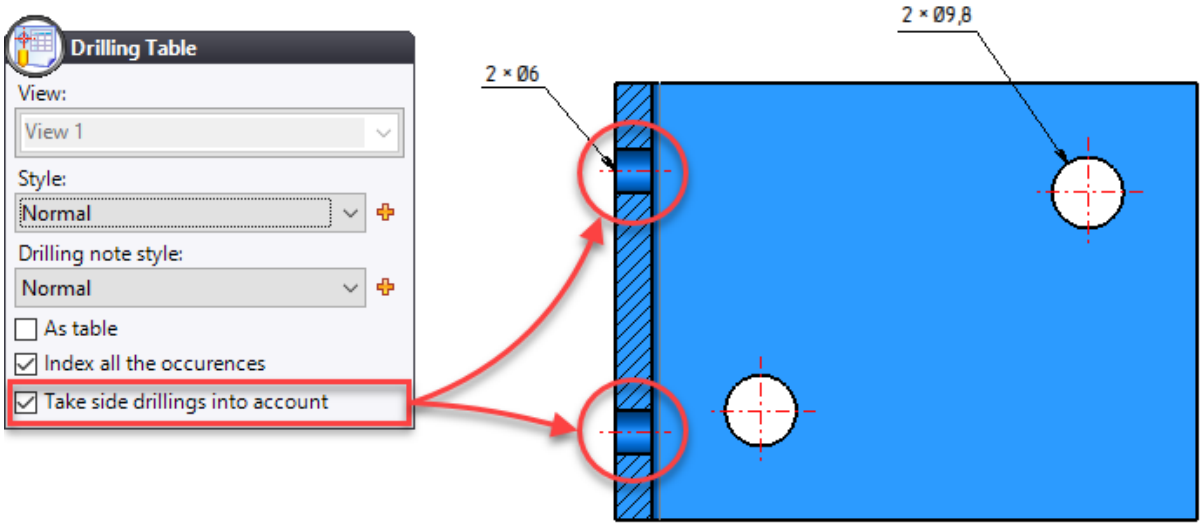
Half part dimensions are now available in the **Detailing** > **Projected Annotations** command.
The type of dimensioning depends on the display mode (half part, large diameter or whole part) defined in the 3D document using the **Display** contextual command on a half part dimension.



Example of projected annotations with half-part dimensions.

Drilling table

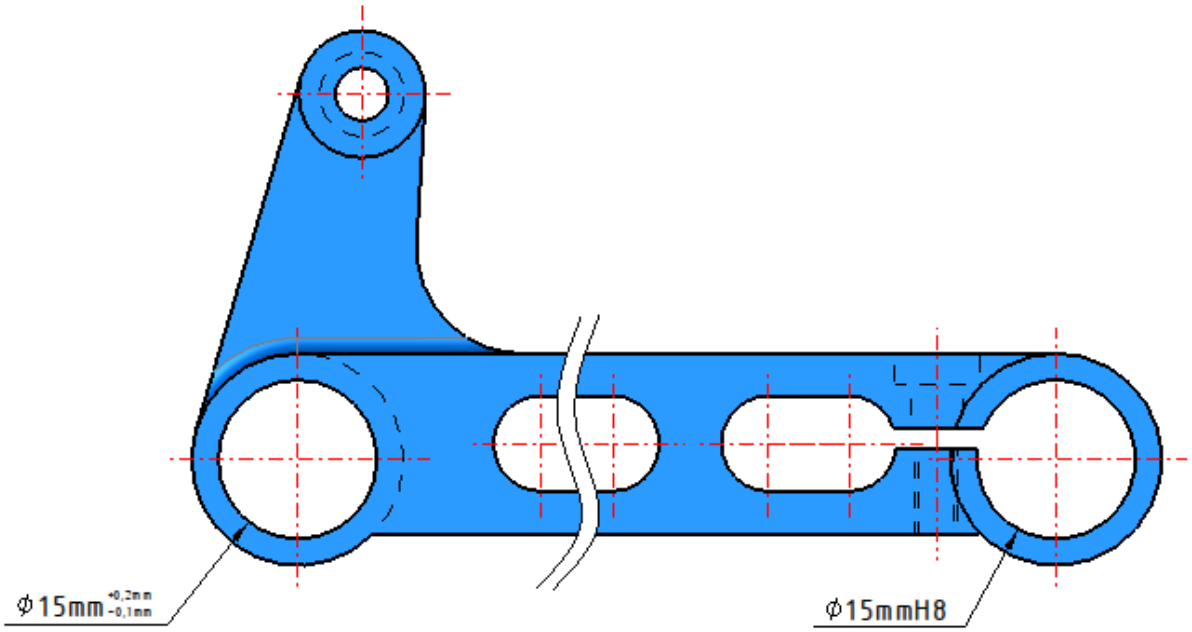
In the **Detailing** > **Drilling table** command, when the **As table** box is unchecked, the new **Take side drillings into account** option allows you to detect side drillings (not represented by a circle). Unlike automatic composite dimensions, you can create the automatic axes on the view beforehand.



Drilling note

When creating a drilling note or a drilling note style, you can now choose to display:

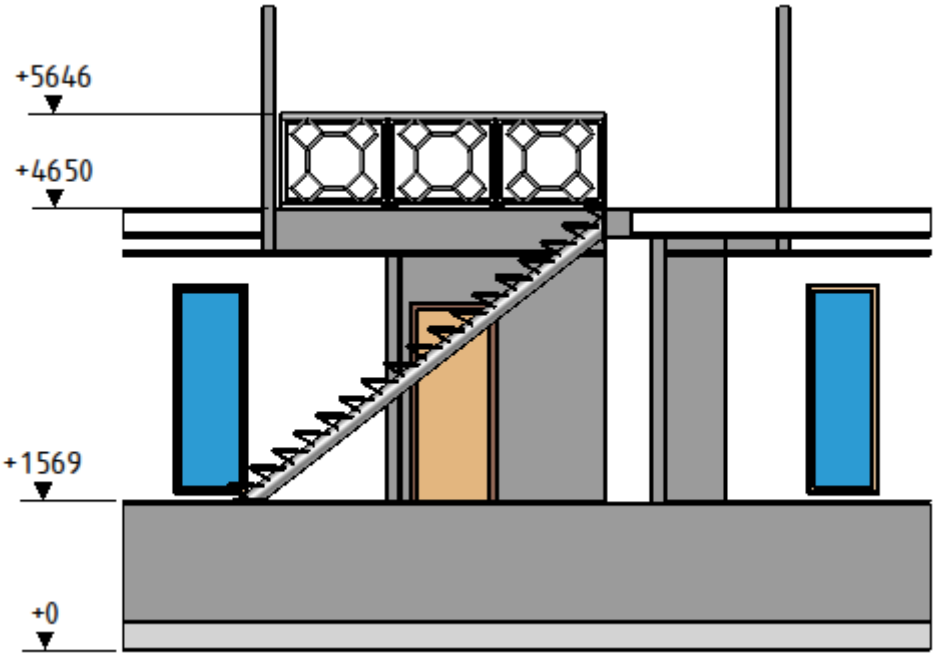
- the diameter effective value (**Nominal**, **Minimum**, **Average**, **Maximum** or **Automatic**);
- quality symbol;
- tolerances;
- effective value symbol.



Example of drilling notes displaying the tolerances of the 3D document.
The same style allows you to create the two drilling notes.

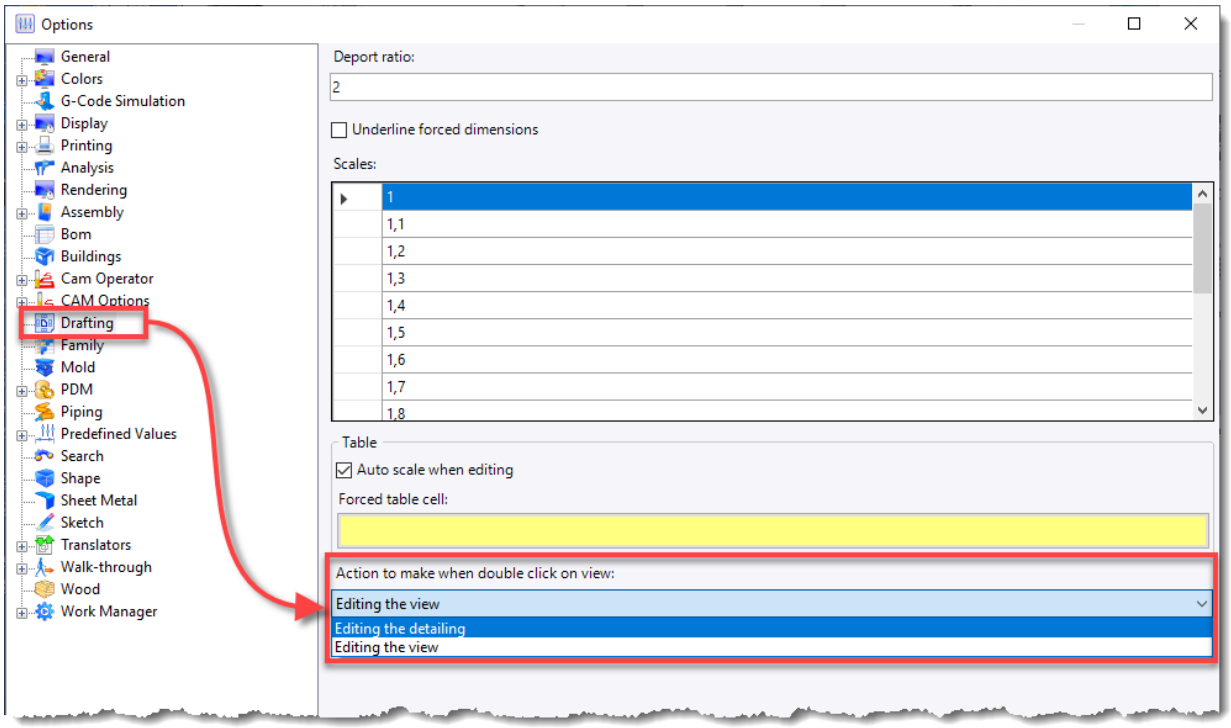
Level note

The new **Detailing** > **Level Note** command allows you to identify the different altitudes of a part or an assembly. The height reference can be defined either locally in the drafting document or by using the **Tools** > **Annotations** > **Level References Management** command available in a part or assembly document.



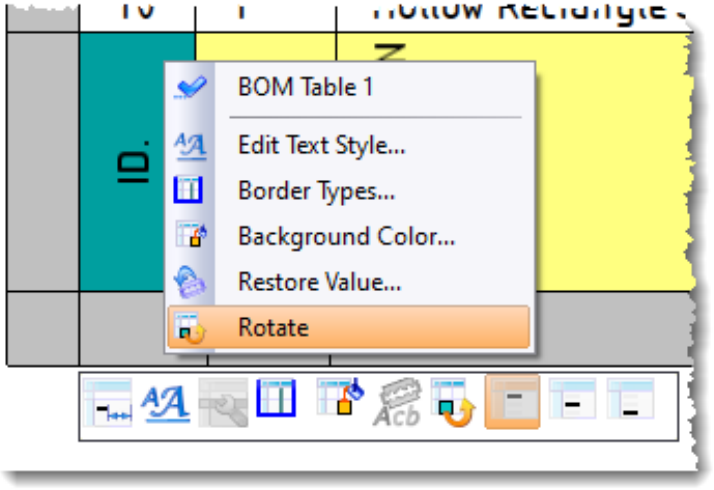
Editing the detailing

In the **Tools** > **Options** > **Drafting** section, you can now choose the action to be performed when you double-click on a view. You can either edit the view (default) or edit the view's detailing.

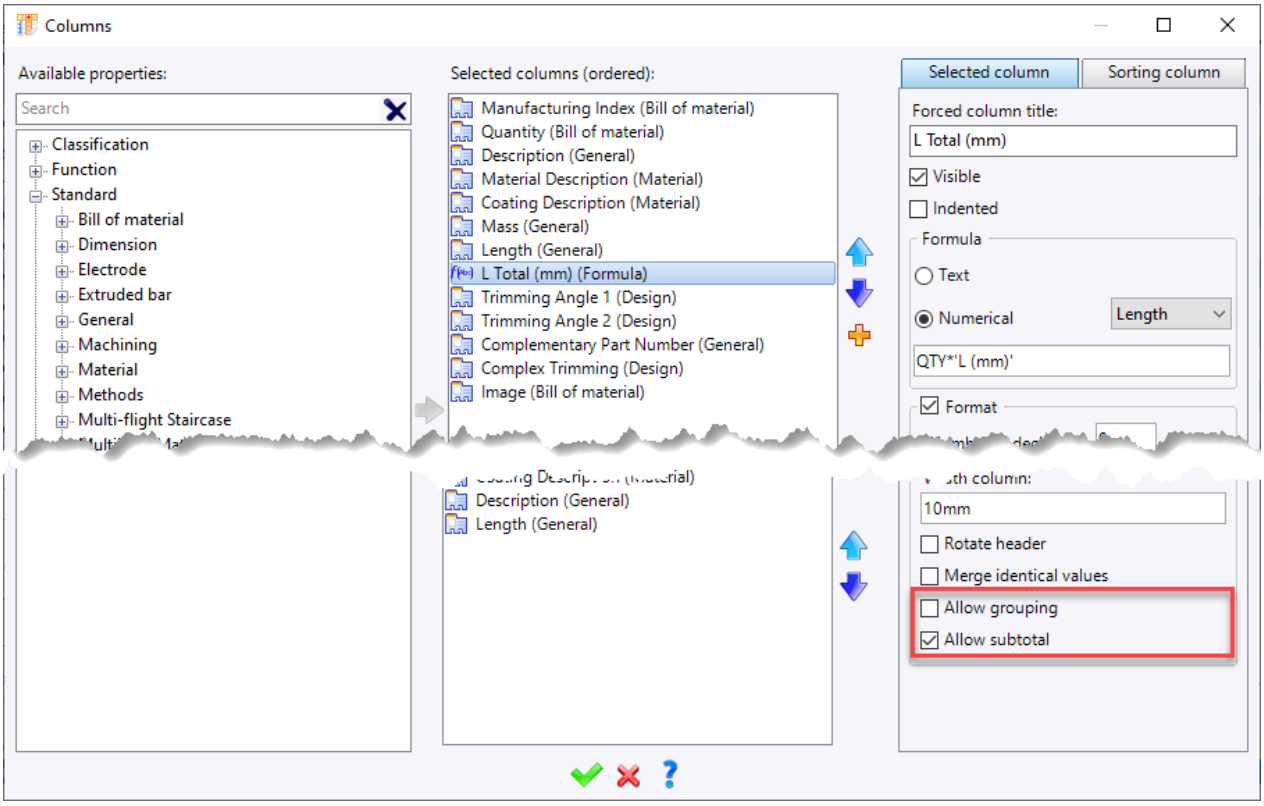


Tables

The content of the headers of the automatically generated tables (BOM table, drilling tables, etc.) can now be modified and the **Rotate** command is also available.

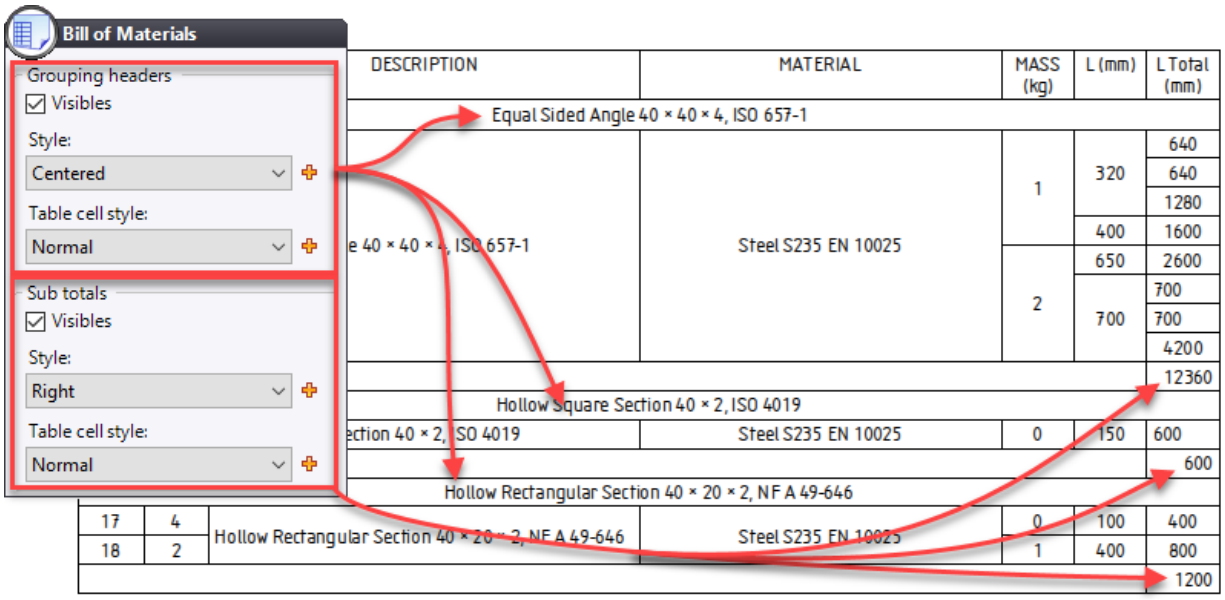


When defining the columns of a BOM, you can now define a column on which **TopSolid** will apply a grouping and possibly to create the subtotal of certain properties.



Example of definition of BOM columns with subtotal.

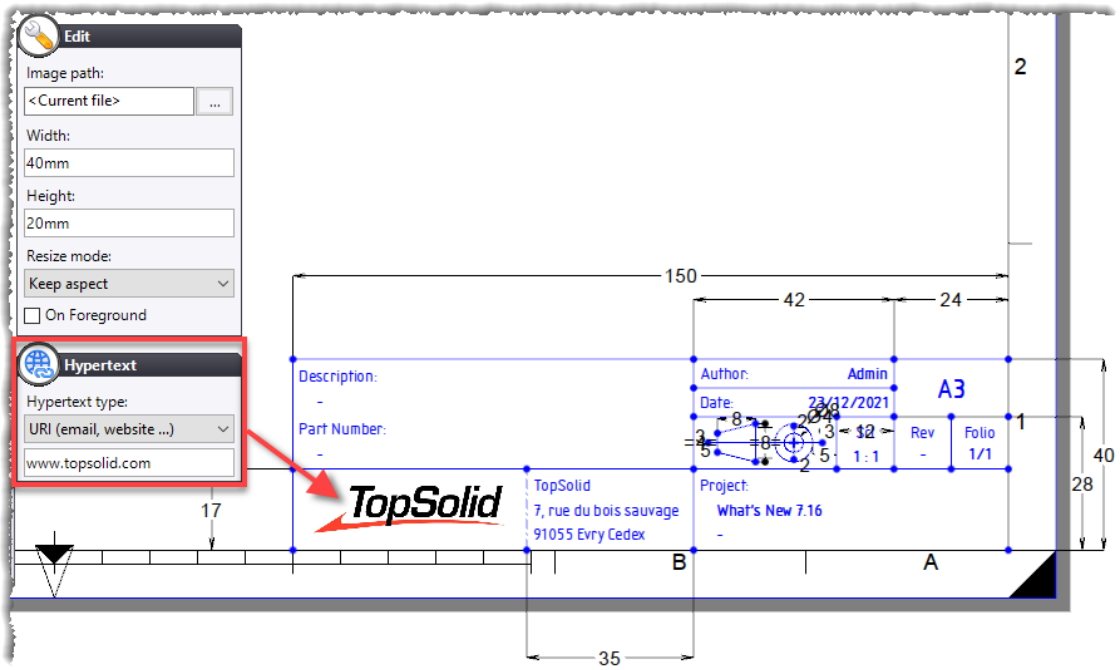
In the drafting document, the new **Bill of Materials** section in the **Detailing** > **Bill of Material Table** command allows you to choose the display of groupings and sub-totals.



Bill of materials table with grouping and sub-totals.

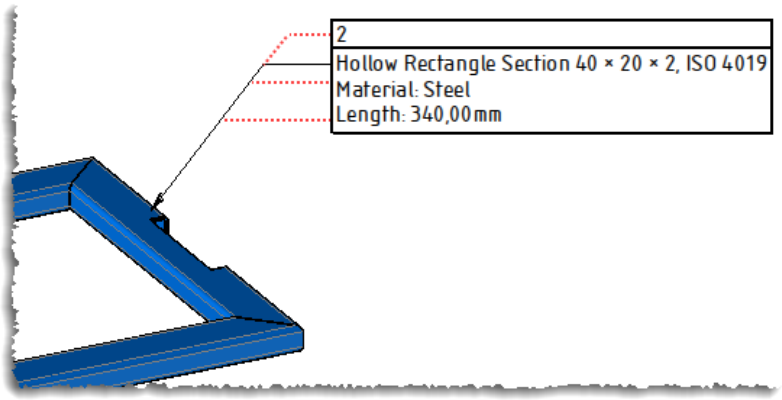
Hypertext link

The **Hypertext** section has been added to the **Sketch** > **Image** command, this allows you, for example, to put a hyperlink on the image of a title block.



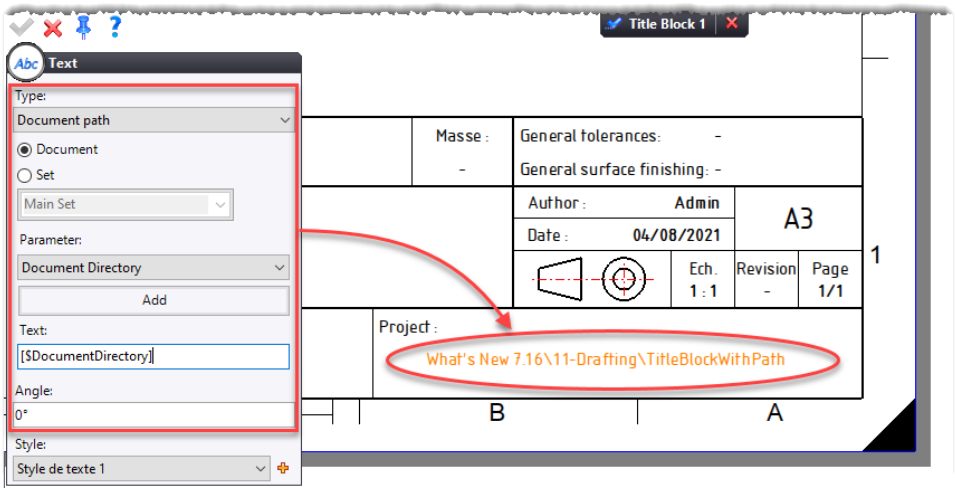
Bill of material index

You can define the starting points of the arrows of the BOM indexes via the **local attributes** setting (as it is already possible to do for rich texts).



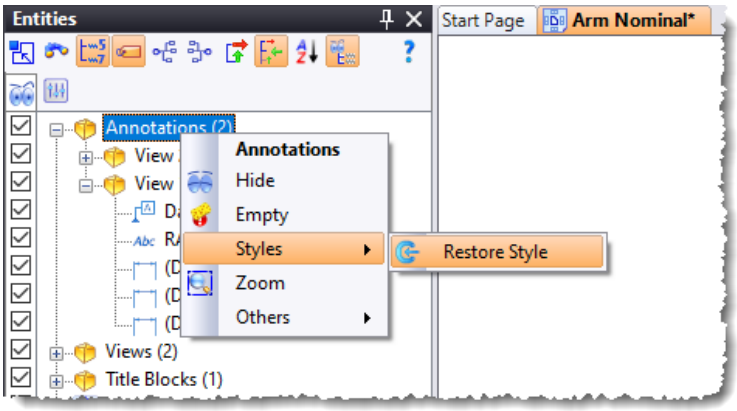
Title block

The **Sketch** > **Text** command now allows you to retrieve the document tree with or without the name and project tree.



Scraping removal

When several annotations or views have been modified without changing their styles, the new **Restore Style** command allows you to undo these modifications. This command is accessible from a selection of annotations in the graphics area or from the Entities tree.



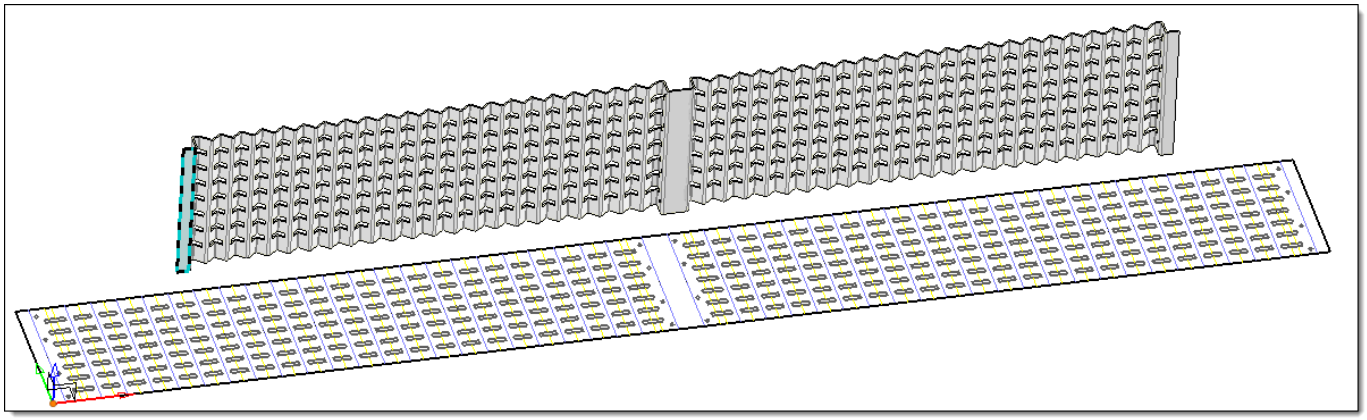
Unfolding

Performance improvement

An important improvement regarding calculation time has been added when creating unfoldings. This improvement is very significant in the case of parts with many bends and/or many holes.
Multiple unfoldings also benefits from this improvement.

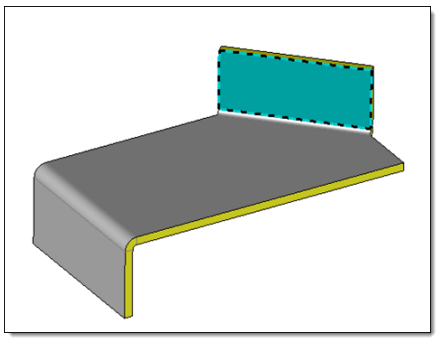
Processing parts with many bends

A memory limitation of the software did not allow you to unfold the parts with many bends.
This limitation has now been extended to parts with more than 100 consecutive bends.
For parts exceeding this bend limit, a message now warns the user, suggesting that he tries to change the original face to balance the unfolding tree.

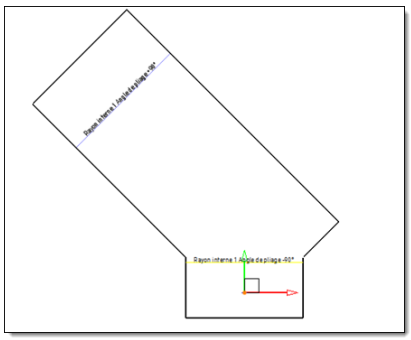


Orient along longest length

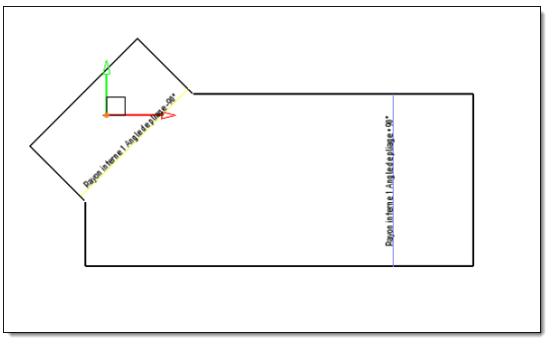
If this new option is enabled, an automatic rotation will be applied to the unfolding so that the longest length of its enclosing box (minimum) is oriented along **the X axis of the destination frame** of the unfolding.



Example of unfolded created with the blue face as the selected reference face.



The unfolded is oriented along this face without enabling the option.



*With the **Orient along longest length** enabled, the unfolded part is automatically repositioned along the **X axis of the destination frame**.*

Work Document and Work Manager

Templates

When performing processes, the templates used in a work document are now copied to a **Templates** folder under the node of the generated **work** document.

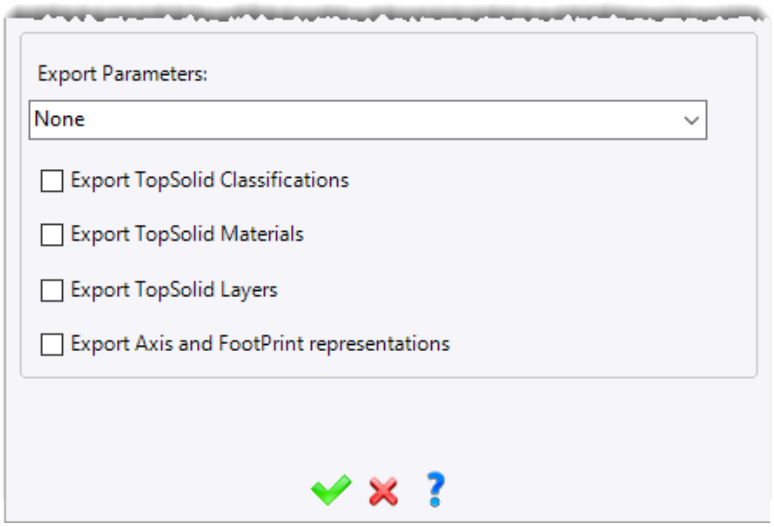
Various improvements

In the list of available processes, the processes are grouped by application.
In the **Work Manager** dialog box, the **Undo Filters**, **Select All** and **Deselect All** buttons have been added.

Translators

IFC Export

The new **Construction** > **Parameters** > **Other parameters** > **IFC Owner History** command allows you to define the property information that will be exported with the IFC document. This information can be defined manually or retrieved from another document.
The new **Export Axis and FootPrint representations** option allows you to export the axes and footprints contained in the representations named **Axis** and **FootPrint**. Axes are used to define the neutral fiber of the walls and the footprints of components such as door components.
When an assembly contains the **Type for BIM** parameter equal to **Simple**, it is then recognized as a multi-layer material in the IFC file provided that the **Export TopSolid Materials** box is checked.
Layers are now compatible with the new **Export TopSolid Layers** option.



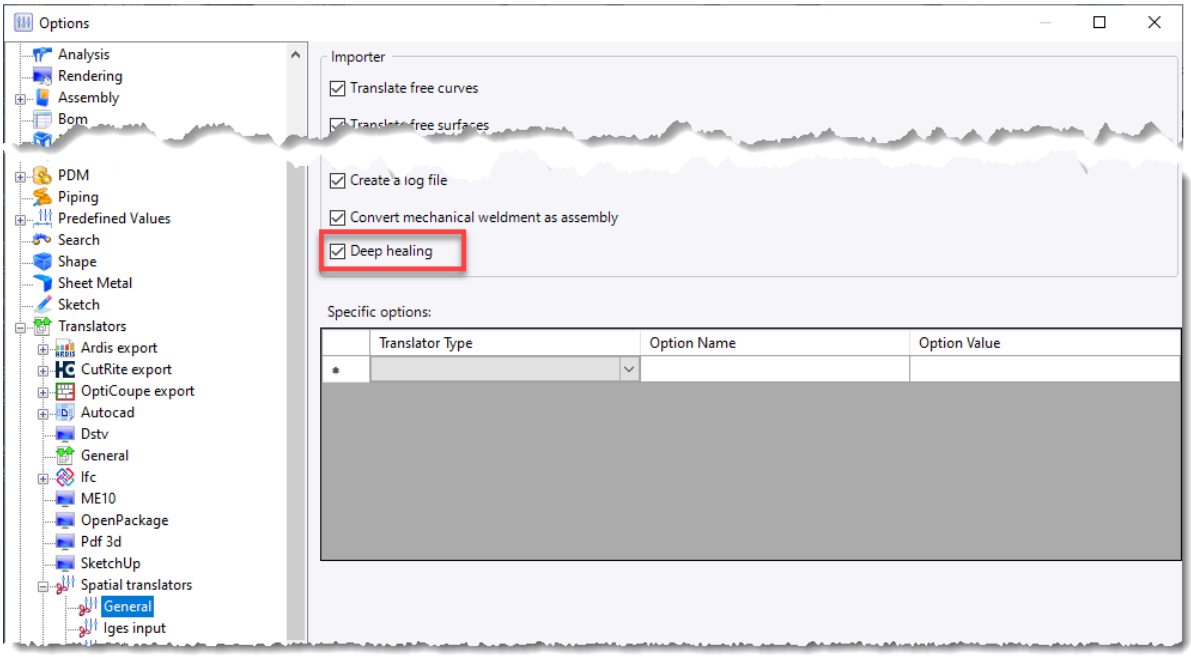
New IFC export options.

TopGmi Import

You can now import configurations from SolidWorks.
In the **Tools** > **Options** > **Translators** > **TopGmi** section, the new **Import as local parts** option allows you to create an assembly document containing only local parts. When this option is checked, the assembly structure of the **Step**, **Parasolid**, etc. file is not kept and all the shapes, including those from a sub-assembly, are imported at the first level of the assembly.
When importing a **TopGmi** file, the drillings are now recovered as inoperative drillings, PMIs and user properties are also recovered.

Spatial translators

The **Deep healing** option has been added to the general options of the **Spatial** translators, it can be very useful when recovering poor quality STEP or IGES files. However, it can significantly slow down the file import time.

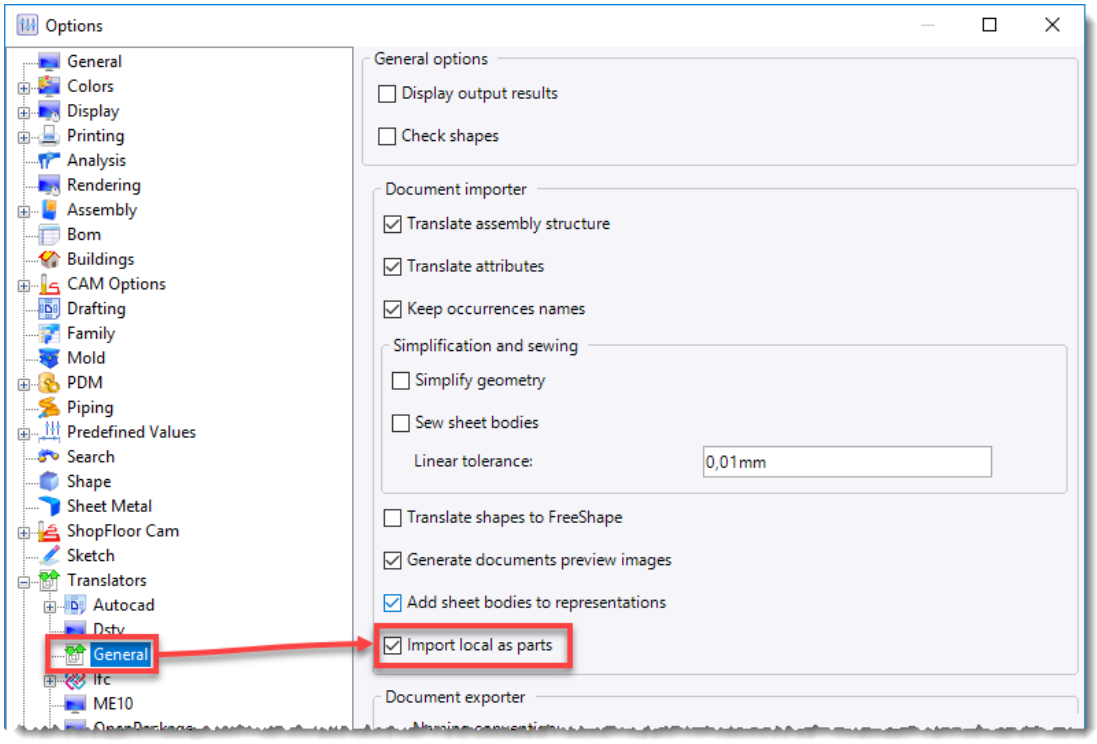


Exporting exclusive sets

When exporting a document, the **Export Sets as Assemblies** option allows you to create a Parasolid, Step, etc. file where only the structure of the selected assembly is exported.

FBX Import

You can now import FBX files, elements can be imported as polyhedrons or faceted shapes.



New export formats

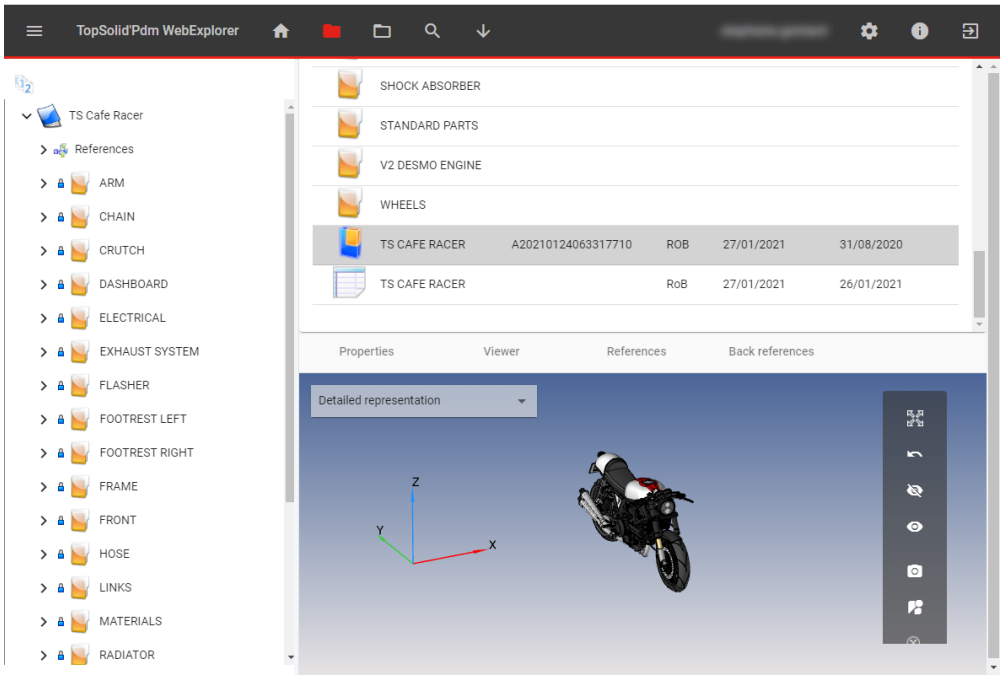
There are two new export formats:

- **GlTF** export (scenes and 3D models) - File extension: **.glTF**, **.glb**
- **Redway** export (3D scenes) - File extension: **.red** (this requires starting **TopSolid** in advanced mode with **-a** in the launch shortcut).

Web Explorer (optional module)

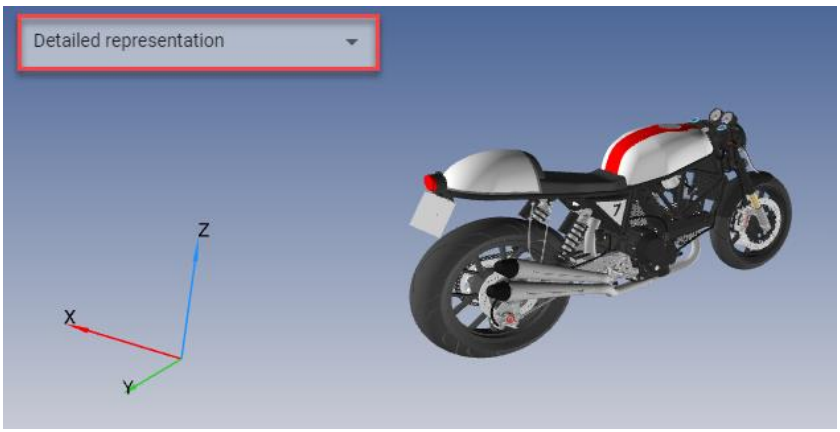
Navigation

The project navigation dialog box has been changed to Windows Explorer, with folders on the left side of the screen, documents or subfolders at the top right of the screen and the dialog box for viewing, properties, references and back references at the bottom right of the screen.



Representations

Document representations are now available in a drop-down list at the top left of the document viewing dialog box.

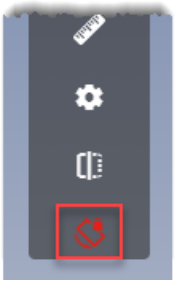


Transparency

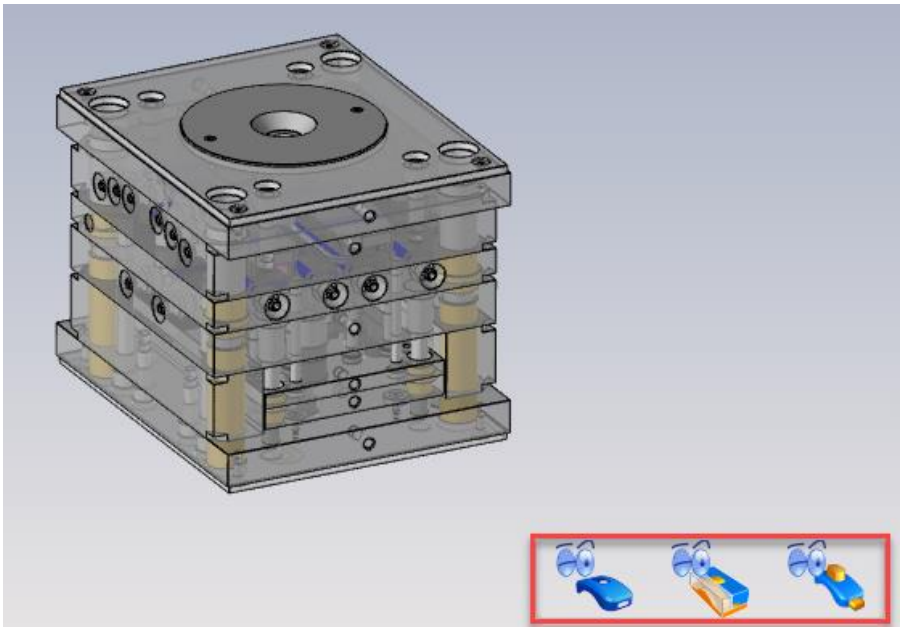
You can now change the transparency of a part or a shape from the contextual menu.

Visualization

The **Vertical Lock** command has been added to the bottom of the icon bar.



For tooling documents (mold, strip, etc.), specific visualization icons have been added at the bottom of the graphics area.

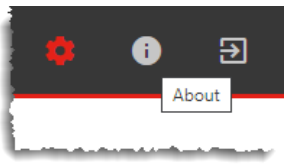


Document sharing

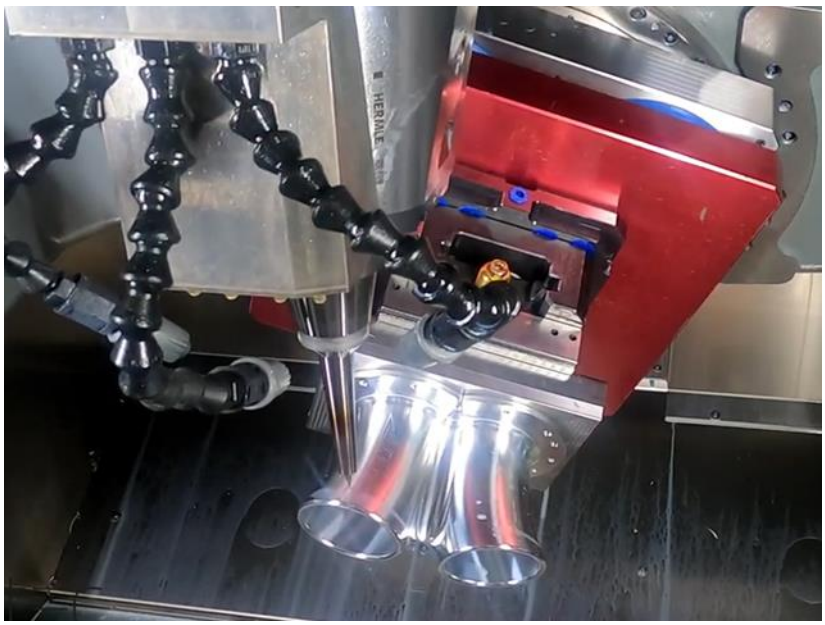
In the **Properties** tab, the **Share document** button allows you to copy the document URL or create an e-mail with a link to the document.

About

The **About** command has been added to the **TopSolid'Pdm WebExplorer** banner. It displays the version of the application, the server version and the number of available licenses.



What's New in TopSolid'Cam 7.16

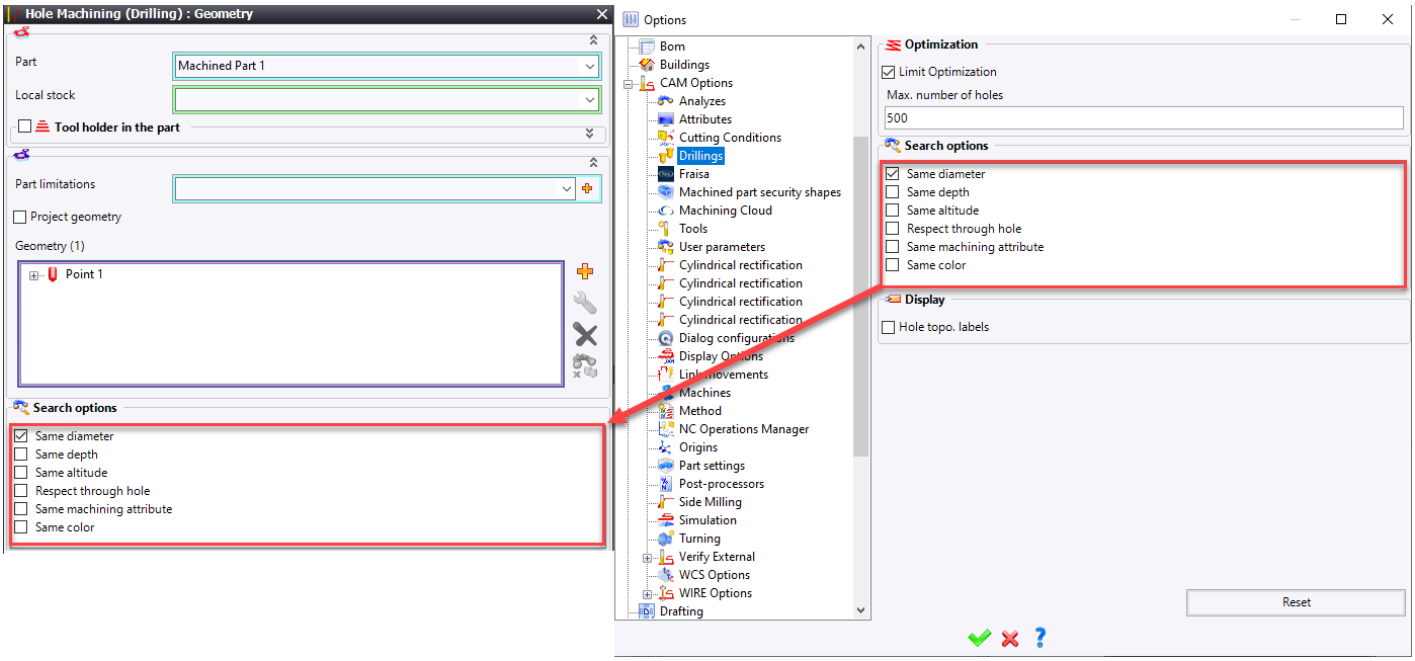


This section describes the enhancements made to the **Machining** application in version **7.16** of **TopSolid 7**.

Drilling

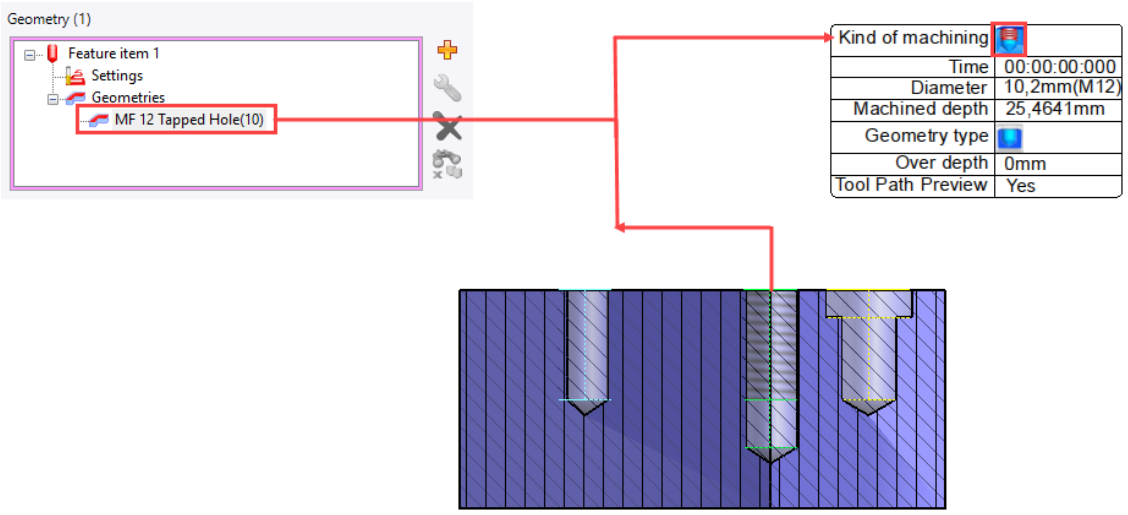
Geometry search (default option)

In the application options, you can now define the search options for similar geometries by default.



Associating the MF type with the drilling primitive

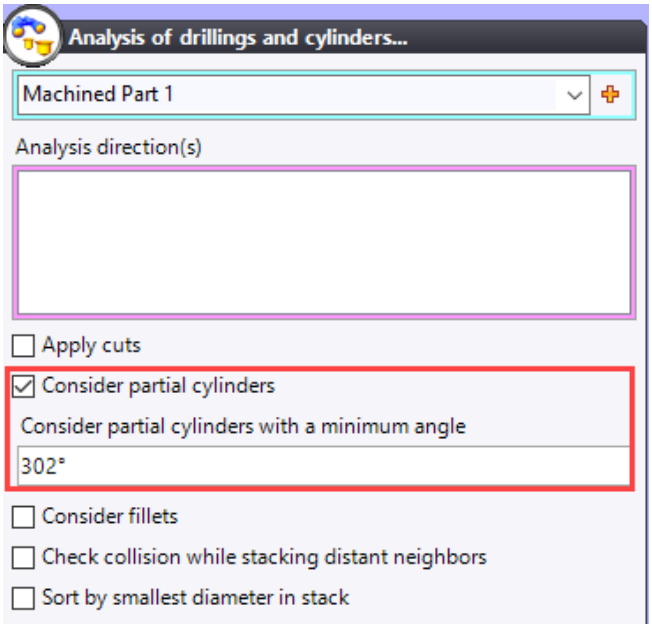
When creating a drilling operation by right-clicking on an MF, **TopSolid** can now recognize the MF type (tapping, drilling, facing) and automatically propose the appropriate operation primitive for the selected MF.



Cylinder analysis

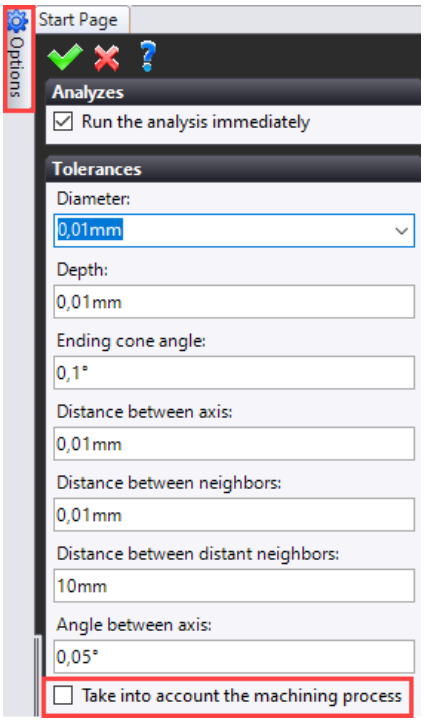
Partial cylinder management

This new option allows you to have an additional criterion when analyzing partial cylinders whose aperture angle is strictly greater than the angle indicated in the **Consider partial cylinders with a minimum angle** field.



Stack/process filter

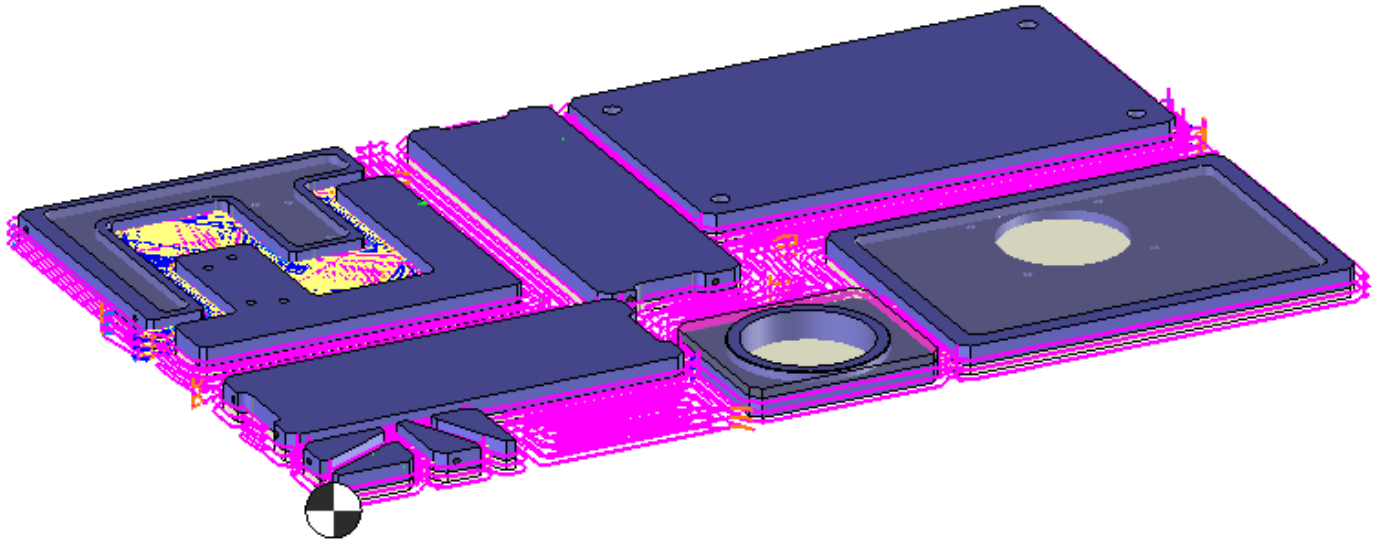
You can regroup the cylinders with the same machining processes in the stacks while excluding those with a different process from this stack. This filter is now available in the document options.



2D Milling

Contour roughing

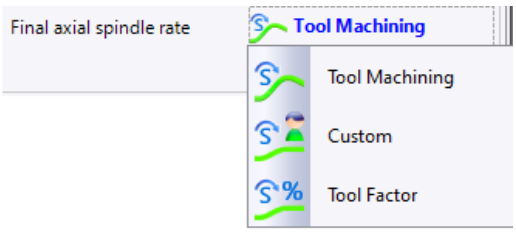
Contour roughing is a new feature for end milling operation that allows you to machine from the outside (the stock) to the finished part with access to the end milling parameters. This new operation works like the classic end milling operation. The difference is that the selected geometry is a vertical face, which allows you to completely machine around the finish (similar to a side milling operation with several radial passes).



End milling

Spindle rate for final path

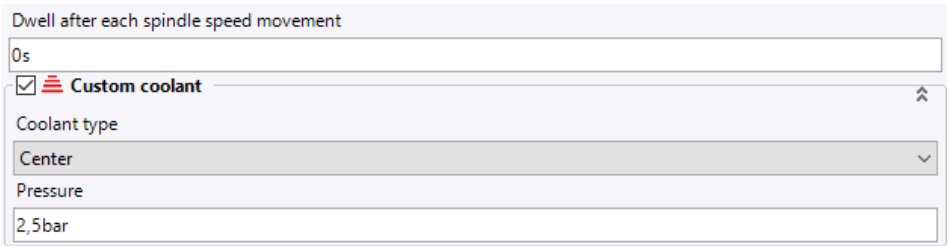
In end milling operation, you can now define a spindle rate independent of the cutting conditions for the final axial path.



Note: This new feature is also available for side milling operations.

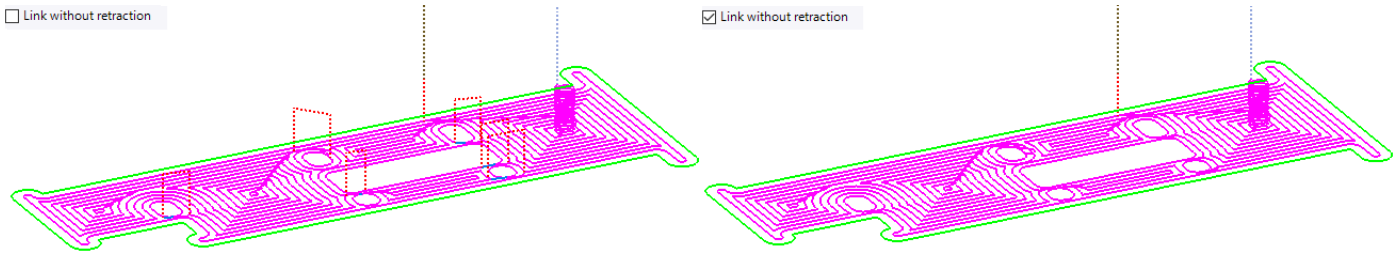
Coolant and delay

As it is already possible in **Boost** mode, the goal is to be able to define a custom coolant for the plunge as well as a delay on the entire end milling strategies.



Retract management in HSM

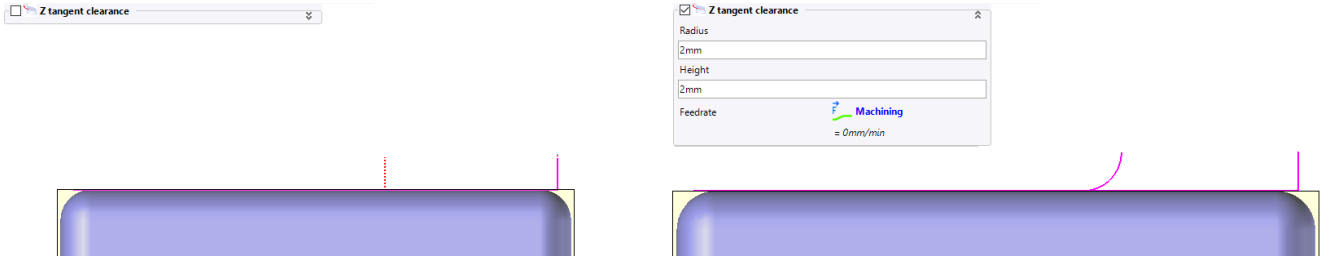
In some cases, you can force the tool to stay at the bottom of the Z pass, so as not to perform retract movements between the areas or different islands to be contoured.
To do this, a **Link without retraction** option is now available in the **HSM** tab.



Restriction: This option is not compatible in **Boost** mode.

Z tangent clearance

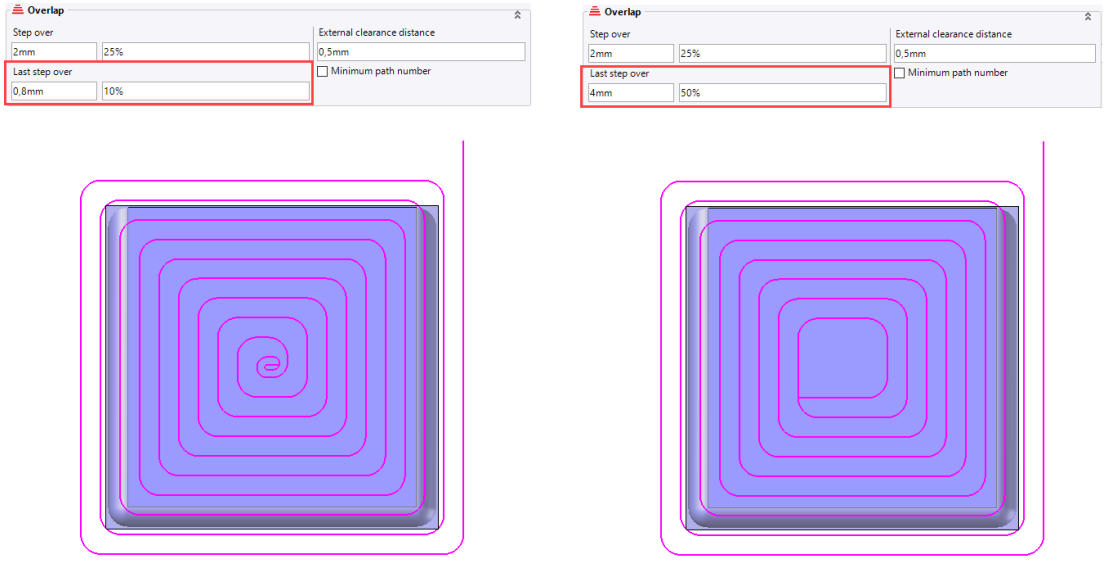
You can perform a vertical tangent retract movement at the end of the machining operation.



Note: This new feature is available in the **Retract** tab for 2D and roughing operations.

Last step over

You can add or delete paths based on the value of the **last step over**.



Side milling

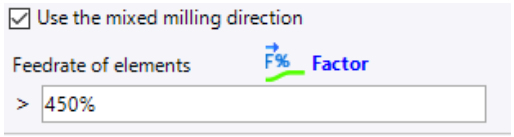
Various improvements in angle management

- **Helix mode**: Shape edges followed by a "consecutive" tangent arc is now possible.
- **Truncated mode**: Open profiles are better handled when using reduced contouring.

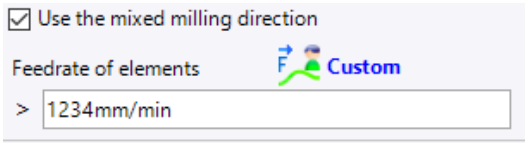
Boost Milling: Reversed machining feed rate management

You can define a custom feed rate for the reversed movements of the tool path. In addition to the work mode, two new modes are now available when the reversed mode is activated:

- **Factor**



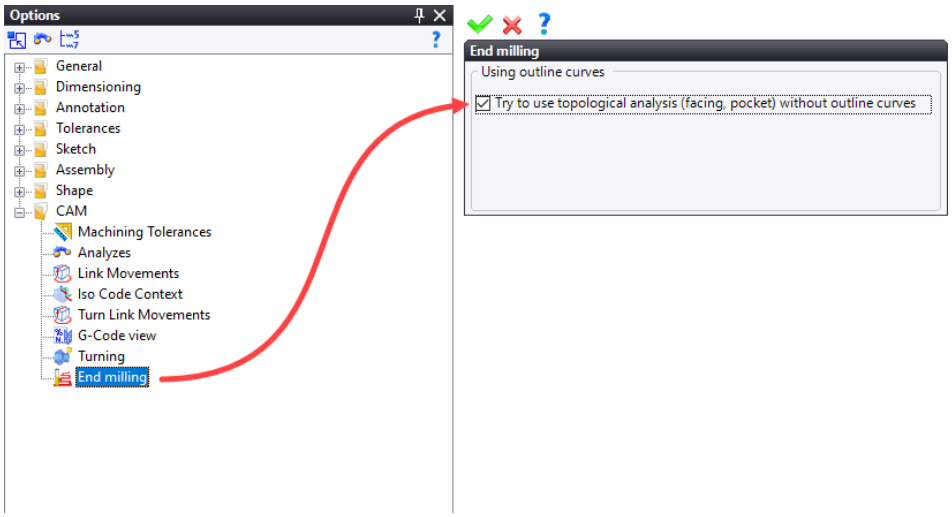
- **Custom**



2D Machining: Various improvements and optimization

End milling

Bottom radius management: When the **Ignore bottom radius** option is checked, the finish shape management is maintained.
Being able to work without a silhouette: Reduces the calculation time of the machining operations. On large parts (or cast parts), the time spent to calculate the silhouettes can be long. In this case, seeing that the **Take into account the finish shape** box has already been unchecked and with the **TopSolid Basics** default values, the calculation of the silhouette can be simplified.



Restrictions:

- This option does not work in open pocket and through pocket.
- Cannot enable the **Take into account the finish shape** option.
- It is also preferable to be in **TopSolid Basics** (default value).

Plunge point in optimized mode: You can now define a user plunge point when performing pocket machining operations in optimized mode.

Information (events): Let the default values drive the plunge type (direct, helix, etc.). If the default values for the plunge options are **Direct** and **Plunge authorized** and the selected tool does not have a center cut, a warning message will appear in the events view.

HSM not alpha loop: Improving the remaining material removal path to avoid having loops in the path if possible.

Slope management: You can apply an angle offset between the Z passes to manage the incremental offset of the stock on the sides of the finish. This was already possible in value mode for end milling operations.

Side milling: Reduce feed rate in angles

The feed rate reduction is applied only on the angles, it was previously applied to the overlaps.

Collision management

The **collision management** options available in 3D (tool holder management) are now available for 2D operations.

Boost Milling: Performance improvement

Pass processing has been optimized to shorten the machining time and optimize the path when a user plunge point is defined.

Slot milling

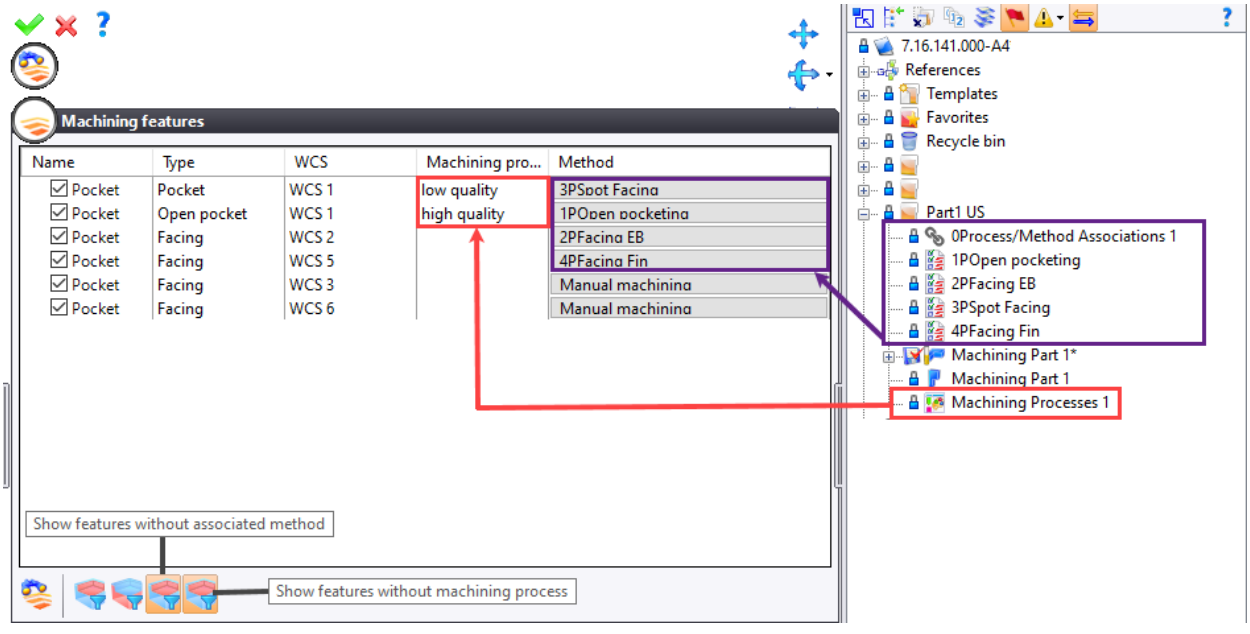
Topological recognition of the slots and integration of the **Boost** mode has been improved.

Machining features analysis: Filter and optimization

The analysis of milling shapes is easier to use in the analysis of machinable shapes thanks to two filters that are now available:

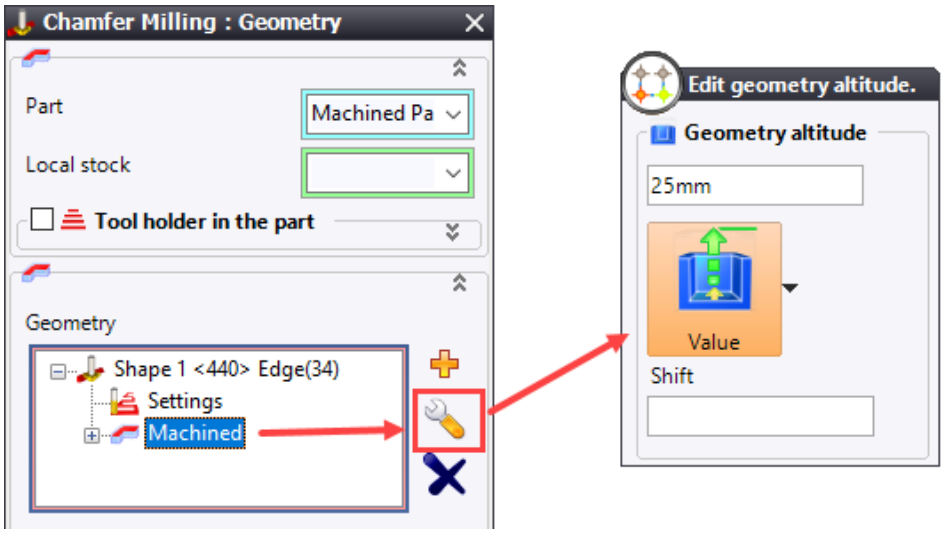
- **Show features without associated method**: When this filter is unchecked, the machinable shapes for which no machining method has been associated are no longer displayed in the list.
- **Show features without machining process**: When this filter is unchecked, the machinable shapes for which no machining process has been defined in CAD are no longer displayed in the list.

The use of these filters requires the use of documents external to the machining document such as the process document and the method/process association document.



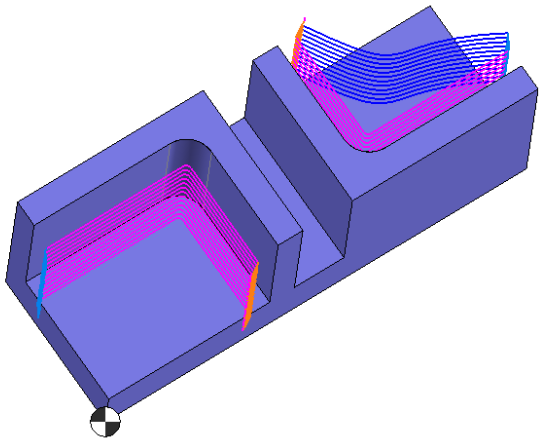
Chamfering: Altitude management in wireframe mode

You can now position the machining of a wireframe geometry at a given altitude and thus avoid touching the altitudes.



Side milling: Staying at the same Z level

You can perform a side milling operation while avoiding rapid retracts in order to reposition between each pass in Z as in 3D roughing.



Machining features analysis using direction: External silhouettes

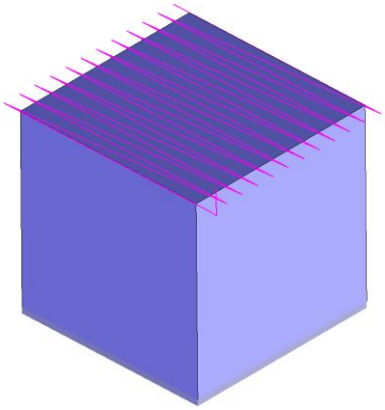
The topological search for feature analysis using a direction generates a machinable outer shape by a contour roughing or side milling operation.

Note: The contouring MF is taken into account in the machining features analysis.

Grinding

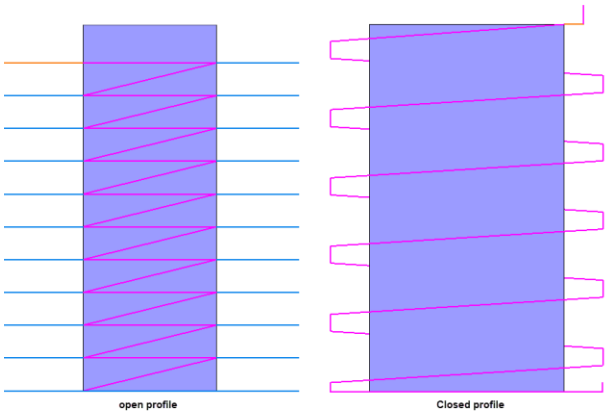
Grinding by facing

There is a new **Grinding by facing** operation. It consists of horizontally grinding the part with a "grinder" tool. This operation is similar to the end milling operation in **Sweeping** mode with a new **Drift cycle**.



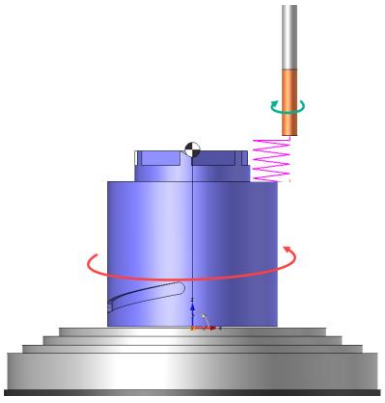
Grinding by contouring

Similarly, you can create a new **grinding by contouring** operation. It consists of grinding the part with a "grinder" tool. This operation is similar to the side milling operation with "drift cycle" tool paths in case of open profile and "helix" tool path in case of closed profile.



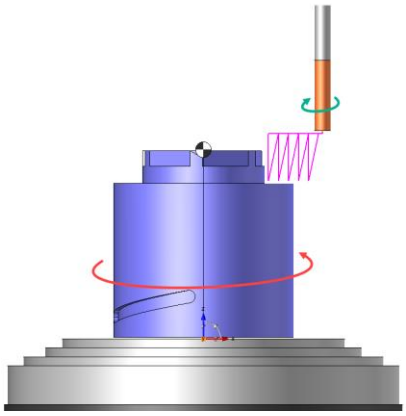
Cylindrical grinding by facing

You can create a **cylindrical grinding by facing** operation. It consists of horizontally grinding the part with a "grinder" tool. This operation is similar to the grinding by facing operation except that the part rotates with the power axis and the tool always rotates in the opposite direction.



Cylindrical grinding by contouring

A new **Cylindrical grinding by contouring** operation is now available. It consists of grinding the part with a "grinder" tool. This operation is similar to the grinding by contouring operation except that the part rotates with the power axis and the tool always rotates in the opposite direction.



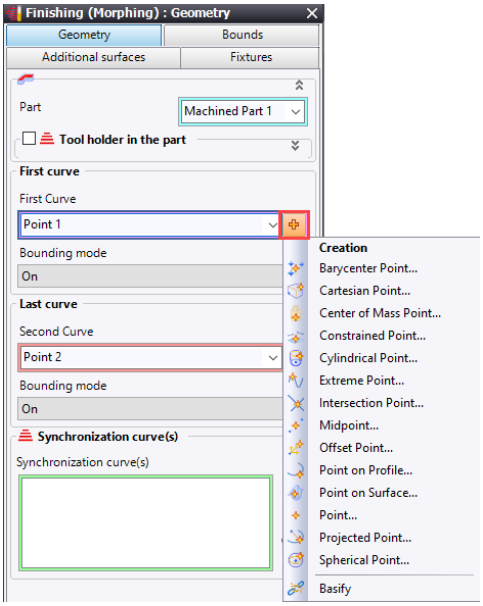
Creating new grinder tools

In response to the addition of new **grinding** operations, tools dedicated to these types of operations are available.

3D Milling

Morphing: Creating points in the operation

You can now create points directly in the operation using the selection control which will be managed by the **first** and **last curve** operation.



Restriction: Creating sketches for the curves using the **First Curve** and **Last Curve** fields is still not compatible.

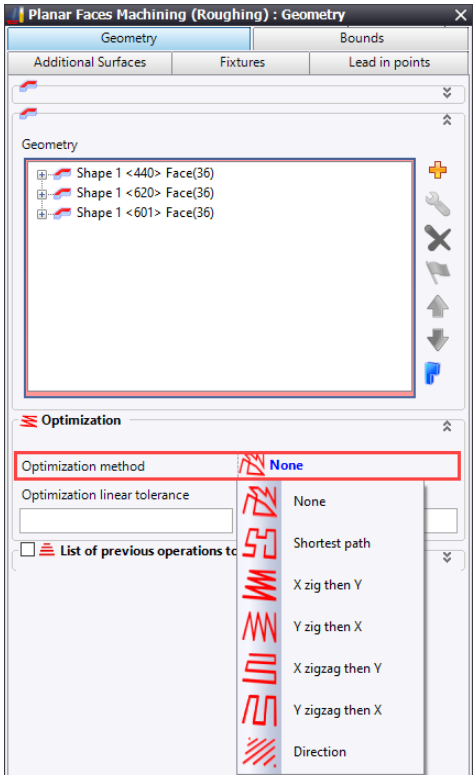
Dividing tool path by tool lengths: Cutting conditions management

You can now define different cutting conditions between each cut of the path independently of the parent operation while dividing the path.

Planar face machining: Geometry optimization

As in end milling operations, the goal is to optimize the order in which the faces are made using the **Planar Faces Machining** command. By default, the planar face operation machines the faces in the order in which they are selected in the **Geometry** tab.

Note: Changing the optimization mode changes this order.

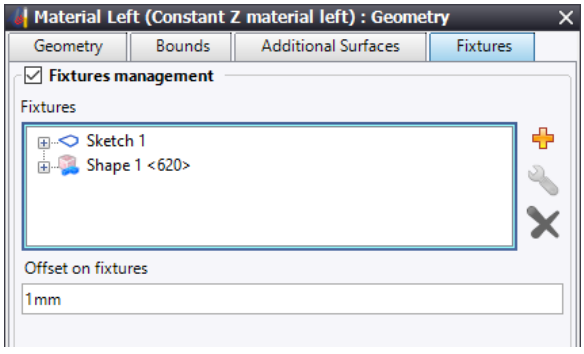


Restriction: This new feature requires that the sorting by altitude/pocket is disabled by default (useful only in case of additional passes).

Material left

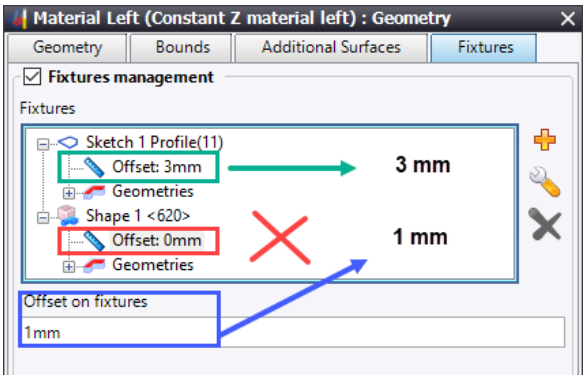
Fixture management

This feature allows you to define the fixtures directly in the operation and to take them into account. In 2D machining, the goal is to force the tool to go up to avoid any collision with the element concerned.



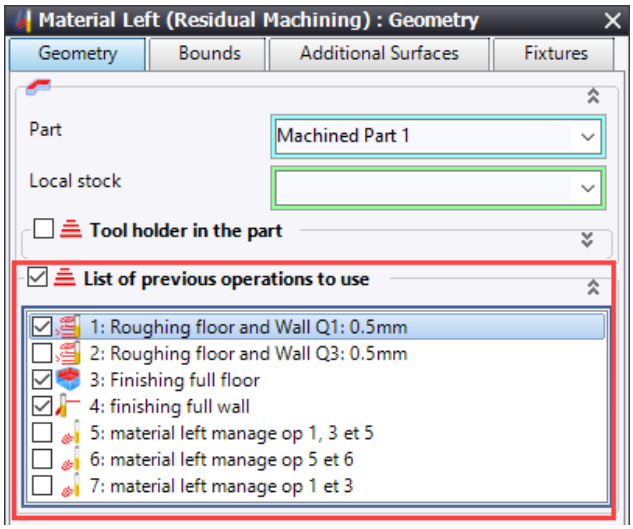
Note: This option is available for all primitives in the **Material Left** operation.

Restriction: When a specific offset is applied to one or more of the fixtures, these will only be taken into account if they are greater than the overall value.



Managing previous operations

You can either take into account the paths of the previous operations or not as with the 3D roughing.



Path management

The goal is to manage the path during the remachining operation. **TopSolid** recalculates the initially calculated path several times with a different offset, therefore allowing the following parameters to be managed:

Last path: Defining the distance between the last remachining path (0mm offset) and the one before it.

Axial depth: Defining the distance between each path.

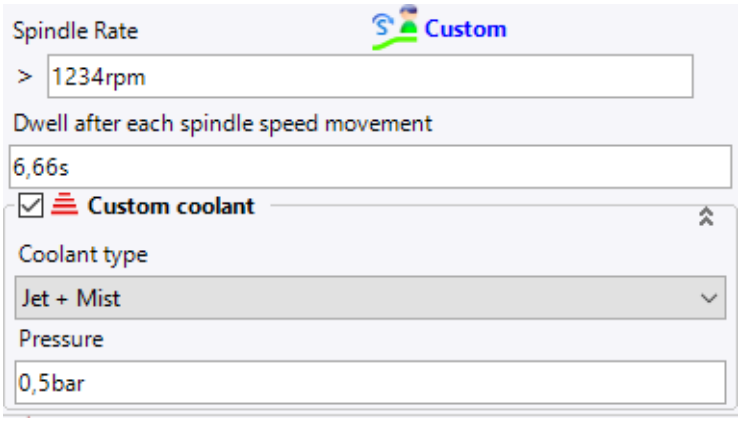
Number of paths: Defining the number of remachining path.

Restriction: We take into account the remaining material and for example, not because you ask for four paths does not mean that you will necessarily have four layers of tool path.
Everything will depend on the amount of material to be remachined.

Roughing

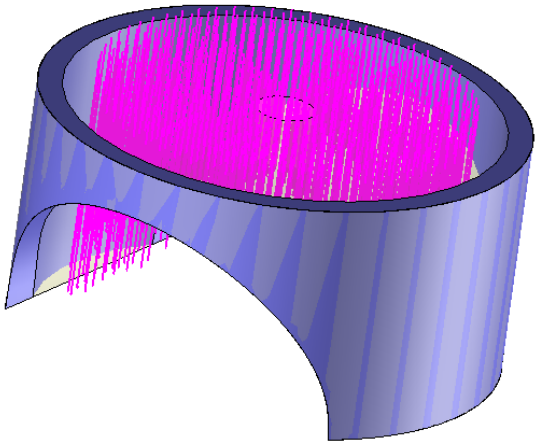
Spindle rate, coolant and custom plunge delay

As in 2D, you can define the spindle rates, coolant and a delay between changes in spindle rotations independent of the cutting conditions for plunge movements.



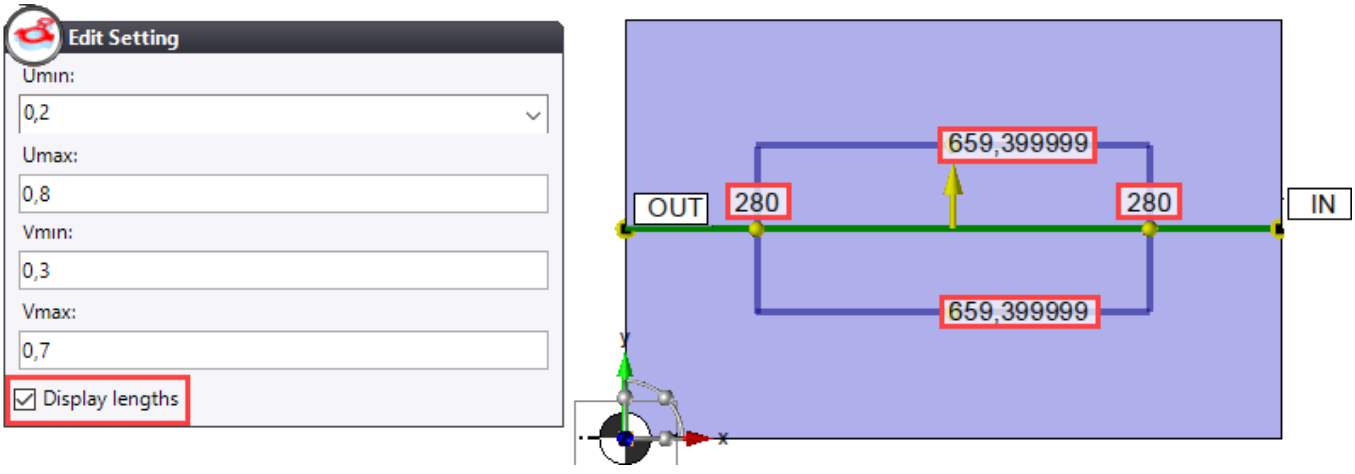
Vertical roughing: Optimized depth

As it is already the case in 2D plunge roughing, you can manage the hollow stocks.



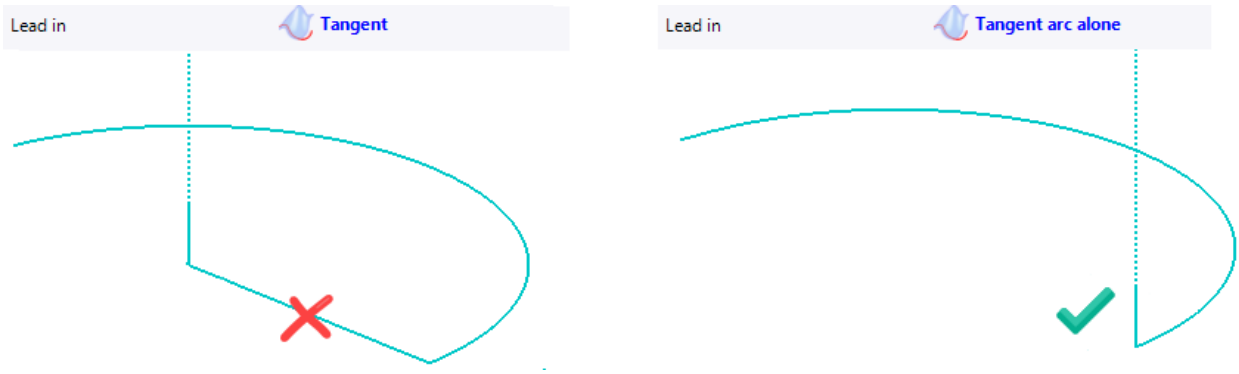
Sweeping: Iso-parametric limitations in millimeters

TopSolid displays the border lengths reduced to the iso-parameters defined in the dialog box. The length is displayed in the document unit.



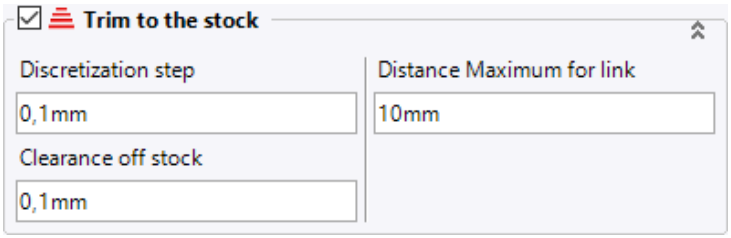
Constant Z Finishing

TopSolid allows you to perform a unique tangential lead-in movement during the approach. A new **Tangent arc alone** lead-in is now available.



3D/5D tool path optimization: Eliminate off stock passes

A **Trim to the stock** option has been added to the operations in order to eliminate all the machining tool paths that are not in the stock. When the path is off stock, **TopSolid'Cam** performs a retract to the security plane or to a direct link below a certain distance.



Restriction: This option is only available for 3D operations that do not support roughing initially, swarf and 5-axis contouring.

Roughing operations: Planar faces machining of the link

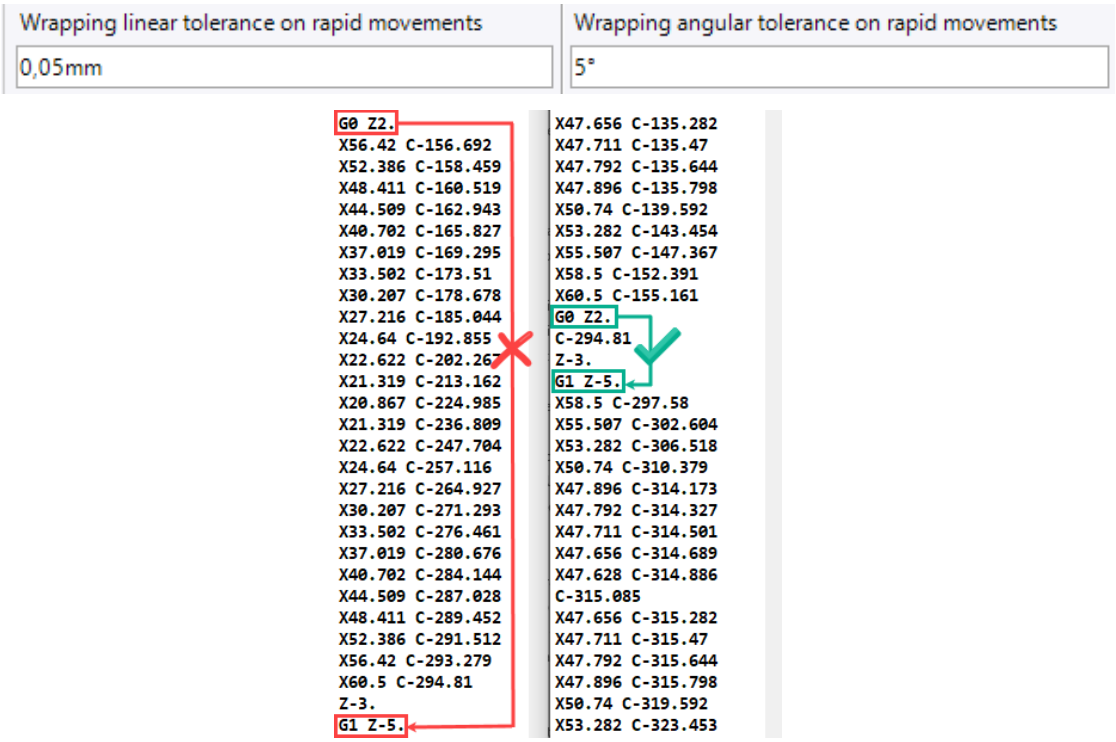
The goal is to be able to choose the value of the link of the planar faces in three different ways:

- **Identical** mode: The link on the planar faces will be equal to the global value of the link of the roughing which is defined in the **Settings** tab.
- **Custom** mode: You can select a custom value different from the global value. By default, the custom value is equal to 70% of the tool diameter.
- **Factor** mode: A percentage of the overall value will be chosen. It allows you to select a factor greater than 100%, but at least equal to 1%. By default, the factor is equal to 100%.

4D Milling

4D machining: Splitting tolerance of the rapid movements

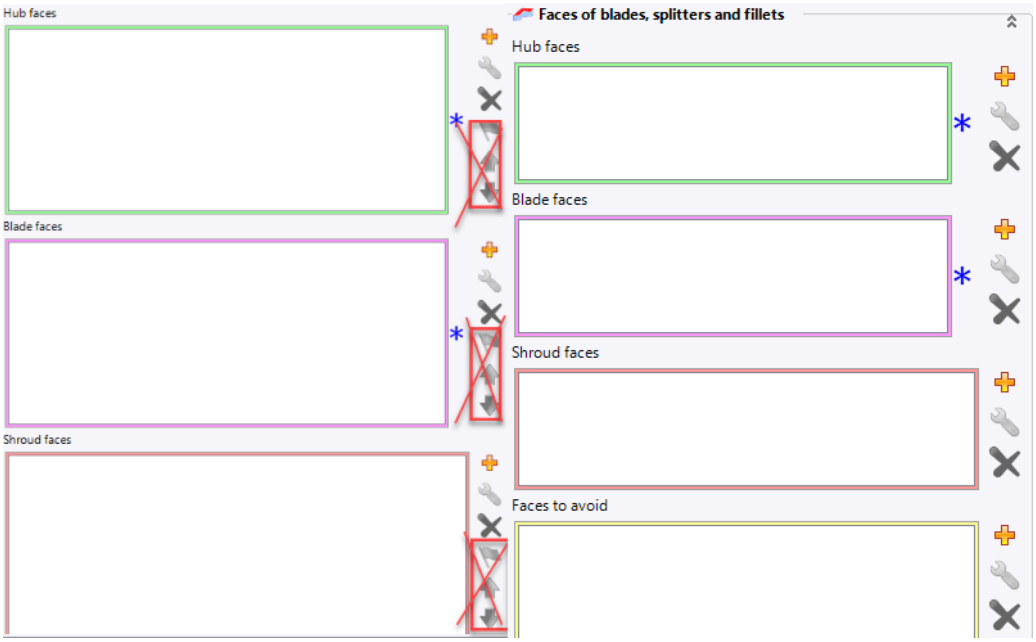
You can now change the splitting tolerance of the rapid movements, in order to obtain more or less points during rapid interpolations in the ISO output.



5D Milling

Blade machining: Simplifying dialog boxes

In order to limit the presence of superfluous icons, the different fields in the **Geometry** option have been refined.

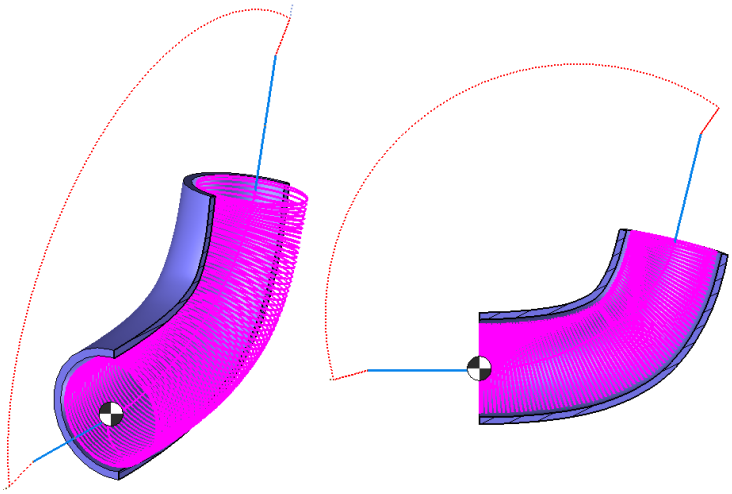


5X deburring

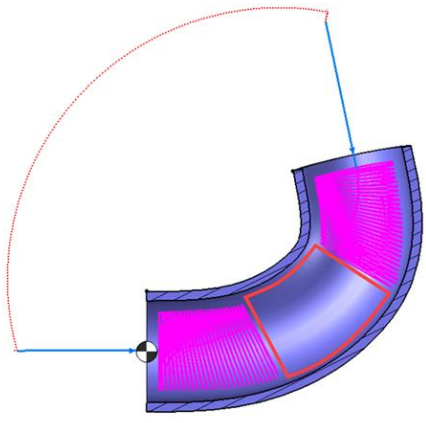
The goal is to automatically deburr shaped parts. This process is similar to the existing 2D edge breaking operation, but applicable to more complex parts that can be machined in 5-axes.

Port machining

Cylindrical cavities and **pipe** shapes can be easily machined.
Topological management will allow you to machine the maximum on one side then on the other once the limit is reached (collision management).



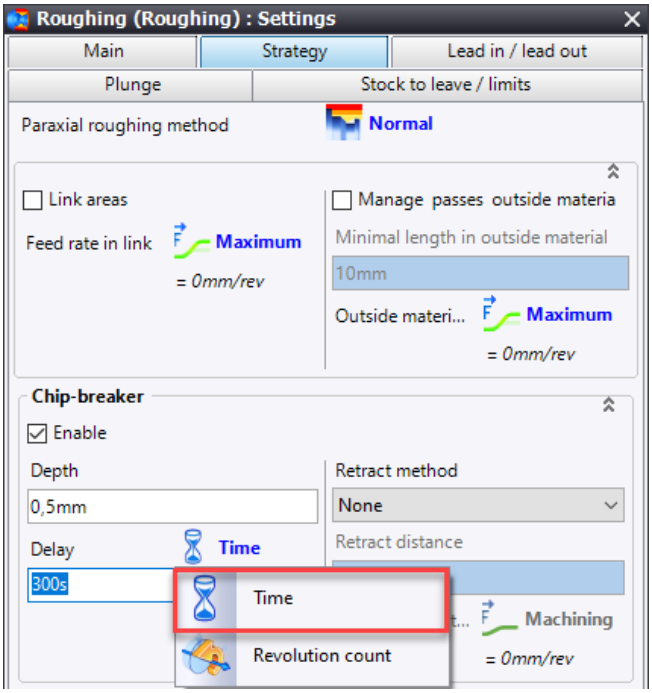
Note: **TopSolid'Cam** will limit itself to the areas that can be reached (see image below) if the tool is too short or if the machine angular limits do not allow the entire area to be machined.



Turning

Roughing: Chip-breaker delay

You can now define a time in the **Delay** field, until now it was only possible to define it in **Count**.



Note: Delay in seconds is applicable to all turning operations.

Machining in the ZY plane

You can work in the ZY plane for all turning operations, when the tool type allows it (tools working in the ZY plane as opposed to conventional tools which work in the ZX plane). The goal is to be able to work with these two tools:

- Y parting tool: CoroCut QD (Sandvik)
- Roughing and finishing tool: FreeTurn (Ceratizit)

Creating turning operations is done like for a conventional tool. The operation automatically detects that it is working with the Y axis. The spindle directions are calculated automatically.
Provided that the machine is capable of withstanding this cycle (in agreement with the post-processor manager), the ISO output, can be done in two ways:

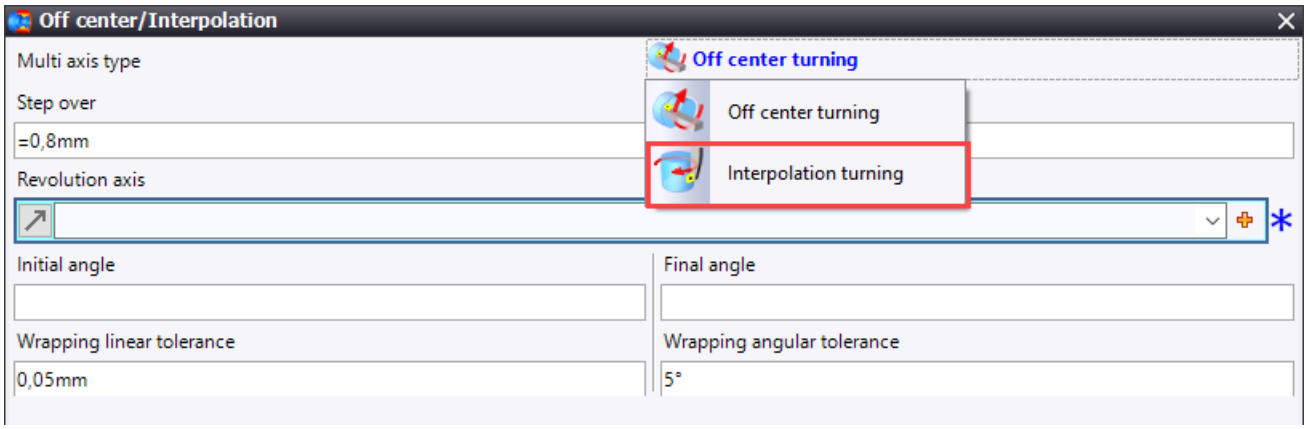
- Conventional WCS and tool path with ZY coordinates.
- Rotated by 90° WCS and tool path with ZX coordinates.

Restriction:

- Verification in programming mode is not compatible.
- To use these tools, you need a Y-axis on the machine.

Machining by interpolation

A new feature allows you to perform **Interpolation turning** operations (tool path wrapped around the part).



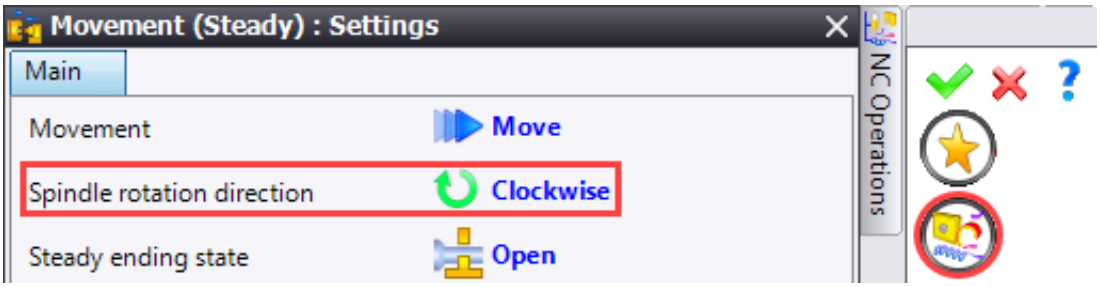
Restriction:

- This option is not compatible for threading operations.
- The spindle axis must be programmable.
- The workstation must allow you to mount the turning tools.

Steady/tail stock movement

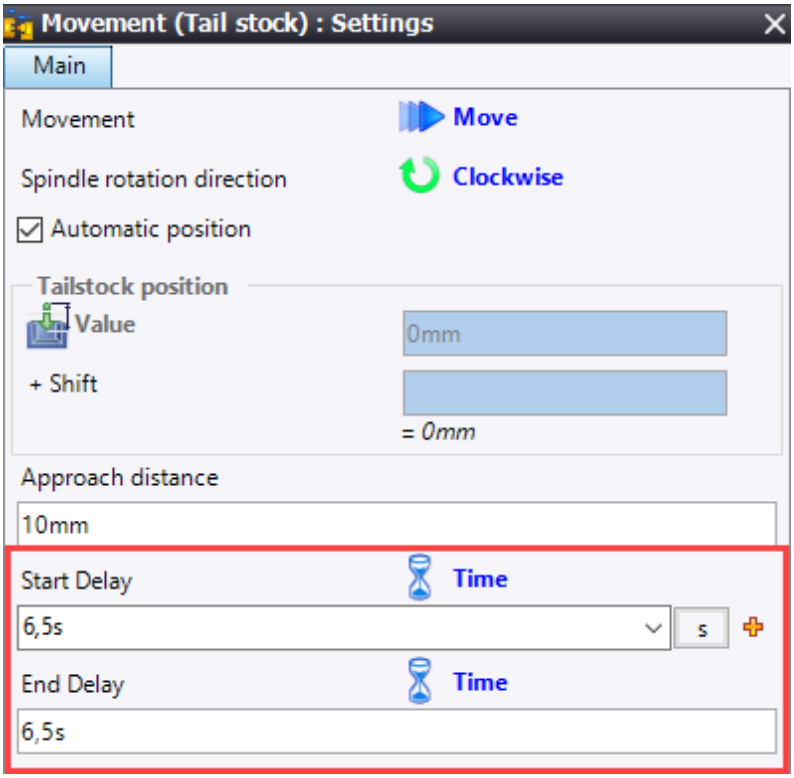
Chuck rotation and coolant

The cutting conditions menu for the **Steady** operation has been added, giving access to spindle speed and coolant options, as well as a **Spindle rotation direction** parameter in order to indicate the spindle rotation direction, which defaults to **None**, corresponding to a stationary spindle.



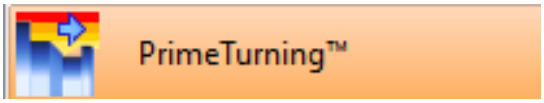
Delay

You can now define a delay for tail stock movements.



Turn roughing: PrimeTurning™

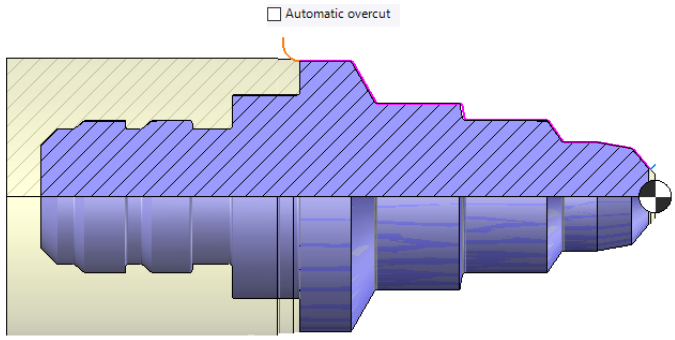
There is a new turn roughing operation: PrimeTurning™. The principle of this new strategy is to perform a back roughing operation. This machining technique is unique to Sandvik and is described on their website: https://www.sandvik.coromant.com/en-us/products/coroplus-toolpath/pages/primeturning.aspx

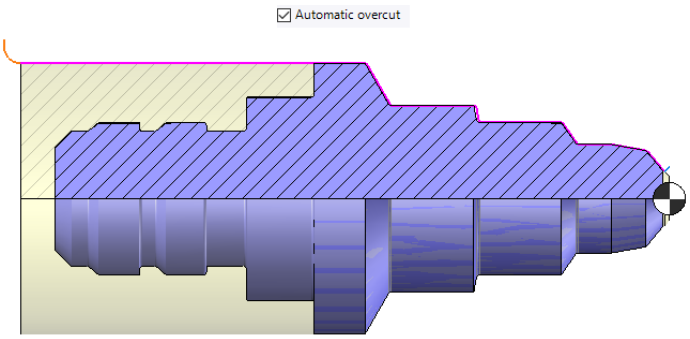


Finishing: Extending the path to the stock

When this option is checked, it automatically adds an overcut to the tool path to reach the end of the stock.

Note: You can also manually enter an overcut which will be added to the **automatic overcut**.

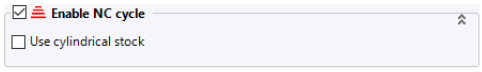


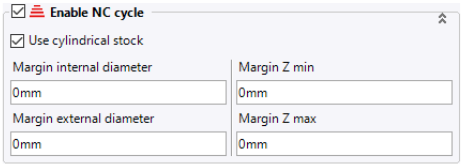


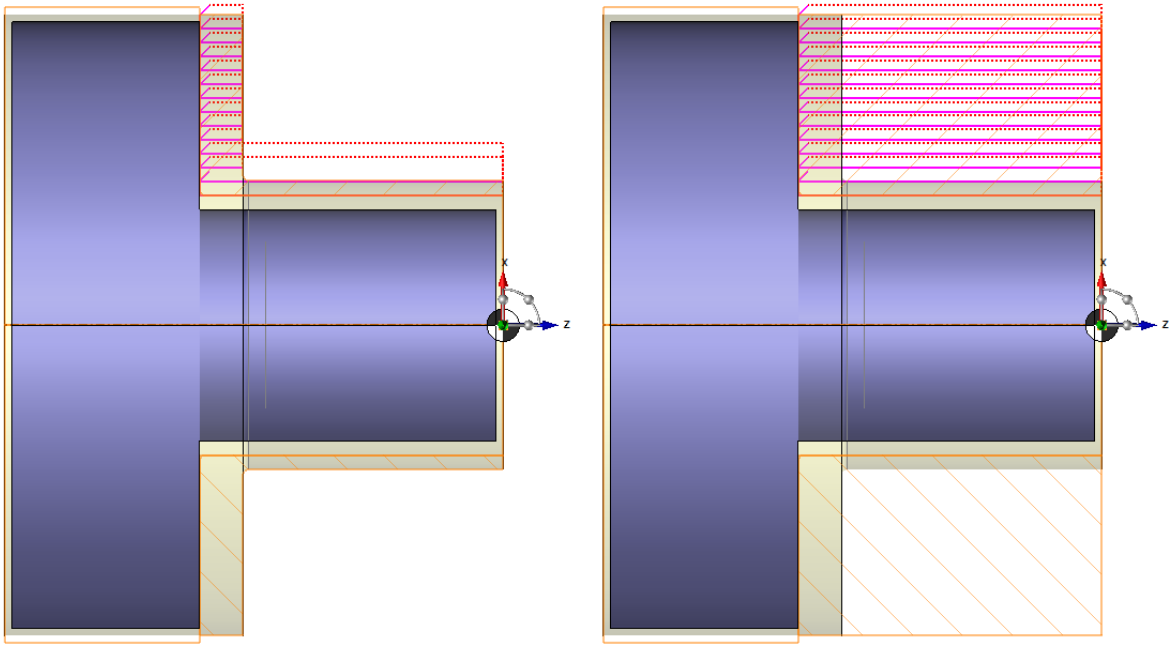
Restriction: The environment is not supported.

Roughing: Using a cylindrical stock in machine cycles

Some machines cannot take into account complex shape stocks in the machine cycle, so in **TopSolid**, we could not see a tool path similar to the one that the machine can take. An option has therefore been added in the parameters so that the calculation of the work area uses a virtual cylindrical stock that includes the real stock.





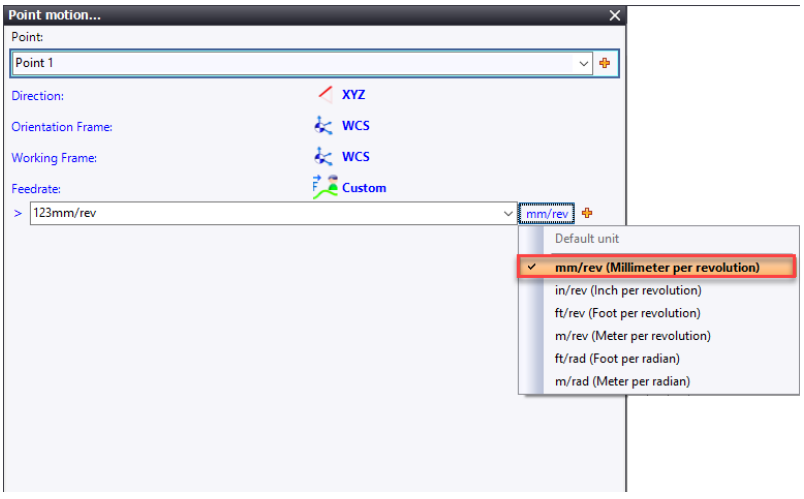


Restriction: You can increase the size of this stock, but only the margins compatible with the **configuration of the finish** will be taken into account.

Virtual Jog

Feed rate movement mm/rev

The **mm/rev** unit is now available for all custom feed rate movements.



Tools and Cutting Conditions

Cutting conditions

Type of operations

In the cutting conditions documents for each specific tool, you can now define the type(s) of operation(s) such as: **Roughing**, **Pre-finishing** and **Finishing**.

Coolant Type	Pression d'arrosage	Tooth feed rat...	Ae	Ap	Quality
None		☐	0mm	0mm	All
None		☐	0mm	0mm	Roughing
None		☐	0mm	0mm	Pre finishing
None		☐	0mm	0mm	Finishing

Ap/Ae

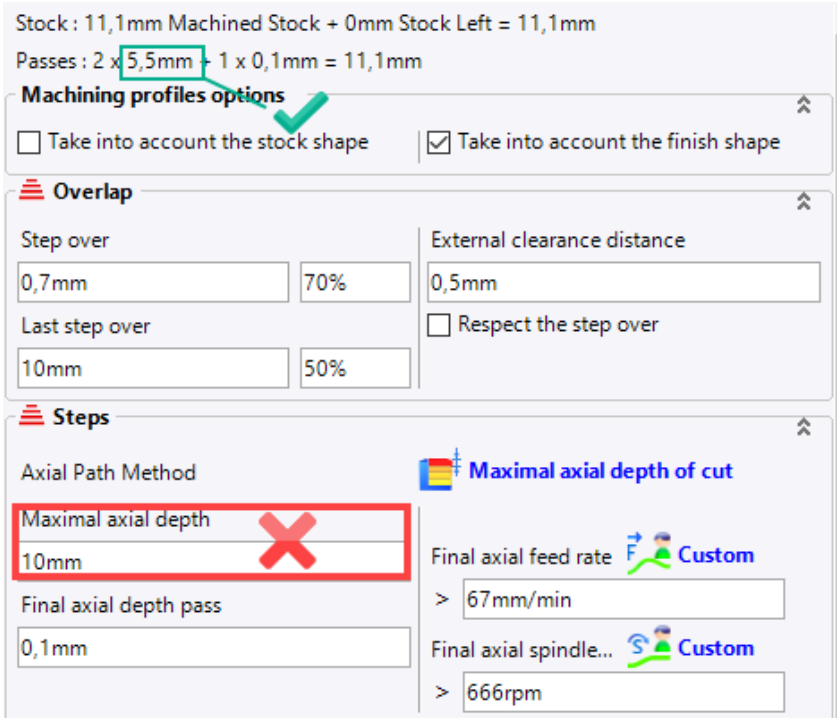
In the cutting conditions documents for each specific tool, you can now define the **Ap**/**Ae** values.

Coolant Type	Coolant press...	Tooth feed rat...	Ae	Ap	Quality
None		☐			All
None		☐			All

Note: This information is not available for **drilling** tools.

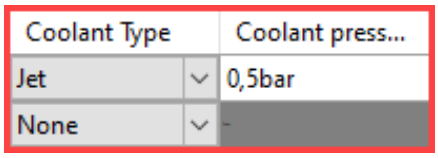
Real Ap

For the choice of values in an abacus, fixed value parameters (maximum pass depth) have been used until now. The real pass depth taken by the tool is often not equal to the values in these parameters. It is recalculated based on other parameters. The new feature consists of recovering this value and using it in the abacus.



Cooling pressure

In the cutting conditions documents for each specific tool, you can define, in addition to the cooling type, the pressure of this one.



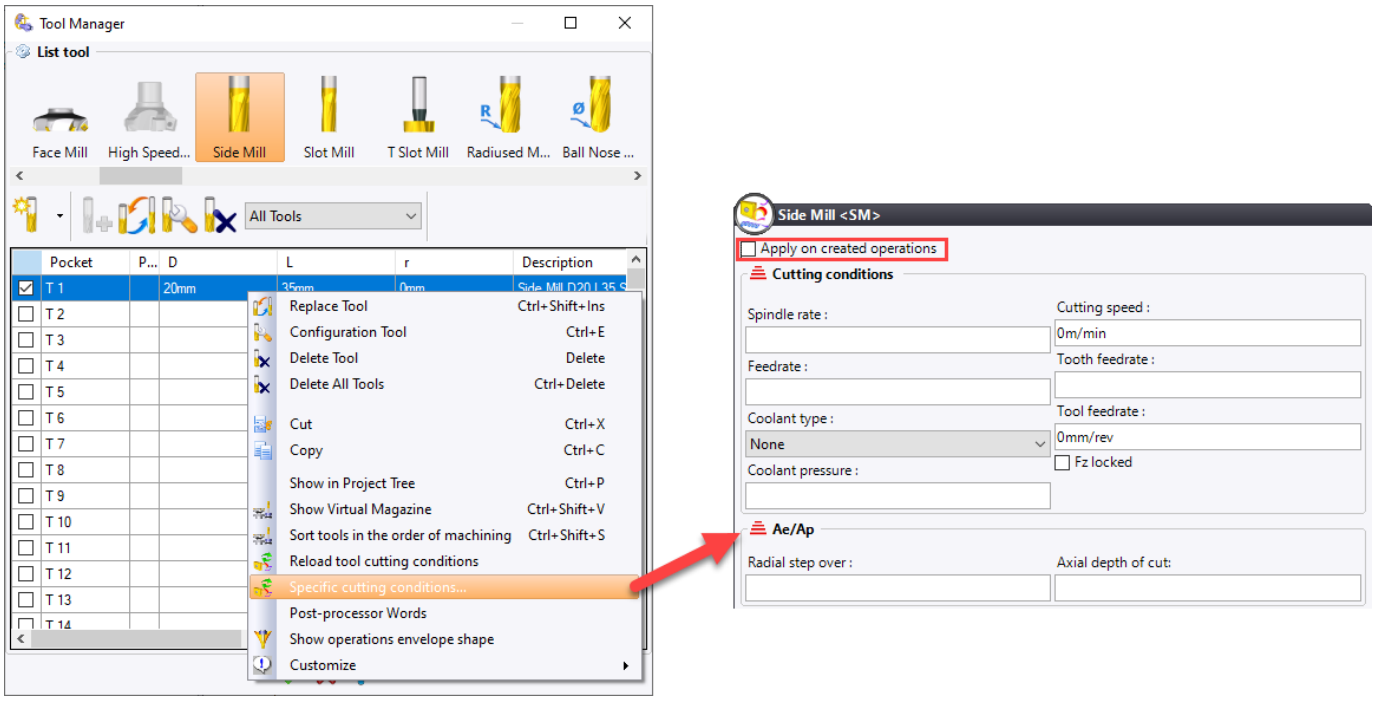
Managing previous operations

Previous operations are taken into account by reapplying the cutting conditions defined when creating a new operation. If the tool is already in use, then we can recover the cutting conditions.

Restriction: This does not apply if the cutting conditions are provided by external elements (example: abacus).

Application from the tool manager

You can define the radial and axial depths as well as all cutting conditions of the tools required for machining directly from the tool manager.



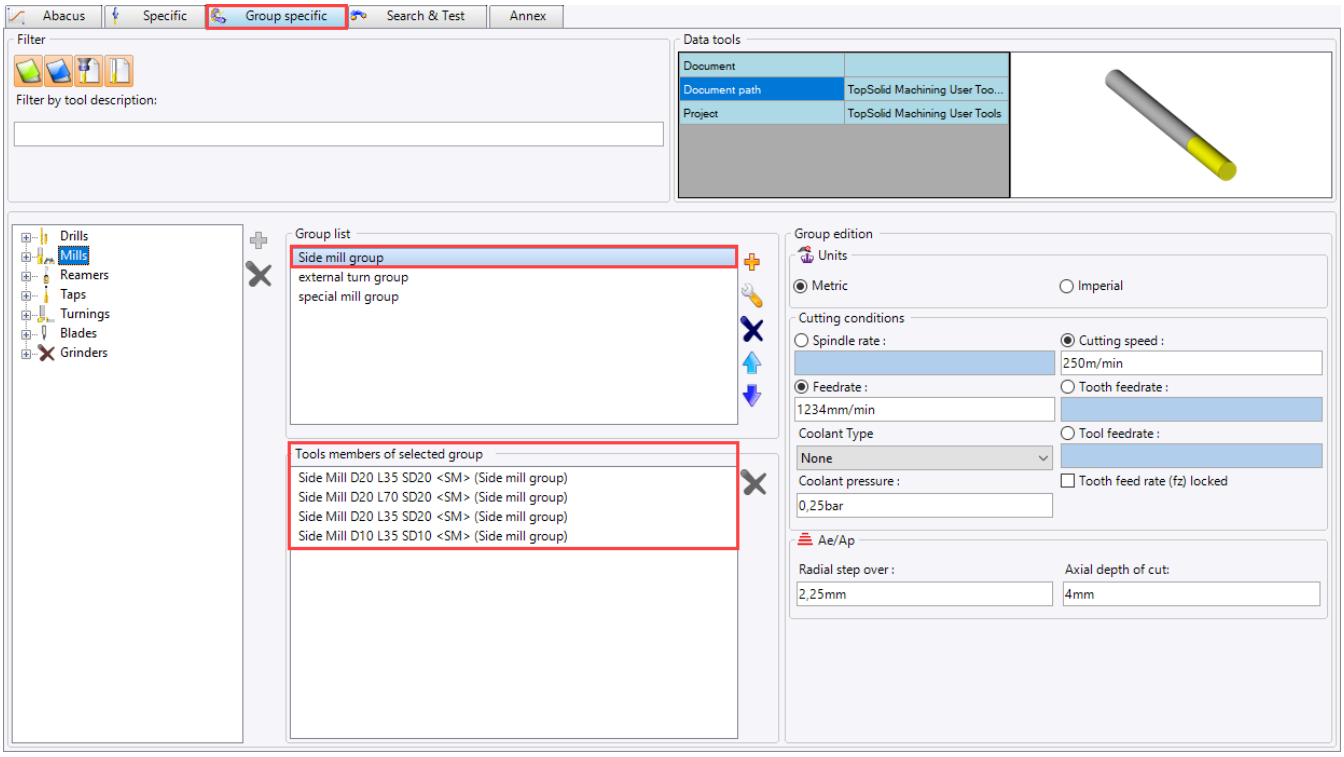
Abacus management in threading (turning) operations

In the abacus, you can now choose the **Speed ISO output**, and therefore enter the spindle rate instead of the cutting speed.

When applying in an operation, if the **Speed ISO output** (in turning operation) parameter exists, it will be taken into account.

Group specific mode

Before this new feature existed, you had to create a specific abacus per tool. However, several tools can have the same cutting conditions. To avoid duplicating these cutting conditions per tool, a new entity has been created. Therefore, a new tab appears in the cutting condition document.



Customized coolant and pressure

You can create a document with different coolant associations and different pressure ranges. If a coolant document is linked to the machine used in **TopSolid'Cam**, the coolant drop-down list will only be filled with the associations available in the coolant document.

Note: You can set up customized coolants in the methods.

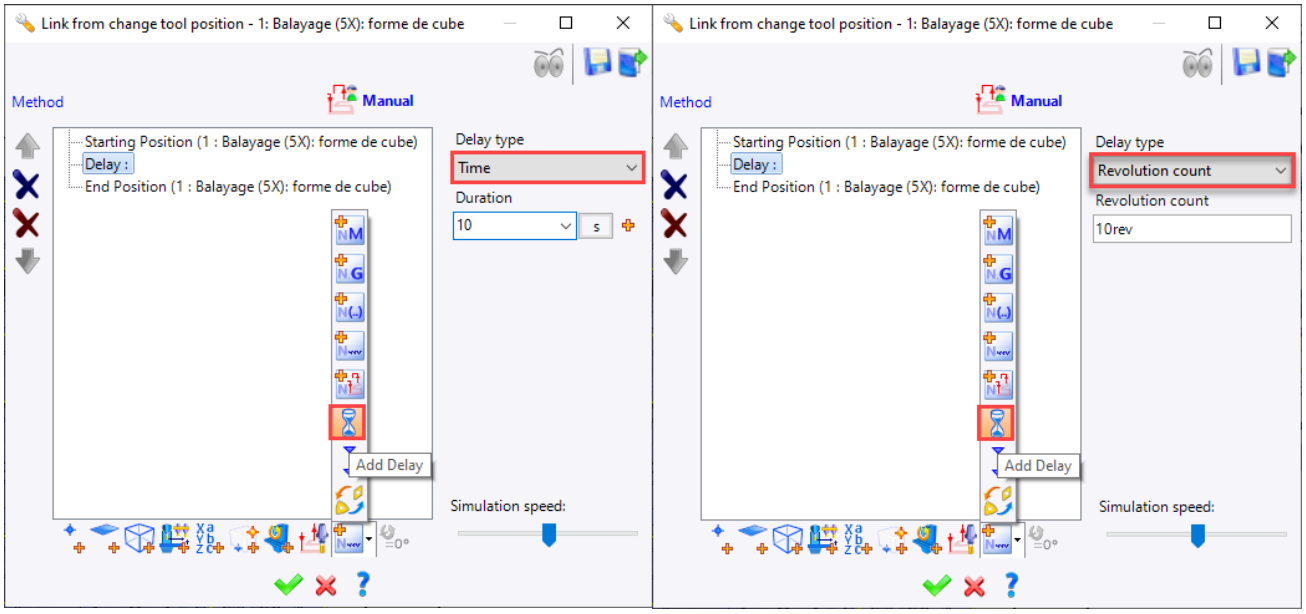
Restriction: These parameters cannot be modified in the machining document. If a coolant document is linked to the machine document, you cannot use a custom drilling document.

In the methods, when both the **coolant** and **custom coolant** are filled in, the custom coolant takes precedence if the chosen association exists in the document linked to the machine. When the step format is not correct, the default values will be used.

Link Movements

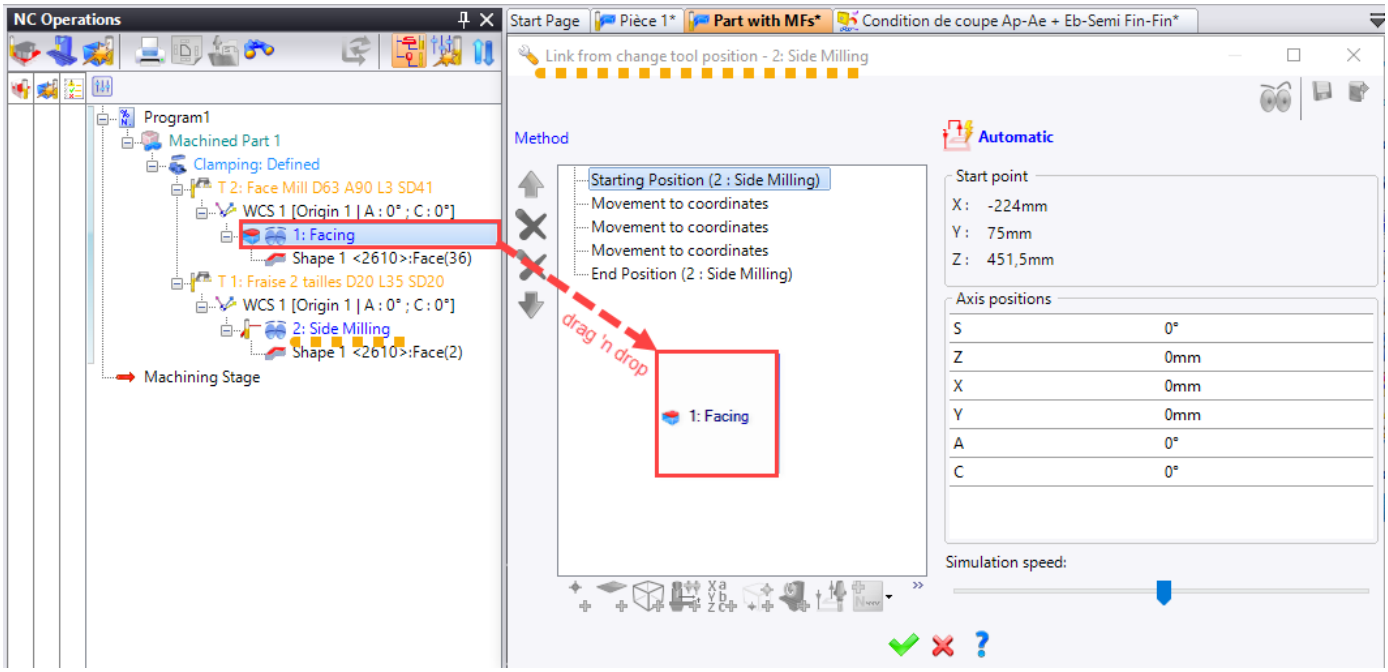
Delay

You can now define a delay (in time or in counts) in the link movements.



Copying and pasting a link movement from one operation to another

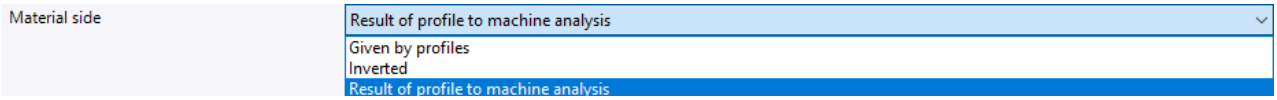
You can now copy the link movements from one operation to another without prior backup. To do this, you simply have to drag and drop the source operation into the dialog box to edit the link movement of the destination operation.



Methods

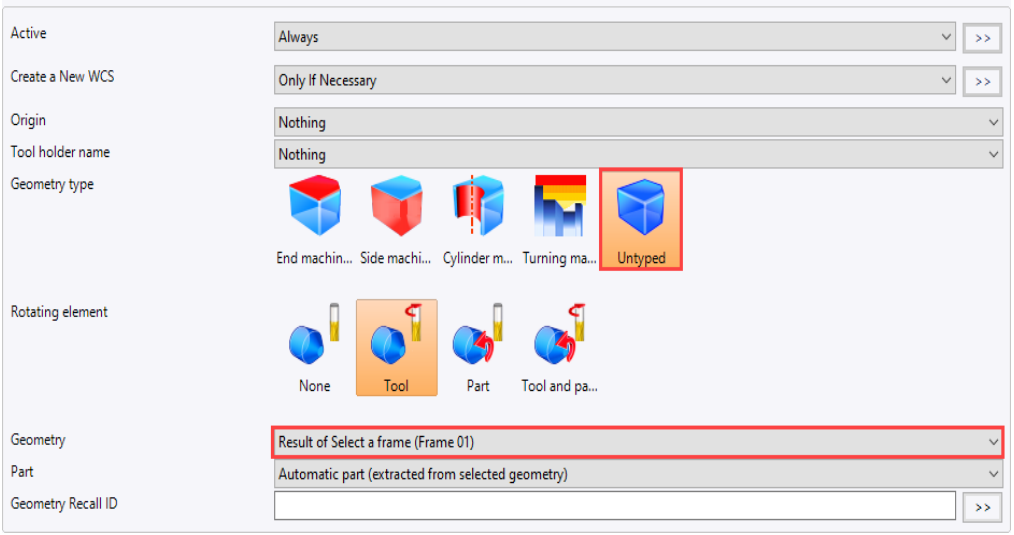
Managing the material side

As is already the case with side milling operation, the choice of the material side is now available for the following operations: **Chamfer milling**, **Radius milling** and **Side plunge**.



Finding work coordinate system on frame

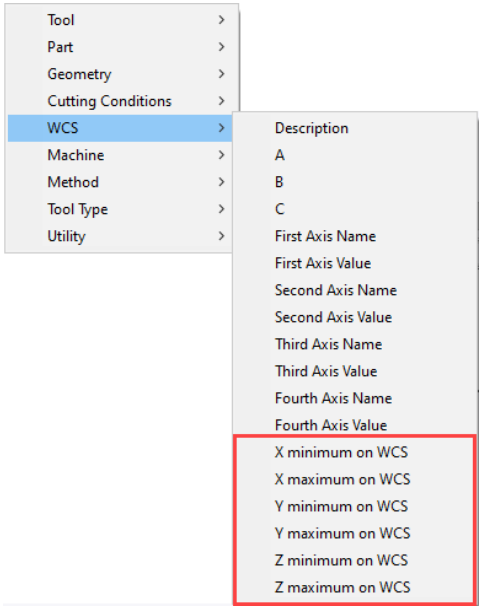
You can now create a work coordinate system on existing frame directly from the method document.



Note: You can also select a solution interactively when performing its method if it includes an "element selection".

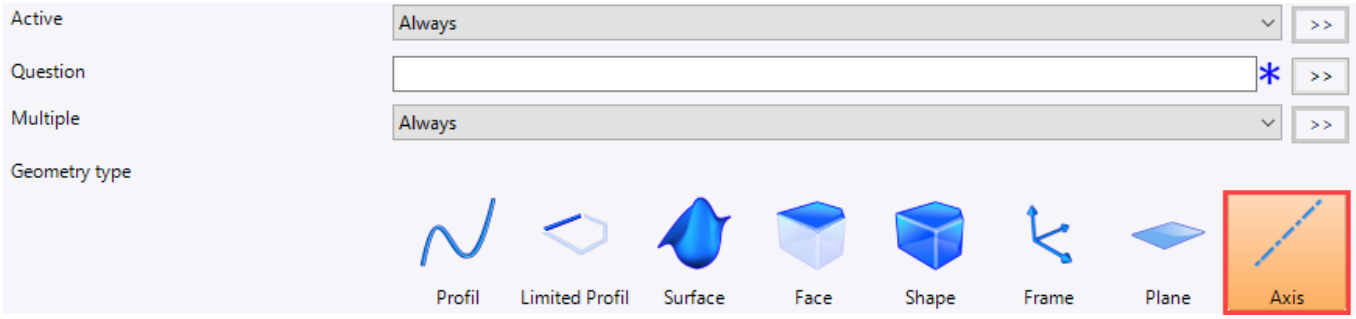
Using the angular values of the work coordinate system as a comment on the operation

This new version allows you to build an operation commentary from the angles of the work coordinate system solution chosen by the machining. The properties of the current solution are now available in the methods.



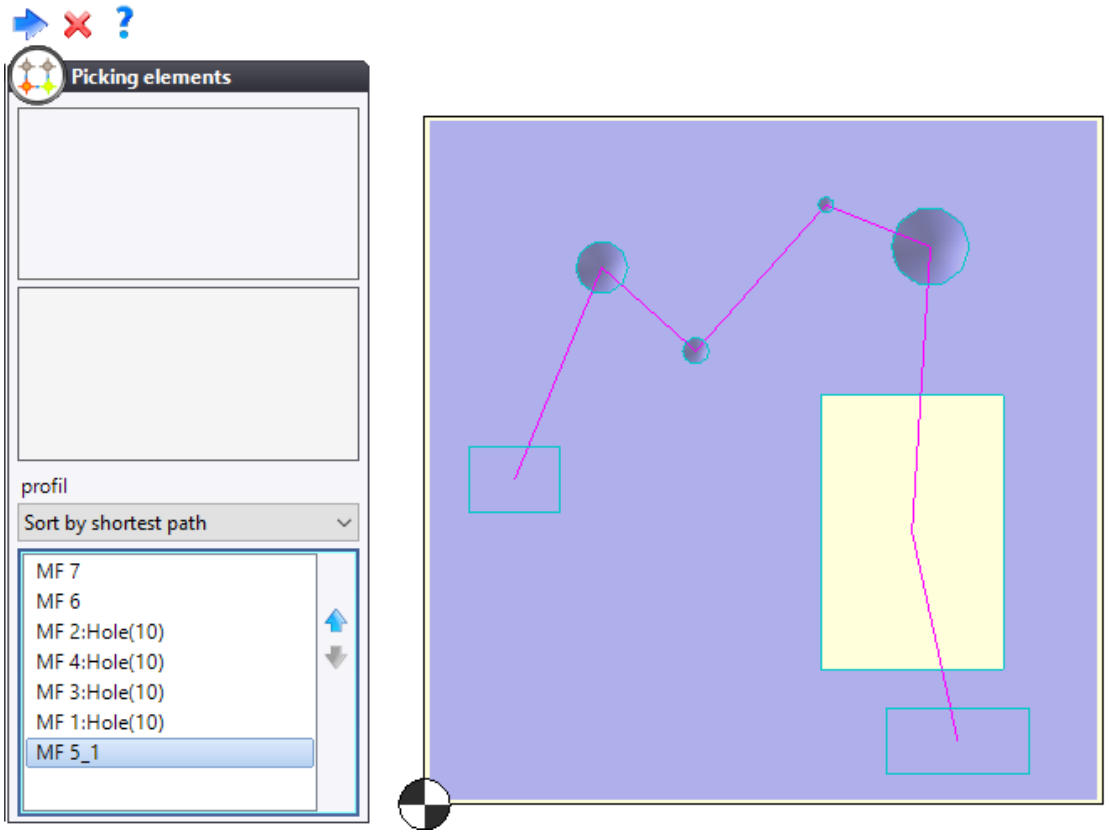
Direction management

In the methods, the goal is to manage the **Direction** parameters present in the operations. To do this, the **Axis** elements are available.



Optimizing geometries

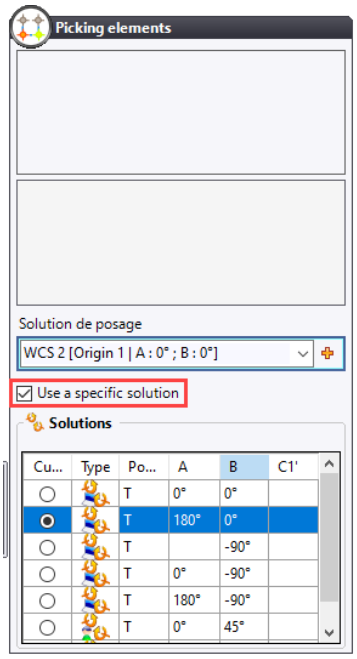
You can sort the selections automatically or manually in the method selection dialog box.



Restriction: Face selection is not supported for automatic sorting. This sorting mode does not take precedence over any sorting criteria specific to the operations.

Modifying or defining the work coordinate system solution when performing methods

You can now choose the most suitable work coordinate system solution from the list of available solutions when performing the method. To do this, a **Use a specific solution** checkbox is now available.

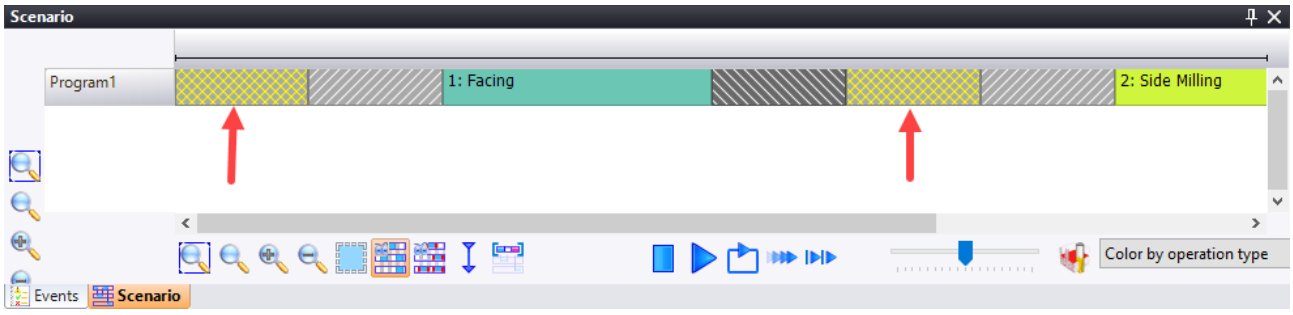


Miscellaneous

Scenario

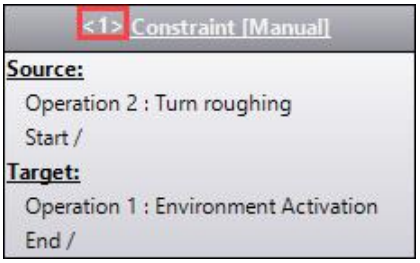
Colored tool change

In the **Scenario** dialog box, the tool change movement now appears in a different color in order to easily differentiate between approaches and retracts tool changes.

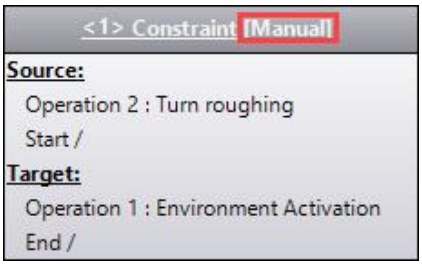


Tooltip on constraint

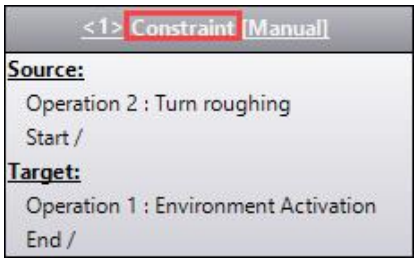
- **Constraint number**: In the scenario, when you place the mouse on an operation, the tooltip now appears with the constraint number in the title in the **<number>** form.



- **Constraint type**: In the scenario, when you place the mouse on an operation, the tooltip now appears with the type of constraint in the title as **[Manual]** for manually added constraints and **[Auto]** for constraints added with a name.

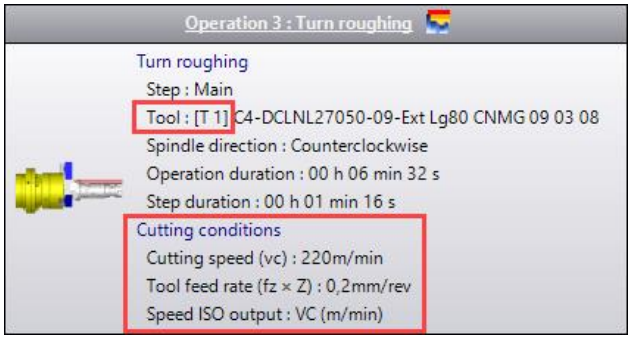


- **Constraint name**: In the scenario, when you place the mouse on a **constraint**, the tooltip appears with the name of the constraint in the title when it has been placed with the name.



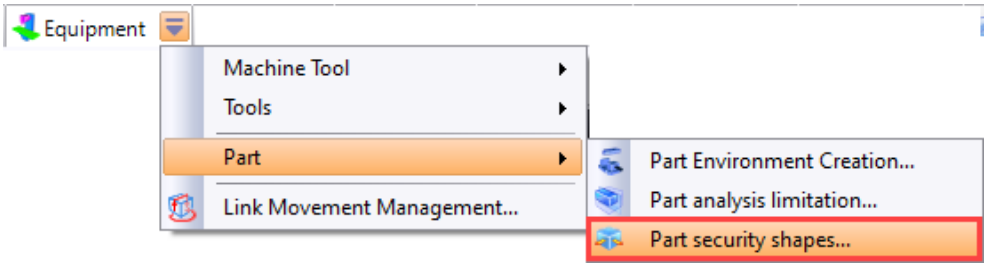
Tooltip on operation

In the scenario, when you place the mouse on a constraint, the tooltip appears with the **cutting conditions** and the **tool [number]**.



4/5-axis machining: Global security shapes

You can define a security shape (block, cylinder, sphere, plane and direction) global to the part, so that all 4/5-axis machining can be based on the same shape. Based on the same principle as the security shape of the operation, these shapes can be defined manually or automatically.

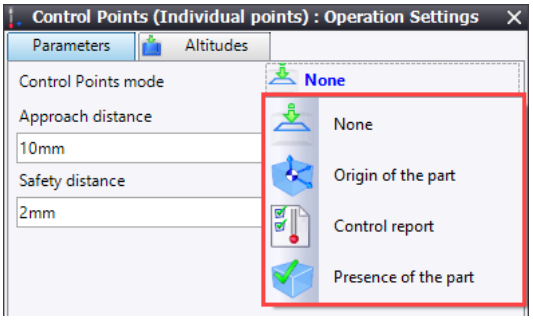


Restriction: Part security shapes can be used in an operation but cannot be modified in this one. You have to edit the entity directly for any modification.

Probing

Probing type

In order to dissociate the different types of machine-programmable probing and to provide the post-processor with the corresponding codes, probing modes have been added to all available probing primitives:



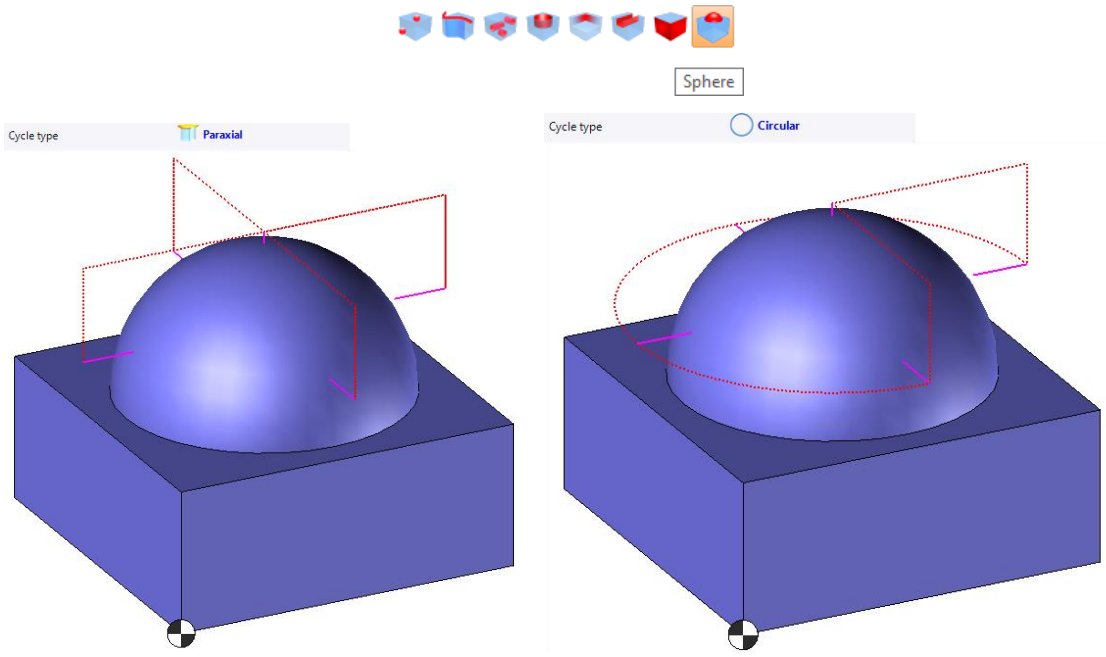
- Presence of part/flange: positioning probe
- Origin of the part: probing for the origin
- Control report: measurement probing

Geometry selection mode

In order to overcome the problems of picking geometry in probing operations, the selection mode has been modified. A new geometry selection dialog box is now available for each probing operation.

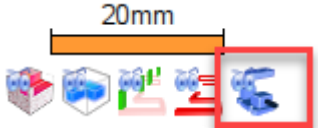
Probing on a sphere

In the list of primitives, a **Sphere** primitive is now available.



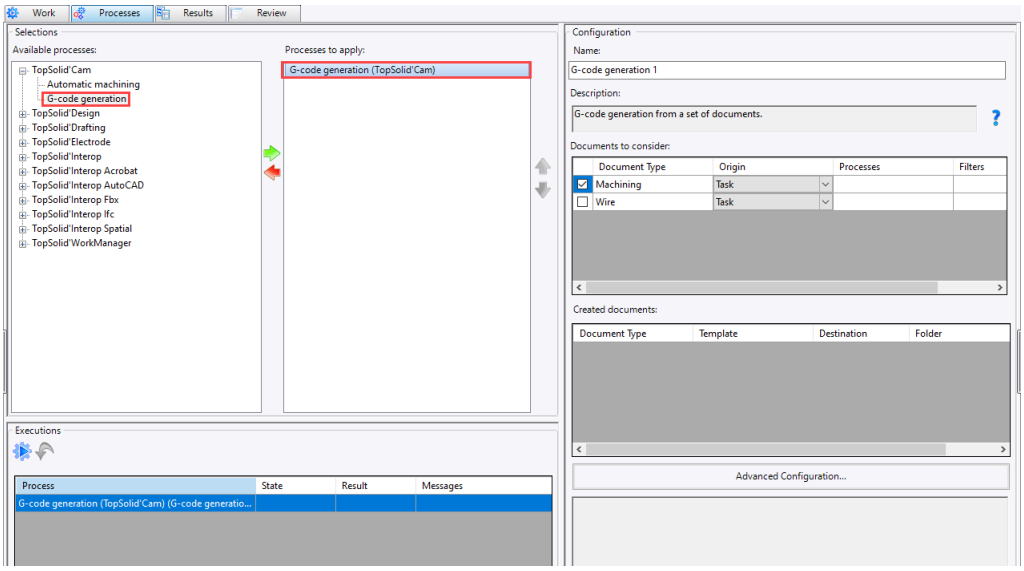
Environment: Hide/show

You can now hide or show environment elements by simply clicking on the icon directly from the graphics area.



Working document: Generating G-codes

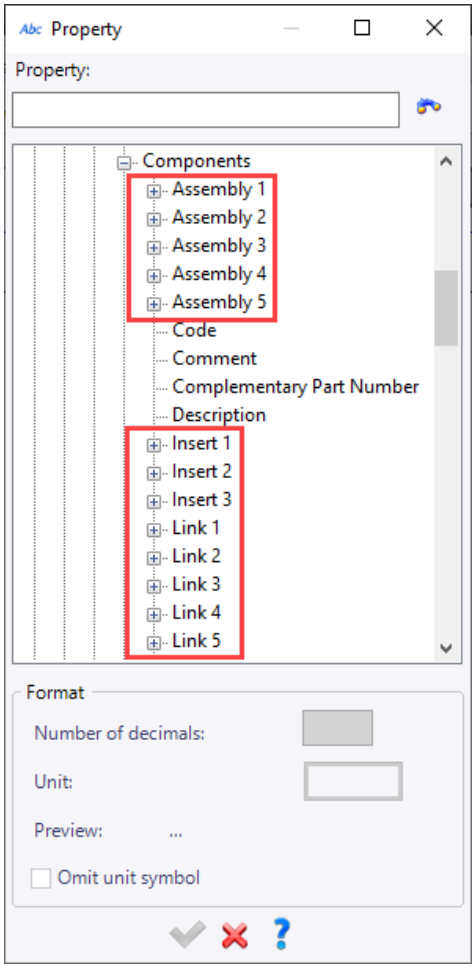
A new process is now available in the work document to generate G-codes from a batch of machining documents.



Shop floor documents

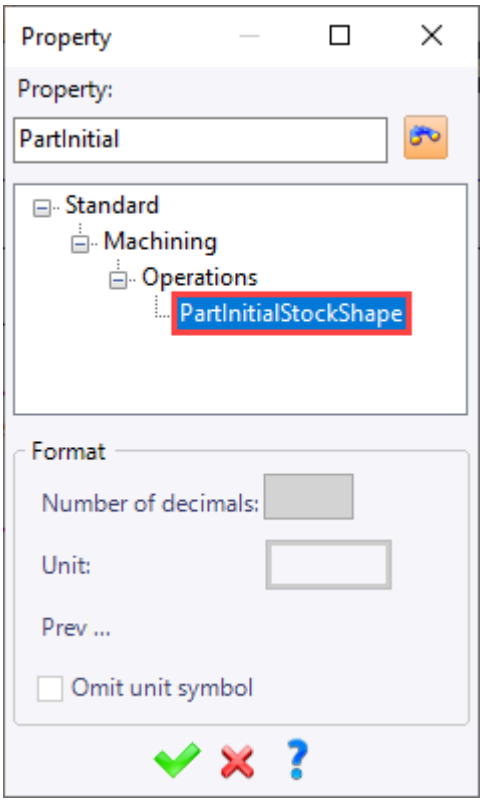
Adding component variables

The number of tool components available in the shop floor documents has been increased. As a result, three inserts, five assembly components and five links are now available.



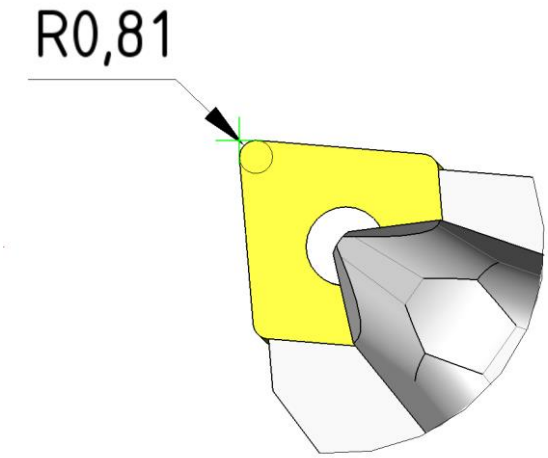
Projecting the stock to the initial state

You can have the initial stock in the shopfloor documents (not the final stock). This allows the operator to have a better view of the assembly, the adjustment on the machine and the preparation to be done. A new property is now available: **PartInitialStockShape**.



View of the tool driven point position

A symbol has been added showing the driven points used for a machining operation. The position of each driven point is displayed by a marker whose style can be configured (green cross in the image below) and a circle corresponding to the plane compensation (2D offset).



TopSolid'Cam Operator

Touch tablet or keyboard/mouse mode

You can choose its preferred configuration, especially according to its screen, whether touch screen or not. If the screen is not tactile, the behavior is similar to the well-known keyboard/mouse behavior used with **TopSolid'Cam**, in particular with the contextual menu that is displayed when you right-click the mouse.

Administrator rights management

This allows the PDM administrator to allocate modification rights to specific parameters for each operation and according to the user.

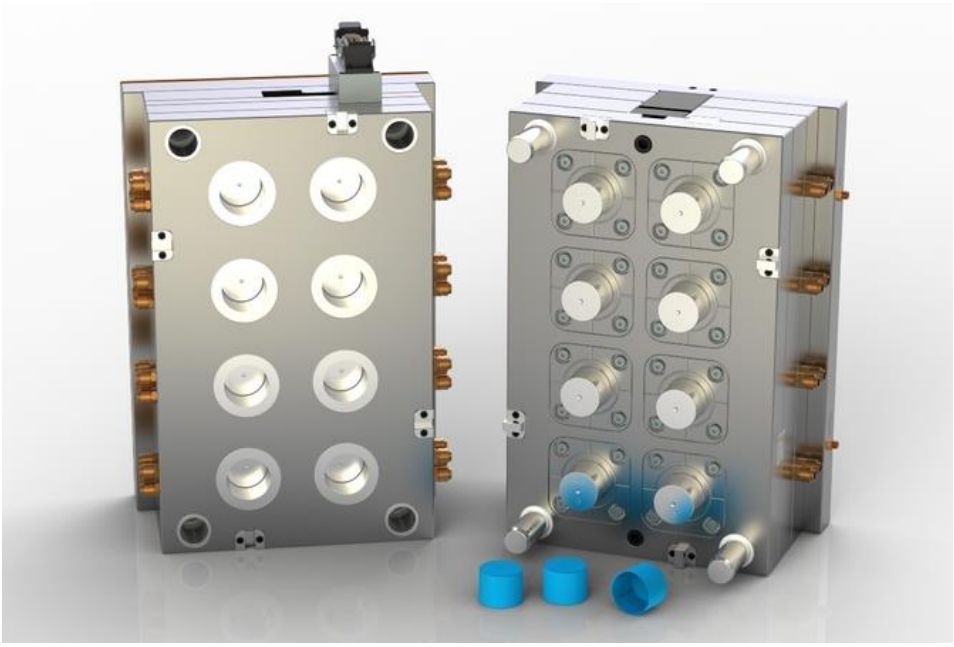
Drafting document

A new command has been added to the machining documents displayed in the search list. This command will only be displayed when the machining document has drafting documents referenced to it.

Operations tree

The filters of the Operations tree are now available: displaying program lines, NC part lines, environment creation lines and displaying tool lines.
Multi-operation modification is now possible.

What's New in TopSolid'Tooling 7.16



This section describes the enhancement made to the **Split**, **Mold**, **Strip** and **Progress** applications in version **7.16** of **TopSolid 7**.

TopSolid'Split

Moving the Split options

Some of the **Split** document options were located in the **Tools** > **Options** dialog box of the application:

- Default **colors**;
- Options for creating the **stock**.

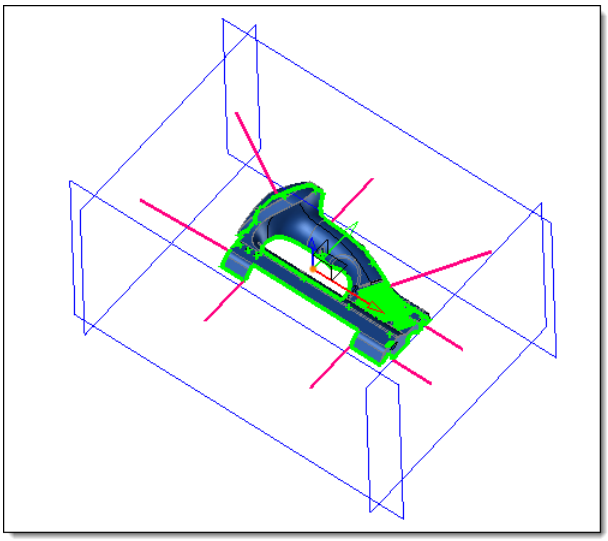
These options have been moved to the **Split Blocks** folder of the options of each split document.

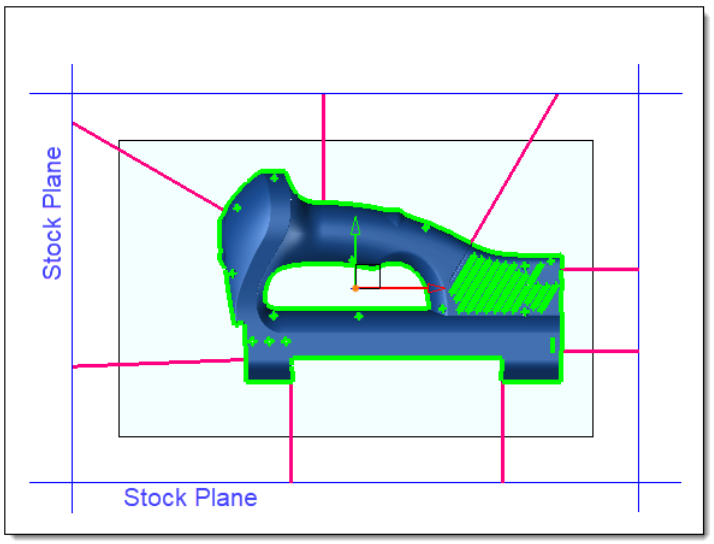
Selecting parting edges

Parting edges selection picking was sometimes confusing depending on the number of selections available: tangent edge path, internal face loops, etc.
A new **Single edge selection** advanced option allows you to manage what is offered in first selection, in this case, systematically the **single edge** hovered by the mouse. Smart selections, available from this edge, are then offered via rotary picking.

Vector profile

You can now define the length of the vector profiles by choosing to trim by **plane**, or by the **stock planes**.
In this case, the vector profiles are certainly long enough to define surfaces entirely trimming the core cavity blocks:





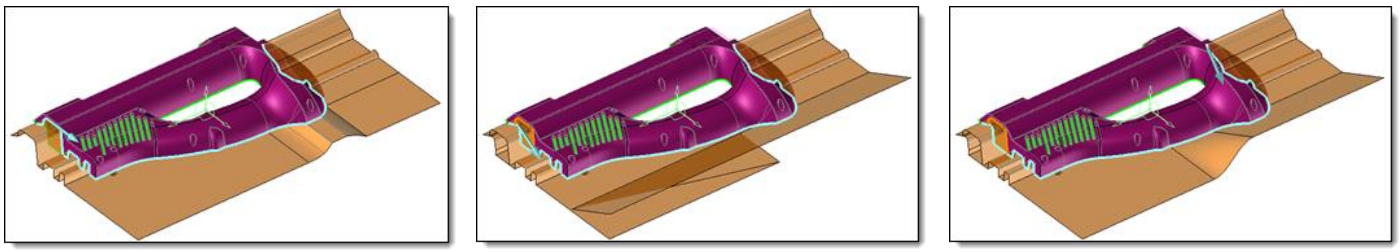
Planar parting surface

The behavior of the plane edge path detection has been modified to make the creation of this type of surface easier:

- When a **start edge** is selected, the longest possible path of edges included in the same plane is automatically detected.
- And the **end edge** of this path is pre-filled with the last edge of the same plane. It is always possible to restrict it by choosing a new end edge.
- Now, when **reversing** the direction of the path, the software automatically recalculates the longest possible path in the other direction from the start edge, and reassigns its new ending edge.

Automatic parting surfaces: start edge and direction of parting lines

The calculation of automatic parting surfaces depends on the start edge in the processed parting line and the direction of this line. In order to stabilize the results produced by the command, this origin and direction are now determined in a constant manner with respect to the mold frame.
In addition, you can now **select the start edge** and **reverse** the direction of the path. These new contextual commands are available by right-clicking on the list of available parting lines.



Various automatic parting surface calculations depending on the start edge and the direction of the selected line.

Shells and previews

Calculating the sewings of each of the parting shells can be quite time consuming and each modification of options in this section results in a recalculation of the selected shell. You can now disable the shell creation preview so that any desired changes can be made more quickly in the case of complex surfaces.

TopSolid'Mold

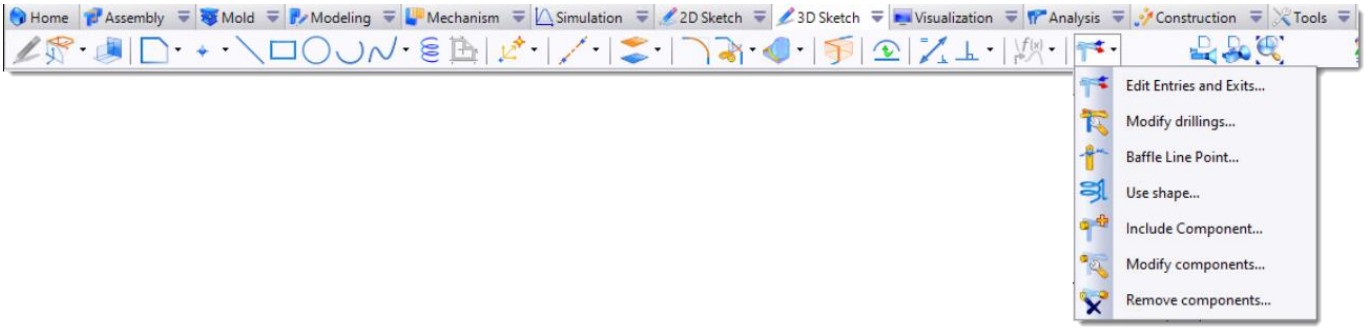
Other Set

A new **Other** set category is now available in the mold.
In addition, its viewing shortcut is available in the quick view bar.

In order to benefit from this new set in a document created in previous versions, you have to launch the **Recreate Die Sets** contextual command, from the **Sets** folder in the Entities tree.

Cooling and reorganizing commands

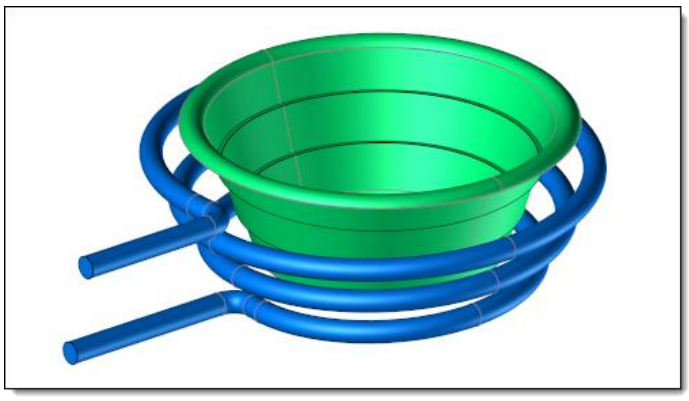
Until now, many of the commands used in the cooling sketch were only available in the contextual menu. They have been grouped in a **Cooling sketch** section to make them easier to find in the contextual command menus.
In addition, these commands are now all grouped at the end of the 3D sketch menu.



Cooling

A new **Use Shape** command available in a cooling sketch, allows any shape to be considered as a cooling positive. Therefore, you can create a conformal cooling after having modeled or imported this positive into **TopSolid**.

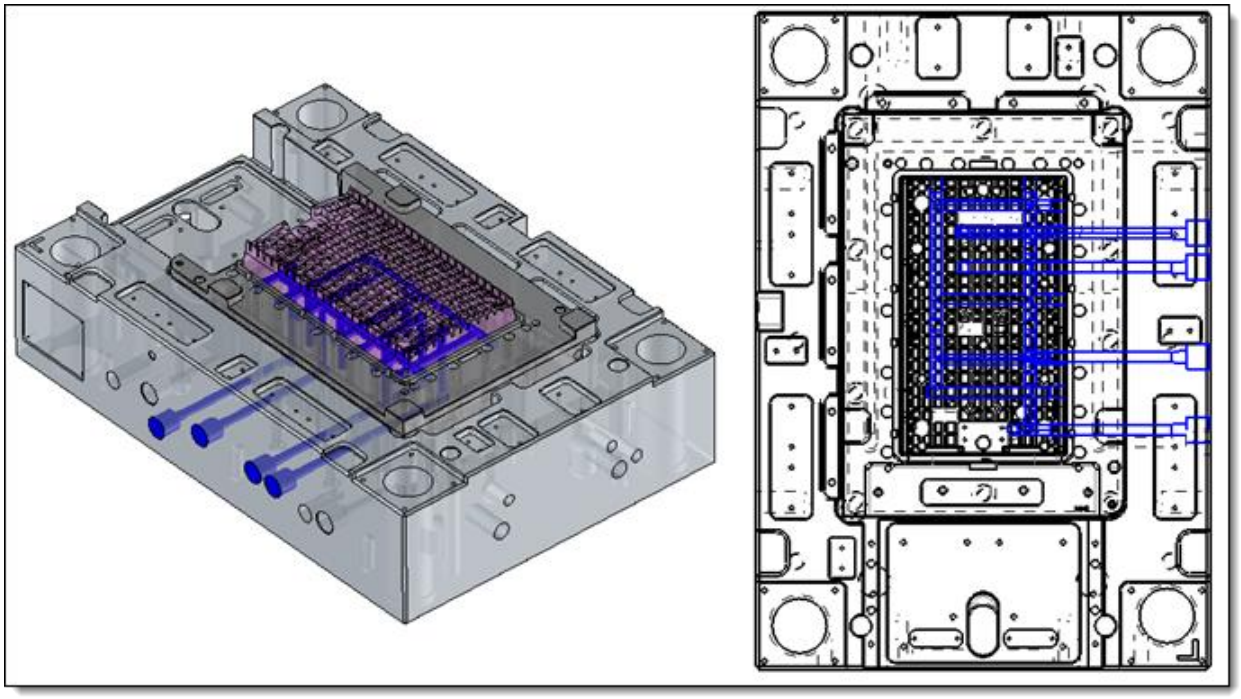
The **Cooling Circuit Process**, **Safety Margin** and **Collision Analysis** commands in the cooling sketch, the **Cooling Circuit Leaks** and **Cooling Circuit Attributes** commands in the drafting also take this new type of cooling into account.



Example of a conformal cooling

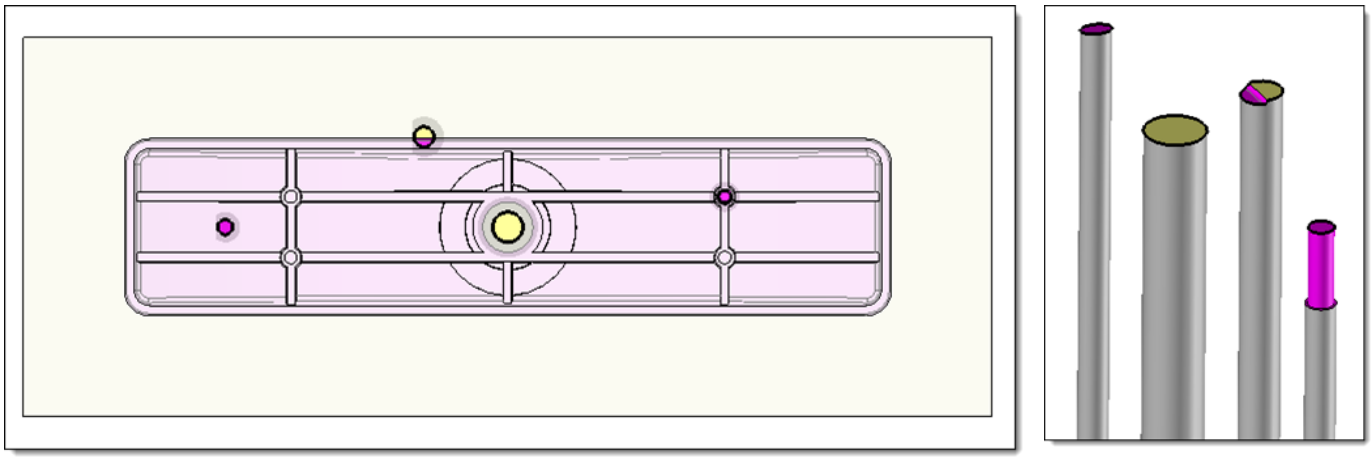
Cooling: Definition of circuit in an imported part

Thanks to this new **Cooling Shape** command available in the **Shape** menu of a part document if the mold is installed, you can define a cooling circuit in the parts of a mold imported via translator in **TopSolid**.
The command allows you to create a cooling positive from the faces of the cooling drillings present in the part.
The positive is then used in the mold to create a cooling circuit and its process. Therefore, the **Cooling Circuits Collisions**, **Cooling Circuits Leaks** and **Cooling Circuit Attributes** commands can be used as if the cooling had been created with the "classic" cooling commands.



Pins: Color the trimmed faces

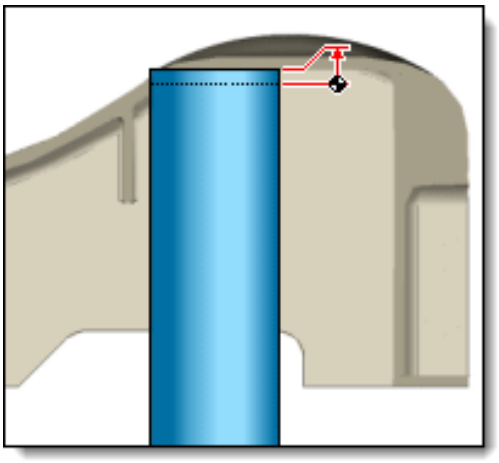
A new **Color faces** option is now available in the section defining the trimming options of the pins.
If this option is enabled, the faces created by trimming the pins will recover the colors of the core cavity blocks that were defined when creating the parting shapes in the **Split**.



This new mechanism can only be applied to new documents created from version **7.16** of **TopSolid**.

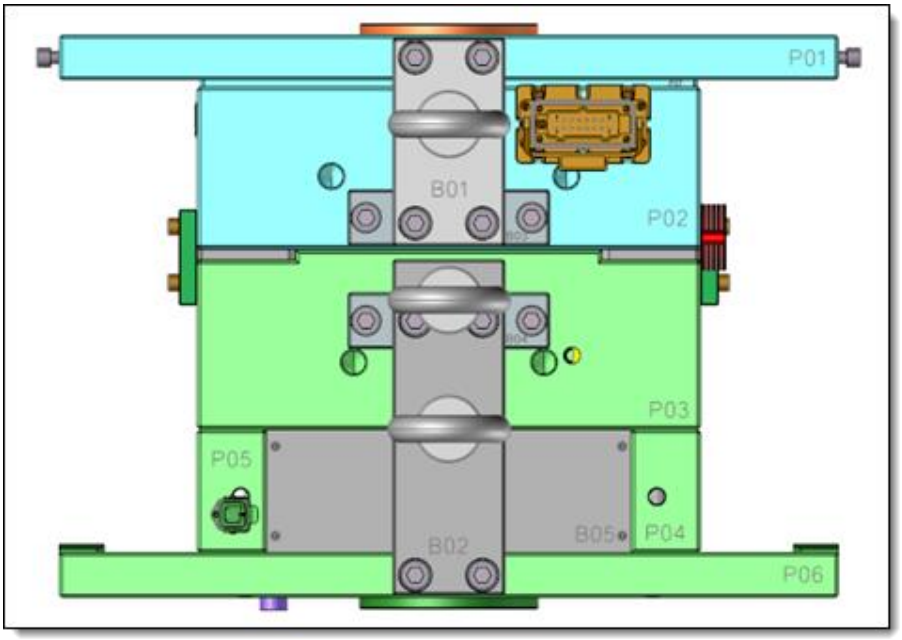
Pin process: Non-operable part

When trimming a pin, you can trim it by the upper plane of the molding part that it meets at the parting plane.
For the files created from this new version, the part, however in collision with the pin is now automatically ignored in the search of the targets to be operated by the process.
You no longer have to create the process in manual mode.



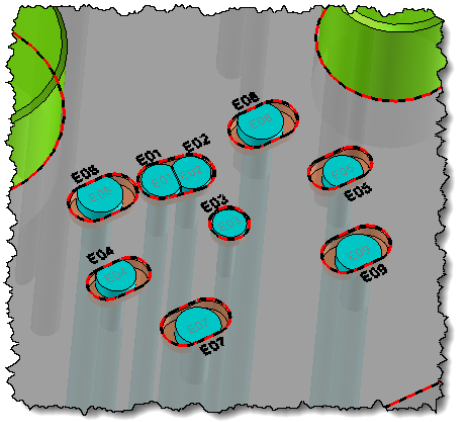
Marking

Thanks to the complete redesign of the **Marking** command, you can now very easy to mark all parts, from the **Mold** document. You can engrave any text or a sketch, etc. as well as properties of the parts, such as the **manufacturing indexes**.



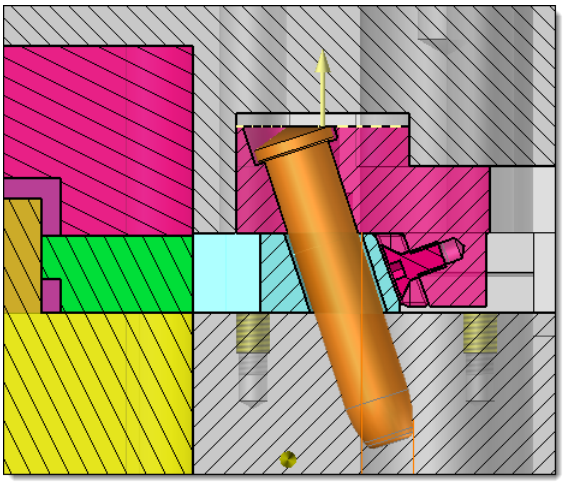
Pins marking

This new command allows you to automate the marking of the pins and/or of the plate in which the pin head housing is made.
By default, the command is used to mark the **Manufacturing index** property. However, you can choose another **property** of the pin, or a **text**.
When the command is launched, the font, size and direction settings are chosen based on the first pin. These settings are then automatically associated to all the pins present in the mold. The positions of each engraving can be adjusted with each pin.



Graphic cut

A new graphic **Cut** command is now available in the **Angle Pin**, **Angle Pin Pocket** and **Slide Joint** commands.
This command allows you to either enable a graphic cut of the mold or not to facilitate the inclusion of a pin, for example. It is then automatically disabled when exiting the command.
You can configure the attributes of this cut in the options of the mold document.



Slide joint

This command has been completely redesigned in order to apply major improvements.

- The profiles created are now simplified to the areas actually in contact between the slide and pin or corner. Therefore, the calculation of the kinematics becomes much less time-consuming, because as soon as the shapes are a bit complex, the intersection profiles are also complex, and the time to solve the kinematics increases.
- You can reverse the direction of the profiles created in order to respect the direction of the material of each part in contact.

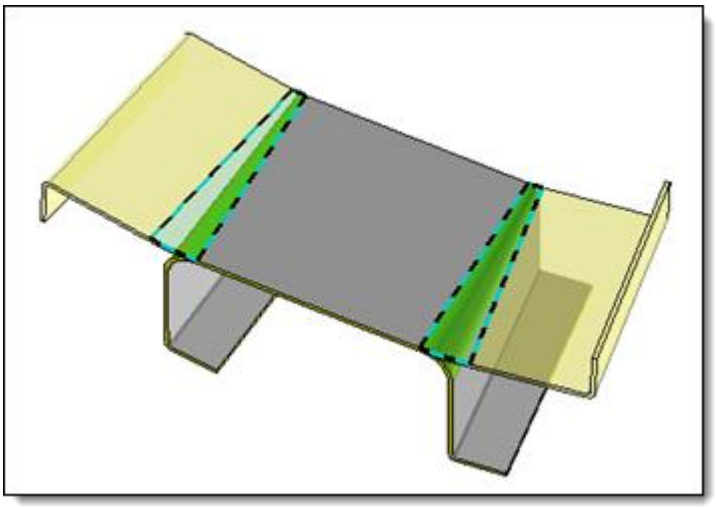
TopSolid'Strip (Strip Design)

Drivers

In order to facilitate the positioning of the pilot symbols on the strip, a **Filter full circles** advanced option has been added that allows you to select only complete circular edges.

Unbending of bend

A new **Developable bend** mode has been added to the command. Therefore, you can now to unroll strictly conical bends to create the different station shapes of the strip.

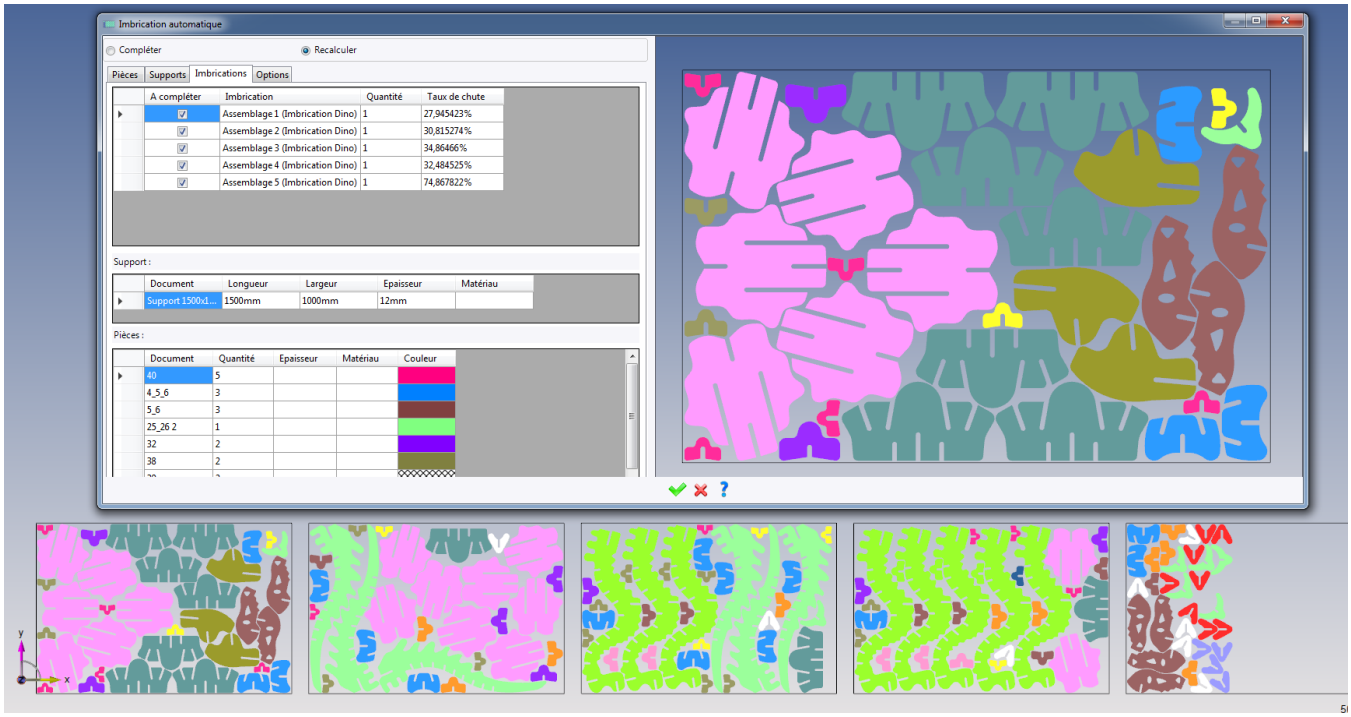


TopSolid'Progress (Progressive Die)

In-place punches

The modeling tolerance set in the options of the template used to create the punch document is now used to create the **Punch** operation.

What's New in TopSolid'Cut 7.16

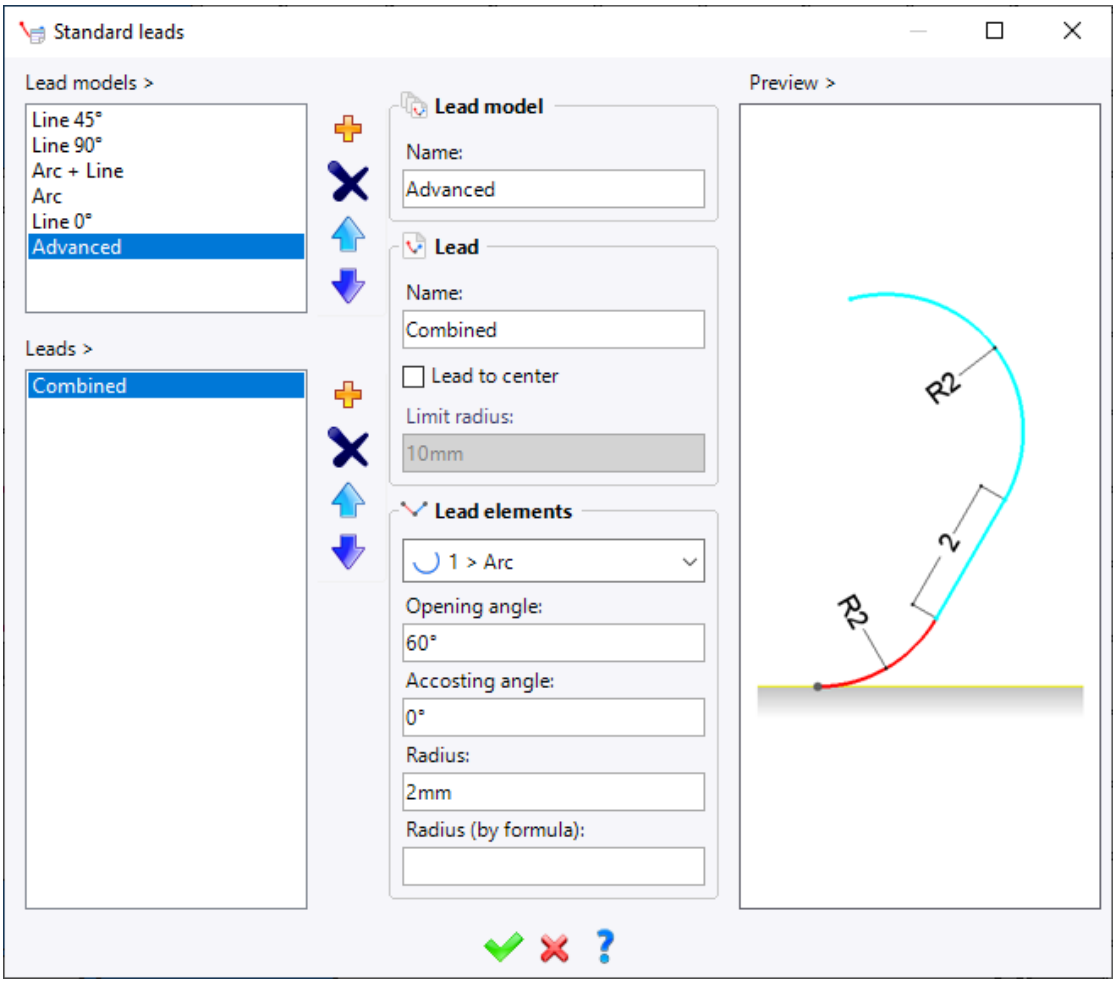


This section describes the enhancements made to the to the **Sheet Metal Machining** application in version **7.16** of **TopSolid 7**.

2D Cutting

Standard leads catalog

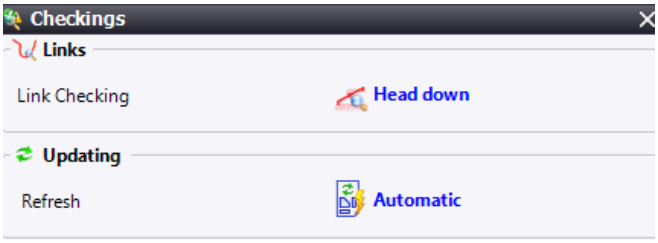
In previous versions, a predefined and not very customizable leads catalog was available.
From now on, the user has total freedom to create his own leads according to his needs. There are always "classic" leads with one or two elements (line, arc and arc + line), with the dimensional parameters.
The user can now create more complex and hybrid lead models. These new models allow you to combine lines and arcs, while managing the link between all these elements with leads angles.
The concept of a model allows you to perform a lead quickly by starting on the preset parameters in the latter.
All these models are stored in the PDM and can be accessed from all the projects in the PDM.



Standard leads catalog.

Management

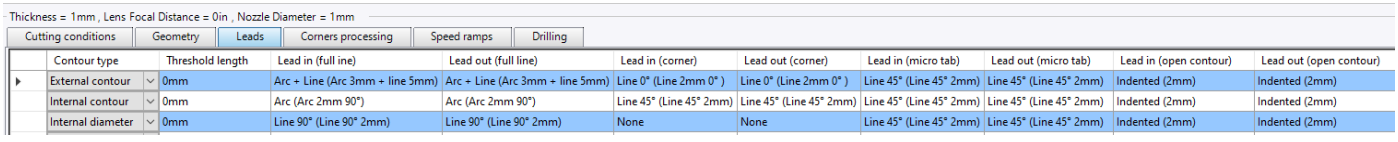
The **Management** menu has undergone a major enhancement in version **7.16**.
The success of the automatic assignments in the process relies entirely on the correct configuration of the management operations. Already present in version **7.15**, the lead, micro-tab and cutting parameter management has been improved at the interface for a greater clarity.
In version **7.16**, six new management operations have been added: standard leads, cuttings management, path optimization, burning, drilling and checkings.



The fine tuning of all these operations, together with the simplicity of the cutting operation and the efficiency of the work document, allows automation to its maximum and reduces the programming time thanks to **TopSolid'Cut**.

Automated management of leads and corner cuttings

Previously limited to one type of lead per machining document, **TopSolid'Cut** now offers unlimited parameterization of each type of lead and corner processing according to materials, thicknesses, technologies, types of contours and cutting qualities.
Concerning the leads, the parameterization of these is based on the database present in the standard catalog.

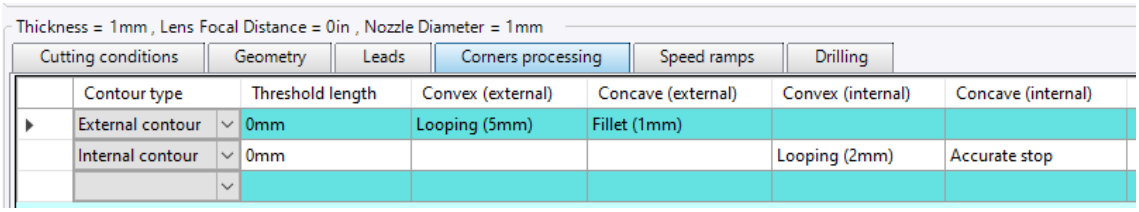

Thickness = 1mm , Lens Focal Distance = 0in , Nozzle Diameter = 1mm

Cutting conditions | Geometry | Leads | Corners processing | Speed ramps | Drilling

Contour type	Threshold length	Lead in (full line)	Lead out (full line)	Lead in (corner)	Lead out (corner)	Lead in (micro tab)	Lead out (micro tab)	Lead in (open contour)	Lead out (open contour)
External contour	0mm	Arc + Line (Arc 3mm + line 5mm)	Arc + Line (Arc 3mm + line 5mm)	Line 0° (Line 2mm 0°)	Line 0° (Line 2mm 0°)	Line 45° (Line 45° 2mm)	Line 45° (Line 45° 2mm)	Indented (2mm)	Indented (2mm)
Internal contour	0mm	Arc (Arc 2mm 90°)	Arc (Arc 2mm 90°)	Line 45° (Line 45° 2mm)	Line 45° (Line 45° 2mm)	Line 45° (Line 45° 2mm)	Line 45° (Line 45° 2mm)	Indented (2mm)	Indented (2mm)
Internal diameter	0mm	Line 90° (Line 90° 2mm)	Line 90° (Line 90° 2mm)	None	None	Line 45° (Line 45° 2mm)	Line 45° (Line 45° 2mm)	Indented (2mm)	Indented (2mm)

Leads by contour/thickness.

The corner processing is based on the parameters assigned in the corner processing management operation, but in a more advanced and customized way.



Corner processing by contour/thickness.

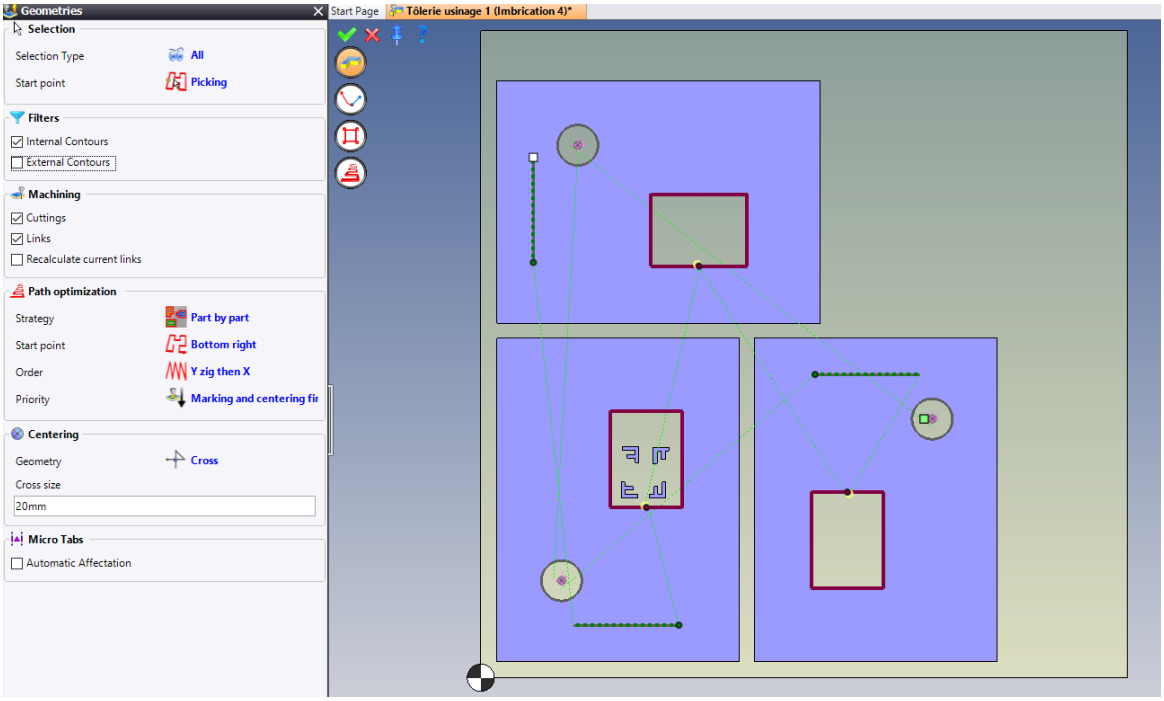
However, the leads and corner processing settings are not always appropriate. In order not to block the user in his programming, he can choose at any time not to take these parameters into account in the cutting operation.

Processing with strategy per part or per sheet metal

The path optimization strategies have been improved. You can now manage the cutting sequence per part or more generally on the entire sheet metal. In addition, other path optimization options are available:

- At the shortest, X then Y, ZigZag, etc.
- By type of operation (such as the one to perform the centering and marking operations on all of the parts before performing the cutting operations).

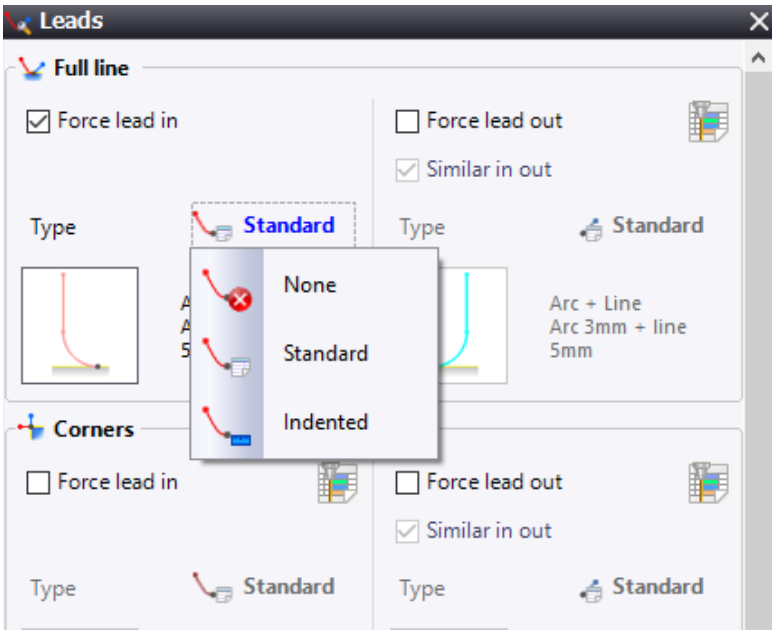
All of these parameters can be very useful to avoid overheating the sheet metal while working in the same area, for example.



Strategy per part with marking and centering.

Forcing lead in/out

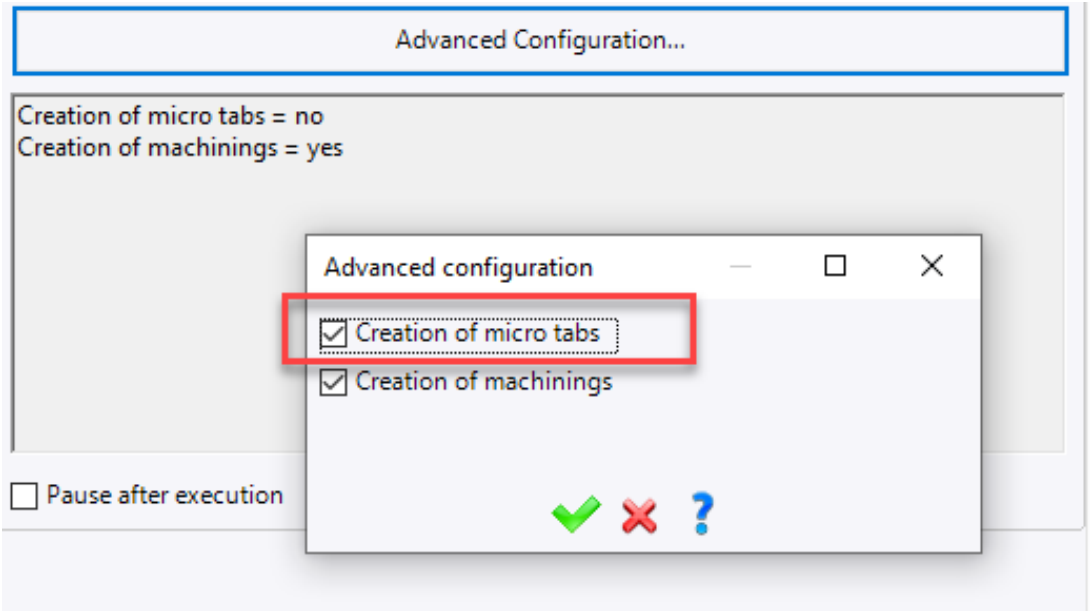
Each part has its own specificities in terms of topology, material, etc. Even a very comprehensive leads catalog can sometimes be inadequate and blocking.
In order to avoid blocking the user, **TopSolid'Cut** now integrates the possibility of forcing lead in and lead out in several places: operation, contextual menu, etc. This possibility can be applied locally or globally to a part or a document. The user therefore has great flexibility in his programming.



*Forcing lead in and lead out in the **Management** menu.*

Assigning micro-tabs in sheet metal processing

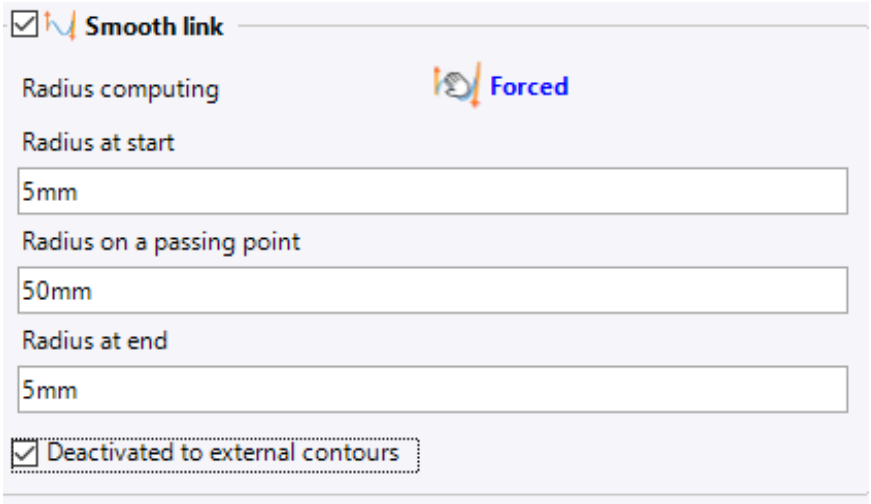
In the previous versions, the creation of machinings was only available. In order to improve the efficiency of the automation in the work document, you can now create and assign micro-tabs in the sheet metal machining process. As with the other processings, it is based on the parameters configured in the template document management operations.

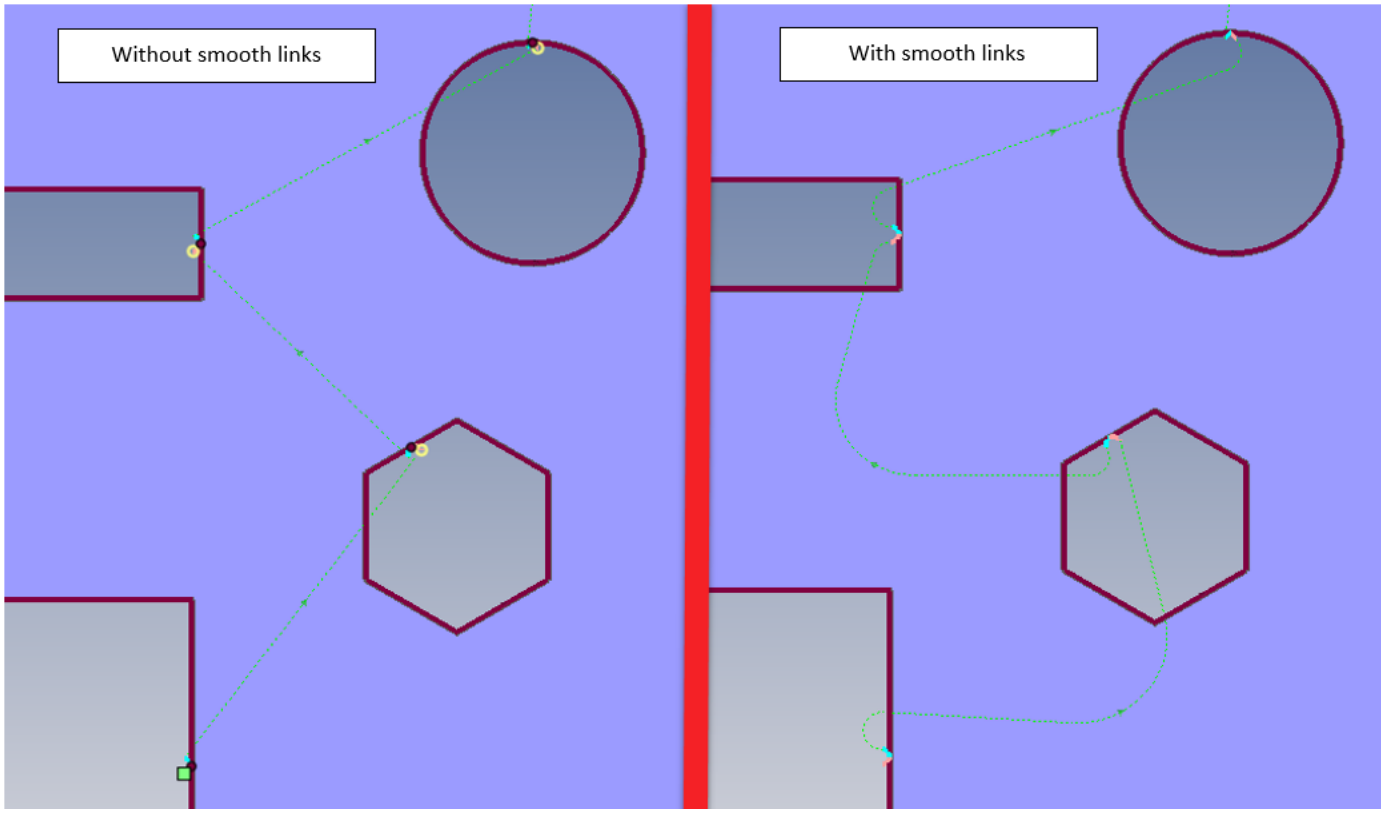


Micro-tabs in the processing.

Smooth links

Machine performance is constantly evolving dynamically, with increasingly feed speeds, particularly for rapid movements other than cutting. **TopSolid'Cut 7.16** now integrates smooth links. These make it possible to round off sudden changes of direction in order to reduce significant deceleration by maximizing passing speeds.
Combined with cutting speed ramps management, and the passing points, the user can therefore control manually and/or automatically all the movements.





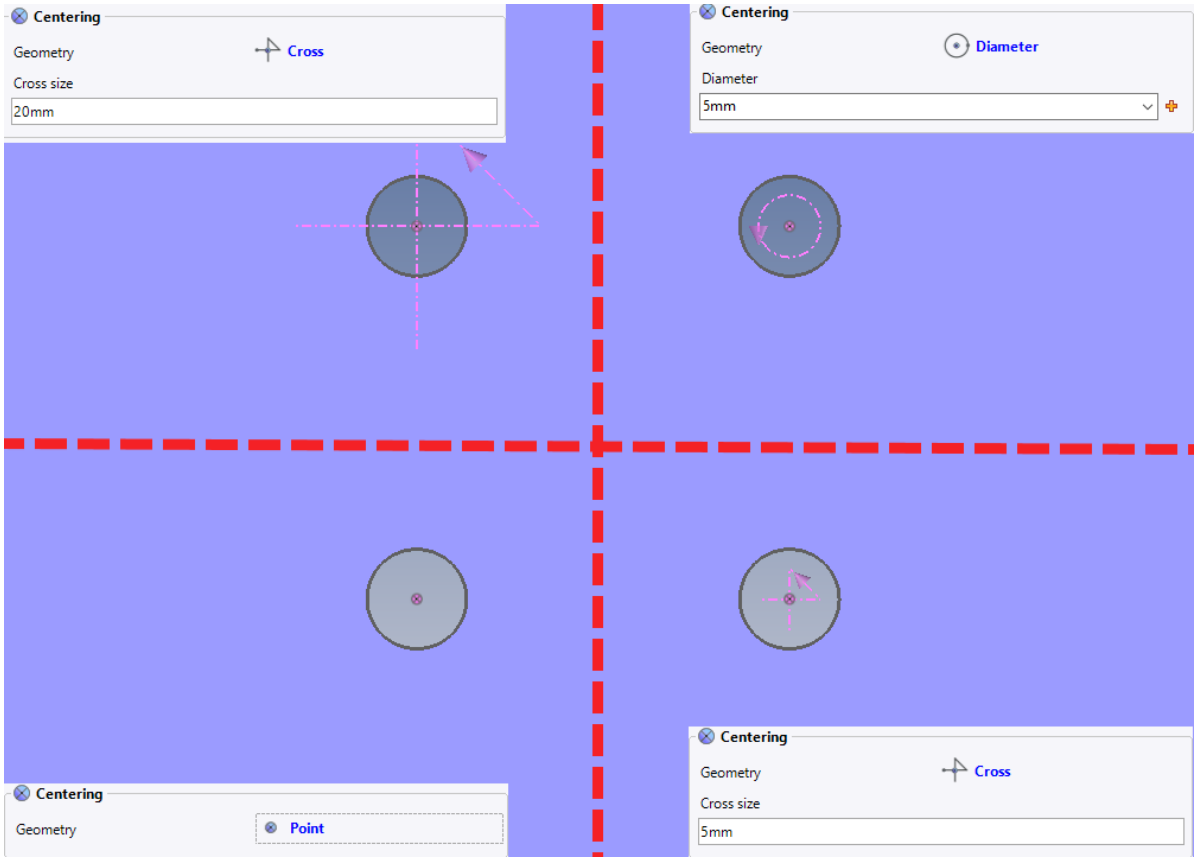
Smooth links

Cutting on sketch

In this new version, you can directly select sketch entities and cut them. This command is completely transparent to the user but gives him a lot of flexibility.
This allows you to add elements directly to the machining document without having to go through the part document.

Centering operation

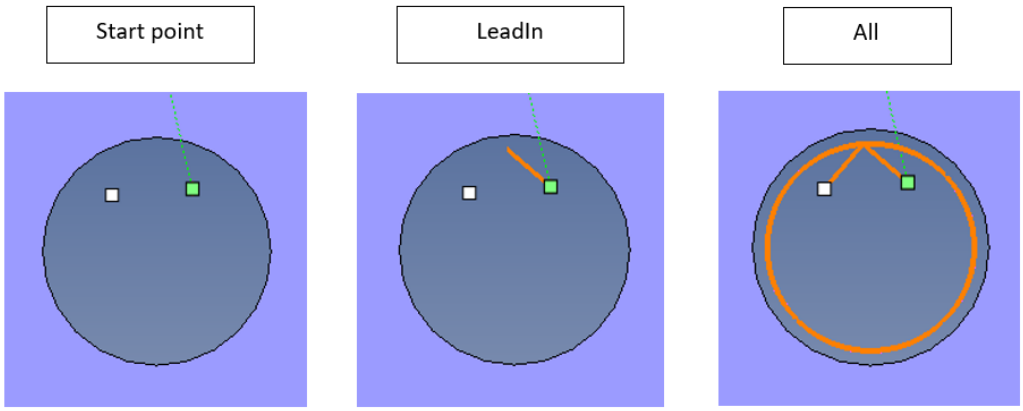
Among the great new features of version **7.16** is the centering operation. It is a full-fledged operation that is fully configurable. It can be applied both manually and automatically, on diameters but also on small internal cuttings. The concerned profiles will be dimensionally configured in the technology document to manage their eligibility for the application of the operation. Three types of centering are possible: point, diameter and cross. Obviously, each centering can be modified locally.



Centering types

Burning operation

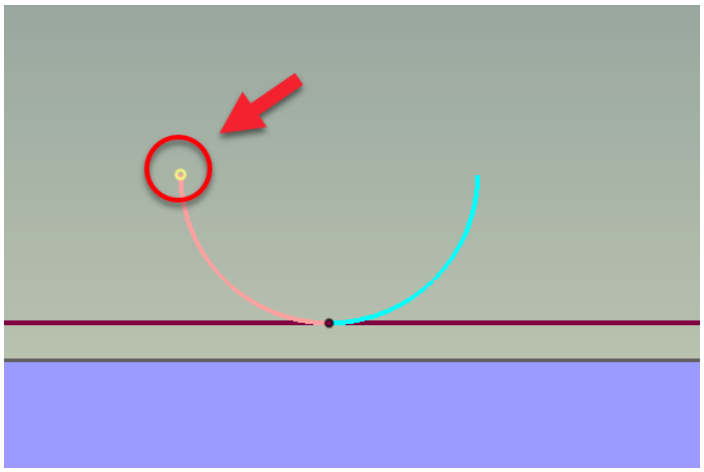
As for the centering operation, the burning operation appears in version **7.16**. Unlike the previous version, it is considered as a preparation operation.
Of course, the burning technology must be created beforehand in order to apply it. The user has several options to apply a burning: none, start point, lead and all the contours. In addition, you can assign burnings to the desired cuttings. However, only one type can be configured throughout the document, unlike a centering operation.



Burning types

Drilling operation

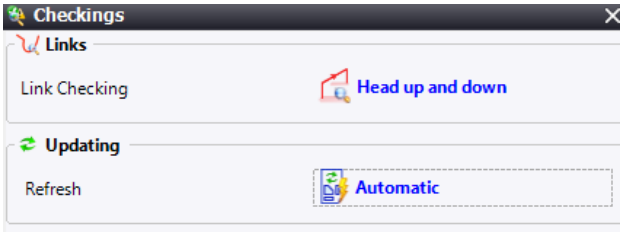
The drilling operation has been integrated in this new version. It can be configured to perform drilling operation before each cutting or to be added on all the contours of the part.



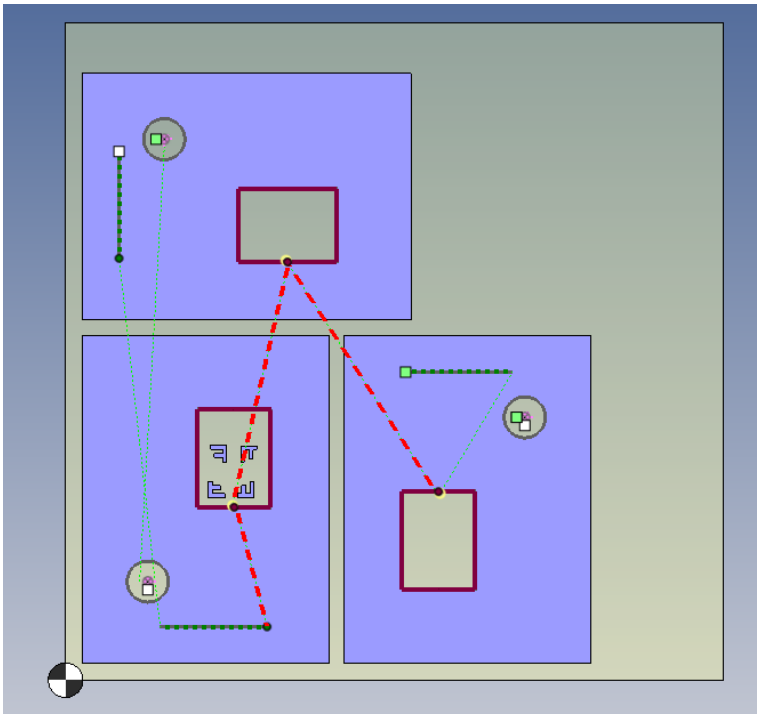
Drilling

Checking operation

During the cycle, the losses and skeletons are likely to switch once cut and collide with the head when linking parts. The control operation therefore allows the user to be informed of these so-called "dangerous" links.
Obviously, this operation takes into account the machining path optimization in order to indicate only the passages on the already cut profiles. These are highlighted with the dotted red lines.



Automatic control of dangerous links



Entity search command

In order to facilitate the identification and location of entities in the sheet metal machining document, a contextual command has been created to search for them.
It is accessible from the Entities tree.

Search entity

Visualization command

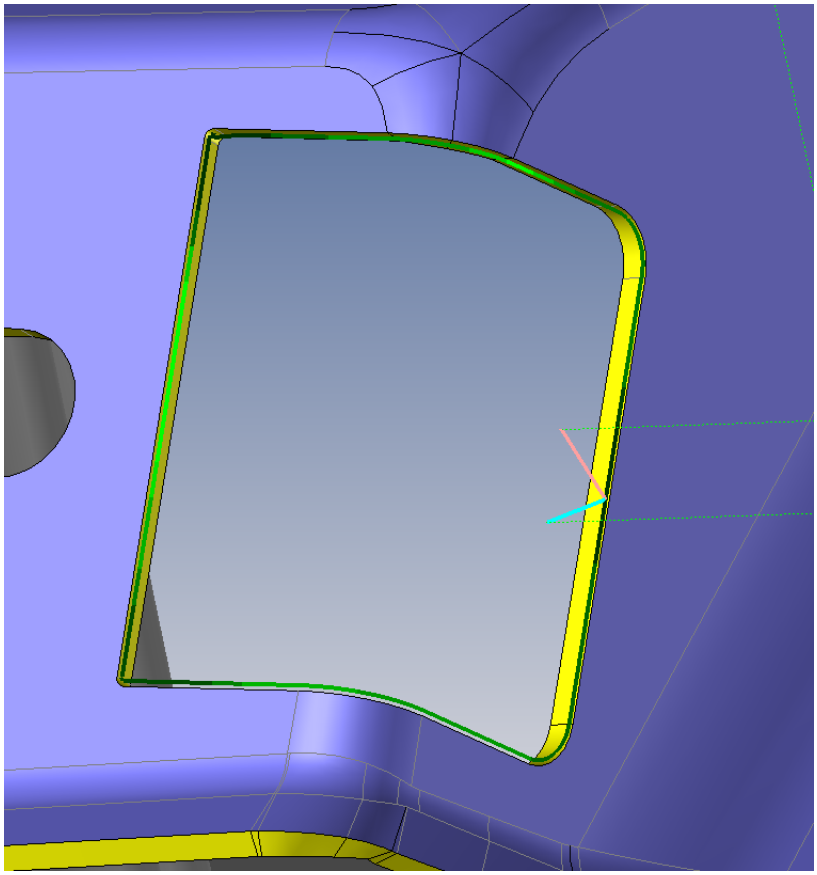
In the case of a nested machining document, the superposition of all the paths can overload the global visualization.

Path visualization

5-axis Cutting

Speed ramps

Based on the operation of 2D operations, the management of speed ramps is now managed in 5-axis cutting operations. It is therefore possible for the user, depending on the topology of the part, to manage the passing speed in the angles independently of that of the radius.
In addition, the levels can be set by cutting quality and thickness. You can assign different parameters depending on the selected entities and profiles.
In order to easily distinguish the application of the ramp parameters, a color gradient is visible on the cutting path according to the speed. The color of the path will become darker and darker as the speed decreases. Therefore, the user can quickly check the behavior of his speed ramps on each path, without having to generate the ISO code.



5-axis speed ramps

This command is available in both 5-axis operations: face cutting and profile cutting.

30mm	0,2in	0,76mm	6s
35mm	0,2in	0,76mm	7,5s

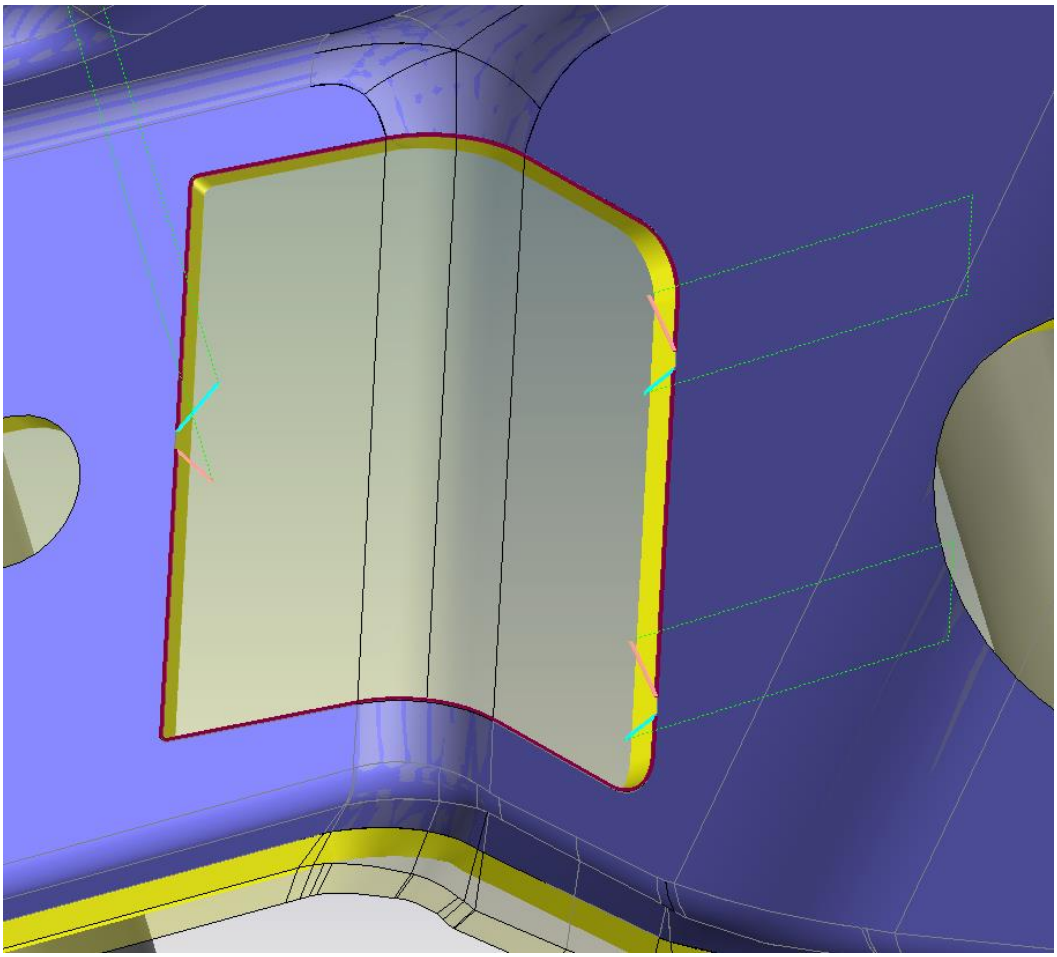
hickness = 30mm , Lens Focal Distance = 0,2in , Nozzle Diameter = 0,76mm

Cutting conditions | Geometry | Leads | Corners processing | Speed ramps | Drilling

Technology	Factor Amin	Amin	Amax	Factor Rmin	Rmin	Rmax	Lmax acceleration	Lmax decceleration	Factor evolution	L after decceleration	L before acceleration
Qualité 1	0,2	60°	180°	0,3	15mm	50mm	10mm	10mm	0,2	4mm	10mm
Qualité 2	0,2	60°	180°	0,3	15mm	50mm	10mm	10mm	0,2	4mm	10mm
Qualité 3	0,2	70°	145°	0,3	1mm	10mm	10mm	12mm	0,2	6mm	9mm
Qualité 4	0,2	60°	180°	0,3	15mm	50mm	10mm	10mm	0,2	4mm	10mm
Qualité 5	0,2	60°	180°	0,3	15mm	50mm	10mm	10mm	0,2	4mm	10mm

Micro-tabs

TopSolid'Cut now allows you to assign micro-tabs in 5-axis paths. This command takes the interface and behavior of the 2D micro-tabs assignment. It therefore includes the different assignment modes: manual, by contour and by segment.



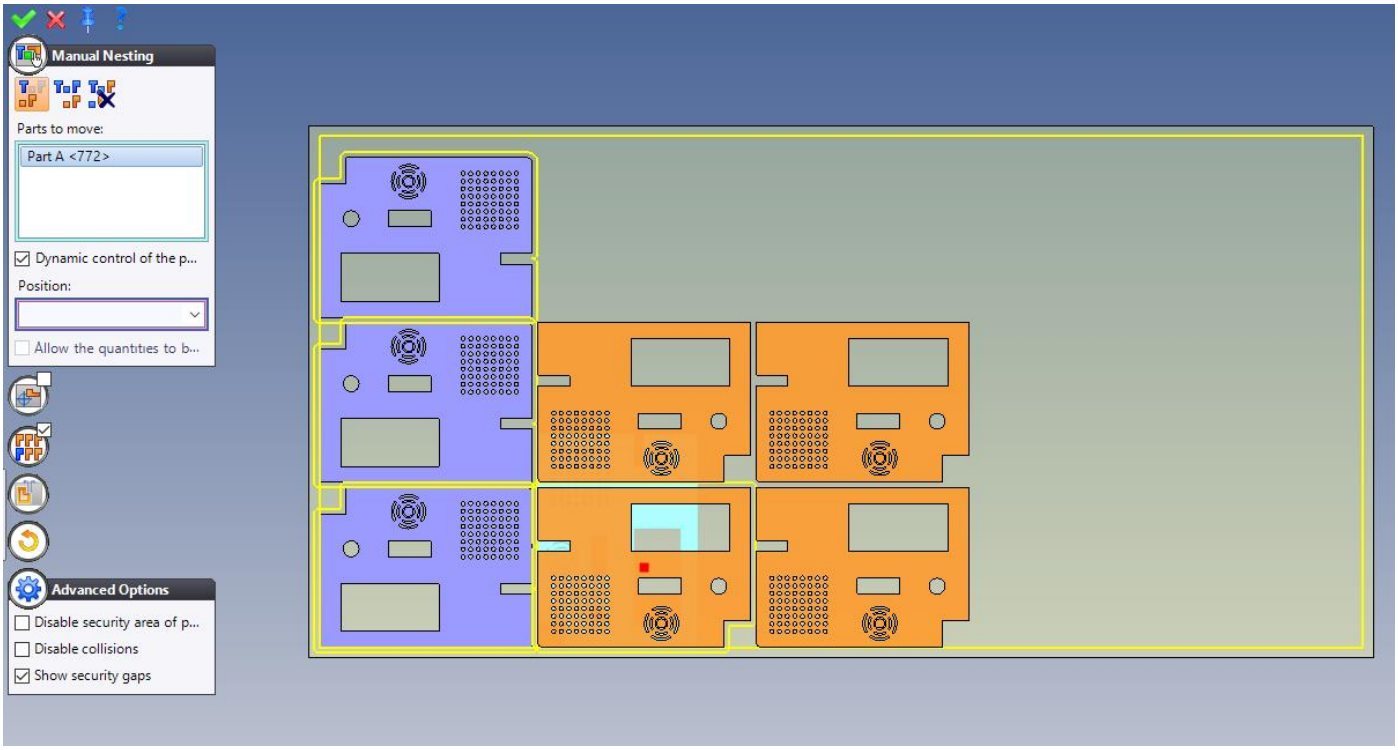
5-axis micro-tabs

This command is available in both 5-axis operations: face cutting and profile cutting.

Nesting

Manual modification

From now on, a nesting modification command is available directly in the sheet metal machining document. This new command can be useful to modify a nesting without replaying it entirely, especially to optimize the gaps between parts and reduce the loss rate.
But the possibilities do not stop at simple modifications. You can also delete, copy or perform a cluster nesting operation from one or more references.
All modifications are made dynamically with collision management, taking into account the differences between parts and nesting characteristics. Therefore, the main parameters present in the nesting document are available directly from the machining.



Manual nesting modification function.

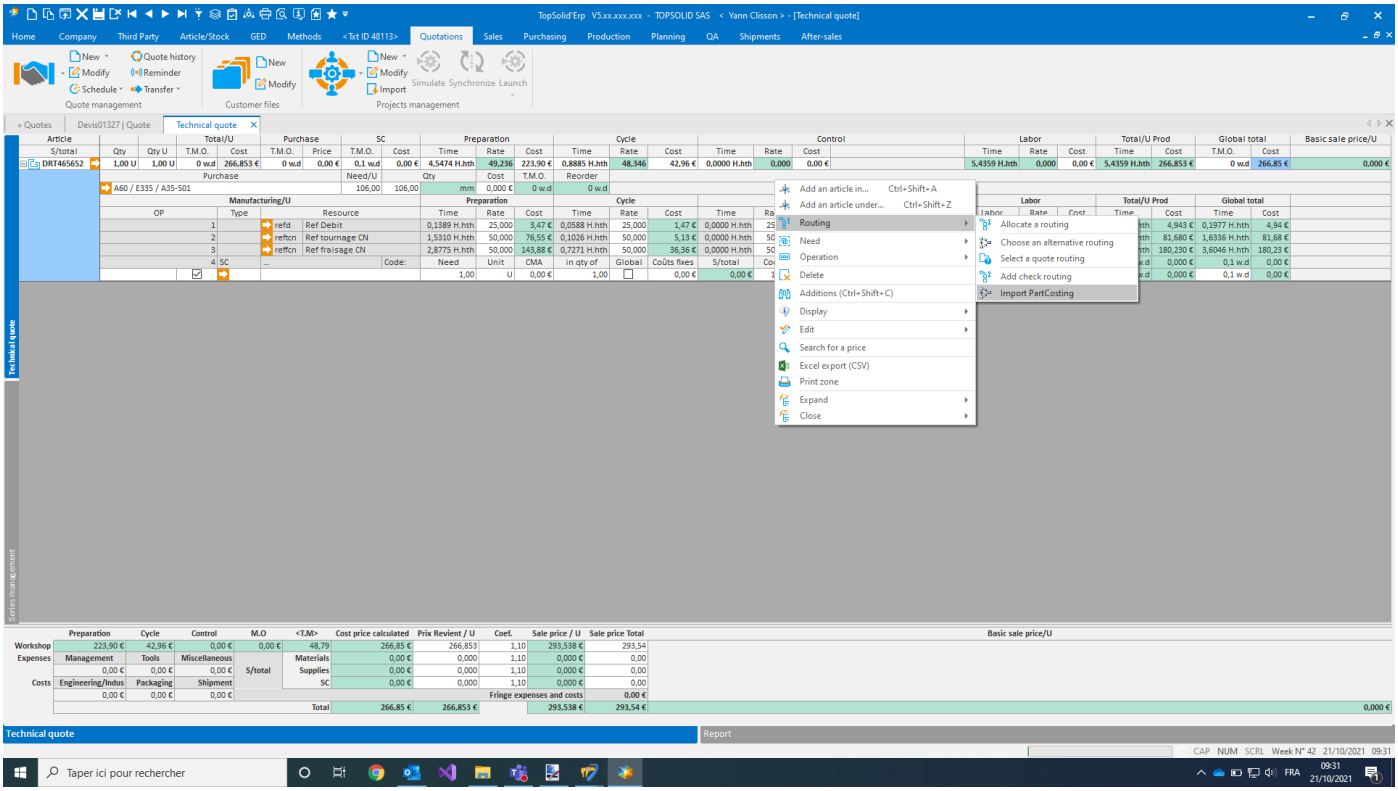
What's New in TopSolid'PartCosting 7.16

This section describes the enhancements made to the **TopSolid'PartCosting** application in version **7.16** of **TopSolid 7**.

TopSolid'Erp link - TopSolid'PartCosting

With **TopSolid'Erp 5.16,** you can now import a study carried out in **TopSolid'PartCosting** from the ERP application.

The import feature offers the list of parts studied in **TopSolid'PartCosting** from a **quote**, a **technical quote** or from the **routings**. The routing with time, material and needs are integrated into the ERP system with just one click. **TopSolid'Erp** is the only ERP application with a machining time calculation system.



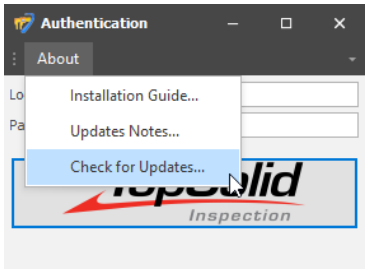
What's New in TopSolid'Inspection 7.16

This section describes the enhancements made to the **TopSolid'Inspection Creator**, **Controller** and **Analyst** applications in version **7.16** of **TopSolid 7**.

General Information

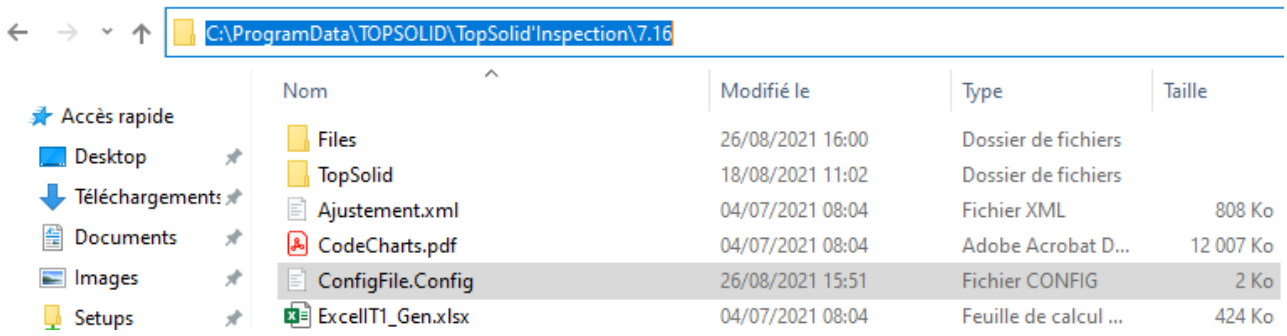
TopSolid'Update

The **check for updates** via **TopSolid'Update** is launched automatically when the applications are started, and it is also accessible via the drop-down menu in the **Connection** dialog box. A link to the **installation guide** and **update notes** is also available.



Remote configuration file

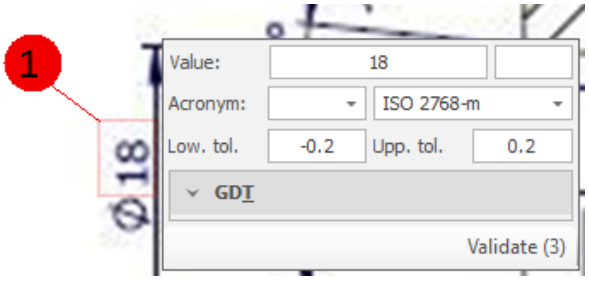
For reasons of access rights to the C:\Program Files\ installation folder, the configuration file containing the **Options** > **Global settings** parameters has been moved to the C:\ProgramData\TOPSOLID\TopSolid'Inspection\7.16 folder. The file is called "**ConfigFile.Config**".



Only the **Parameter files folder path** and **PDF files folder path** remain in the program configuration file (by default C:\Program Files\TOPSOLID\TopSolid Inspection 7.16\Softwares\TopSolid.Inspection.exe.Config) and require administrator rights to be modified.

OCR version update

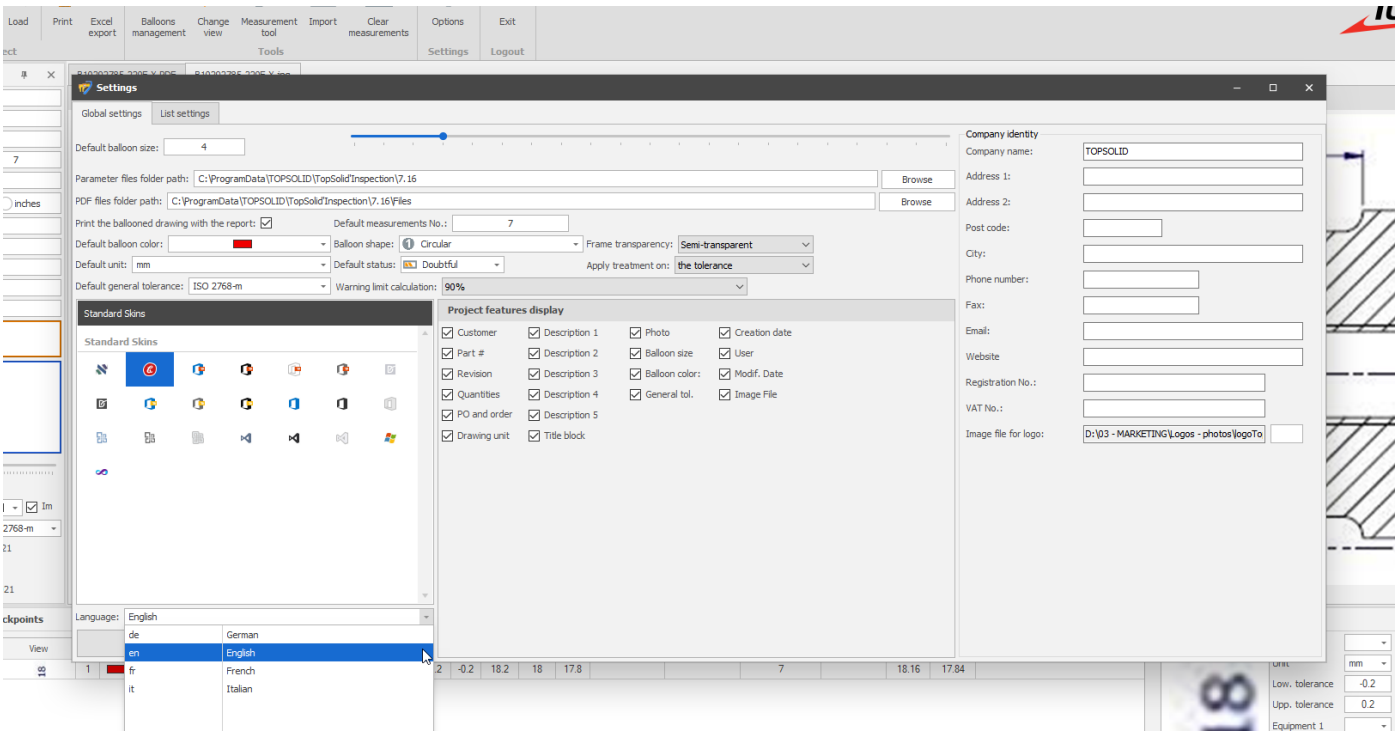
A new version of OCR has been implemented, it is more efficient in recognizing dimensions in image files, especially on vertical texts.



New Features

Language selection

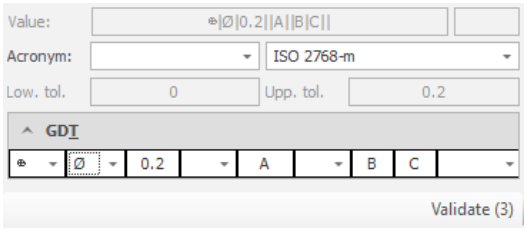
In the **Options** > **Global settings**, a drop-down list allows you to choose the language. This list is automatically provided according to the DLL languages in the localization folders of the installation folder.

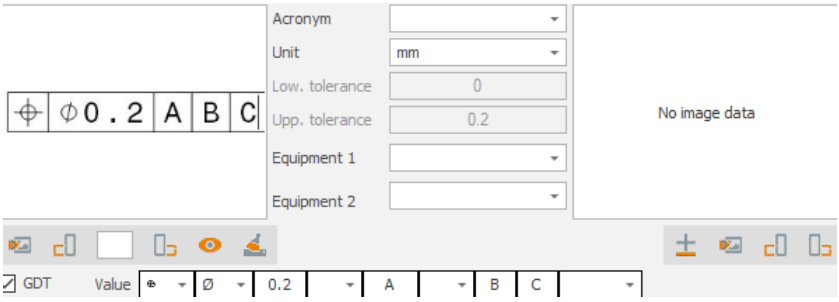


The layouts files that save the layout of the lists and the custom modification of the labels are deleted when the language is changed.

GDT

The GDT standard for geometric tolerances has been implemented. In the verification dialog box and in the checkpoint details, it becomes possible to enter the dimension as a geometric tolerance thus enabling the pre-provided construction system.





The values available in the drop-down lists can be configured in the **GDT1.xml**, **GDT2.xml**, **GDT3.xml** files in the **Parameter files folder path** folder indicated in the **options**.

Phase management and project locking system

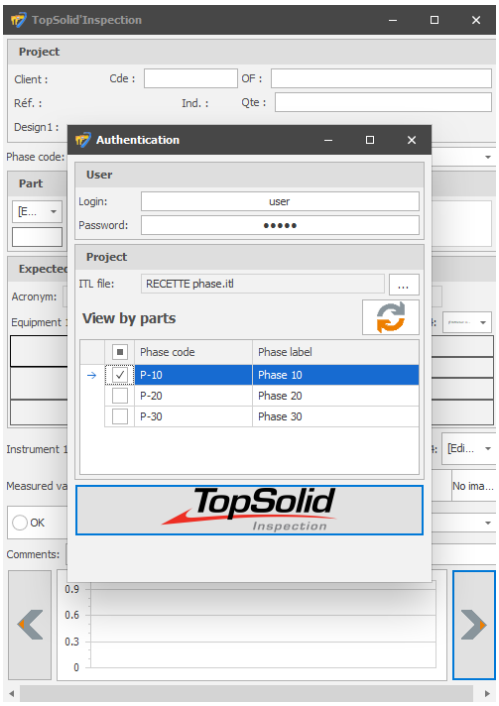
Phase management allows you to flag a checkpoint as belonging to a certain stage of operation plan. This then allows you, in **controller** mode, to select the phase(s) that you wish to control and therefore to display only the dimensions concerned.

An inspection project can now contain the dimensions of several phases. This development goes hand in hand with a project locking system to prevent several users from opening a project at the same time. The locks are stored in the **Lock.xml** file located in the same place as the other data files.

Opening a project in **Inspection Creator** generates a **Project** lock that locks the entire project.

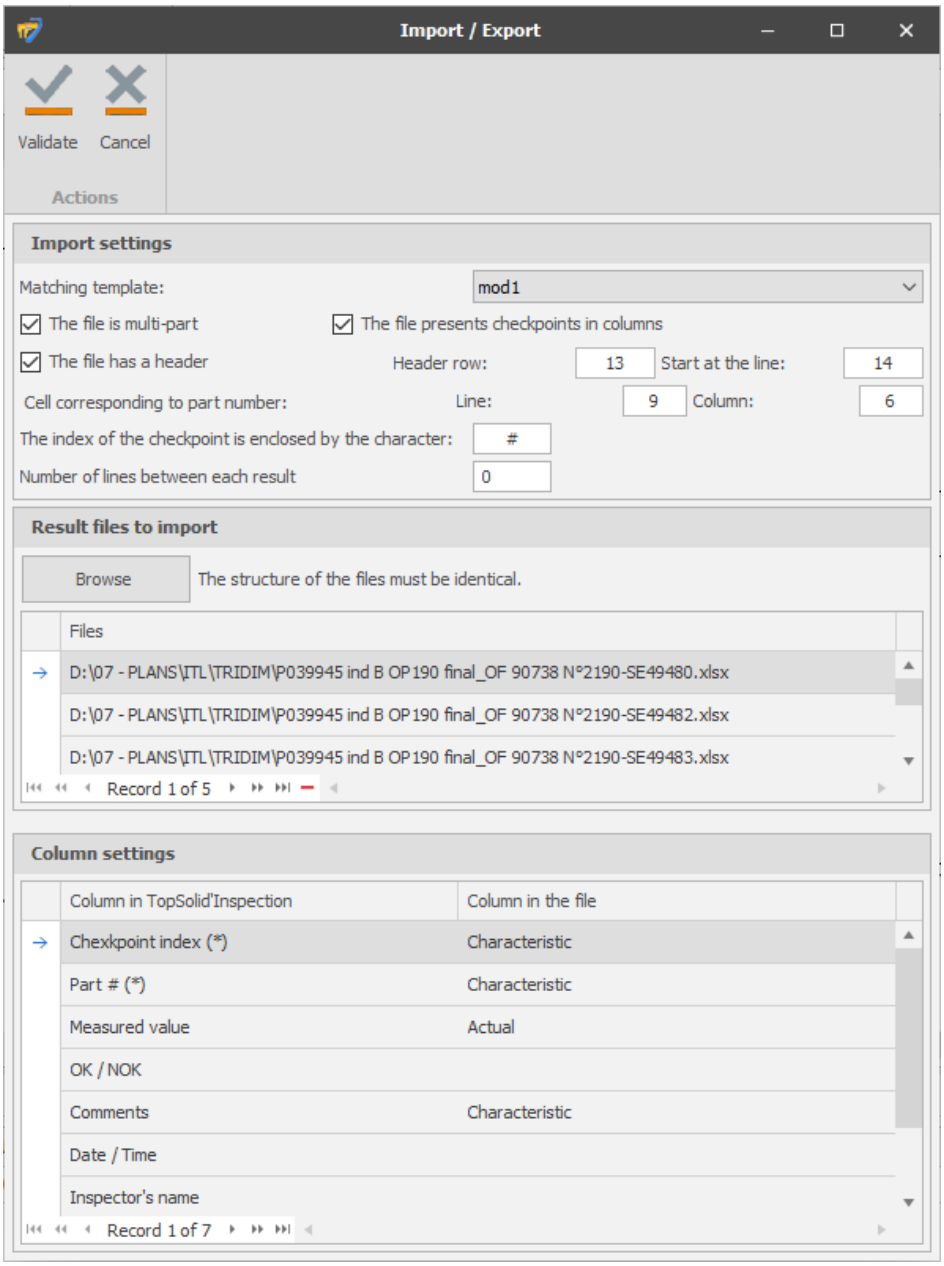
Opening a phase in **Inspection Controller** generates a **Phase** lock which locks the opening of the project in **Creator** and only the phase concerned in **Controller**. Another **controller** can therefore open another phase.

The **controllers** work with a partial project in memory and are able to update the original project with the collected data when closing.



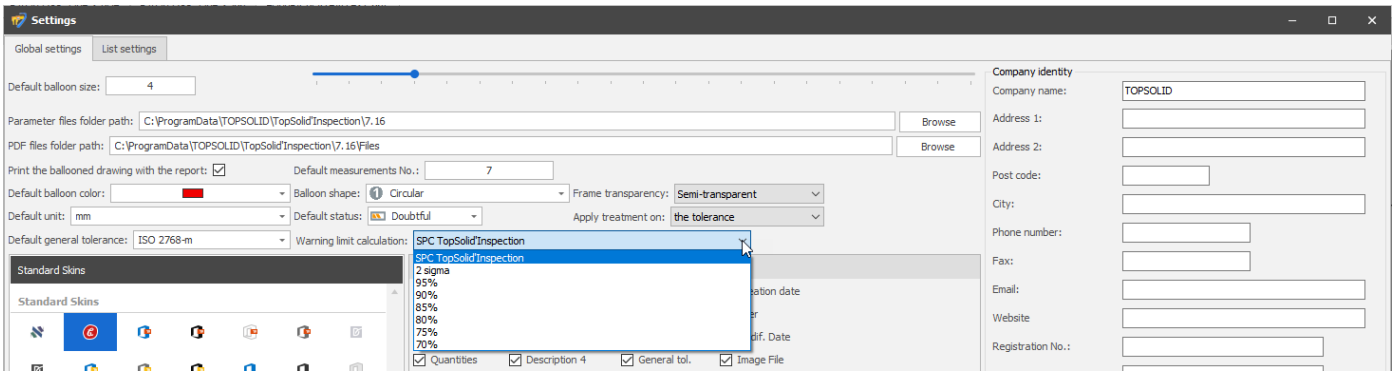
CMM import

The CMM import results from an Excel file that now allows you to use the multi-part files, the results of which are arranged in columns. The **Import/Export** dialog box now has two new **The file is multi-part** and **The file presents checkpoints in columns** options as well as a new **Number of lines between each result** field for the offset step.



Warning limit calculation

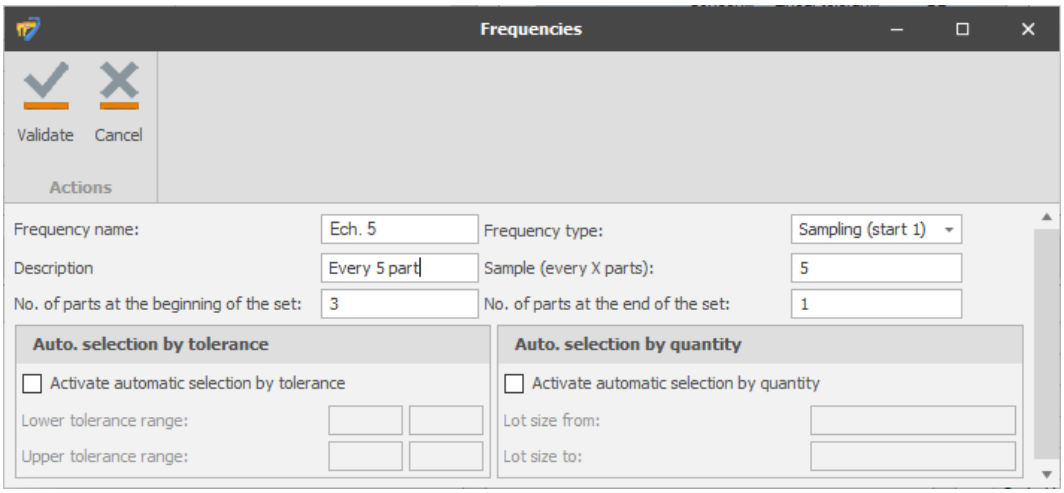
When ballooning a numerical value, the warning limit calculation is carried out according to a formula specific to **TopSolid'Inspection** using a process statistics formula. Several other formulas have been added to the warning limit calculation: the **2 sigma** mode, and several **percentage** modes.
The chosen calculation formula is configured in **Options** > **Global settings** > **Warning limit calculation**.



Please note that for a checkpoint marked GDT, the monitoring limit is calculated in 80% mode.

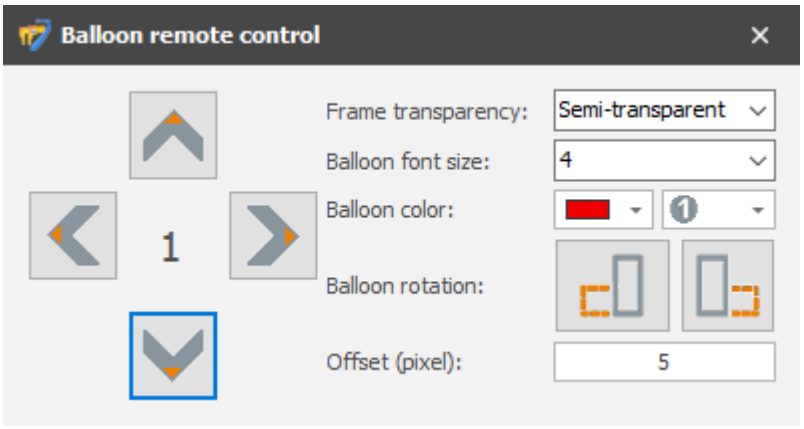
First and last parts in the frequency control

In terms of control frequency, a new setting is available that allows you to take into account the first and last parts to be controlled. In the example below, the controlled parts will be the first 3, then all 5 parts from index 6 (start 1), then the last one.



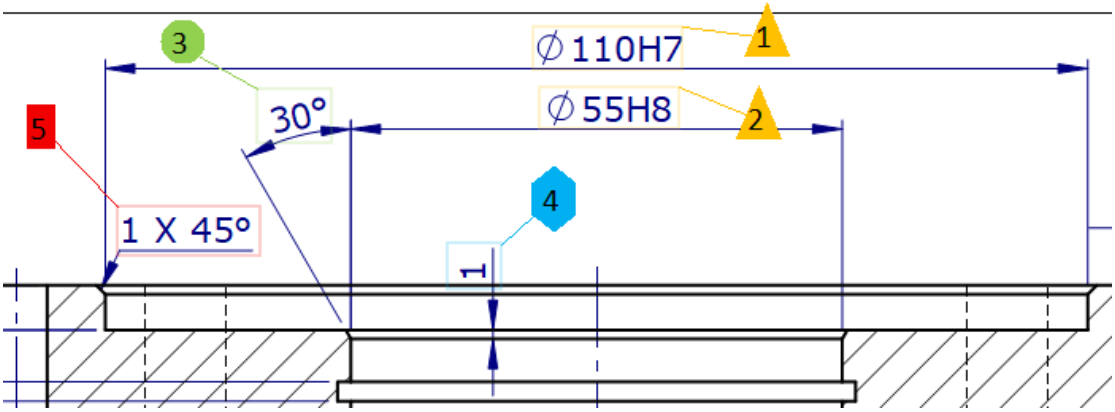
Moving the frame of a checkpoint

The balloon remote control now allows you to move the frame (not the balloon) from a checkpoint.



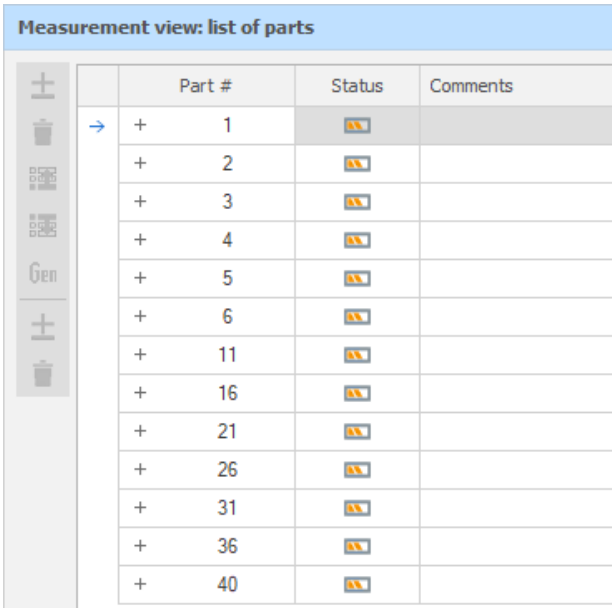
Additional balloon shapes

Triangle and **Hexagon** shapes have been added as possible balloon shapes.



Sorting the parts to be controlled

In the **Creator** and **Controller** programs in **View by parts**, the part list is sorted in ascending order based on the part numbers.



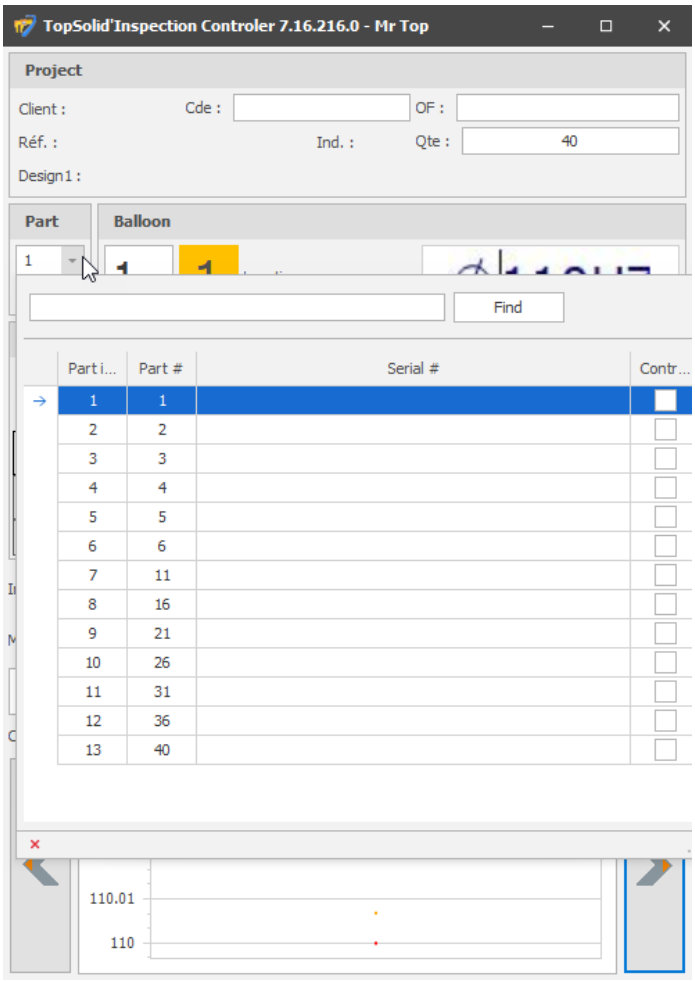
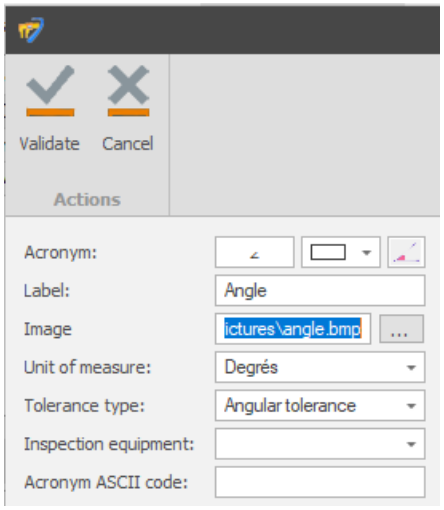
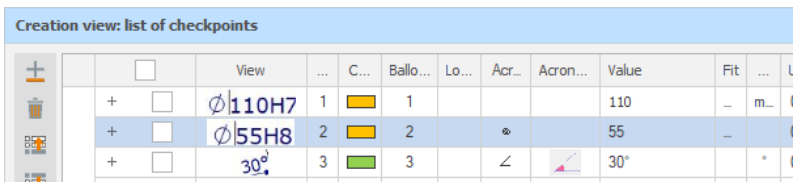


Image on acronyms

An image can be assigned to the **acronyms** in **Options** > **List settings**. The corresponding column appears in the list of checkpoints.



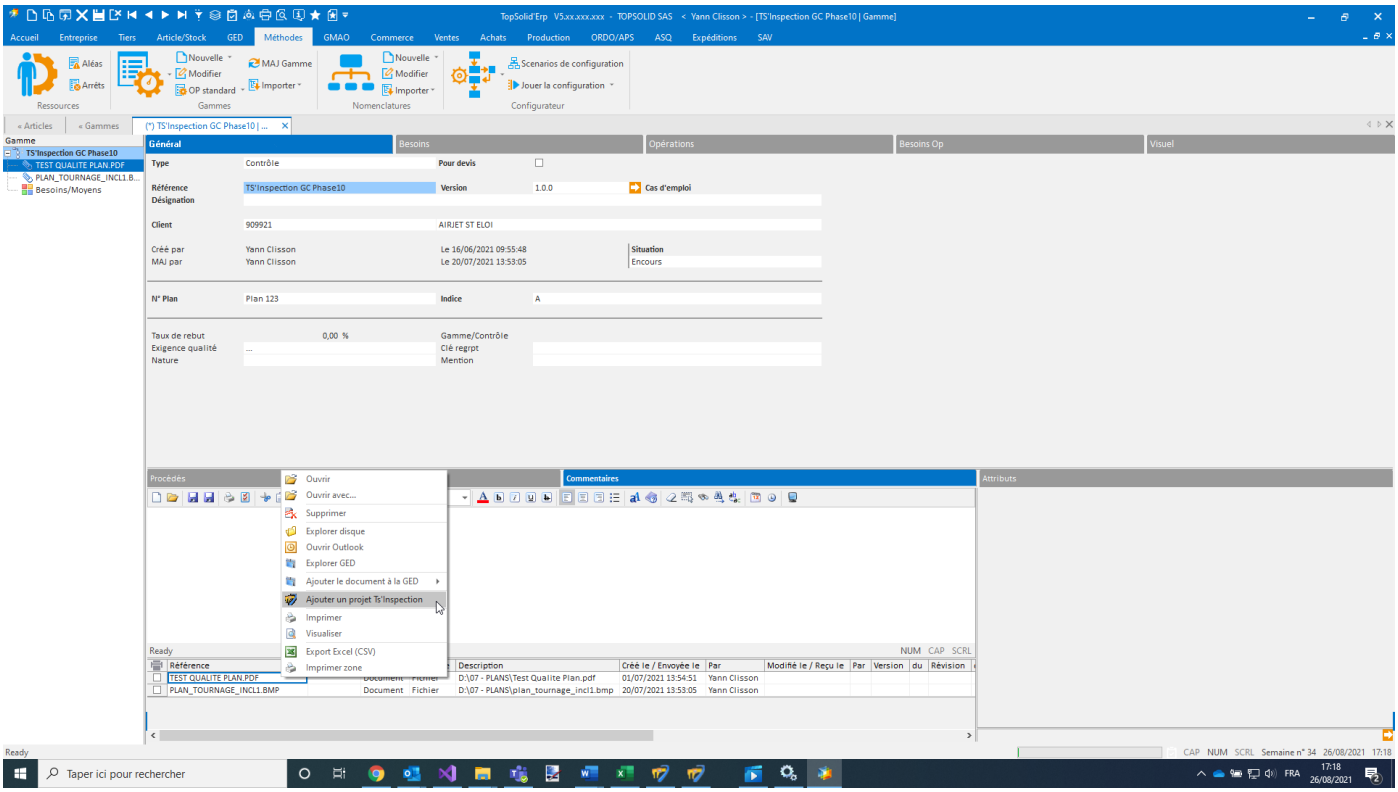


TopSolid'Erp - TopSolid'Inspection link

With **TopSolid'Erp 5.16** comes a close interaction with **TopSolid'Inspection**, driven by the ERP.

Generating the inspection project

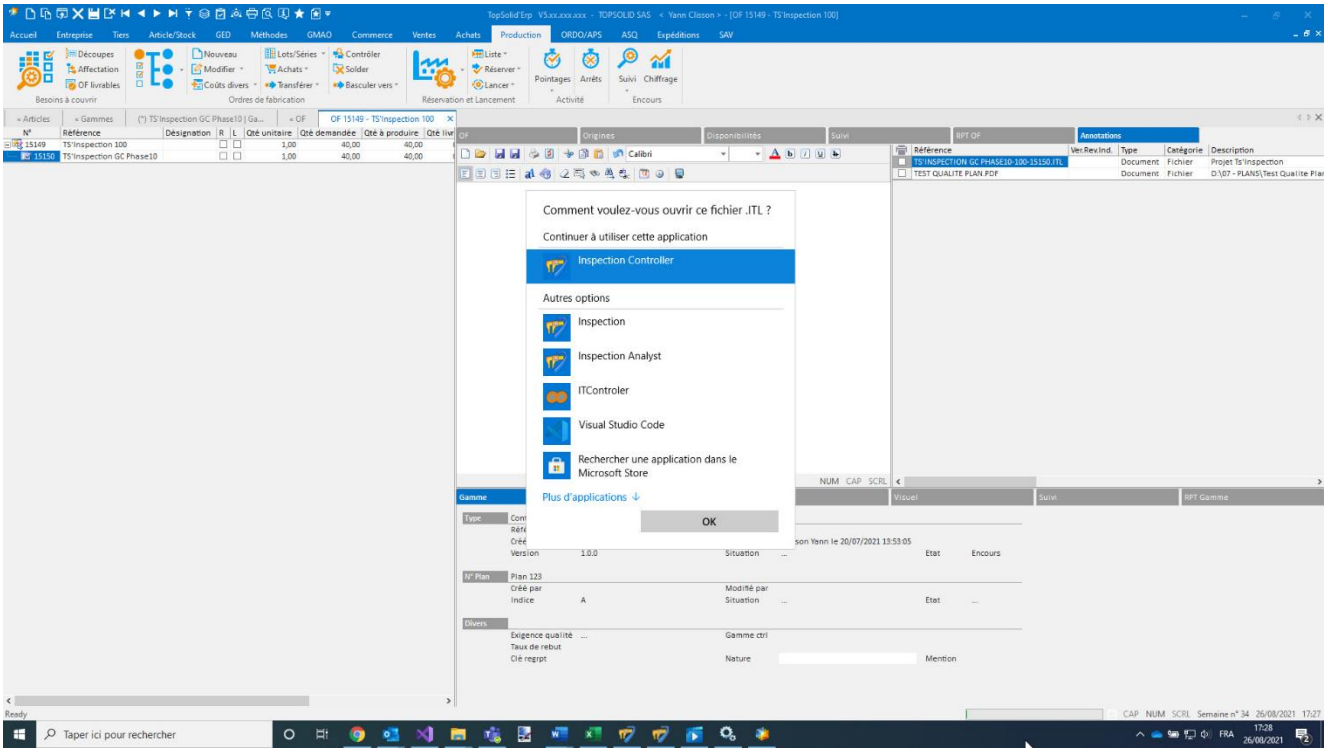
From an **inspection plan** in ERP, the inspection project (ITL file) is automatically generated, therefore including the information in the operation plan.



The project can be opened in **TopSolid'Inspection** from **TopSolid'Erp** to balloon the dimensions and create the control drawing. This is the "model" project.

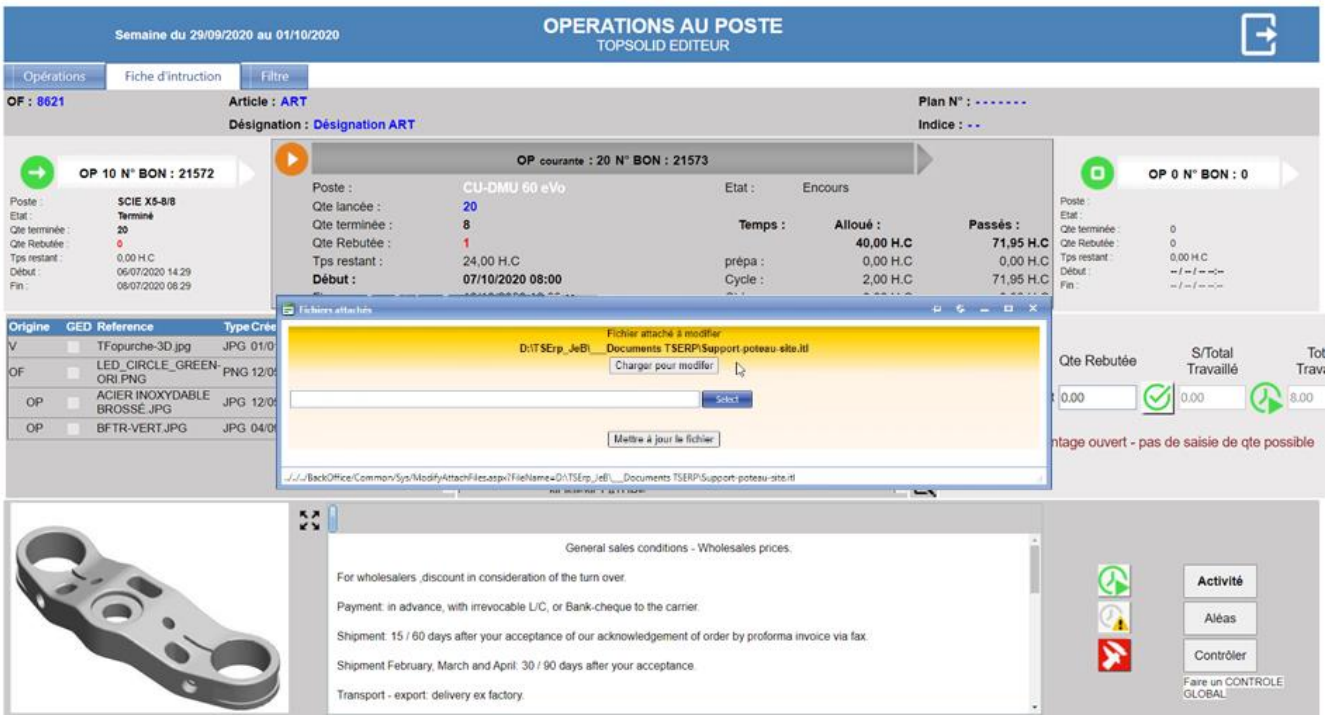
Copying the inspection project

When creating a PO in ERP, the "template" inspection project is automatically copied and updated with the PO information (PO number, part quantity, serial numbers). The quantity of part to be inspected for each **Checkpoint** in the **Inspection Project** is recalculated according to the quantity of part to be produced from the PO and the **control frequencies**.
The inspection project can be opened with the **Creator** or **Controller** program directly from the ERP application.



Operation with IProd

The Inspection project is accessible from IProd for local use. It should then be downloaded to update the ERP server.



Synchronizing the database

Each time **TopSolid'Inspection** is opened from the ERP application, the list of inspection equipment is updated on the **TopSolid'Inspection** side with the ERP items belonging to the **groups** attached to the **Tools** functional group.

Groups | Families

N°	Label	Codification	Functional group
1	Compo. Fabriqué	CF	Other
2	Conditionnement	CDT	Other
3	Fourniture	FOUR	Other
4	Ensemble	ENS	Other
5	Logiciel	SOFT	Other
6	MATERIEL	MAT	Other
7	Matière	MAT	Material
8	Outillage	OUT	Tools
9	Fantome		Material
10	Outillage Controle	3	Other
11	Peinture	1	Material
12	Prestation	PRES	Service
13	Produit	PR	Other
14	Prototype	2	Tools
15	Formation	4	Service
16	Semi-Fini	SF	Other
17	BOIS		Other
18	PC_NDF		Other

The **acronyms** are also synchronized.

N°	Symbole		Code	Libellé	Obsolète
1	⌀		01	Diamètre	☐
2	⟼		02	Côte_ou_Distance	☐
3	◎		03	Concentricité	☐
4	⌖		04	Localisation_ou_Position	☐
5	⏤		05	Rectitude	☐
6	○		06	Circularité	☐
7	⌒		07	Ligne_quelqonque	☐
8	//		08	Parallelisme	☐
9	⊥		09	Perpendicularité	☐
10	∠		10	Inclinaison_ou_Orientation	☐
11	⌯		11	Symétrie	☐
12	↗		12	Battement_circulaire	☐
13	⌰		13	Battement_total	☐
14	▱		14	Planéité	☐
15	⌭		15	Cylindrique	☐
16	⌓		16	Surface	☐
17	√		17	Etat_de_surface	☐
18	⌲		18	Conicité	☐
19	⌔		19	Defaut_de_forme	☐
20	⌒		20	Profil	☐
21	⌓		21	Arc	☐
22	⌯		22	Palpage	☐