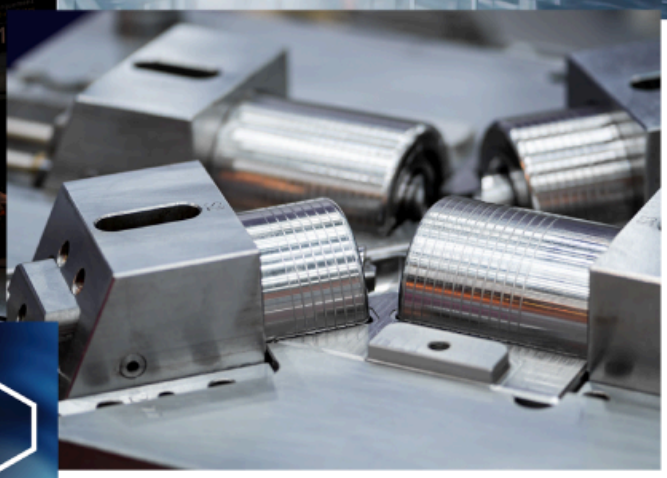
TopSolid

2024



What's New
TopSolid 7.18

TopSolid

INTEGRATED
DIGITAL
FACTORY

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Version 7.18

Rev. 01

> **Note:** If you are experiencing problems using this document, please feel free to send your feedback and comments to edition@topsolid.com.

Contents

Welcome to TopSolid 2024

This document presents the main improvements and evolutions introduced in version **7.18** of **TopSolid 7**. The innovations described here are only part of the new innovative features incorporated in the software.

To find out what else is new, check our e-learning platform (from March 2024) or contact your local agency.

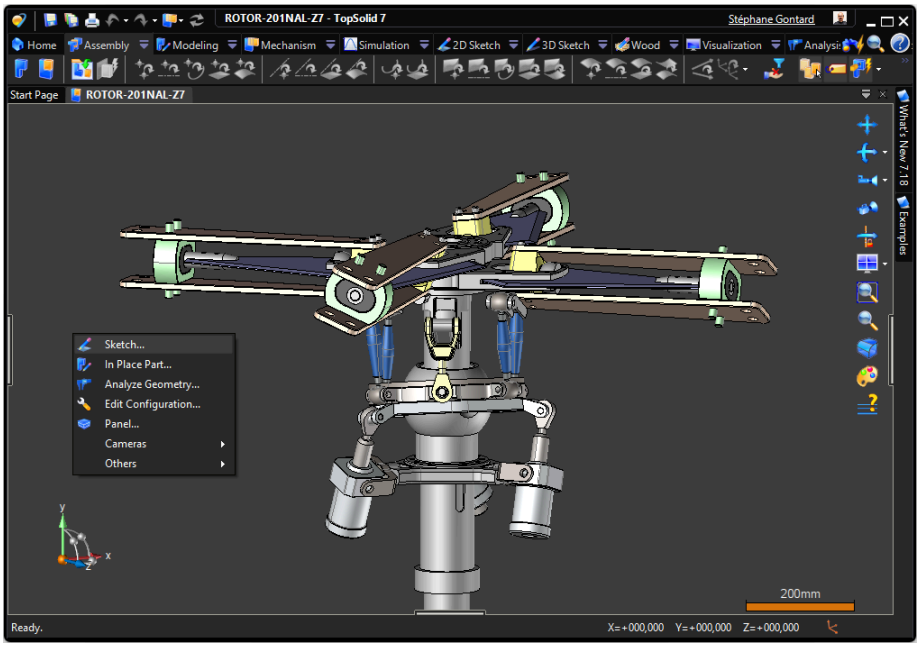
What's New in TopSolid'Design 7.18

This chapter describes the new features of the **CAD** applications in version **7.18** of **TopSolid 7**.

User interface

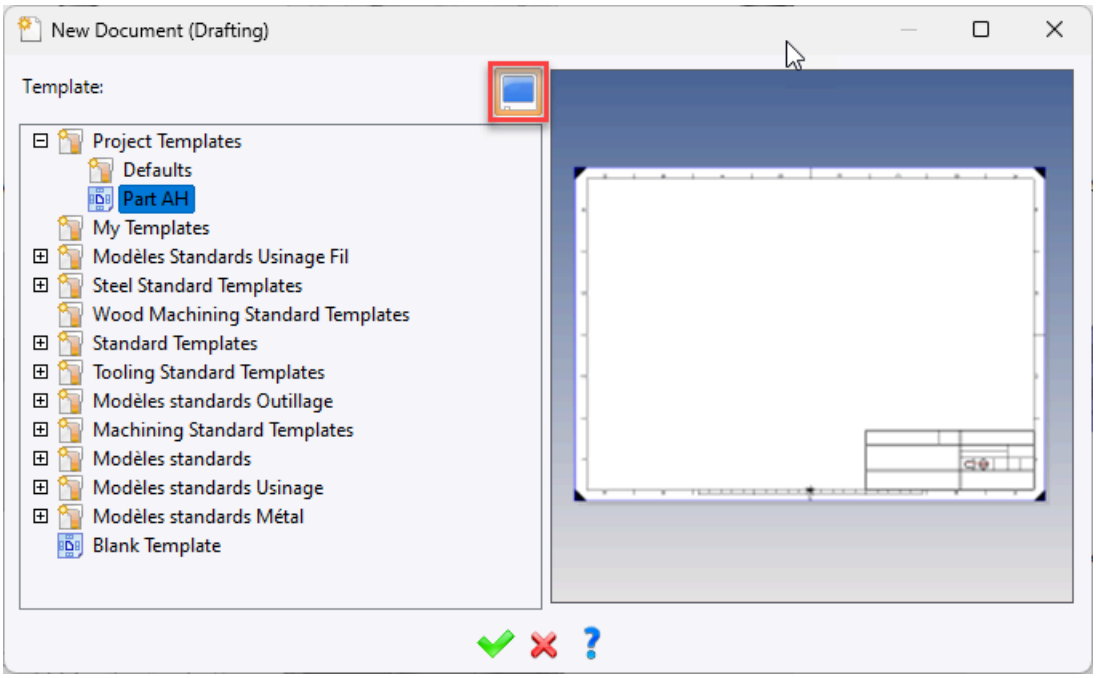
Dark mode

The **Theme** section of the **Tools** > **Options** command now allows the user interface to be displayed in dark mode to reduce eyestrain and power consumption.



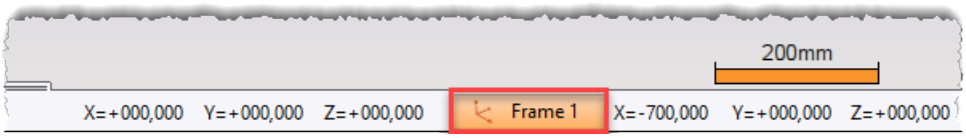
Preview

An icon for previewing the document has been added to the new document creation dialog box.



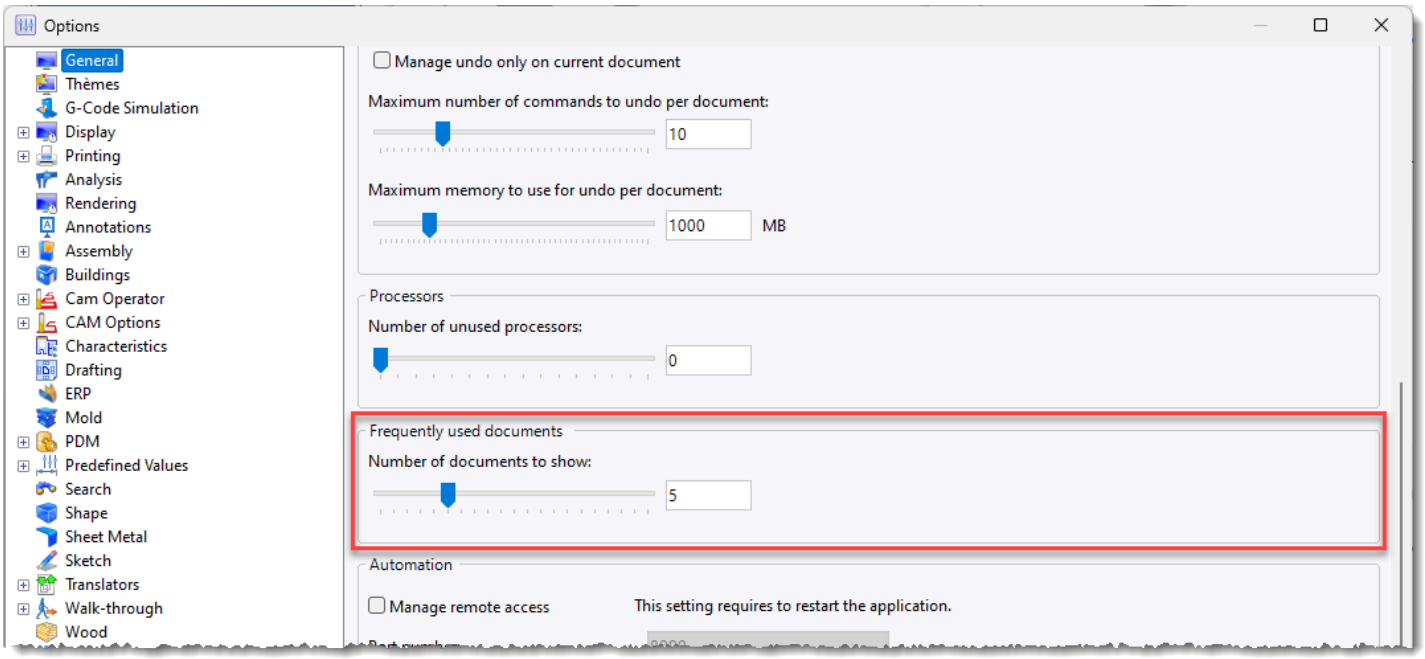
Coordinate display

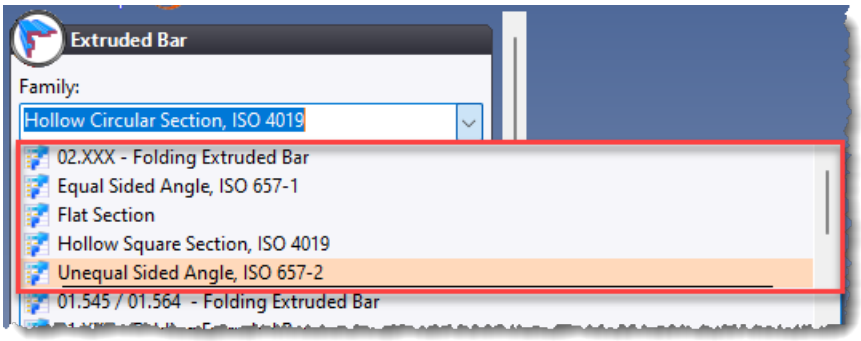
Cursor coordinates can be displayed from a specific frame using the frame icon located below the graphics area. Cursor coordinates are displayed to the right of the selected frame name.



History of last documents used

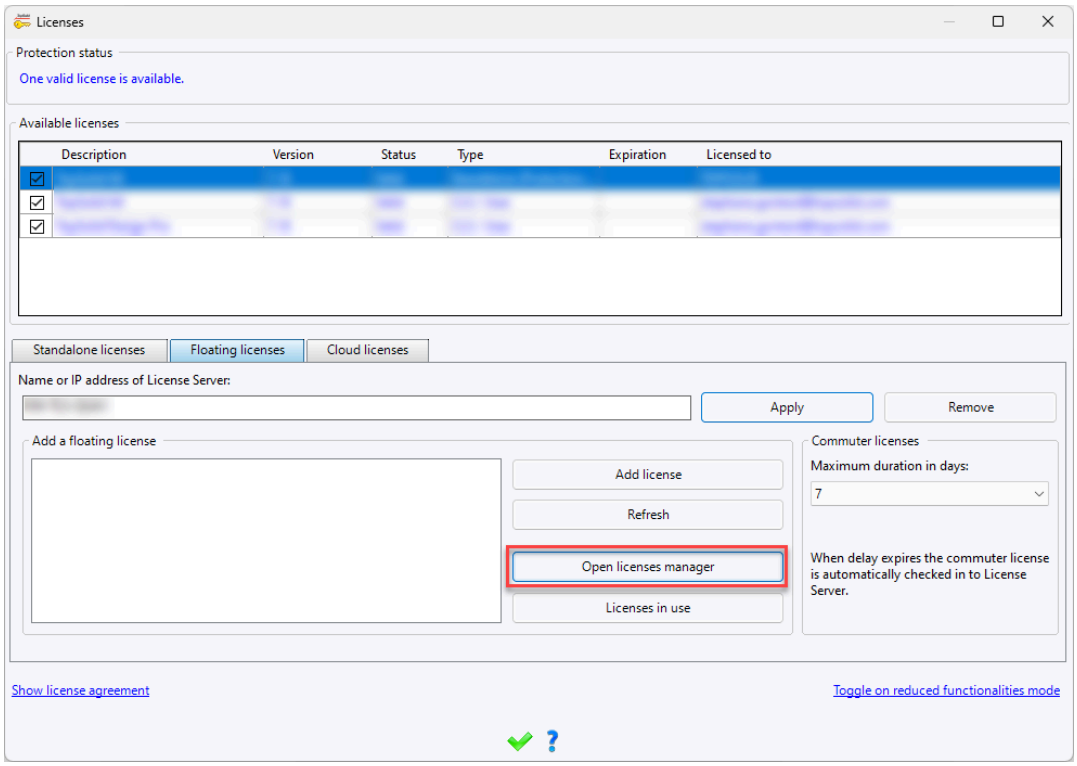
The history of the last documents used is now saved, and these documents can be displayed at the top of the list. The number of documents to be displayed can be set in **Tools** > **Options** > **General**.





Licenses

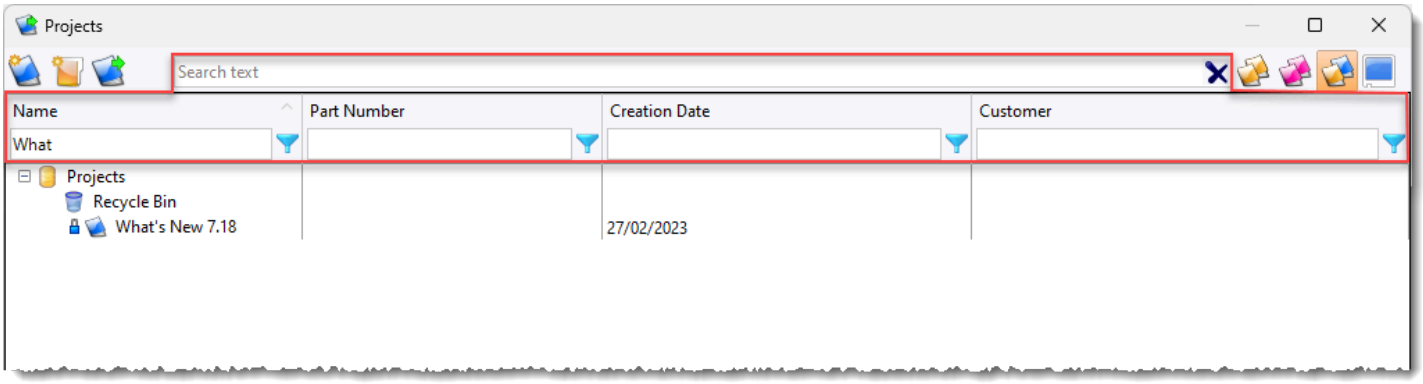
In the **Floating licenses** tab of the license management window, the **Open licenses manager** button opens the **Wlmadim.exe** utility depending on **Sentinel** to add licenses.



PDM

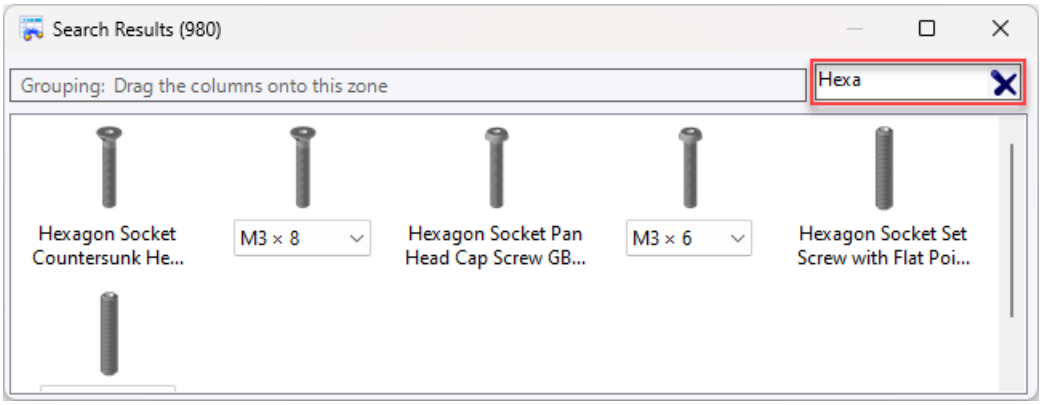
Project manager

The project management window has been enhanced with search fields and filters for each column.



Search

A filter field is available in the search results window.

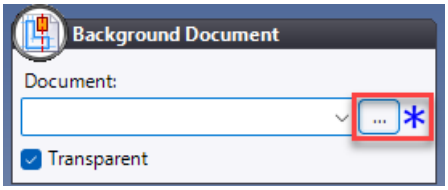


Security

Passwords are now hashed in the SQL database, whereas they were encrypted in previous versions. The hash mechanism ensures enhanced security by preventing the decryption of passwords directly from the information stored in the SQL database.

Background document

The **TopSolid 7** > **File** > **Background Document** command features a new button for browsing the current project tree.

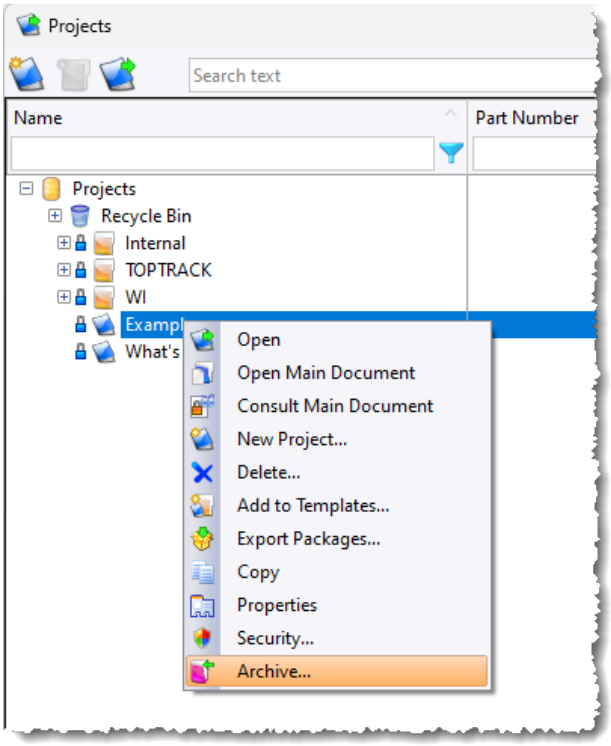


Properties

The **Edit** button has been deleted from the document properties window. Properties are now directly editable.

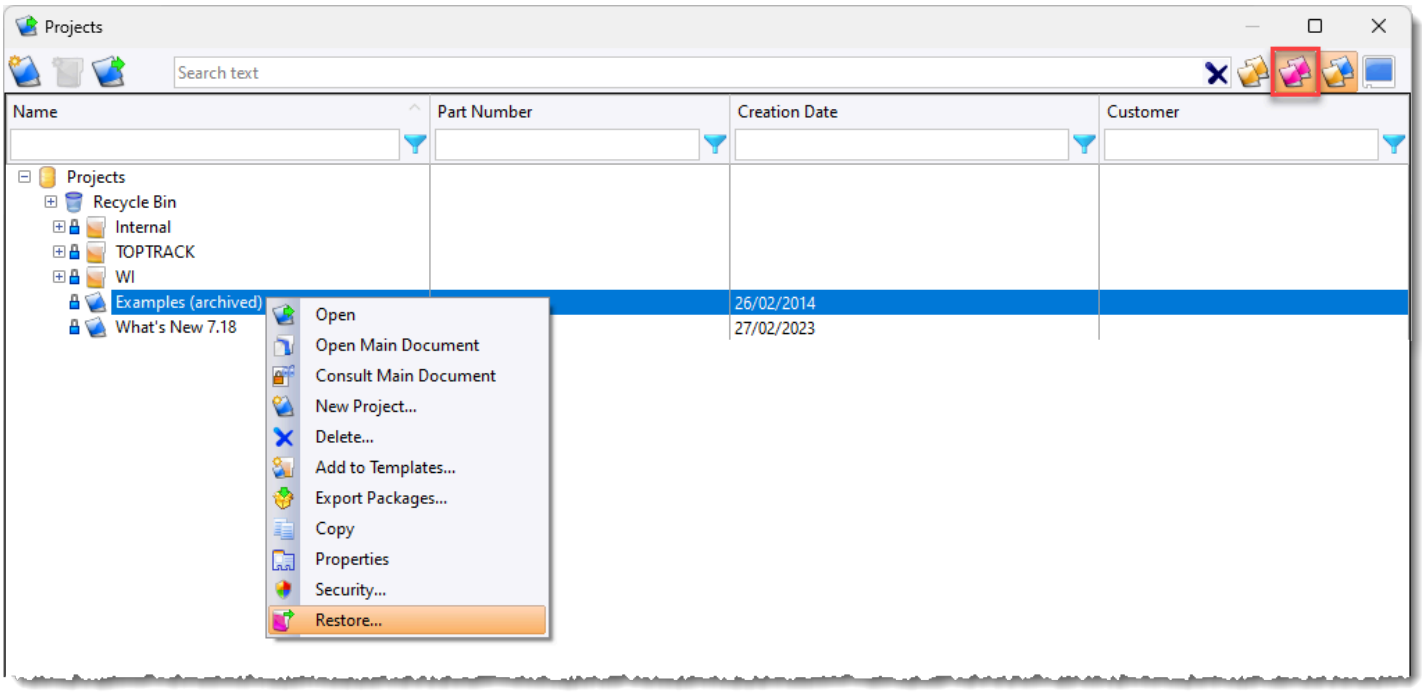
Archiving

The new **Archive** command can be accessed either from the context menu in the project manager or directly on a project in the **Others** context menu.



Projects are archived asynchronously and stored in a specific folder defined in the vault management window of the PDM (Local or Server). Each archive has an identifier recognized by the issuing PDM. Archived project files are then automatically deleted from the vault.

In the project manager, a new icon at the top right of the window shows the archived projects. You can restore an archived project by right-clicking on it and selecting the **Restore** command from the context menu.



Multisite

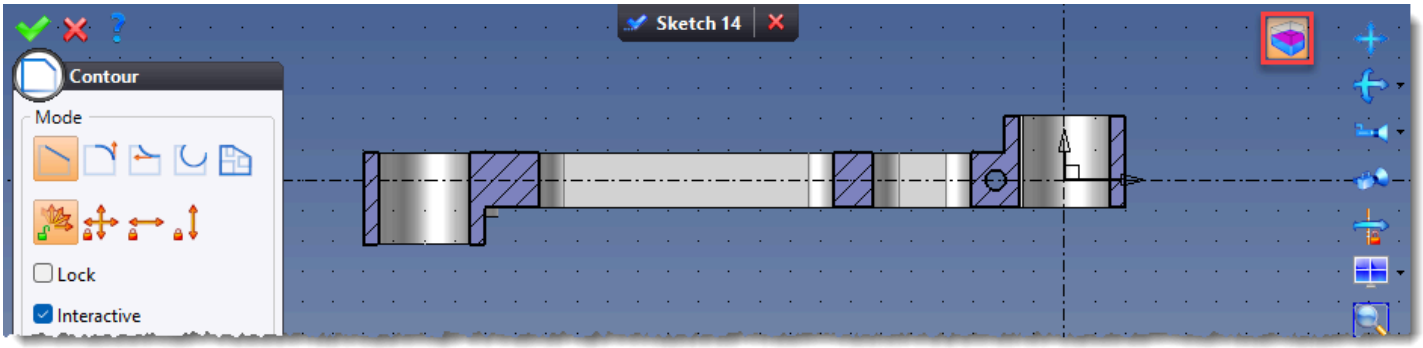
Each time a user connects to a primary or secondary server, files, in the cache vault, whose last reading date exceeds the specified number of days set by the administrator, are purged This purge is performed asynchronously to ensure that it does not extend the session startup time. The number of days can be set in the advanced management of the PDM Server.

The cache vault can be manually purged by the PDM Server administrator using the **Purge cache** button. In this case, cache files are purged without regard to the last read date.

Sketch

Graphic cut

When creating or editing a sketch, a new icon at the top right of the graphics area allows you to make a graphic cut to facilitate modeling.

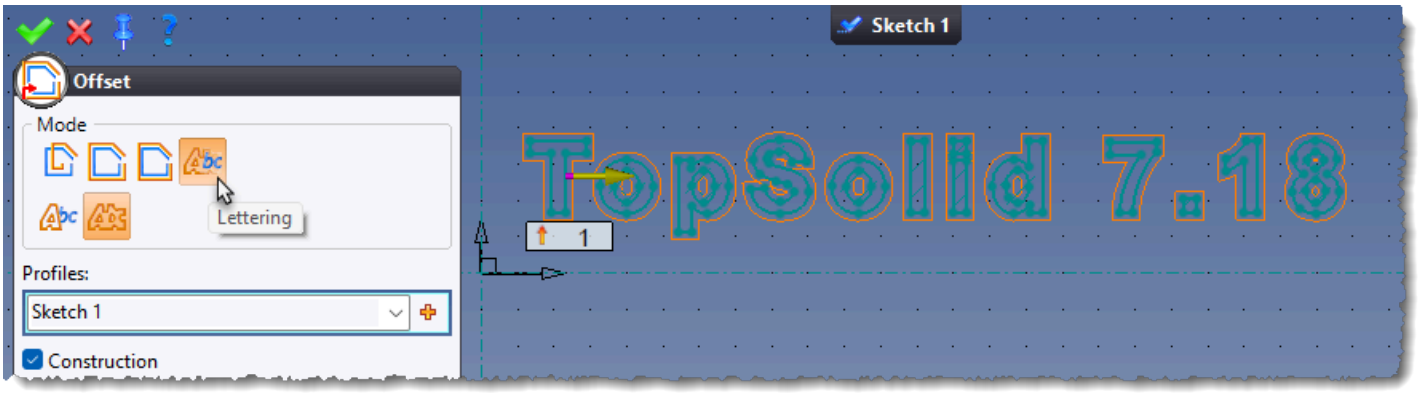


Hook deactivation

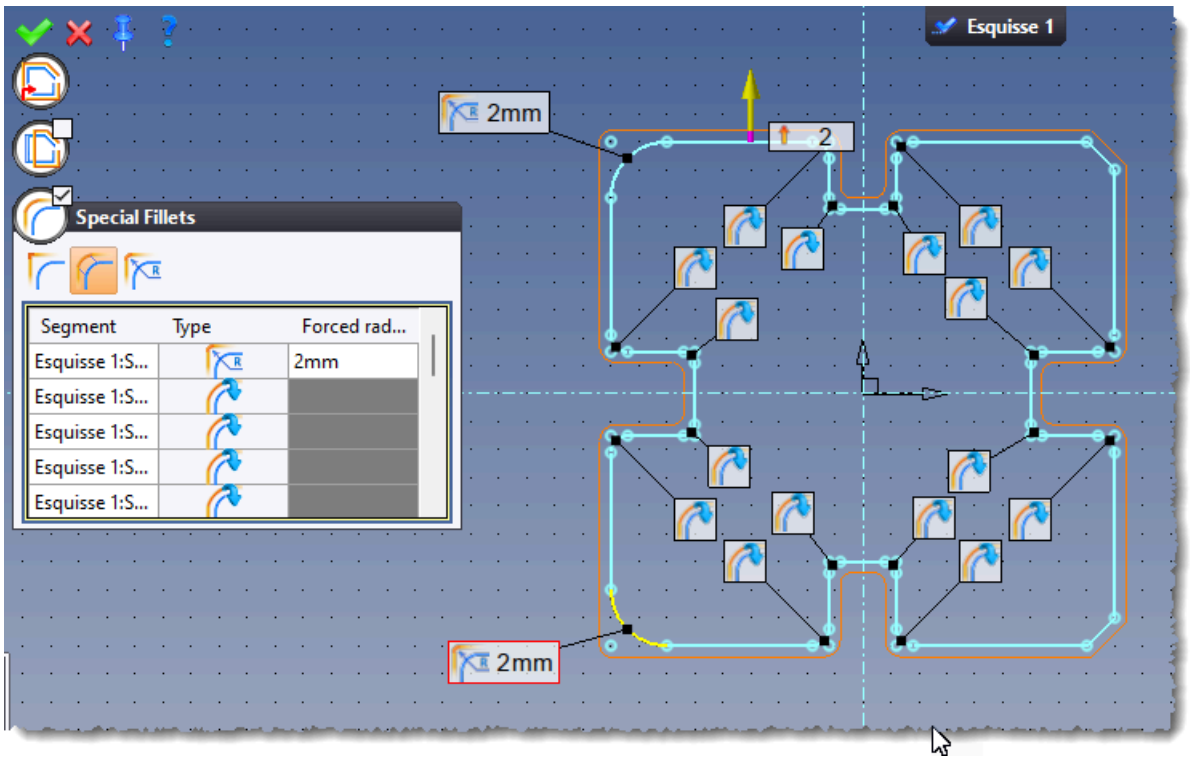
When creating an entity (contour, circle, etc.), press **Shift** to deactivate constraint creation or hook on grid.

Offset

The new **Lettering** mode is available in the **2D Sketch** > **Operations** > **Offset** command. It can be used to create parallels between several closed profiles in a very short time.



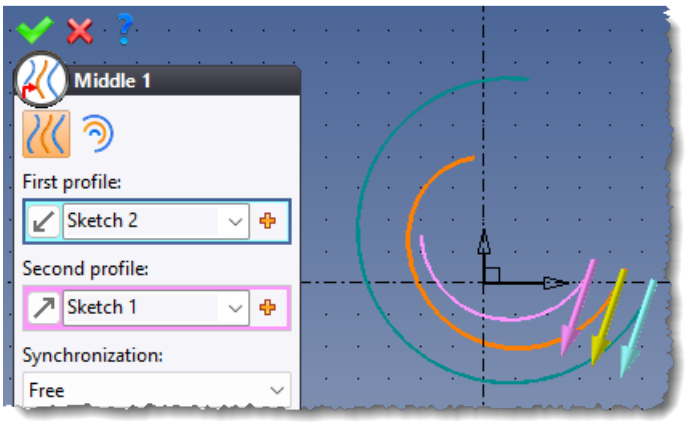
The **Special Fillets** section has also been added, allowing selected fillets to be extracted, kept or replaced globally or locally.



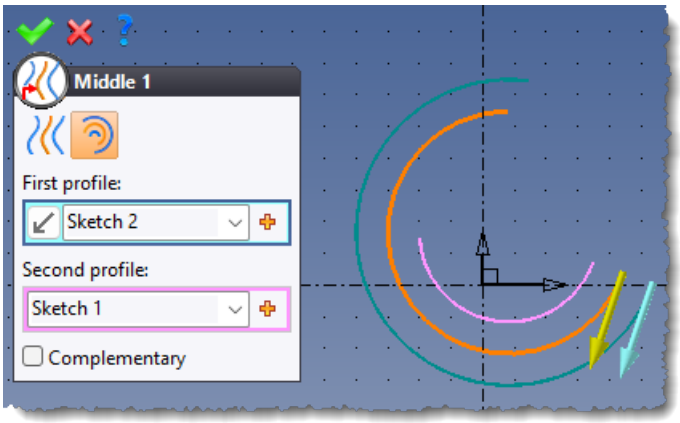
Middle profile

The **2D Sketch** > **Operations** > **Middle** command features the new **Circular arc** mode, which creates a median diameter arc from two concentric arcs.

An option is provided to create the complementary arc.



***General** mode*

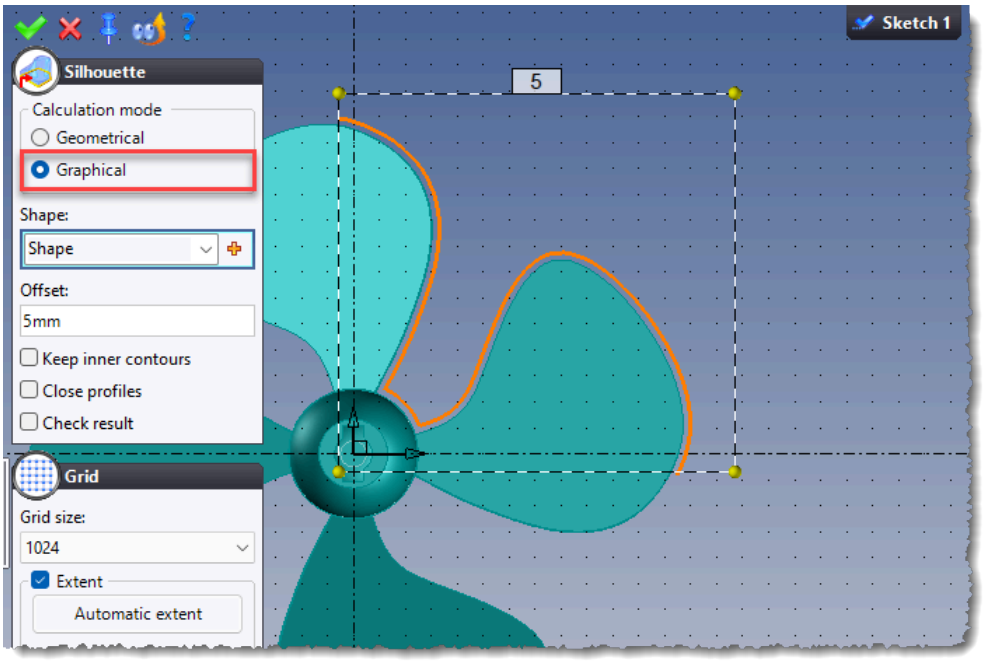


***Circular arc** mode*

Silhouette

The new **Graphics** mode is available in the silhouette's computing operation. It allows computation to be carried out by the graphics card, reducing calculation time for complex shapes.

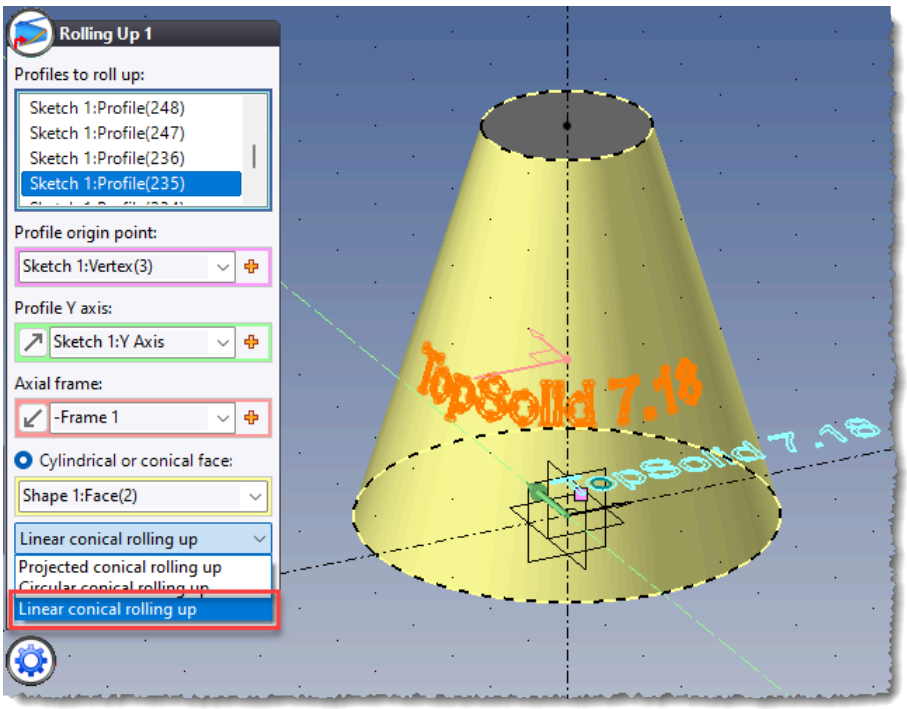
In addition, this mode features an option to perform a silhouette offset, as well as an option to restrict the calculation to a specific area.



Example of a graphic silhouette trimmed by an area.

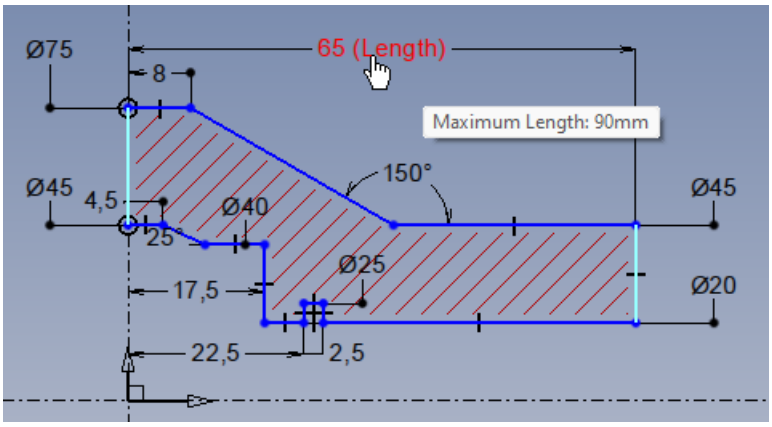
Rolling up

In the rolling up command, the new **Linear conical rolling up** mode allows, for example, to obtain letters that follow the isometric curves of the conical face.



Information

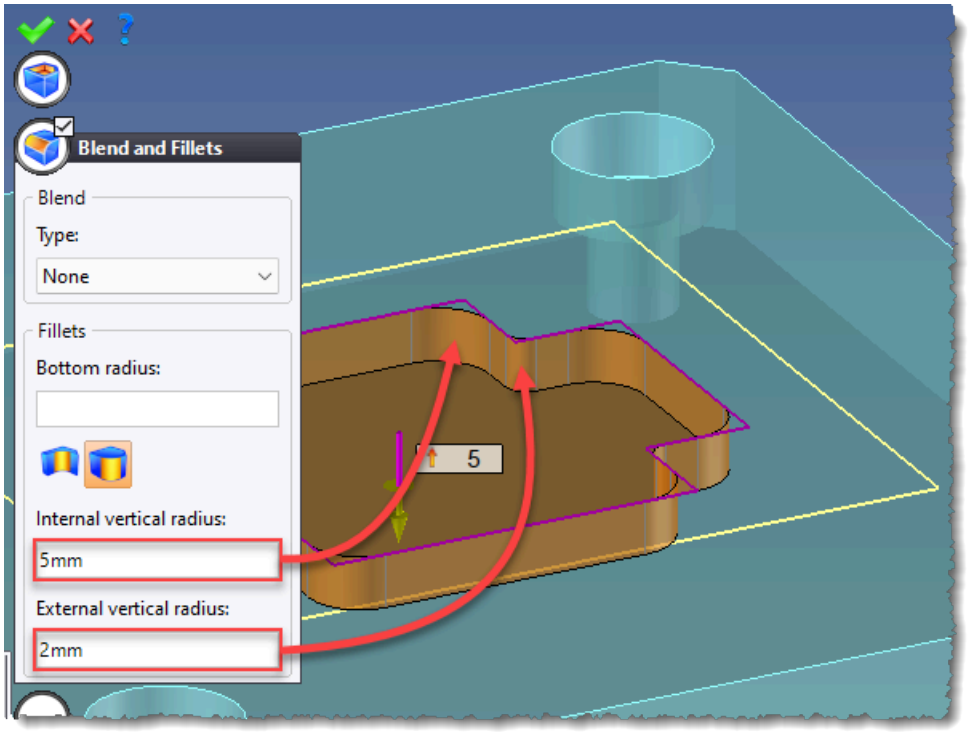
When adding information with the **Others** > **Information** context command, the information description is now displayed in brackets after the dimension value, and the comment is displayed when hovering over the dimension with the cursor.



Shape

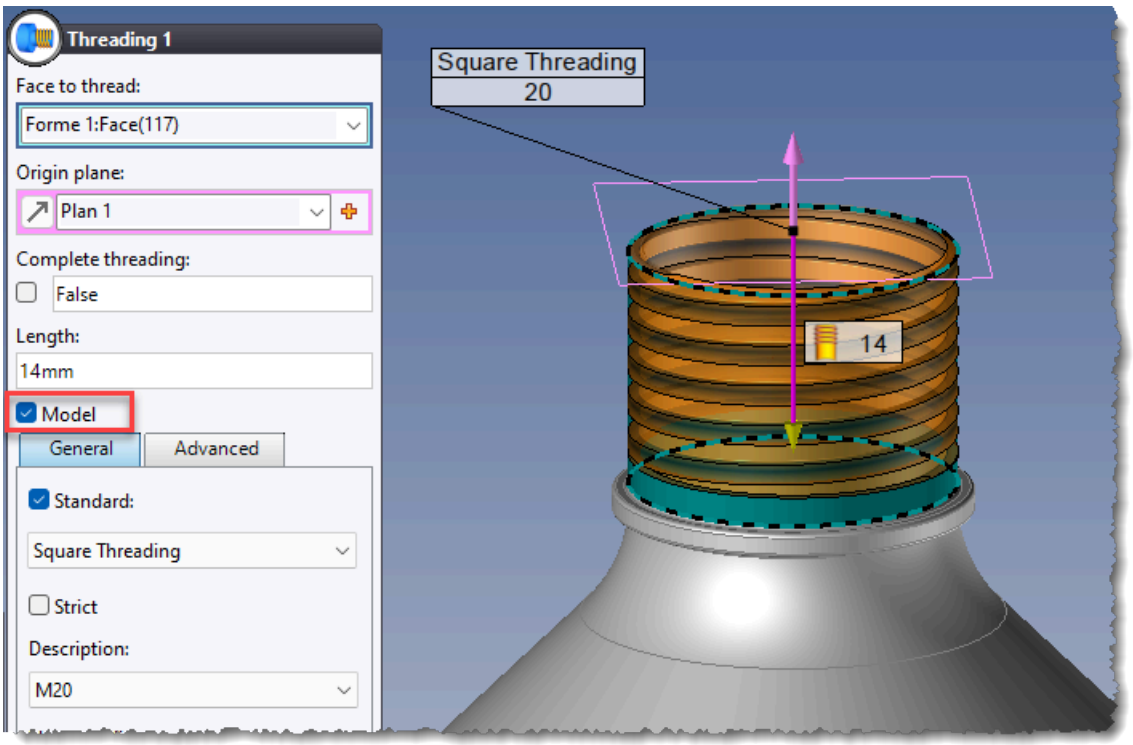
Pocket

When the pocket contour contains concave and convex portions, you can now specify different fillet values.



3D threading

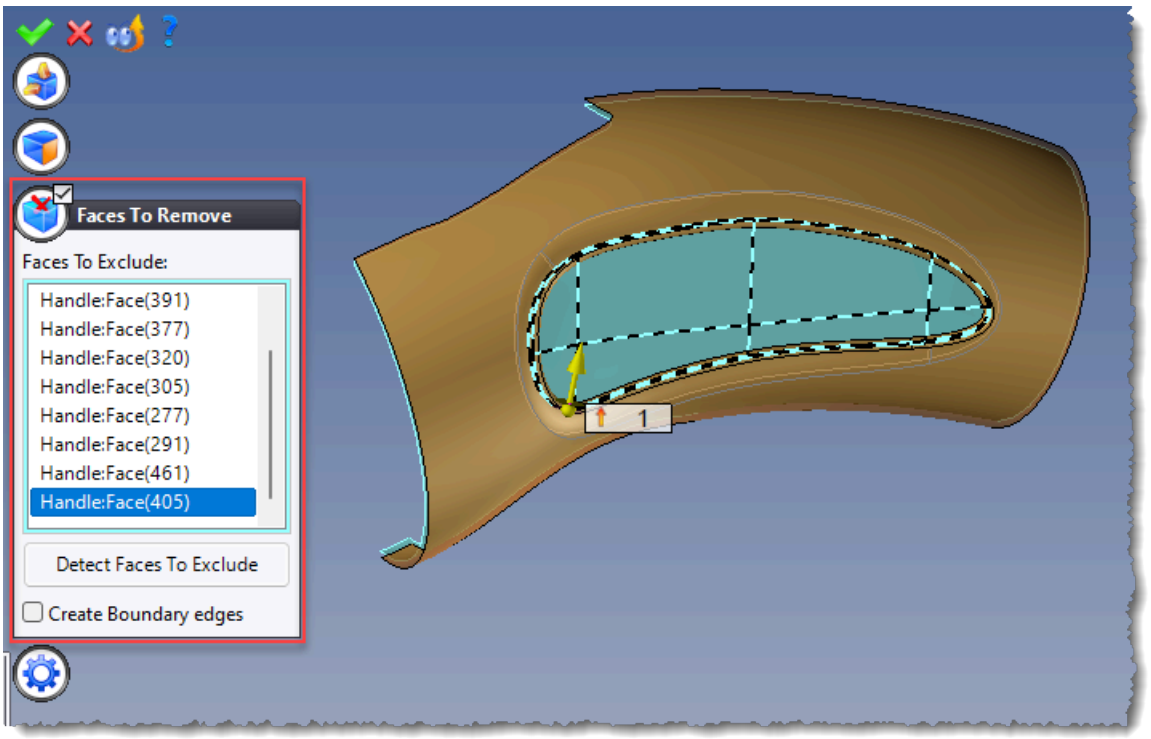
The **Model** option is now available in the drilling (tapped hole) and threading commands. It allows you to create the actual shape of the tapping or threading for 3D printing.



Offset

The dialog box for the **Offset** command now includes a new section for manually excluding faces.

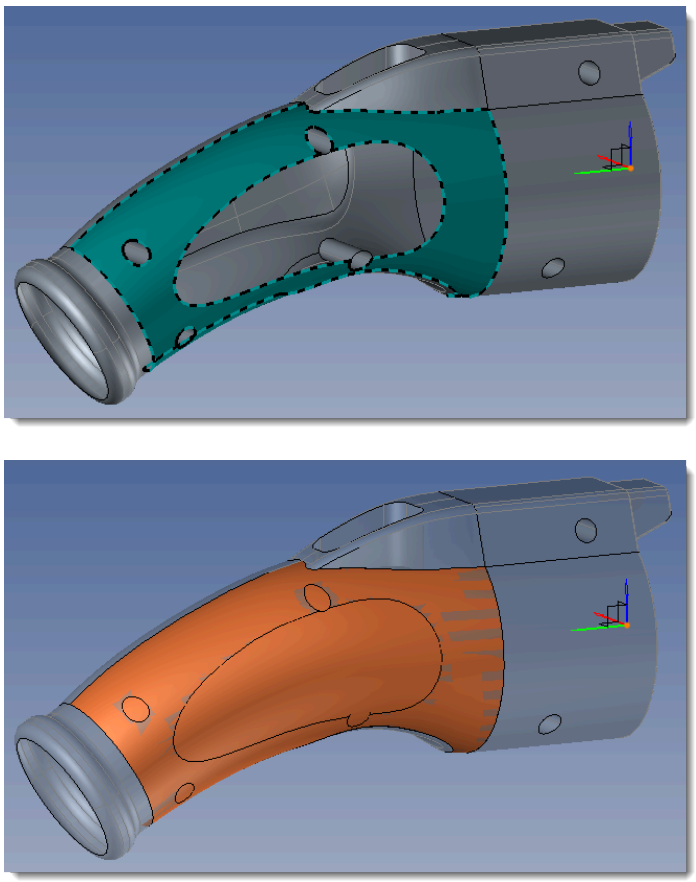
Moreover, the **Detect faces to exclude** button automatically deletes faces that cannot be offset.

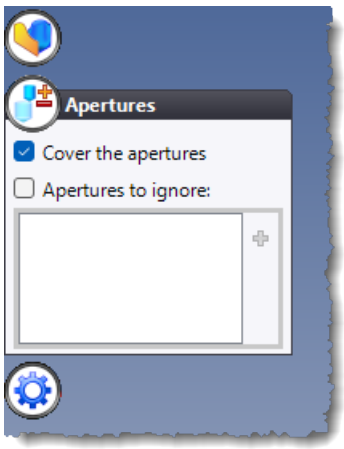


Surface

Face copy

In the **Surface** > **Faces** command, the new **Apertures** section allows you to copy a face in restricted mode without taking holes into account.





Removing

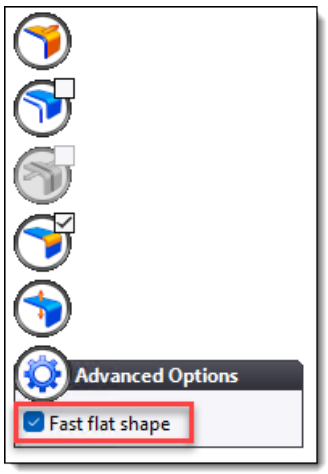
The new **Bodies to remove** and **Bodies to keep** modes are available in the removing command. The **Bodies to keep** mode can be very useful when you want to keep only one body from the shape.

Sheetmetal

Sheet metal on sketch

In the **Sheet Metal on Sketch** command, the new advanced **Fast flat shape** option significantly improves creation times for sheet metal on sketch in the **Flat shape** mode. This can save up to 80% of time in the case of a sketch containing a large number of profiles (a grid with holes, for example).

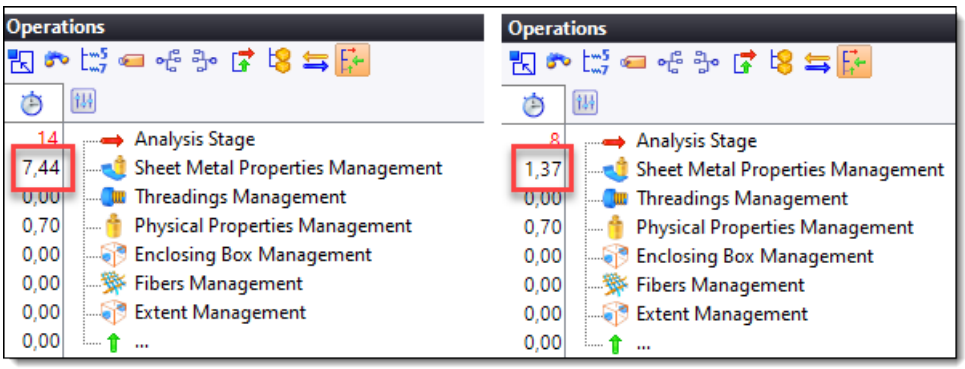
This option is checked by default in the command.



Improved sheet metal properties management

Sheet metal properties management significantly boosts performance, reducing calculation time by up to 80%.

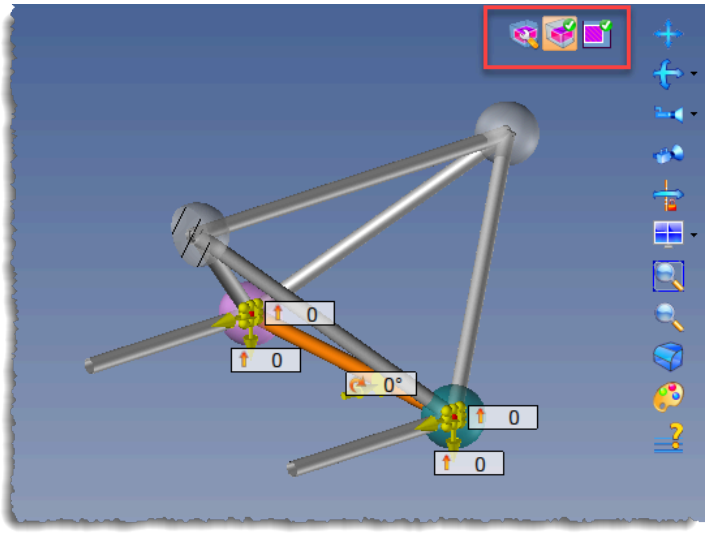
To benefit from this optimization on old files, just regenerate the operation named **Sheet Metal Properties Management**.



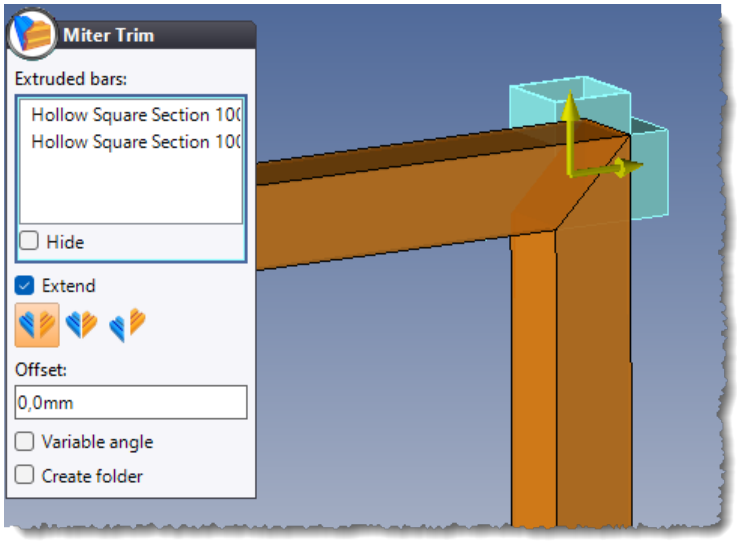
Assembly

Extruded bars

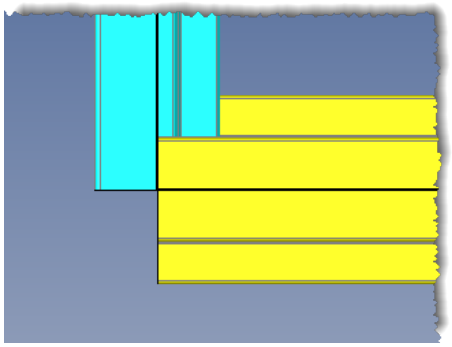
When installing extruded bars, three new icons at the top right of the graphics area allow you to make a graphic cut to facilitate positioning.



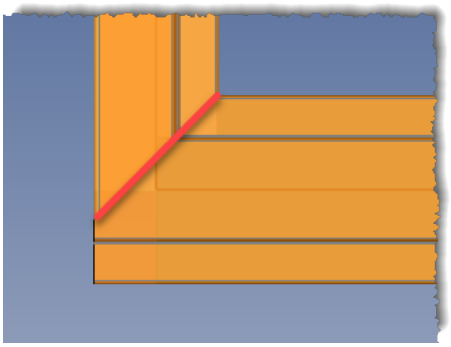
In the **Modeling** menu, the **Miter Trim**, **Main Trim** and **Trim** commands now integrate a preview of the result.



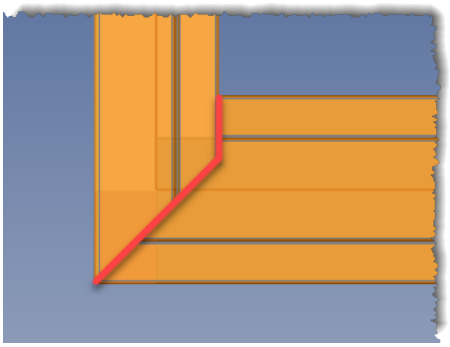
The **Variable angle** option has been replaced by five icons to manage trims between complex extruded bars.



Basic extruded bars



Inside trim



Outside trim

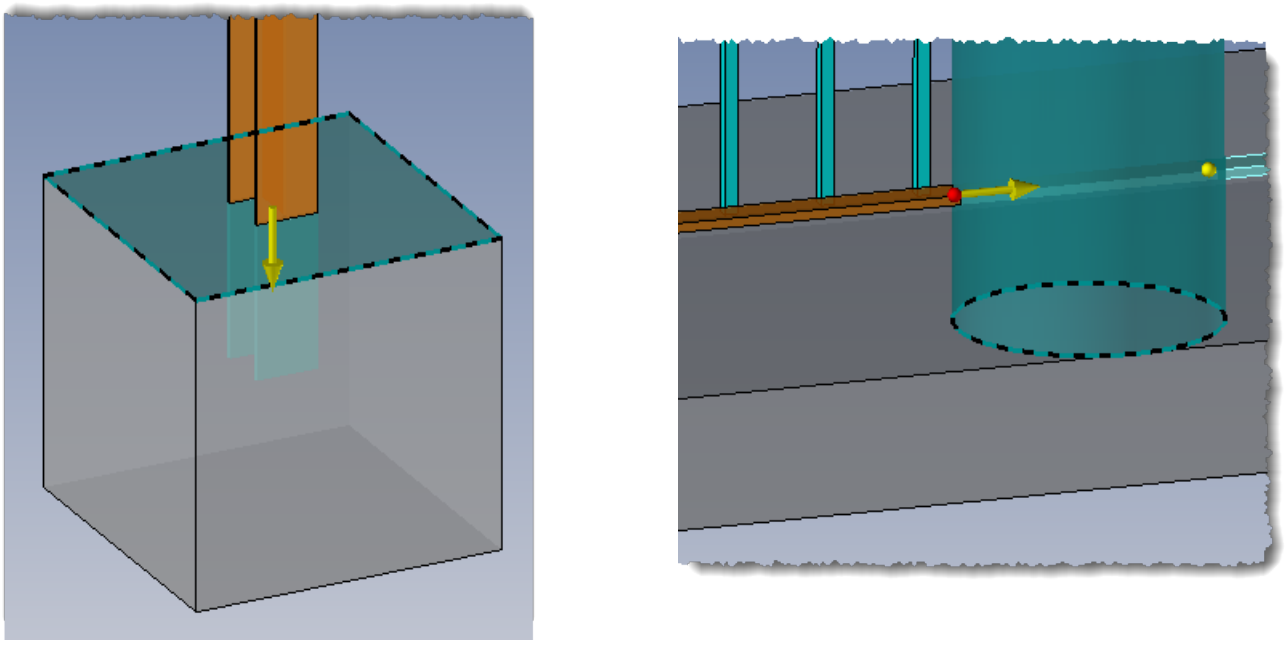
The **Modeling** > **Planar Trim** command has been renamed **Trim** and now offers four trimming modes:

- **Trim by plane**
- **Trim by surface**
- **Trim by faces**
- **Trim by profile**

New handles are available:

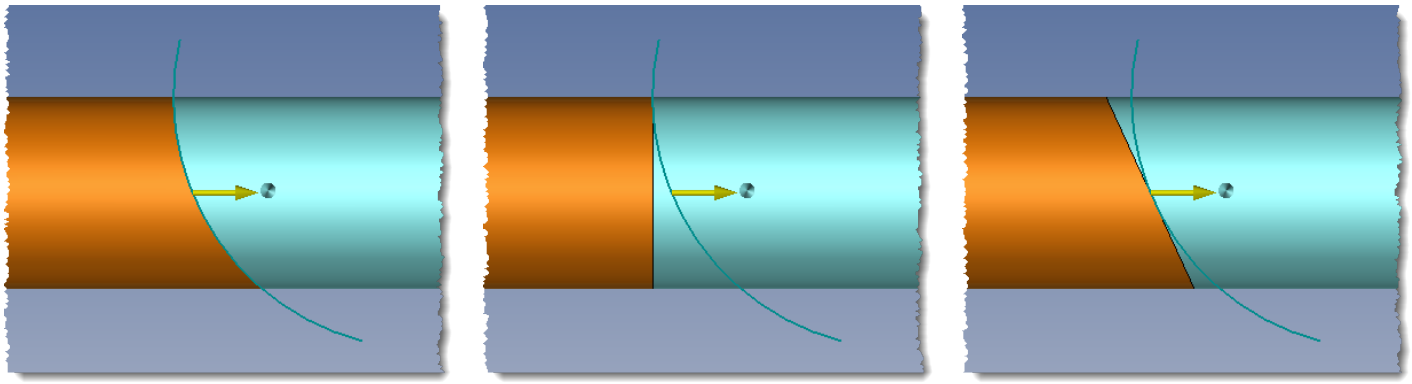
- The yellow handle is used to select the portion to be retained. In previous versions, only the longest portion of the extruded bar was retained.
- In ambiguous cases, like the trimming of an extruded bar passing through a cylindrical face, yellow and red spherical handles allow you to choose which side of the cylindrical face to use.

Note that these handles are also available in the **Modeling** > **Main Trim** command.



Examples of profile trimming with the new handles.

The **Trim** command also features three new straightening modes.

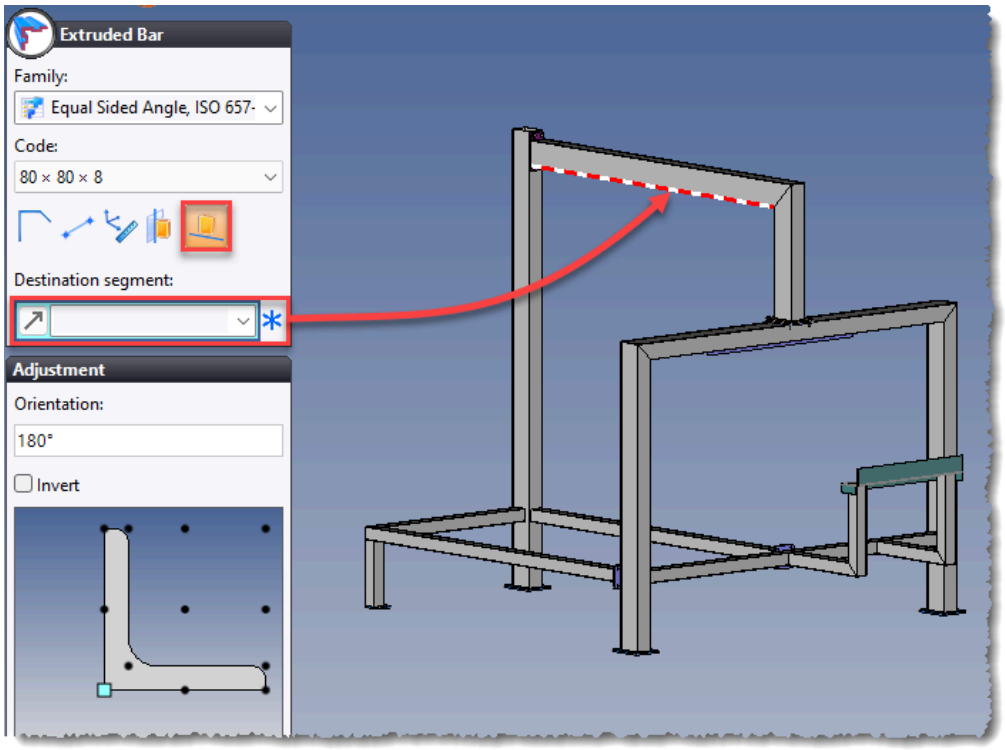


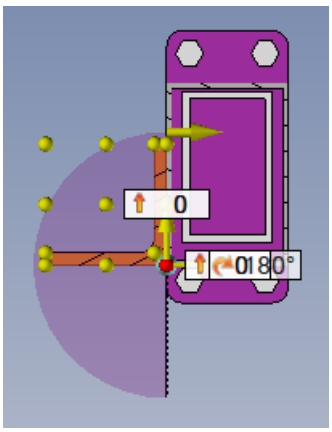
None *Straight* *Tangent*

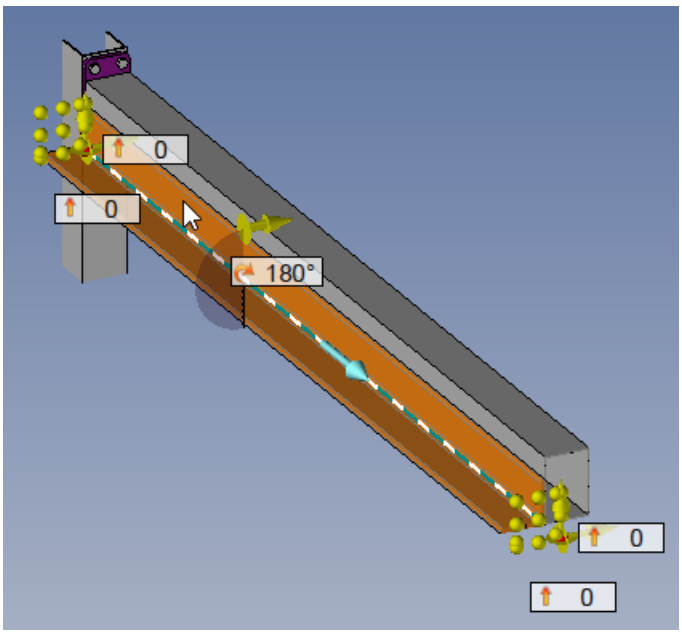
Extruded bars imported and converted via the **Modeling** > **Sheet Metals Recognition** command are now supported by extruded bar trimming commands.

The new segment-based positioning mode automatically performs the following operations:

- positioning an extruded bar on the selected segment;
- create a cut by box with the length of the selected segment and the dimensions of the extruded bar section;
- create a cut by plane passing through the selected point on the segment;
- orientate the camera normally to the cut;
- zoom in on the section.

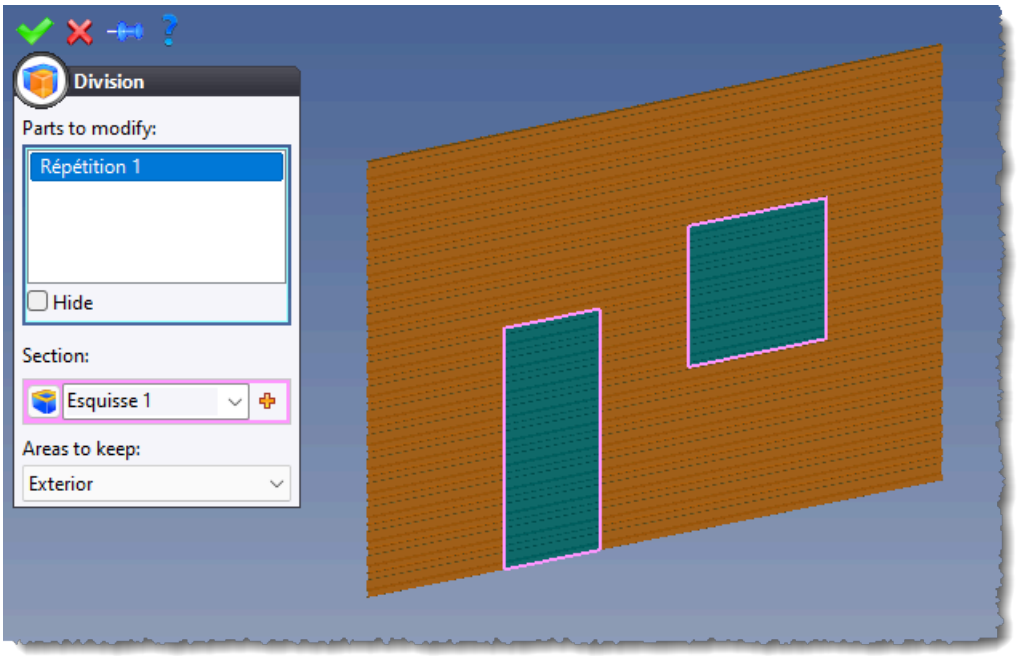






Division

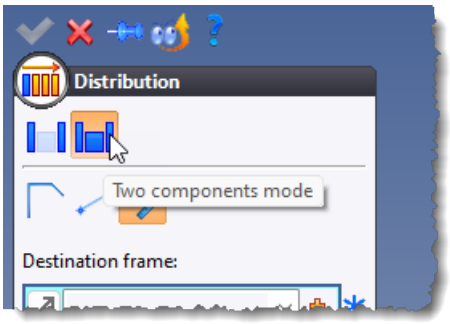
The new **Division** command is available in the **Modeling** menu. Similarly to a part document, it can be used to divide a part or assembly into several parts.



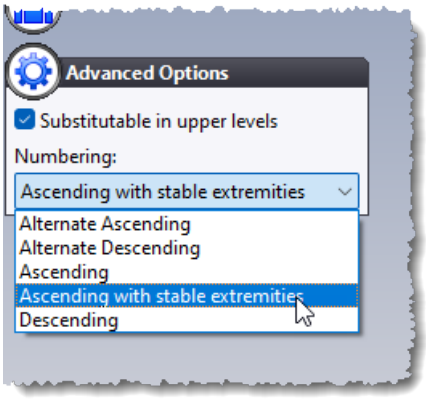
Distribution

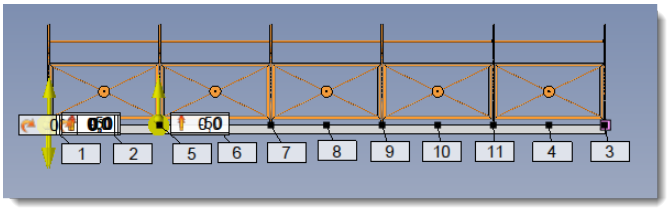
A number of modifications have been implemented in the component distribution command:

- As with extruded bars, three new icons at the top right of the graphics area facilitate graphic cuts by box streamlining the positioning process.
- Two modes for selecting the number of components are now available at the top of the dialog box.



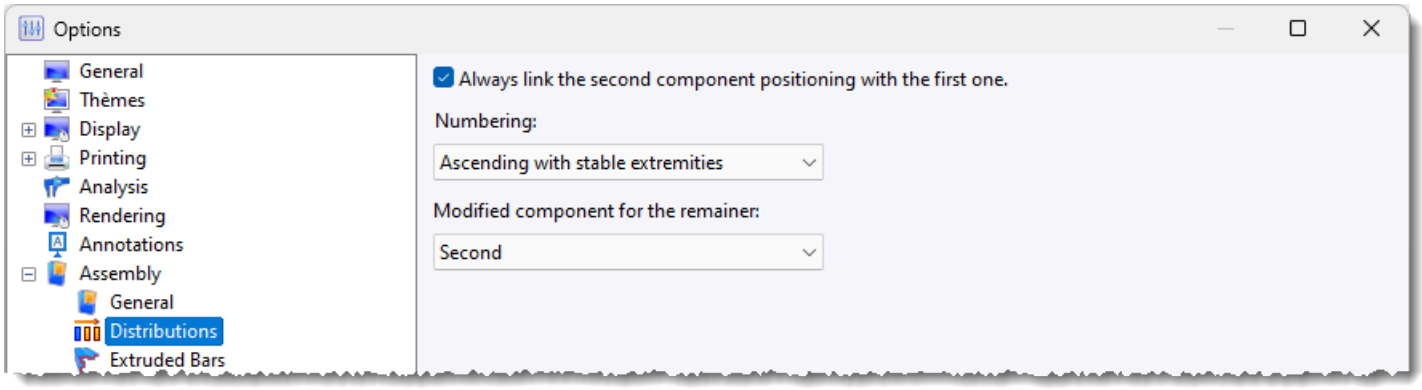
- In the **Second component** section, the **Link the second component positioning with the first one** option applies the same settings (orientation, inversion, offset) as those defined in the **First component** section.
- The numbering type choice has been added to the **Advanced Options** section at the bottom of the dialog box.





Example of ascending numbering with stable extremities.

- In the **Tools** > **Options** > **Assembly** command, the new **Distributions** section is used to define default values for the **Link the second component positioning with the first one**, **Numbering** and **Modified component for the remainer** options.

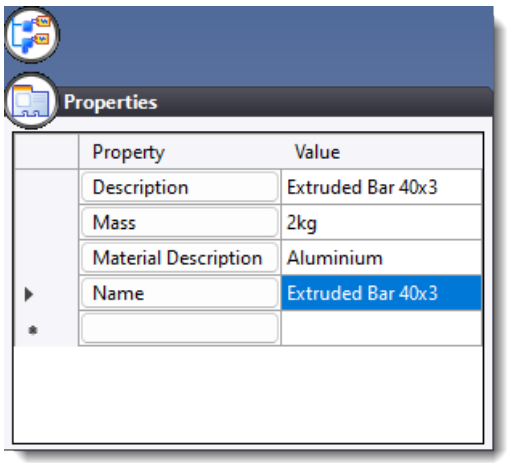


Unsynchronized operations

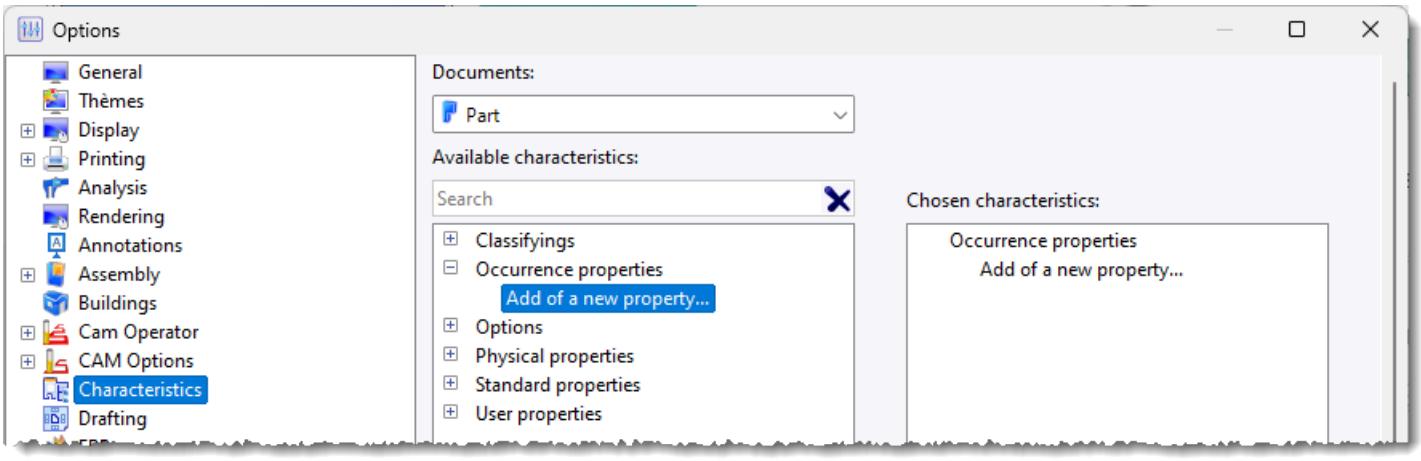
You can now regenerate a selection of unsynchronized operations from the Operations tree.

Occurrence properties

It is now possible to replace the system properties present in definition documents. This allows you, for example, to modify the description or part number of a component included in the assembly.



In addition, the **Occurrence properties** section has been added to the **Tools** > **Options** > **Characteristics** command. This allows the modification of the properties of a part's occurrences from a bill of materials displayed in the Parts tree or from the **Tools** > **Characteristics** command.

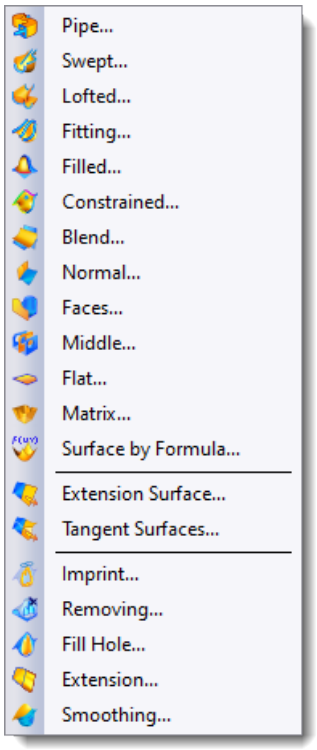


Subcomponent processes

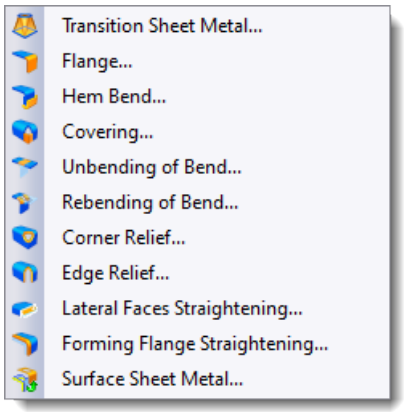
Subcomponent processes are now carried out automatically.

Local shapes

The **Local Shapes** sub-menu has been renamed **Shapes** and the commands for creating surfaces and sheets have been added to the **Surface** and **Sheet Metal** menus.



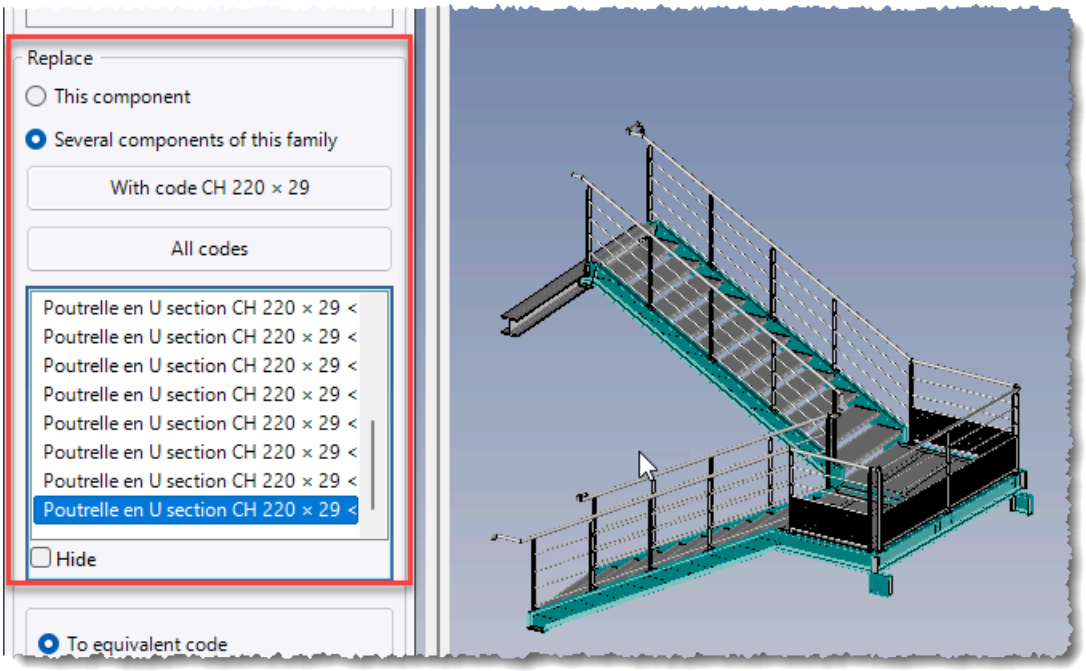
*Menu **Modeling** > **Surface***



*Menu **Modeling** > **Sheet Metal***

Replacement

In the **Replace** section, check boxes have been replaced by buttons which pre-fill the list of selected elements, giving a preview of the elements to be replaced.



Replacement wizard

In the **Replacement Wizard** dialog box, the choice of **Redirect Hooks** or **Replace Geometries** modes has been removed. The default mode is now **Replace Geometries**.

However, it is possible to re-display the choice of modes by starting **TopSolid** in obsolete mode. To activate this mode, add the "-o" argument to the launch shortcut.

Local parts and assemblies

Symmetry of local parts and assemblies is now supported.

In addition, the context **Provide Function** command is now available on a local part or assembly. This allows you, for example, to define the nesting characteristics for a local part.

Building

Staircase layout

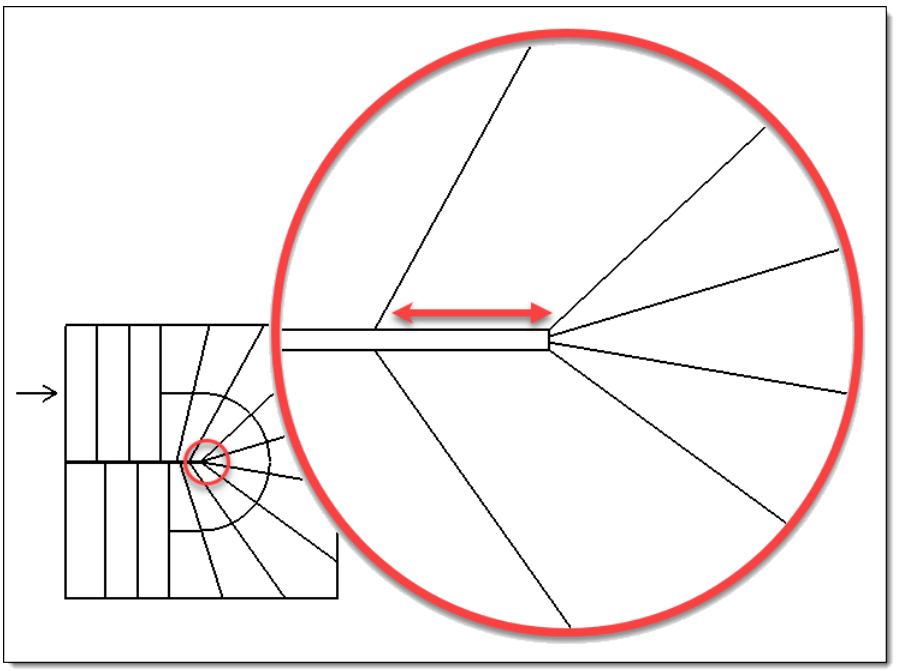
Height to cross

The **Height to cross** information is now available and can be set in the graphics area label. It is possible either to enter a length, in which case the value can be edited in the graphics area label, or to select a point in the graphics area, in which case the height cannot be modified.

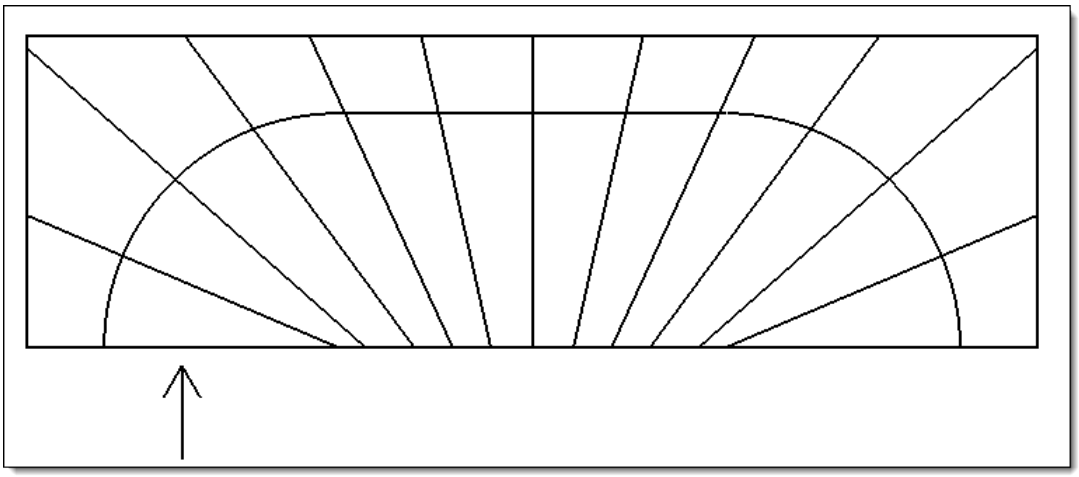
Winder staircase with short flight length

When creating a staircase with a winder landing, the collar of the treads increases progressively as moving away from the change in direction.

With the new version **7.18**, the minimum collar is given to the first treads if the flight length is sufficient; otherwise, the minimum collar is only applied to the first tread of adjacent flights.



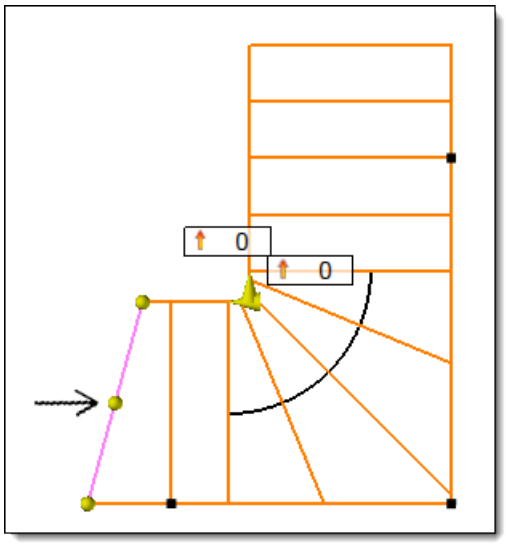
The minimum collar is only applied to the flight permitting.



In version 7.17, the above case could not be created with a collar value other than 0mm.

Modify 2D layout

The new mode **Modify 2D Layout**, accessible via the icon in the graphics area, allows you to locally modify a tread on the 2D sketch. When this mode is activated, it is possible to select a tread segment and rotate it around the treading line by moving the yellow spheres.

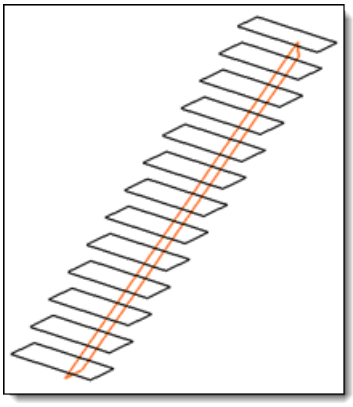


In the example above, the first tread has been modified.

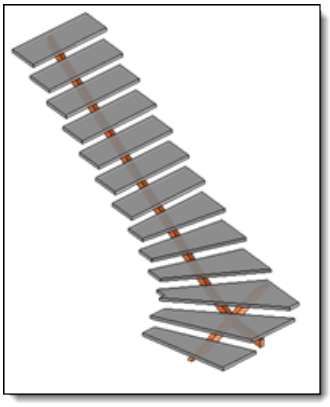
The exception is marked violet.

Stringers

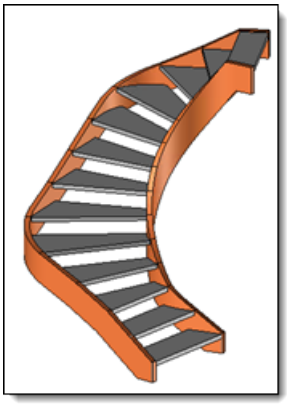
The new **Stringers** section allows you to create sketches and local shapes for stringers.



Sketch for central stringer under treads



Local shapes of stringers under treads

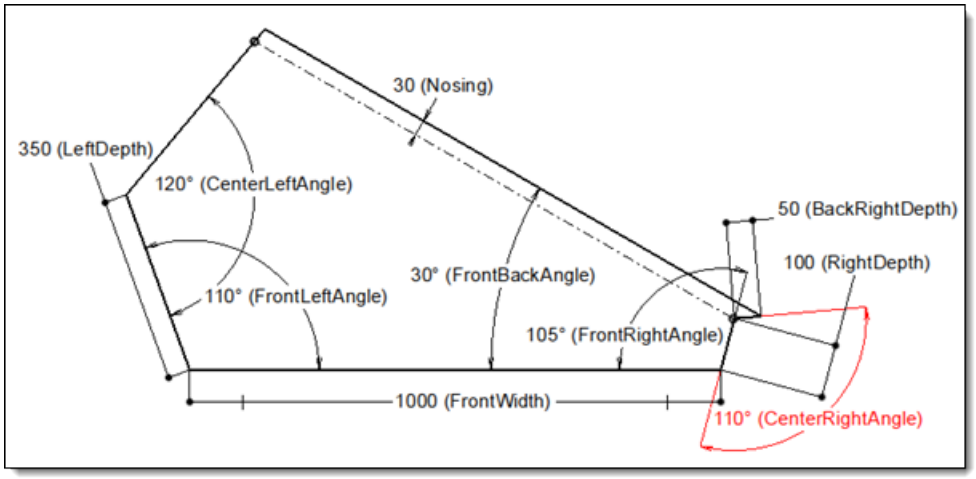


Local shapes for lateral stringers

Staircase treads

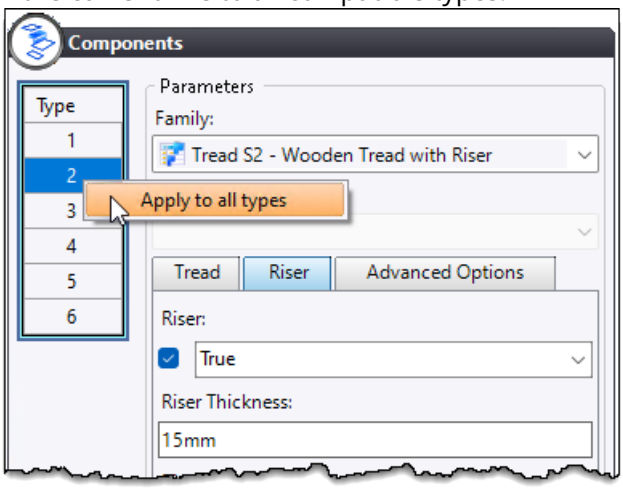
Family driving

The **Center Right Angle** driver has been added to the **Tread family** driving family for T7 type. This means more tread geometries can be matched to the T7 type.



Apply to all types

The new context-sensitive **Apply to all types** command is available from any line in the treads table. It applies the parameters (family and drivers) of the current line to all compatible types.



Modeling

Marking

Font instances used by the marking command are now referenced; these are exported when the document is exported and cannot be purged if used by a marking.

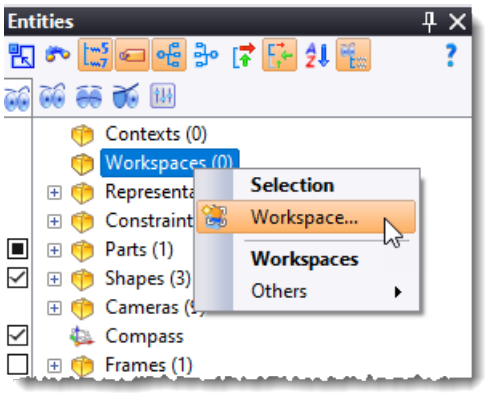
Workspaces

Menus

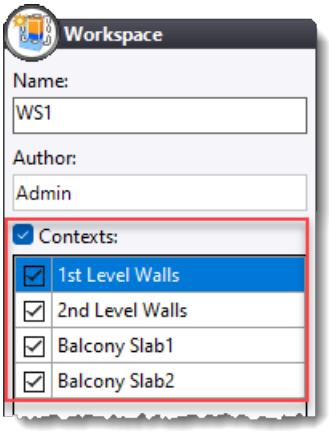
In a **Workspaces Manager** document, the **Shape** and **Surface** menus have been simplified and the **Construction Shapes** command has been added to the Construction menu.

Creation

Workspaces can be created directly from the context menu on the **Workspaces** folder of the Entities tree.

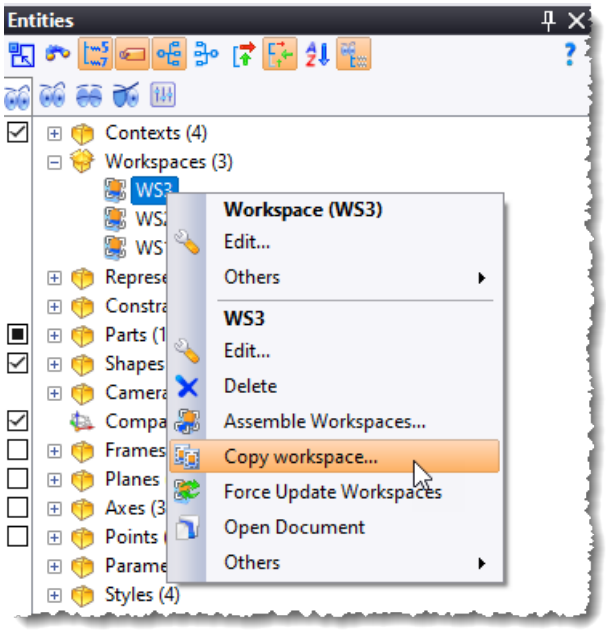


In the creation dialog box, the new **Contexts** check box allows you to select or deselect all contexts.



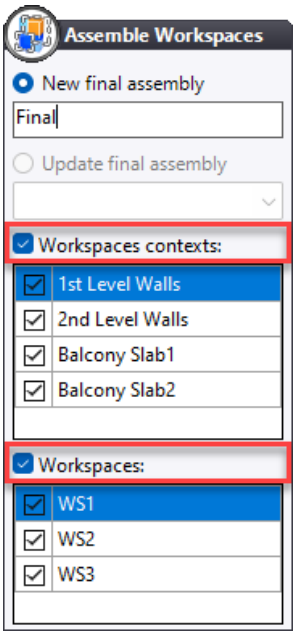
Copy of workspaces

The new **Copy workspace** context command can be used to duplicate a workspace with its background document. It facilitates, for example, the creation of several versions of the same workspace.



Assembly

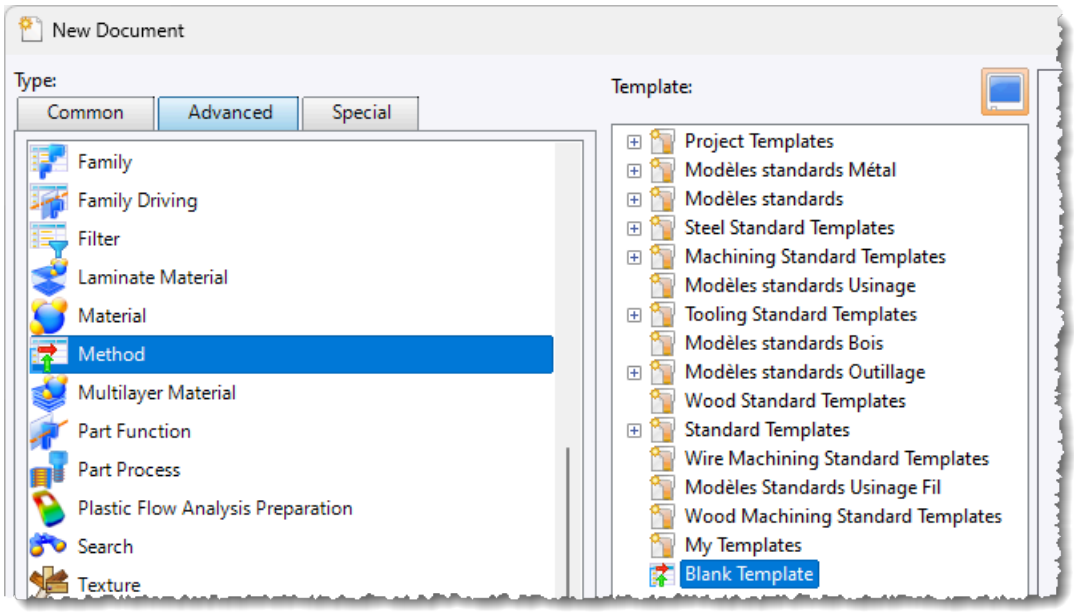
Similarly to the workspace creation command, the workspace assembly command features a new check box for selecting all contexts and workspaces with a single click.



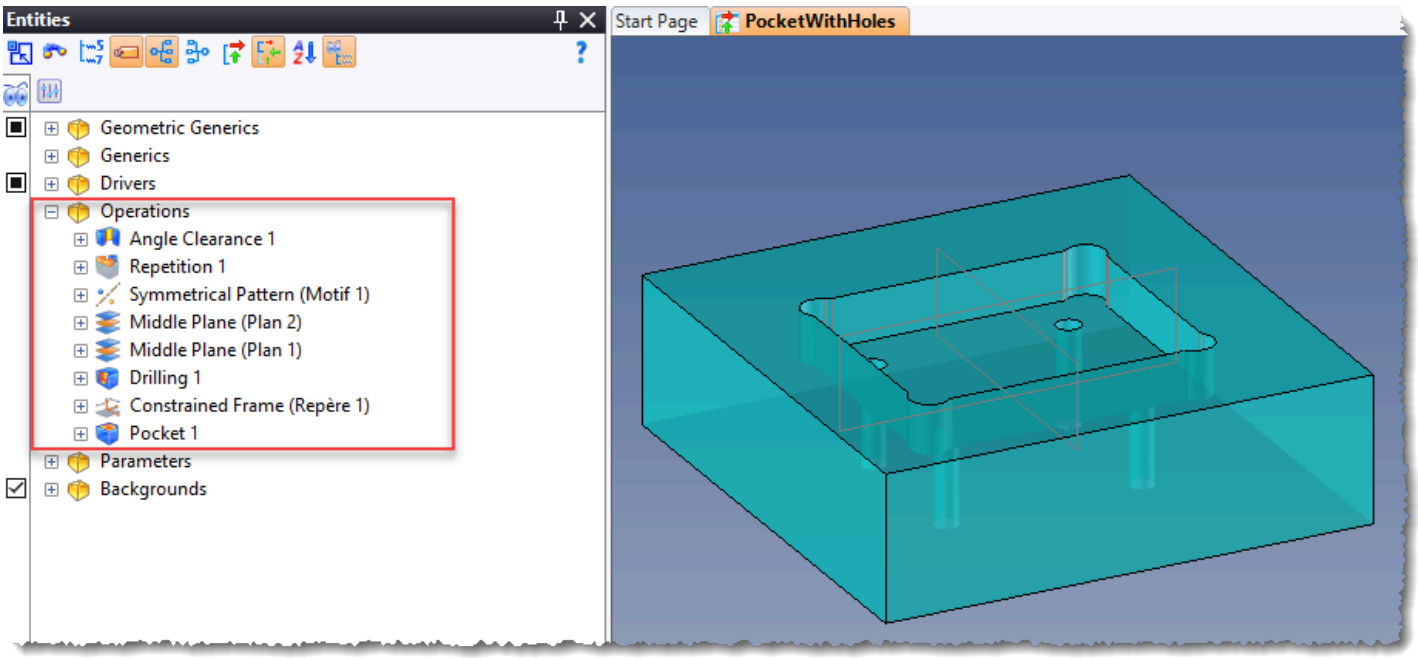
Method

A method is used to define a sequence of operations and apply them to an element of a **Part** or **Assembly** document. This allows you, for example, to create a rectangular pocket with drillings on each vertex simply by dragging and dropping a **Method** document into a **Part** document.

This new document type is available in the **Advanced** tab of the document creation window. It functions similarly to the **Family** document.



When the method is applied to an element, the operations are recreated on it without creating a link to the method. They can therefore be modified independently.



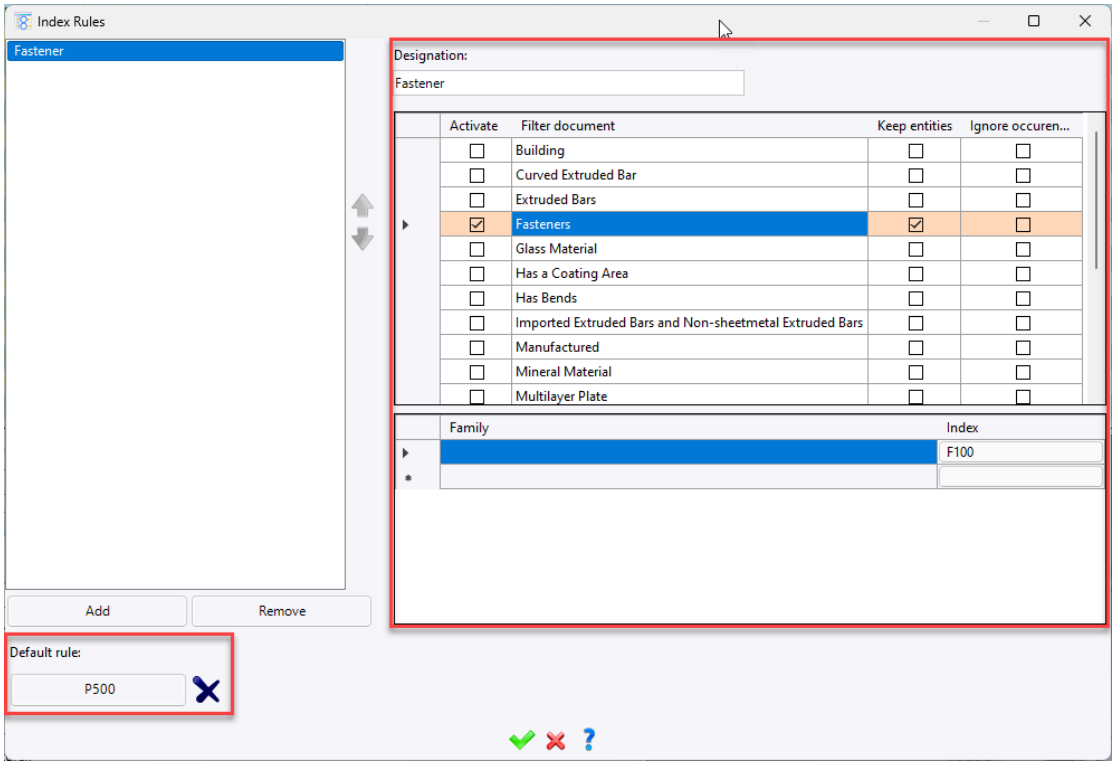
*When including the **Method** document, all operations framed in red are automatically recreated on the destination shape.*

Bill of materials

Index rules

Index rules are no longer defined solely in relation to **Function** documents. They can use **Filter** documents to define the parts to be considered.

It is also possible to define a default indexing for parts that do not correspond to any of the defined rules.



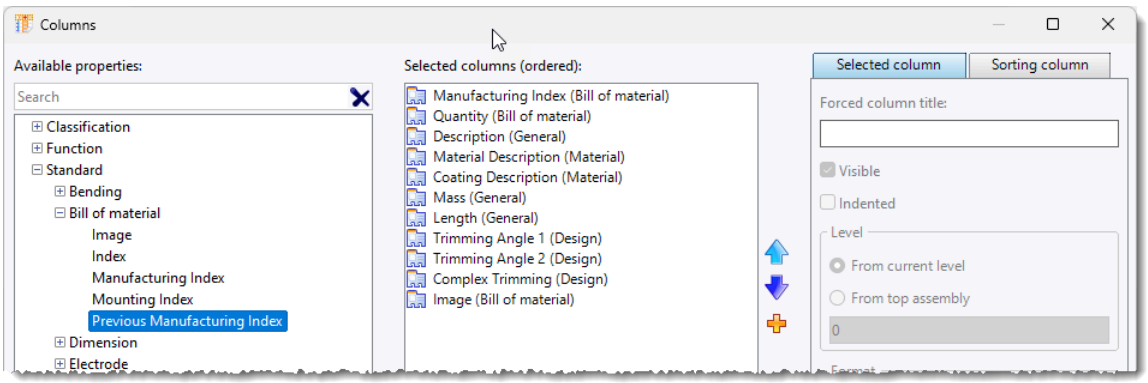
In addition, the two new **Assembly Type For BOM Rules** and **Part Type For BOM Rules** commands allow you to change the value of the **Type for BOM** parameter in the bill of materials only.

Property filters

Properties assigned to part occurrences are now supported by property filters.

Manufacturing indexes

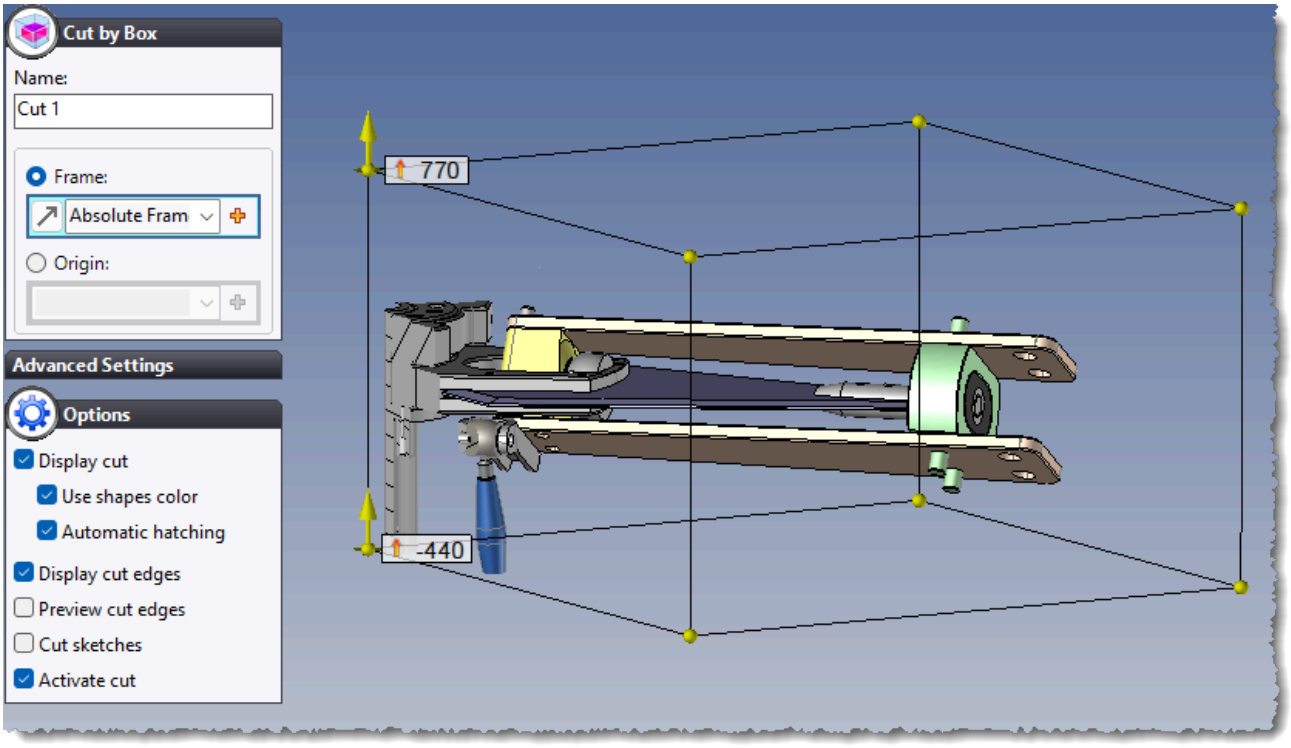
It is possible to obtain a list of modified manufacturing indexes by displaying the new property **Previous Manufacturing Index** in a column. Similarly to the manufacturing indexes, these indexes can be reset via the **Bill of Material** > **Clear Previous Manufacturing Index** command.



Visualization

Cut by box

The new **Cut by box** command in the **Visualization** menu allows you to create a graphic cut from the element bounding box.



Example of a graphic cut by box.

Realistic rendering

If no rendering camera is present in the document, the **Visualization** > **Realistic Rendering** command now proposes to create a rendering camera when it is launched.

Studio

Scattering volume

The new **Scattering Volume** command is used to define a volumetric effect. This matches the effects induced by light passing through particles like gas, smoke, dust, etc.

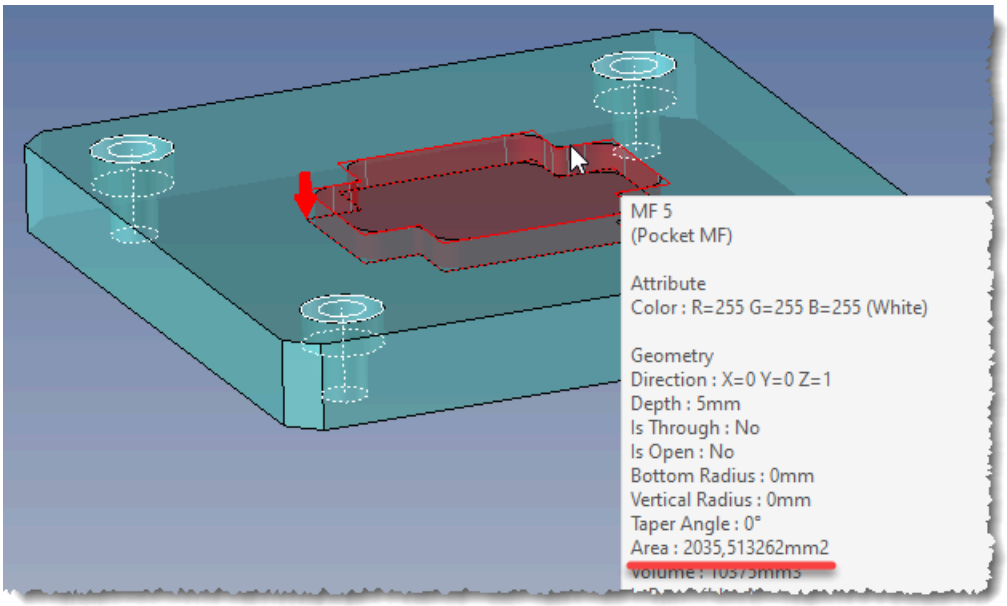
Without scattering volume

With scattering volume

Analysis

Area

The **Analyze** command can now be used to display the area of a pocket MF.



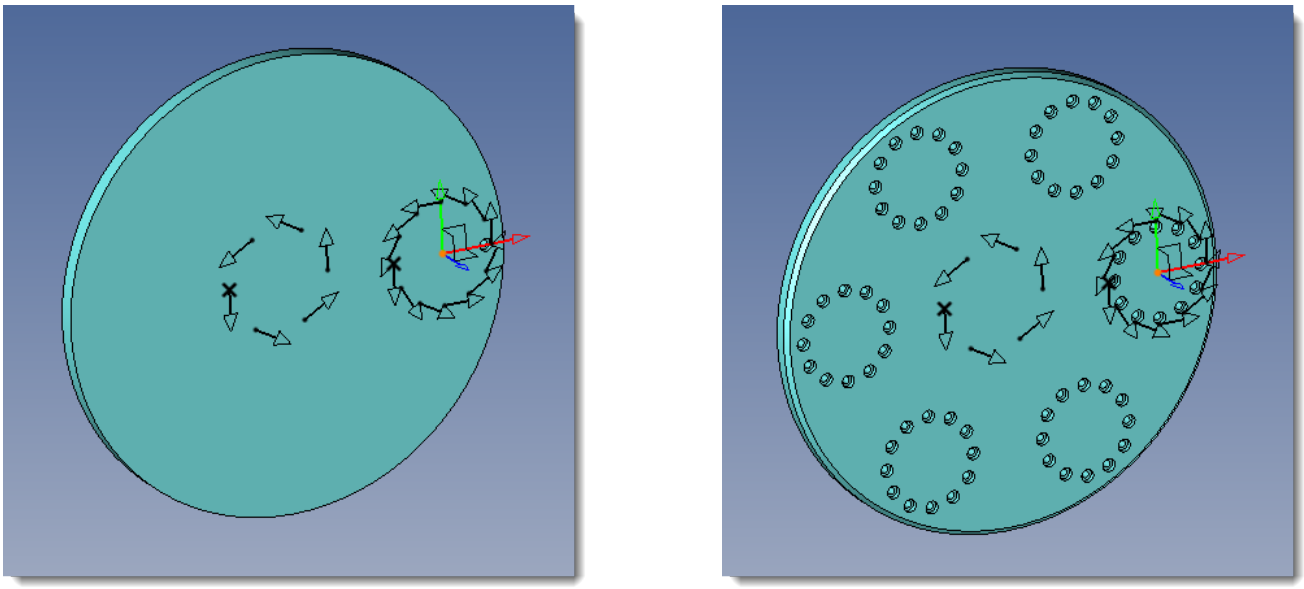
Construction

Color parameters

The **Color Parameter**, **Table Color Parameter** and **Switch Color Parameter** commands have been added to the **Construction** > **Parameters** > **Other Parameters** menu. They can be used to set the color of an element in the **Material**, **Coating** and **Finishing** documents. The parameter created can then be used as a driver in a **Family** document.

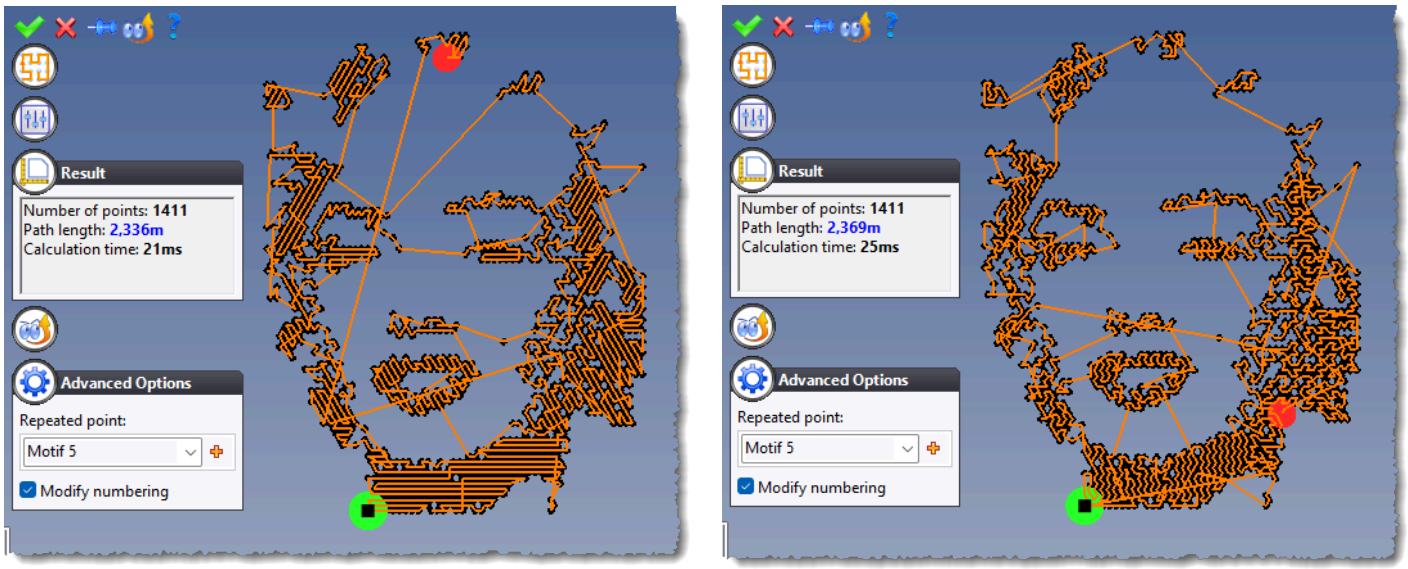
Combined pattern

The **Construction** > **Patterns** menu has been enhanced with the new **Combined Pattern** command, which allows you to multiply and sum multiple patterns.



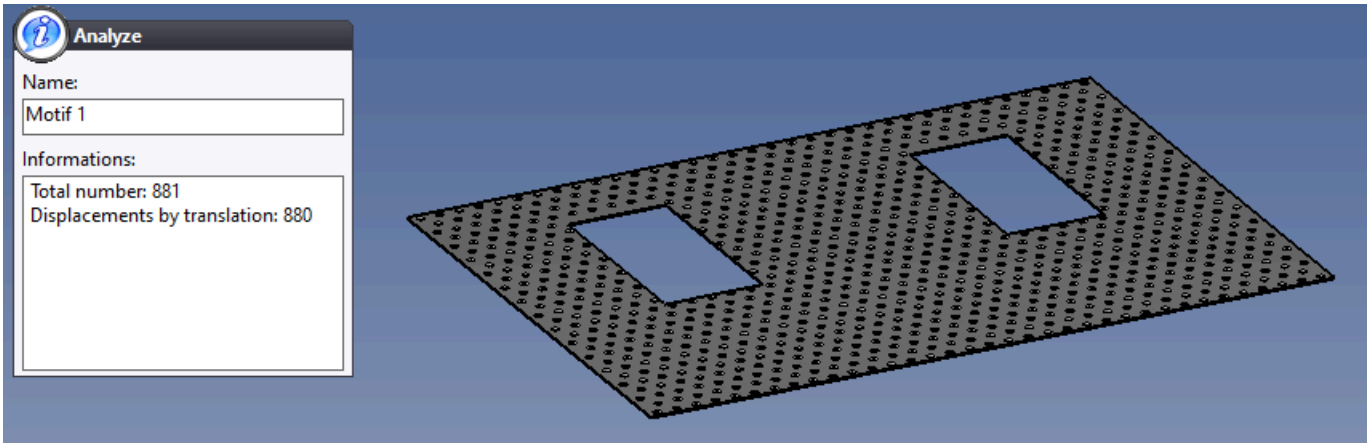
Example of a multiplication pattern combination (right) from two circular patterns (left).

In addition, the new **Construction** > **Patterns** > **Pattern Optimization** command can be used to modify the order of pattern occurrences in order to control the machining path.



Example of pattern optimization from different modes.

In addition, the **Analyze** command is now available in the context menu, providing the number of elements in the pattern.



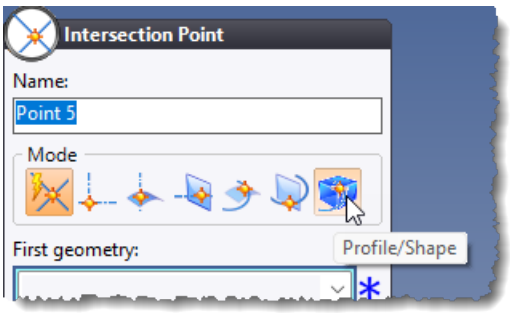
Repetition

The **Repetition** command's performance has been improved.

For example, repeating a point in a 100 x 100 matrix pattern now takes just 2 seconds, instead of more than 20 seconds in previous versions.

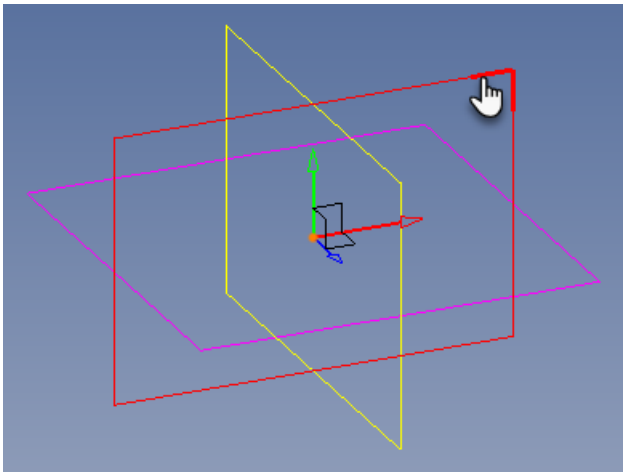
Intersection point

You can now create an intersection point between a profile and a shape. This ensures greater point stability when the shape is modified, or when the intersection point is likely to move to another face on the shape.



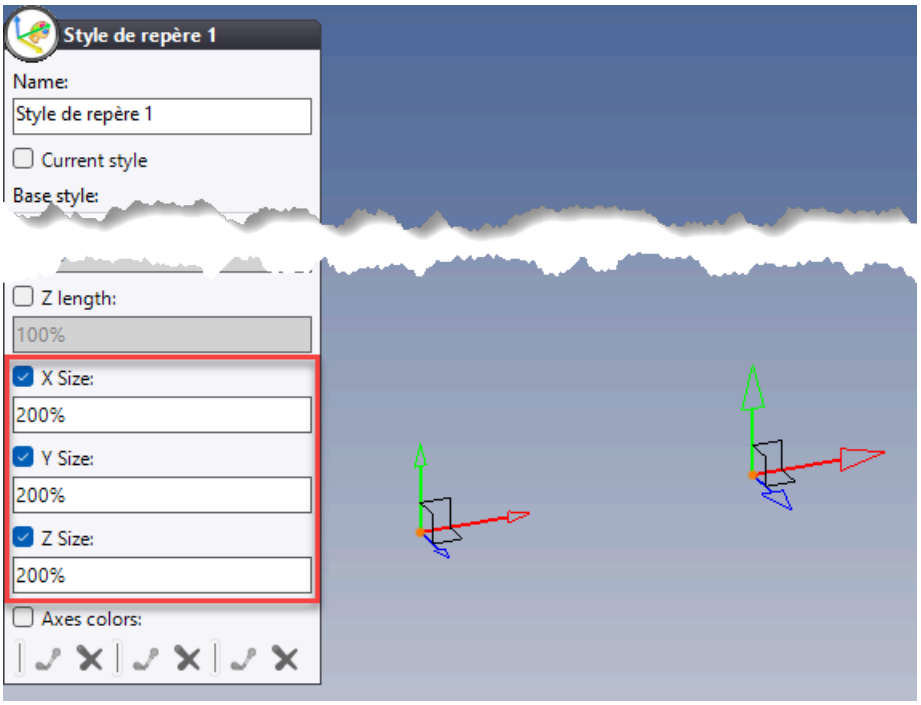
Plane

When hovering over a plane, modifiers are now displayed at the corners and on the middles to change the size of the plane.



Frame

Fields for adjusting the size of frame extremities are now available in the frame style dialog box or when creating a frame.

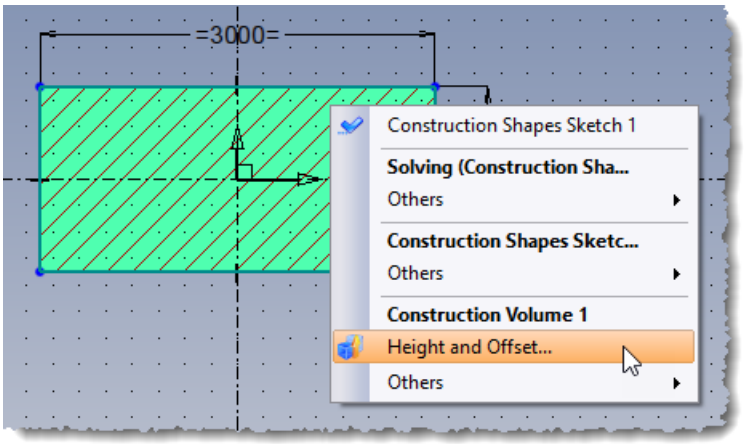


BOM index

The new **Construction** > **Parameters** > **Bill of Material Index** command allows you to define the bill of material index for the part or assembly. For example, this allows you to retrieve this index in the document name using the **[$BOMIndex]** property.

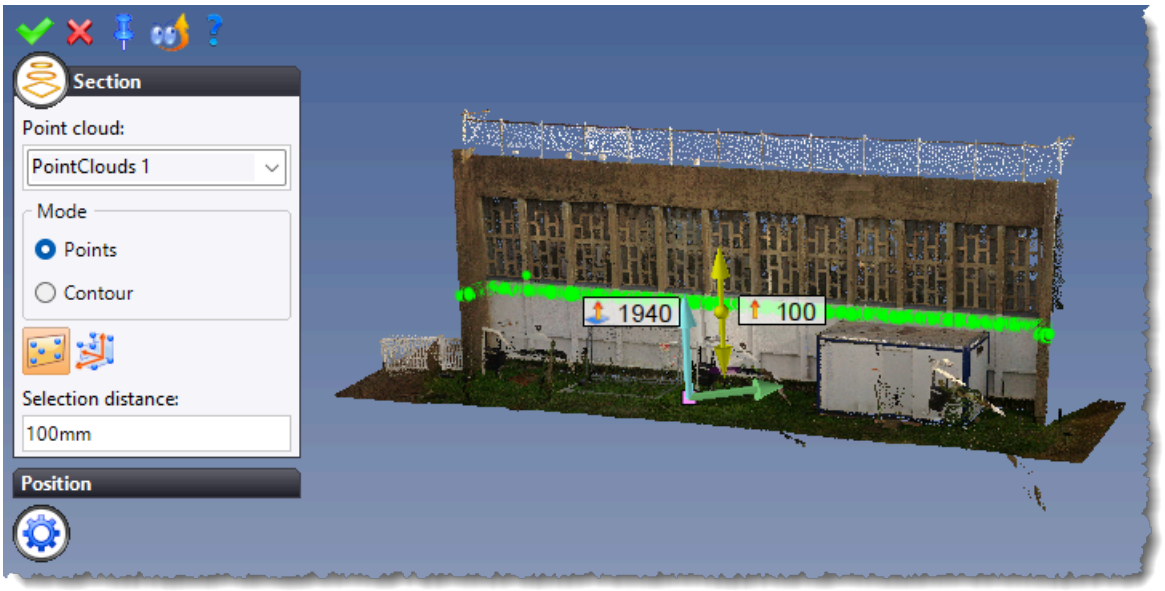
Construction shapes

When creating or editing a construction shape sketch, the new **Height and Offset** context command can be used to force the height and/or offset of each construction shape in the sketch.



Point cloud

On a point cloud, the new **Section** context command isolates a slice of the point cloud by indicating two planes or a box. A mode allows you to either keep the points or to rebuild a contour.



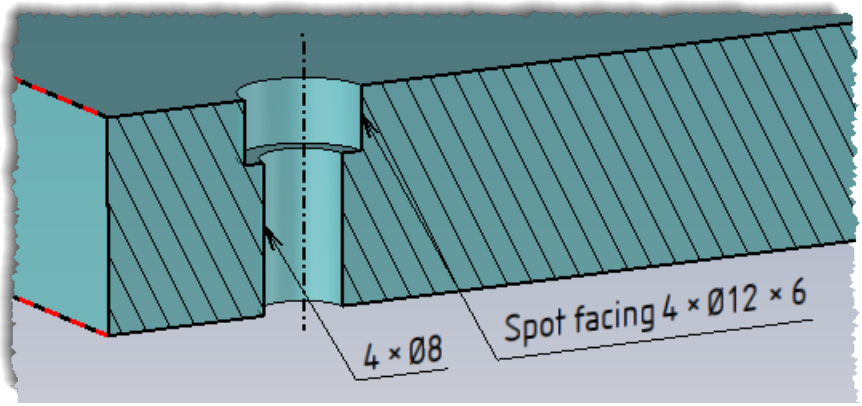
Tools

Annotations

In a 3D document, the operation of the **Tools** > **Annotations** > **Dimension** command is now homogeneous with that of the **Detailing** > **3D Dimension** command from a **Drafting** document.

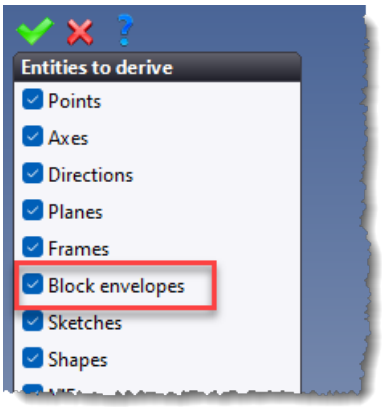
The dimension plane is determined automatically and can be modified manually using the **Change plane** button or the **Ctrl** + **Space** keys on the keyboard.

In addition, the **Drilling Note** command has been added to the **Tools** > **Annotations** menu. Similarly to a **Drafting** document, this command facilitates the retrieval of drilling information.



Derivation

Block envelopes are now proposed in the dialog box of the derivation command.



Processing parts and repeated processing parts

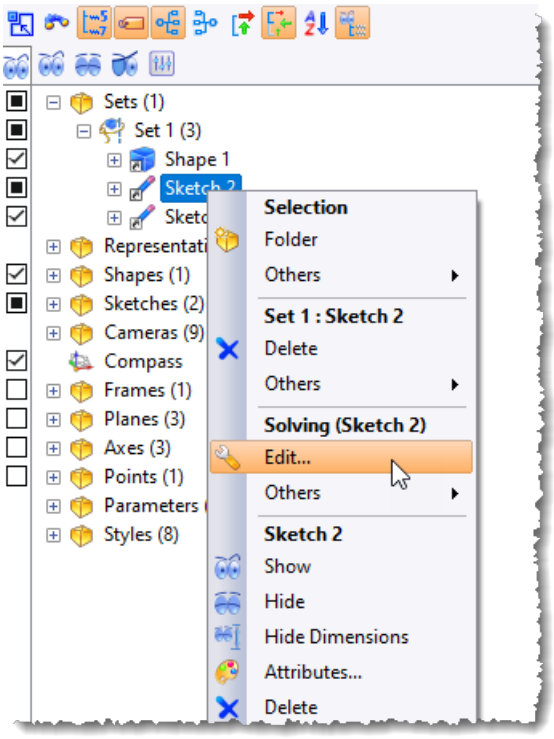
The **Create Process** and **Processing Parts** commands have been moved to a new **Processes** sub-menu.

This sub-menu also contains the new **Processing Parts Repetition** which allows you to create processes for a component, repeat it and define the result as a processing part.

This new command boosts performance by at least 50% (test carried out with a component containing a repetition of 247 screws).

Set

The **Edit**, **Attributes** and **Show/Hide dimensions** commands are now available in a set's context menu.



Company options

The **Predefined Values** section can now be accessed via the **Tools** > **Company Options** command.

Materials and textures

The user interface for the **Material**, **Coating** and **Finishing** documents has been improved:

- The **Common** tab has been renamed **Appearance**.
- The **Advanced** tab has been renamed **Properties**. It also contains the BOM information (description, part number, category, etc.) of the document.
- Rendering can be previewed with the help of a sphere in the graphics area.

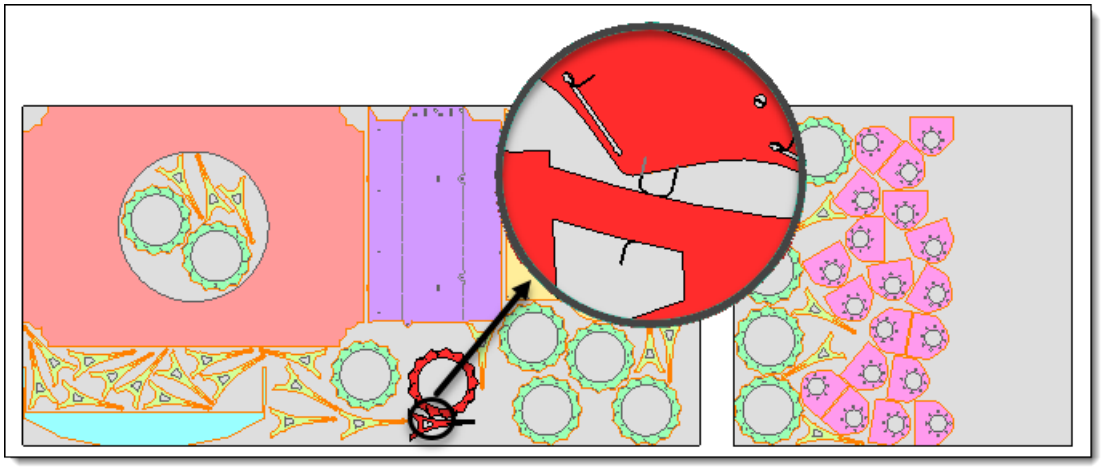
In addition, it is now possible to use parameters in the fields, which can then be driven in a **Family** document.

Nesting (optional module)

Integration of lead in/out geometries

In the case of nestings of machined parts, the lead in/out geometries are now correctly taken into account by the nesting or collision checking.

Moreover, collision management between these geometries and parts has been improved.



Bar nesting

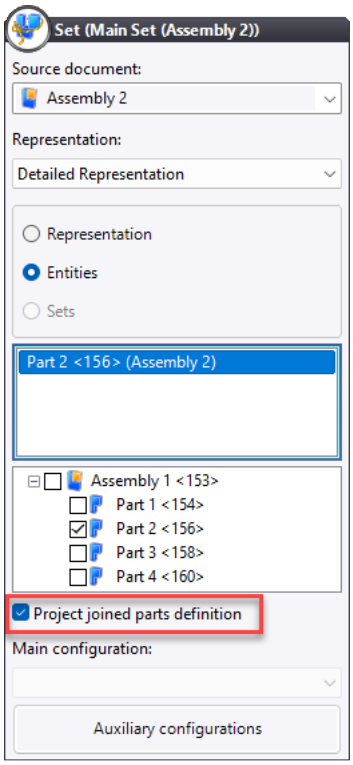
Simplified dialog box

The **Parts** and **Supports** tabs have been lightweighted, and the **Section**, **Material** and **Thickness** columns have been removed as unnecessary.

Drafting

View

The new **Project joined parts definition** option allows you to draft the part as defined in its generic document, but with the dimensions after inclusion in the assembly.



Cross section

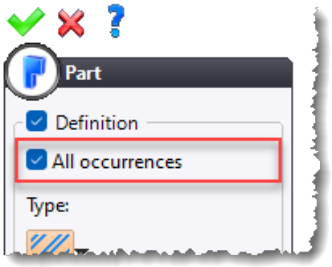
In the dialog box of the **View** > **Cross Section** command, the **Geometry** field is no longer mandatory when the **Depth** option is checked. This allows a cross section to be performed with only one additional depth.

Bill of materials

The **Open Document** command is now available in the context menu on a BOM table or on a BOM index in the Entities tree and in the Operations tree.

Hatching

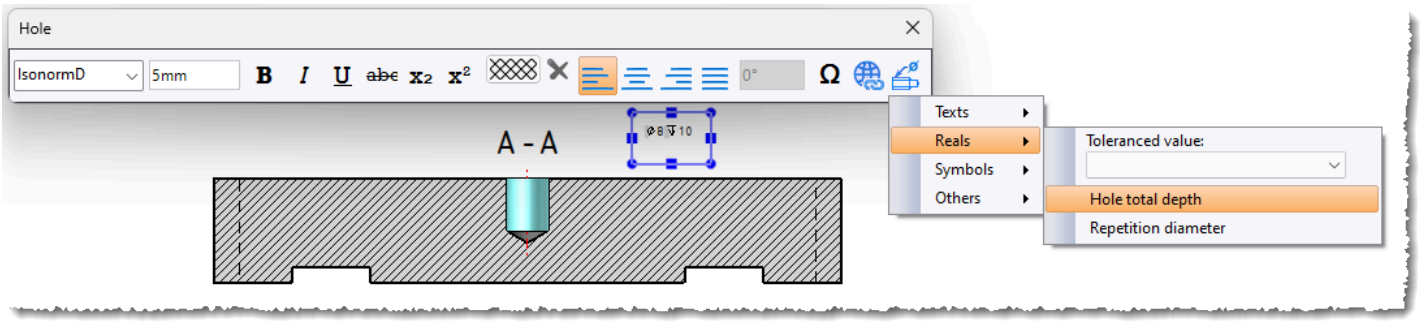
When modifying a hatched area, the new **All occurrences** option allows modifications to be applied to all hatched areas (in all document views) of all occurrences in the same part.



Drilling note

In a drilling note style, it is now possible to retrieve the point depth.

This value can also be defined from the drilling note creation.



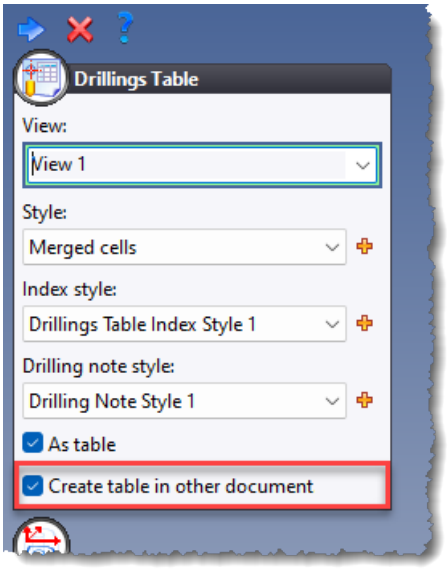
Tables

Table position

The table creation dialog box has been transformed into a wizard.

The **Create table in another document** option positions the table to another document, whether new or existing, with a different paper format. This option is available in coordinates tables, drillings tables, dimensions tables and bends tables.

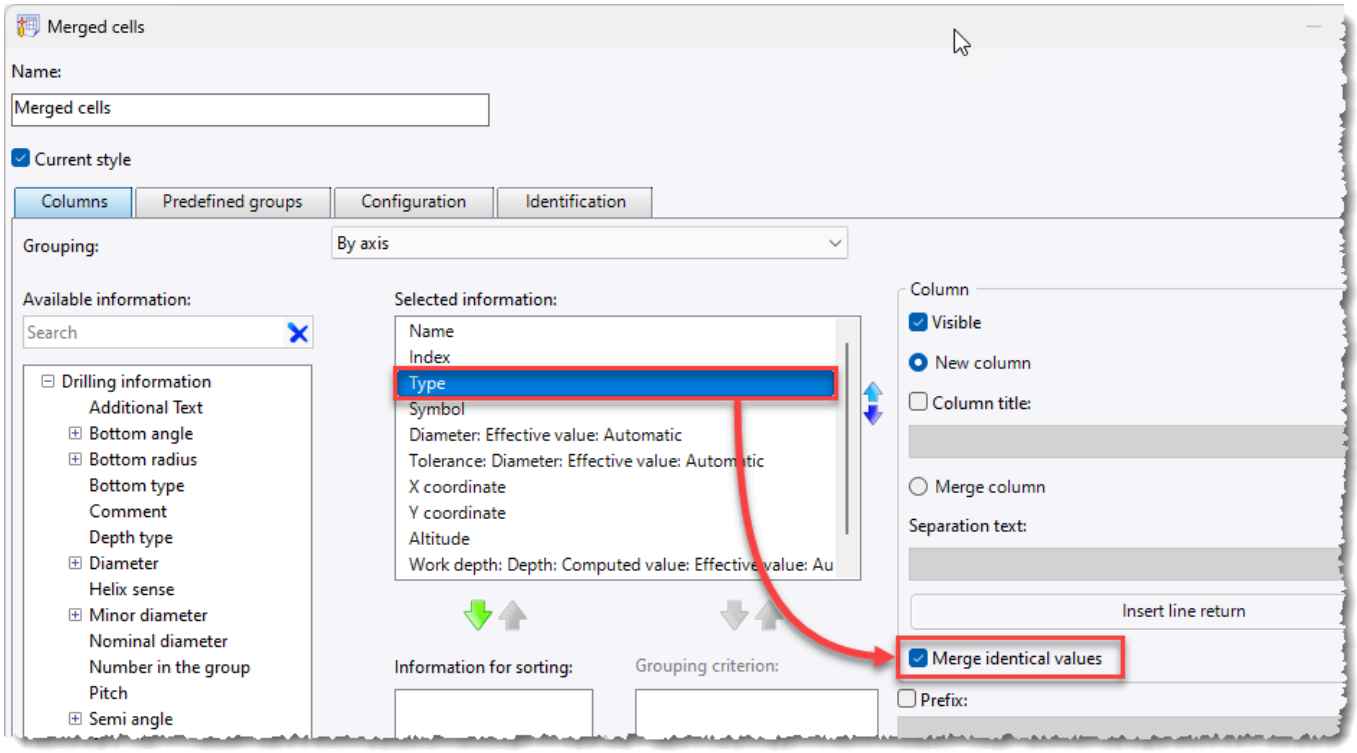
In addition, the new **Detailing** > **Tables** > **Table As** command can be used to create a copy of a table in another document.



Drillings table

Several improvements have been implemented in the drilling table style definition window:

- The window can now be resized.
- Grouping options have been moved to a drop-down list.
- A new information search field is available.
- The new **Merge identical values** option allows you to cluster cells with similar values.



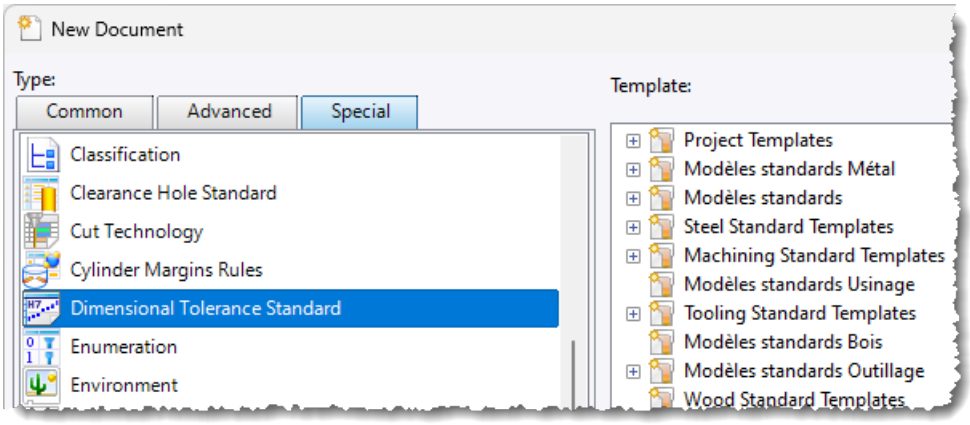
NAME	TYPE	SYMBOL	DIAMETER	COORDINATES	ALTITUDE	WORK DEPTH	NOTES
SPOT FACING 10	SPOT FACING		18MM	216MM X 166MM	0MM	10.5MM	18 10.5
HOLE 9	HOLE		11MM			THROUGH	11
SPOT FACING 8	SPOT FACING		18MM	216MM X 30MM		10.5MM	18 10.5
HOLE 7	HOLE		11MM			THROUGH	11
HOLE 6			8MM	176MM X 126MM			8
SPOT FACING 5	SPOT FACING		18MM	30MM X 166MM		10.5MM	18 10.5
HOLE 4	HOLE		11MM			THROUGH	11
HOLE 3			8MM	70MM X 70MM			8
SPOT FACING 2	SPOT FACING		18MM	30MM X 30MM		10.5MM	18 10.5
HOLE 1	HOLE		11MM			THROUGH	11

In addition, the following modifications have been implemented:

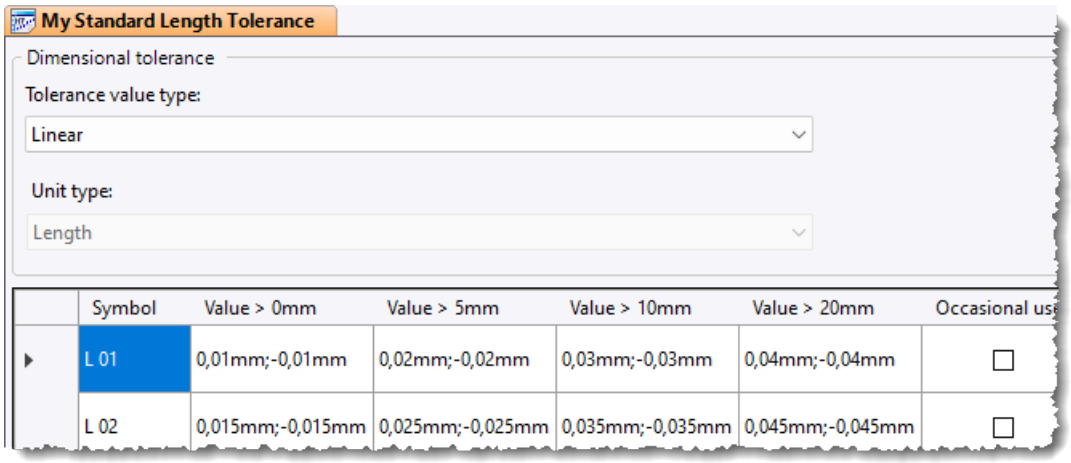
- Entry points defined during pocket creation are automatically retrieved.
- It is now possible to retrieve the nominal, maximum, minimum or average value of a toleranced value.
- For derived parts, the **Automatic** mode retrieves values according to the defined mode (nominal, maximum, minimum or average) in the **Tools** > **Derivations** > **Redimensioning** command.
- The depth can be either theoretical (value input when creating the drilling) or computed (when drilling through a cavity).

Dimensional tolerances

In the **Special** tab of the document creation window, the new document type **Dimensional Tolerance Standard** allows you to create your own linear or angular tolerance standards.

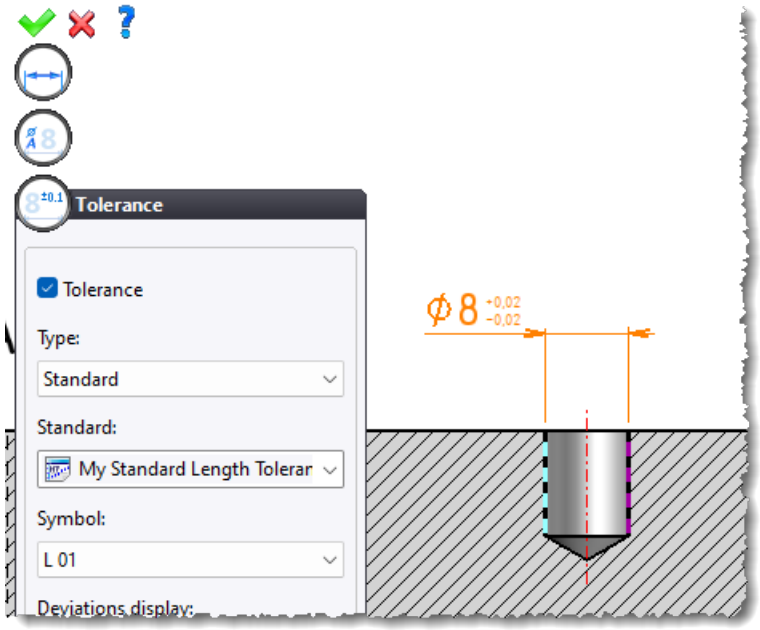


Dimensional tolerance standard *document creation.*



Dimensional tolerance standard *document example.*

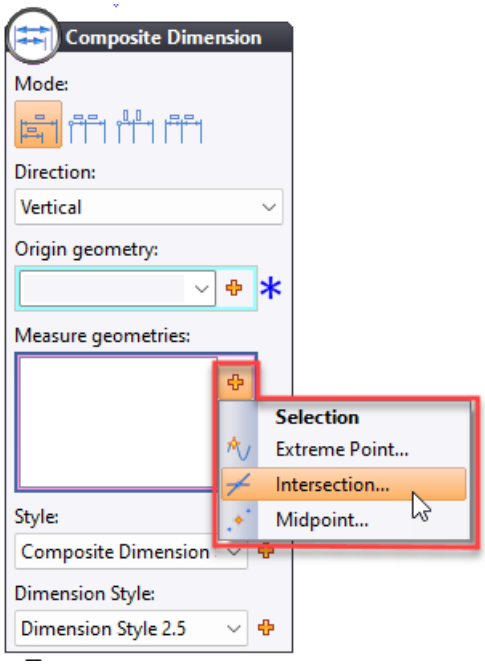
These standards will be visible in the **Tolerances** section of the dimensioning commands.



Use of dimensional tolerance standard in dimensioning.

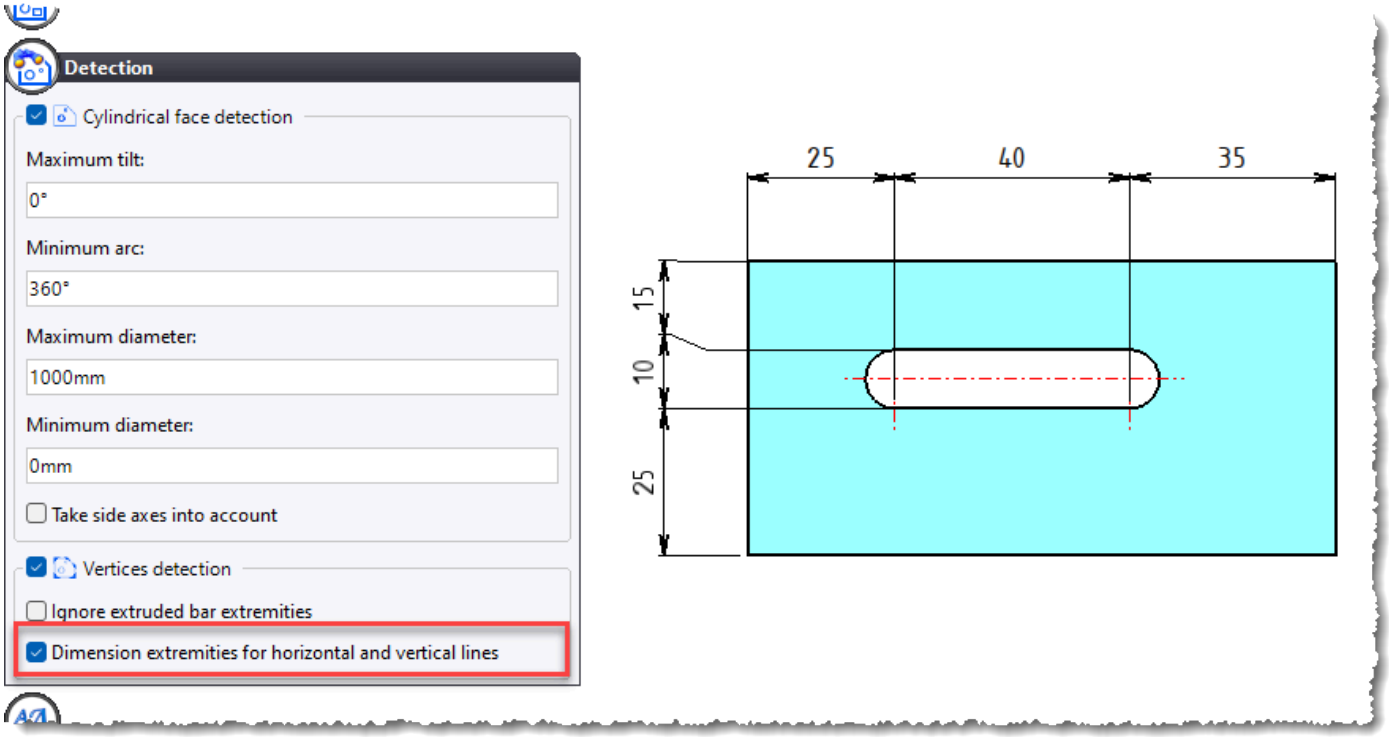
Composite dimensions

The icon for special inputs is now available in the **Composite Dimension** and **Automatic Composite Dimensions** commands.



Automatic composite dimensions

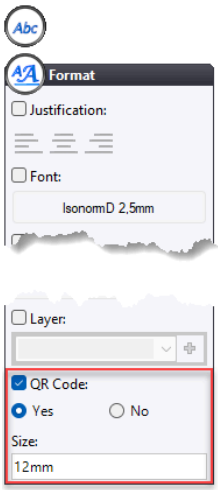
The new **Dimension extremities for horizontal and vertical lines** option is available in the **Detection** section of the **Automatic Composite Dimensions** command. It selects the vertices of all horizontal and vertical lines in one view.

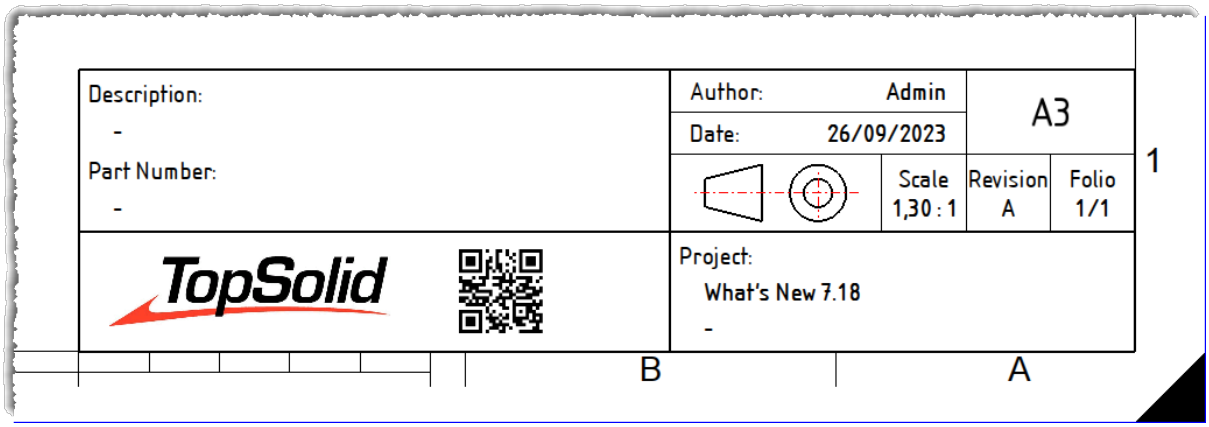


In the above example, the detection of the extremities of the horizontal and vertical lines automatically dimensions the oblong hole.

QR code

The **Detailing** > **Text** and **Sketch** > **Text** commands (**Part** and **Drafting** documents) feature a new option for converting text into a QR code.





Unfolding

Automatic flip over in case of imprint

When manually creating an unfolding or when generating multiple unfoldings using the matching command or an unfolding process, the origin face is automatically selected by **TopSolid**. Previously, this referred to the largest planar face on the part, or otherwise the largest developable face.

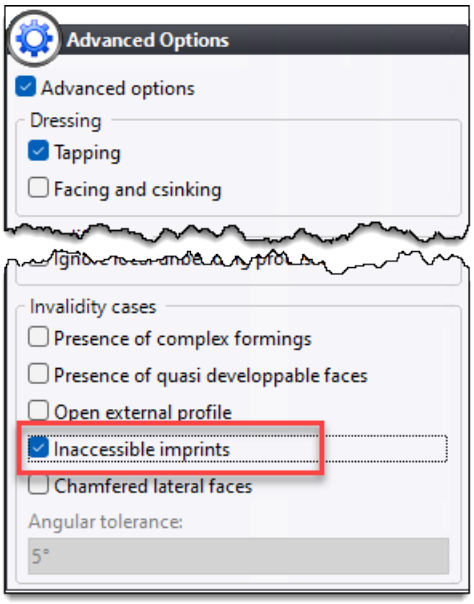
Now, if the part has imprints on only one side of the sheet, this side is given priority for the selection of the origin face. The **Turned over** mode is automatically activated for positioning the unfolding. This guarantees the inclusion of imprints in the unfolding, positioned in the correct orientation for drafting or machining.

Inaccessible imprints

The new **Inaccessible imprints** option is available in the advanced options of the **Unfolding** and **Unfolding Style** commands.

When this option is checked, the unfolding becomes invalid if:

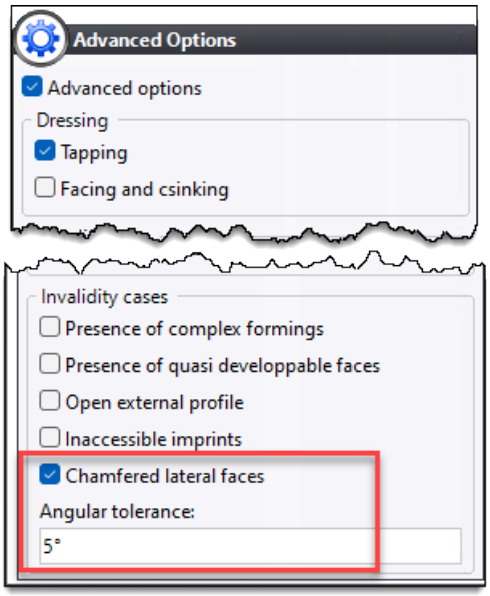
- the sheet has imprints from both sides;
- the sheet has imprints on one side only, but the original face is selected on the opposite side.



Chamfered lateral faces

In the advanced options of the **Unfolding** and **Unfolding Style** commands, the new **Chamfered lateral faces** option is available for invalidity cases.

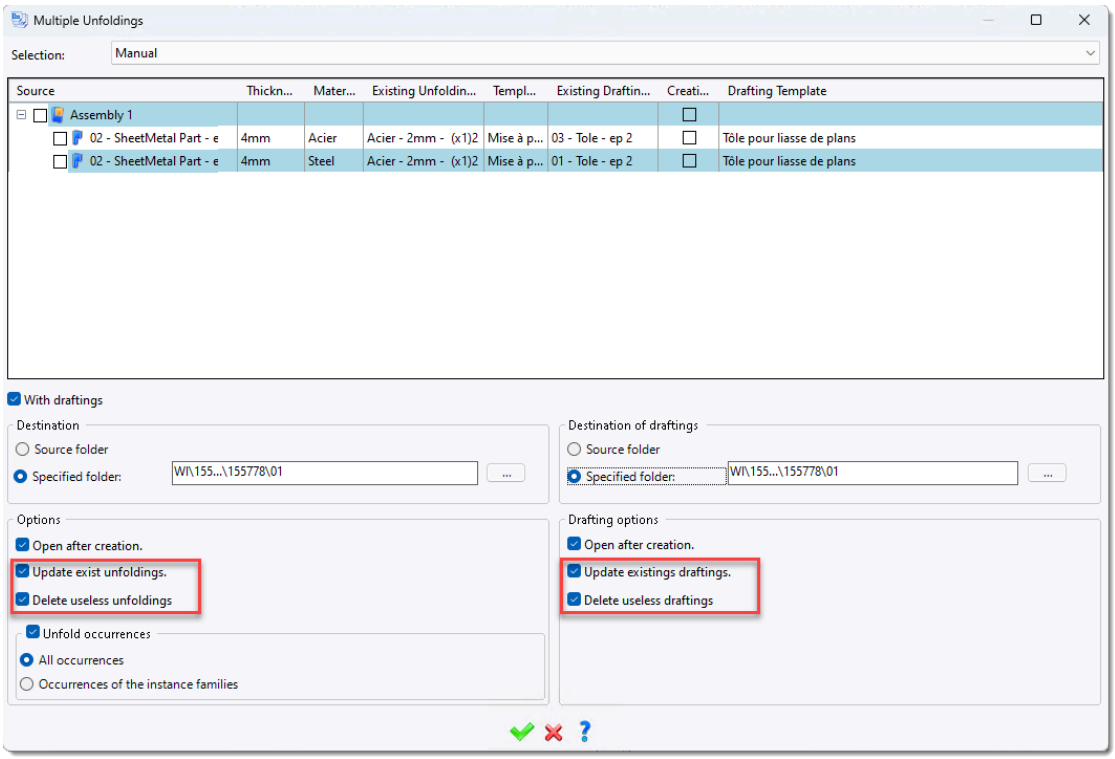
When this option is checked and the sheet has chamfered lateral faces on the original face side, unfolding becomes invalid. This prevents the generation of too short unfoldings because of chamfers.



Multiple unfoldings

The search for existing unfoldings or draftings to be updated now accurately identifies the unfoldings generated on the occurrences. In addition, if there are no existing unfoldings or draftings to be updated, the corresponding options are grayed out.

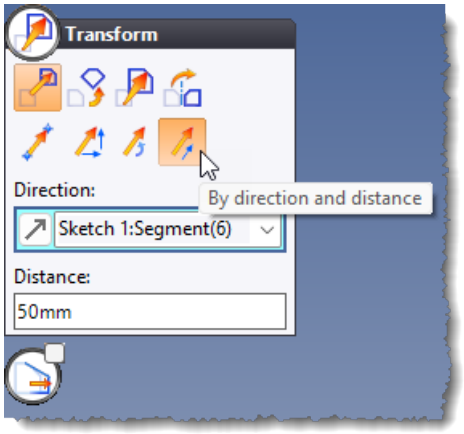
As with multiple draftings, two new options are now provided for deleting unnecessary unfoldings or draftings. The eligible documents for deletion include unfolding or drafting documents located in the specified destination folders that are not part of the documents to be updated. These options are automatically disabled if no candidate document is detected or no folder is specified.



Drawing

Transformation

The **Sketch** > **Transform** command features the new **By direction and distance** mode.



Work document and Work manager

Update

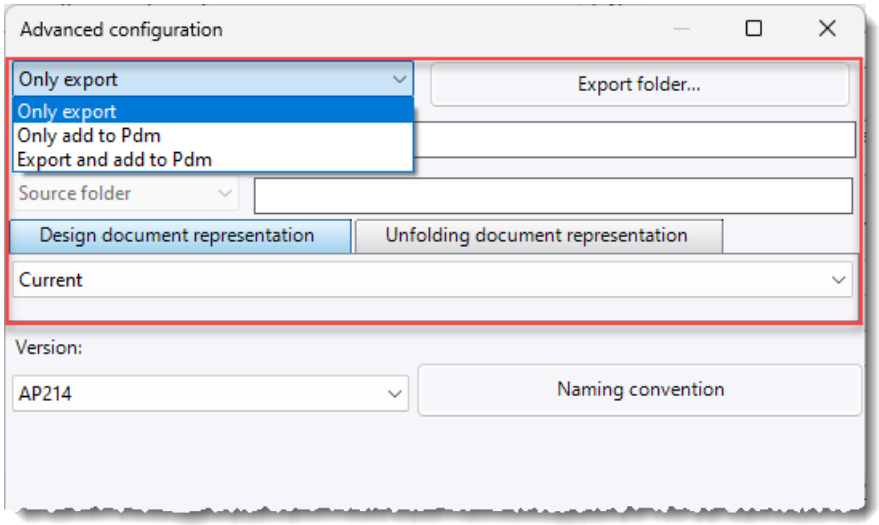
When new parts or assemblies are added, or when repeat quantities are changed, the documents created by a work document can be updated when running the program again.

Moreover, it is now possible to assemble documents of the **Unfolding** type.

Exports

In document export tasks, a drop-down list now allows documents to be exported either to a PDM folder, to an external folder, or to both.

In addition, two tabs allow you to choose which representations to export. These are also available in the **Export Several Documents with Conversion** command window.



2D modeling

Parameters, points and frames

To facilitate the creation of parameterized symbols, the following commands are available in the **Construction** menu from a **2D model** document:

- **Cartesian point**, **Polar point**, **Point on profile**, **Extreme point**, **Intersection point** and **Projected point**
- **Angle parameter**, **Distance parameter**, **Distance in direction parameter** and **Length parameter**
- **Axis by point and direction**, **Middle axis**, **Offset axis**, **Pivoted axis** and **Orthogonal axis**
- **Frame by point and direction**

Interfaces

Spatial translators

An explanatory message is now displayed when the **Sewing faces** is checked.

Family instances export

The **Export Several Documents with Conversion** command is now available in the context menu of a family document. It allows you to export one or more instances of one or more **Family** documents in any standard exchange format (Step, DXF/DWG, etc.).

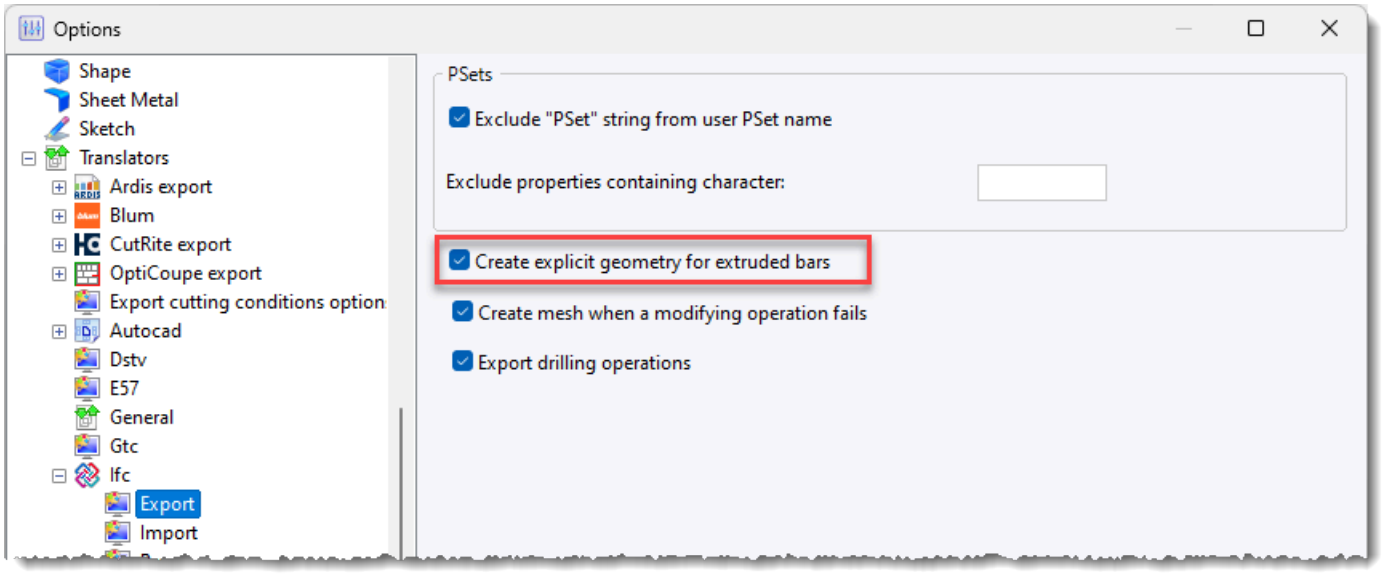
STEP export

Properties of text type (text parameter, user property, enumeration parameter and material parameter) are now exported.

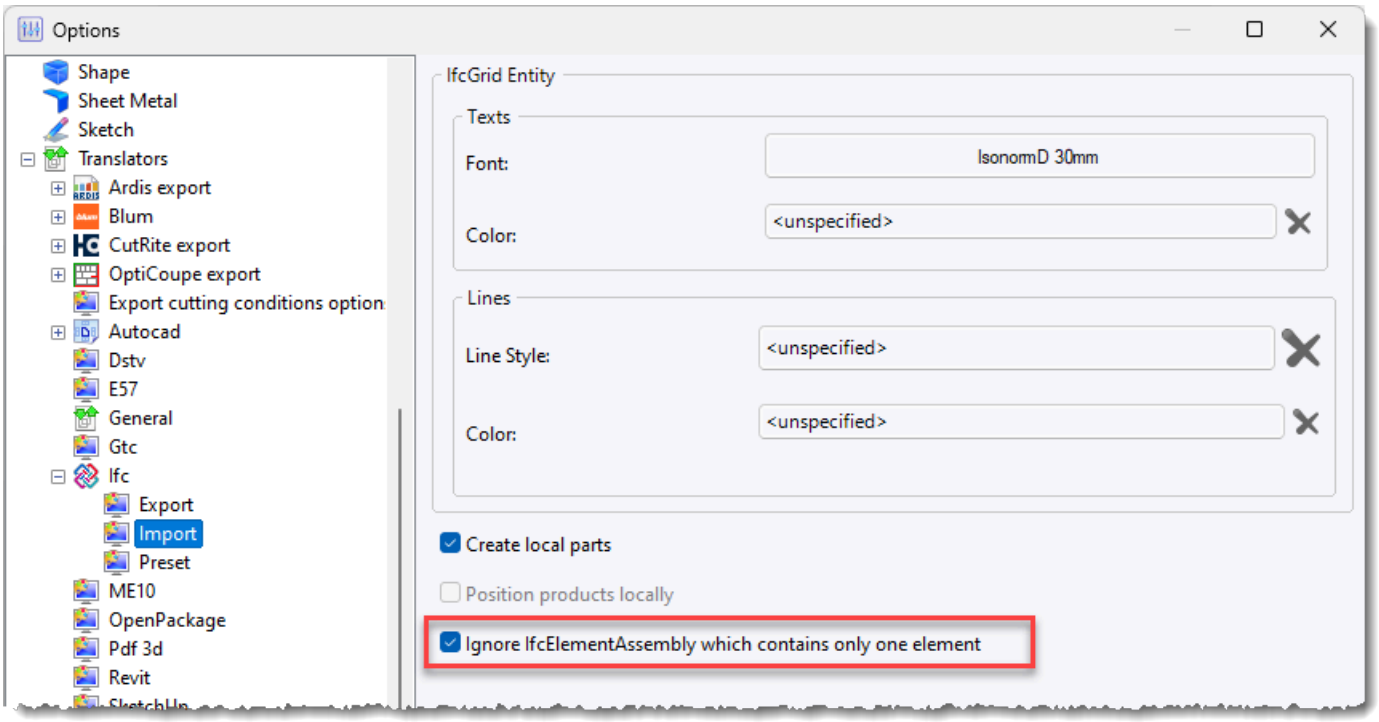
From a bill of materials, the **Import / Export** > **Export Several Documents with Conversion** command now exports each part in Step format.

IFC export

An extruded bar that does not provide an FEA function will be processed like an extruded element, provided that the **Create explicit geometry for extruded bars** option is checked.



The new **Ignore IfcElementAssembly which contains only one element** allows assemblies containing only one part to be imported as a part.



PDF3D export

Layers are now supported when exporting a drafting document, so that they can be shown or hidden in the PDF viewer.

DSTV export

It is now possible to export an extruded bar's type and subtype as well as the trims on an imported profile.

Importing ASC and e57

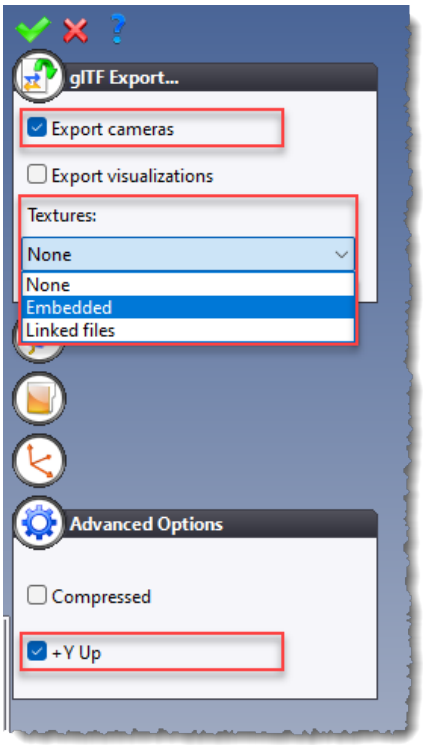
Files in **ASC** and **e57** formats can now be imported as point clouds. Note that a single .e57 file can contain several point clouds. Importing with conversion imports all point clouds, whereas the **Construction** > **Point Cloud** command imports only the first point cloud.

FBX export

All properties (color, roughness, metalness, etc.) of a PBR material are now exported.

glTF export

Camera export, texture export mode selection and vertical direction are now available from the command dialog box.



TopSolid'Virtual

When the **TopSolid'Virtual** application is installed, and if it has been declared in the **Tools** > **Options** > **Translators** > **TopSolid'Virtual** section, the new **File** > **Translators** > **Export to TopSolid'Virtual** command transfers the topology and textures of parts (or assemblies) to **TopSolid'Virtual** and launches the application.

Translator

Column

It is now possible to add the **Document type** column to the translation table.

What's New in TopSolid'Cam 7.18

This chapter describes the new features of the **CAM** applications in version **7.18** of **TopSolid 7**.

Hole machining

Drilling: New tools for checking collision points

Checking collision points now supports the following six types of drilling tools:

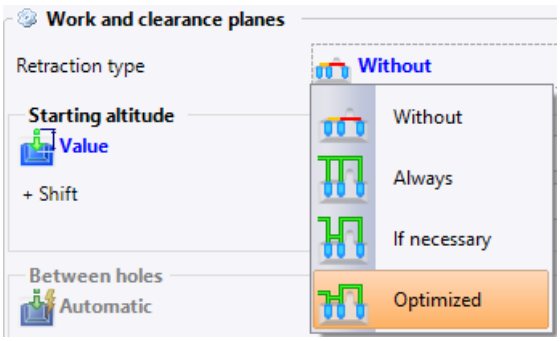
- **Twist drill**
- **Flat drill**
- **Gun drill**
- **Spotting drill**
- **Center drill**
- **Countersink drill**

When a collision points check is performed on a multi-function tool, a warning message is displayed in the Events window to indicate that only the current tool function is being checked.

Drilling: Collision management with environment

The environment management operation is now independent of the drilling retraction management operation.

The old **Always rise up** and **Check collisions** options have been replaced by the **Retraction type** choice list which offers the following four retraction types: **Without**, **Always**, **If necessary** and **Optimized**.

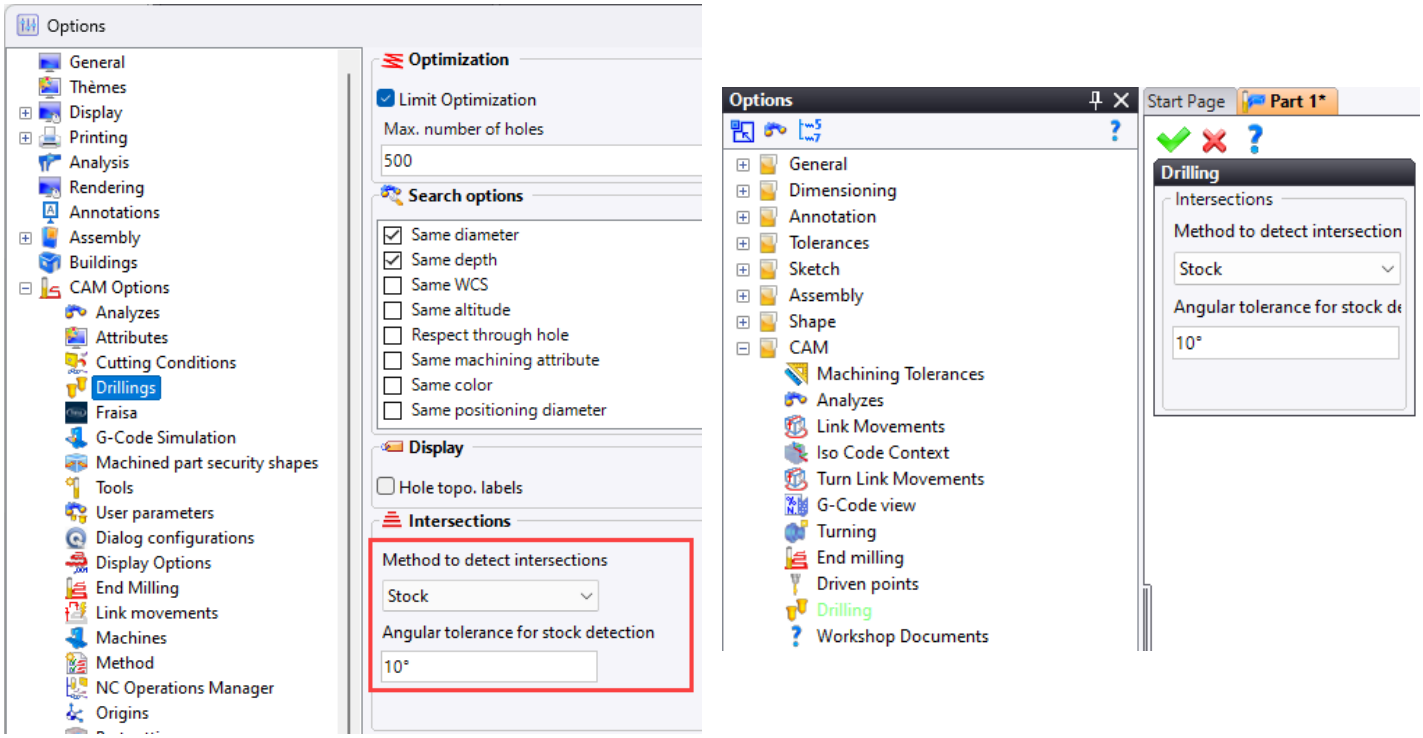


Drilling: New intersection detection mode

A new intersection detection mode is available under **CAM Options** > **Drillings** > **Intersections** in the general options of **TopSolid**, as well as in the machining document options.

The **Operations** mode, which was the default in previous versions, only takes into account previous drilling operations in the document and computes their intersection(s) with the current drilling operation.

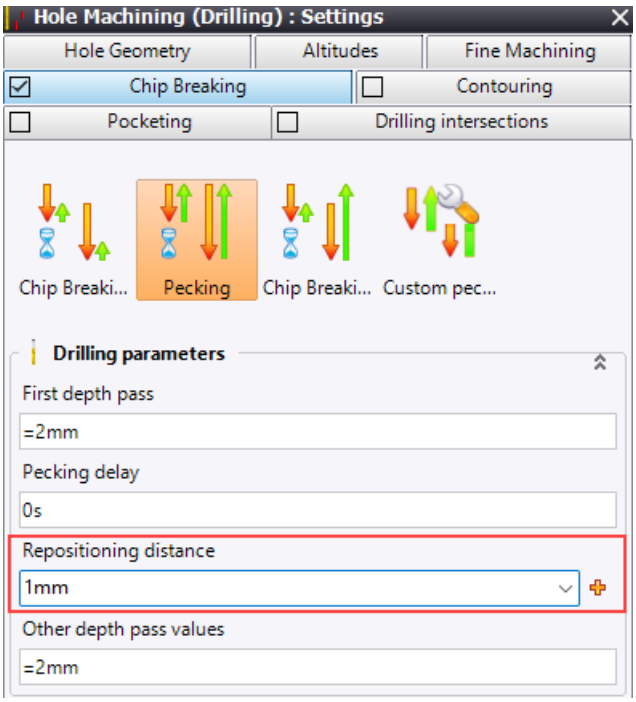
In the new **Stock** mode, the stock is probed to detect an approximation of intersections from ongoing drilling with material free pockets.



Drilling: Repositioning distance for drillings with pecking

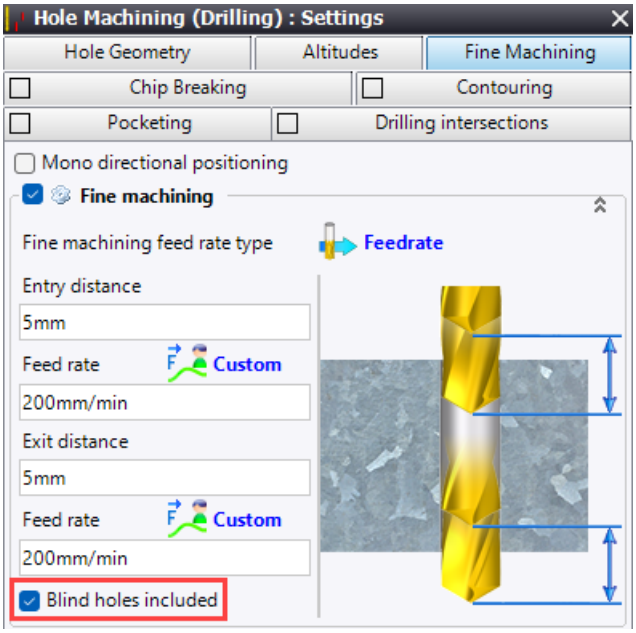
When drilling with pecking, once the tool is out of the hole, it goes back down at rapid feed rate at a certain distance above the position where it stopped in the previous step. Until now, this was safety distance.

It is now possible to enter a **repositioning distance** in the **Pecking** and **Chip Breaking/Pecking** modes.



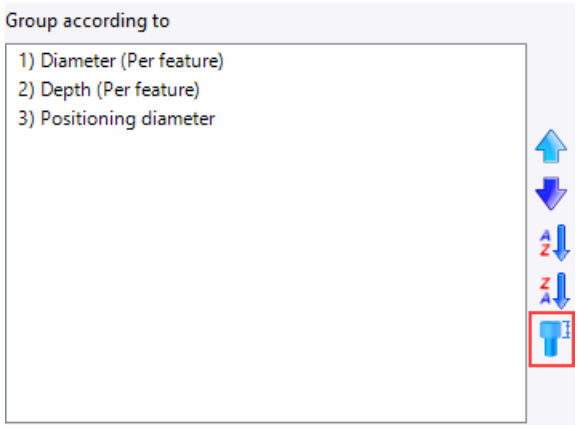
Precise drilling: Exit feed rate for blind holes

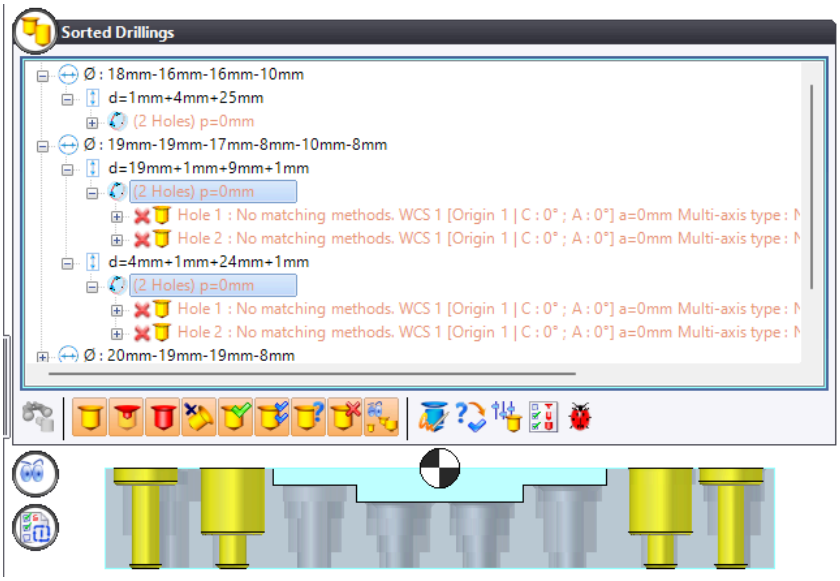
For precise machining, you can apply the exit feed rate change for blind holes at the end of drilling.



Cylinder and drilling analysis: Sorting drillings per feature

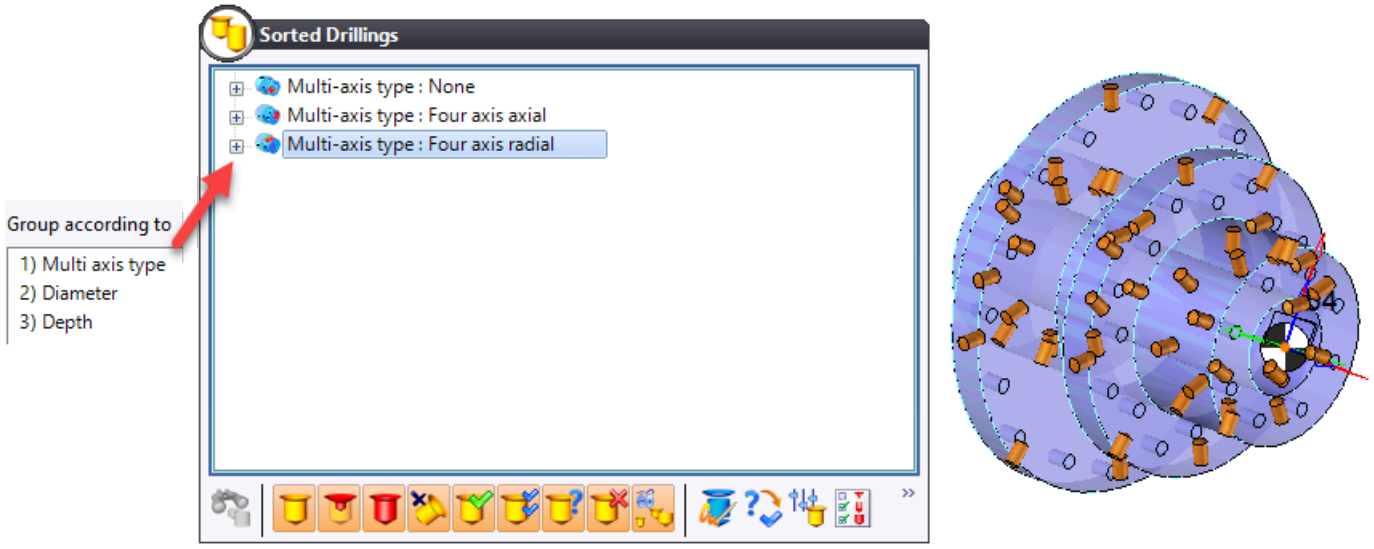
In the analysis of cylinders and drillings, the new **Per feature** option allows you to sort drillings according to the diameters and depths of the different features of these drillings.





Cylinder and drilling analysis: Hole analysis in 4X axial and 4X radial modes

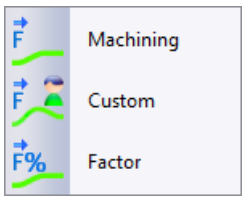
The new **Multi-axis type** sort mode allows axial/radial holes to be grouped together to offer a unique WCS for each group.



2D milling

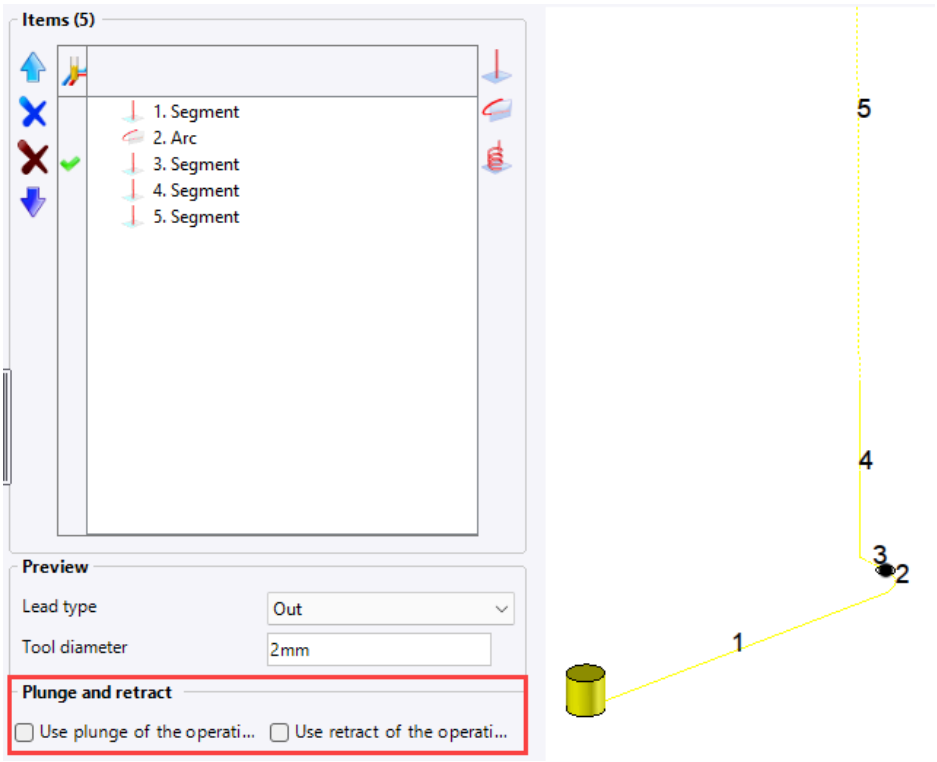
2D milling: Feed rates in angle management

In angle management for side milling, chamfer milling, edge breaking and Tslot milling operations, it is now possible to select the feed rate for broken edge segments.



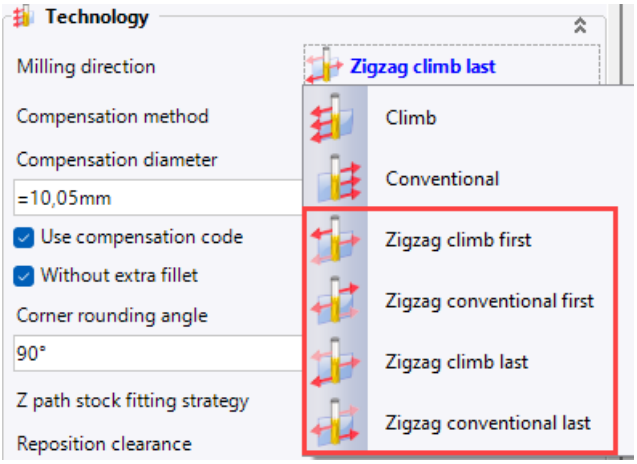
2D milling: Using plunge and retract of the operation

In the **Customized lead in/lead out** document, the new **Use plunge of the operation** and **Use retract of the operation** options allow you to use, when necessary, plunge and retract as defined in the operation, following those defined in the lead in/out document.



2D milling: New zigzag milling directions

Four new zigzag milling directions are available for side milling, chamfer milling and Tslot milling operations.

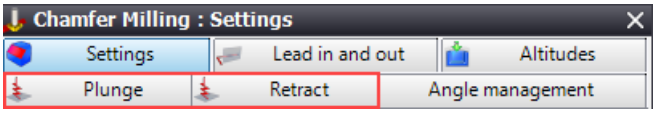


2D milling: Forcing the last axial depth

You can force a last spring pass even if there is no path.

2D milling: Management of plunge and retract in chamfer milling operations

Similarly to side milling, the **Plunge** and **Retract** tabs are now available in chamfer milling and radius milling operations to manage the plunge, in and out of material, and retract.

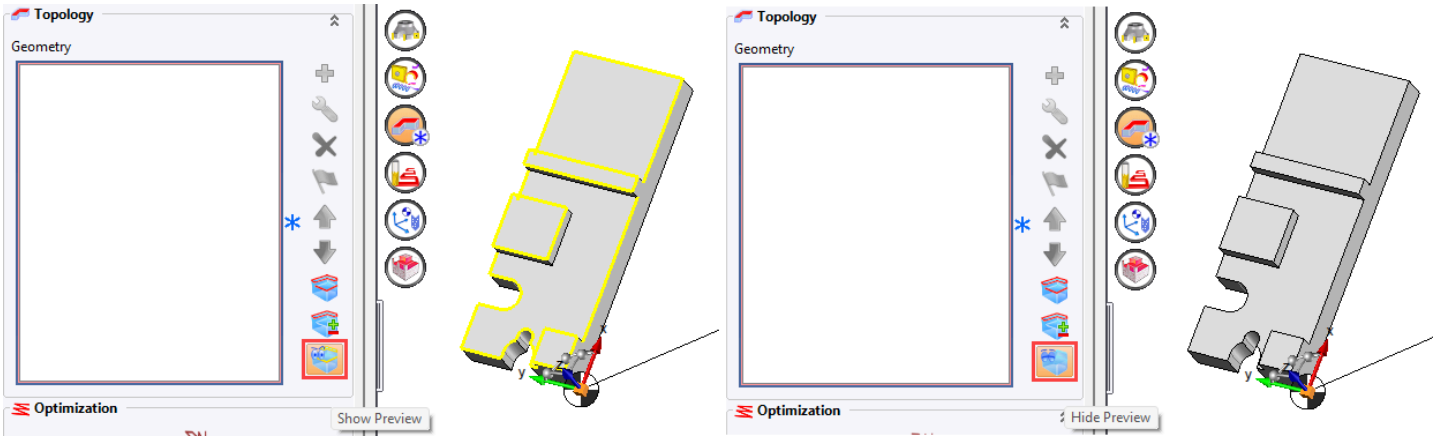


2D milling: Chamfer milling in Z axis and on slope

A chamfer milling operation can now be performed either along the chamfer slope, or in the Z axis, like a classic contouring operation.

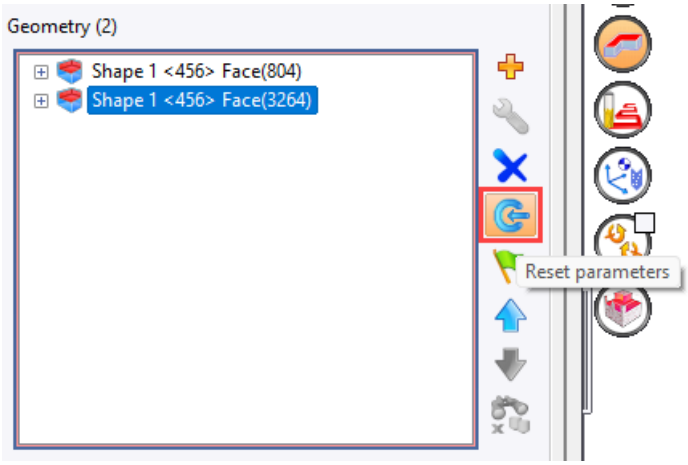
2D milling: Preview in automatic search for breaking edges

In the **Geometry** section of the breaking edges operation, a new icon allows you show or hide the preview of generated edge sequences.



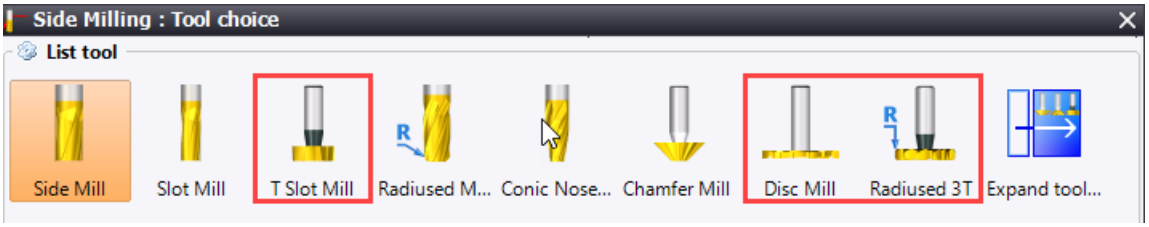
Multi-geometry: Reset global operation values

The new **Reset parameters** icon is available in the **Geometry** section and resets the values of a geometry, i.e. the global values of the operation.



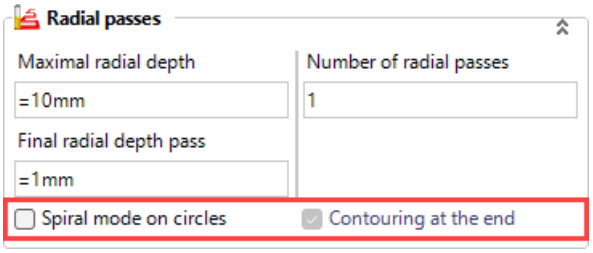
Side milling: New tool types supported

The **disc mill**, the **radiused staggered teeth mill** and the **T slot mill** are now supported in the side milling operation.



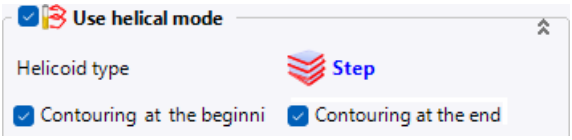
Side milling: Flat spiral contouring

In the side milling operation, the new **Spiral mode on circles** and **Contouring at the end** options are available in the **Radial passes** section of the **Settings** tab. They enable flat spiral contouring based on circle geometry, with the option of final contouring at the end of the spiral, while respecting normal contouring operation.



TSlot milling: New helical mode

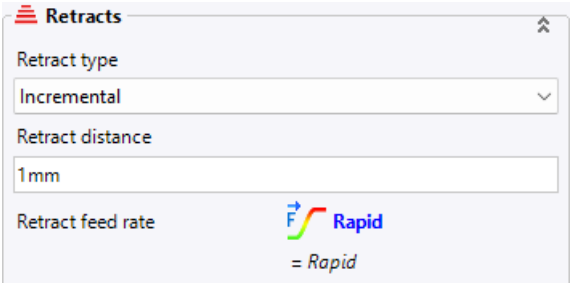
As with the side milling operation, the **Helical mode** option and its associated parameters have been added to the **Settings** tab of the TSlot milling operation.



Broaching: Retract management

Three types of retracts are now available in the broaching operation: **Incremental**, **None** and **Value**.

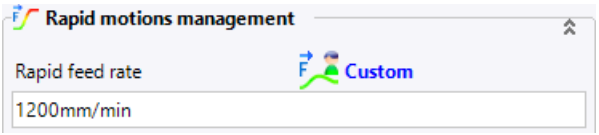
You can also enter a retract distance and select the retract feed rate.



3D milling

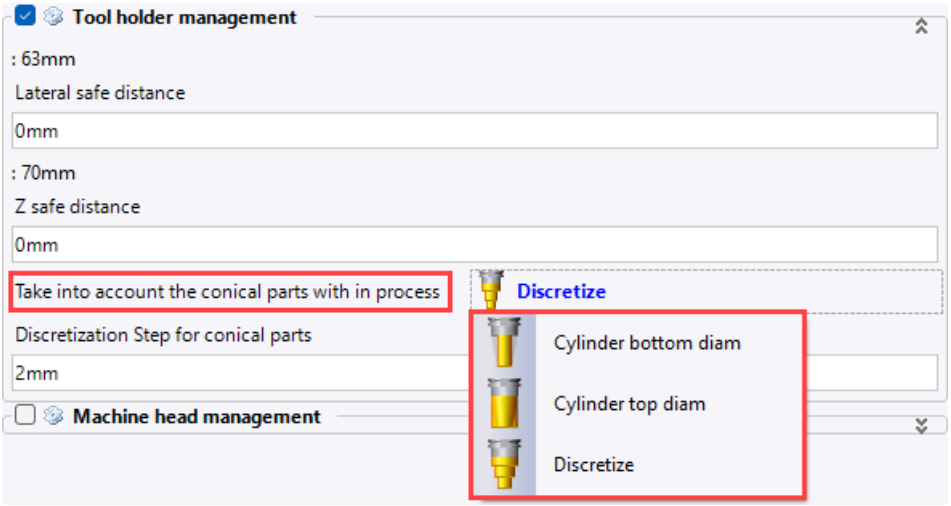
3D milling: Rapid link feed rate

In finishing, superfinishing and material left operations, the new **Rapid motions management** section allows you to adjust the rapid link feed rate. Rapid movements (G0) are transformed into feed rate movements (G1) once the toolpath is calculated.



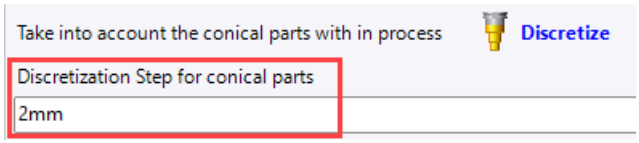
Roughing: Computing modes for the conical part on current stock

In the 3D roughing operation settings, the old **Take into account the conical parts with in process stock** checkbox now offers three computing modes.



Roughing: Discretization of conical parts when using tool shape

The **Discretization step for conical parts** option can also be set in the **Tool definition** tab of the 3D roughing operation.

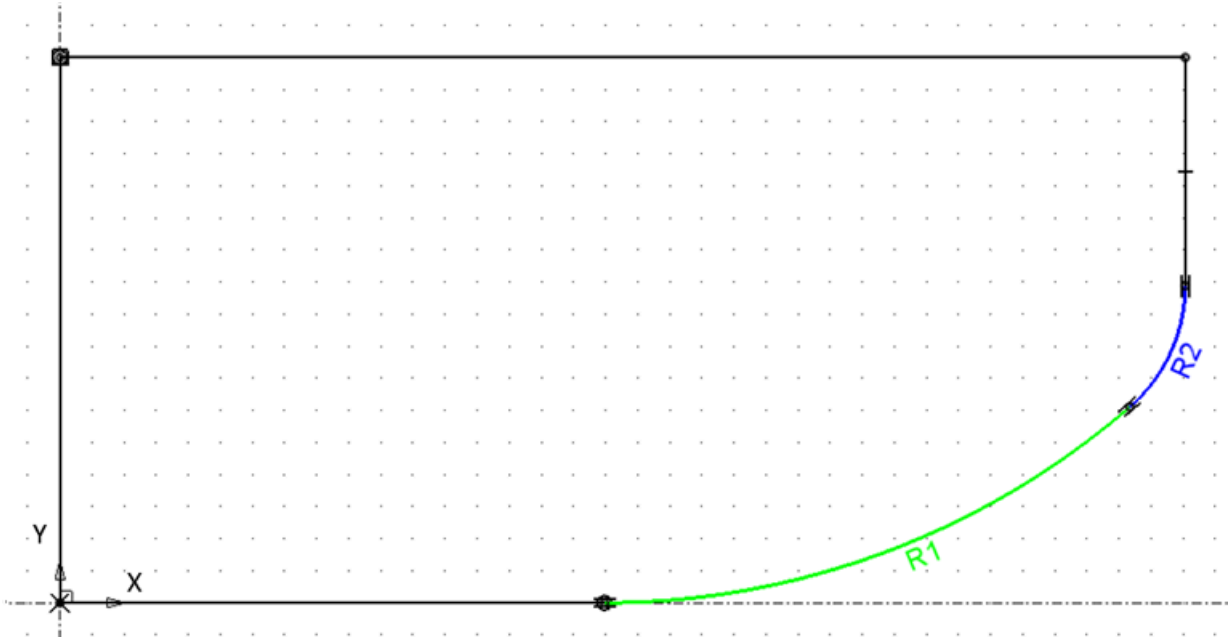


Roughing: Modification of feed rate on approach/retract movements

The new **Approach feed rate** and **Up feed rate** parameters allow you to modify the feed rate on approach and retract movements.

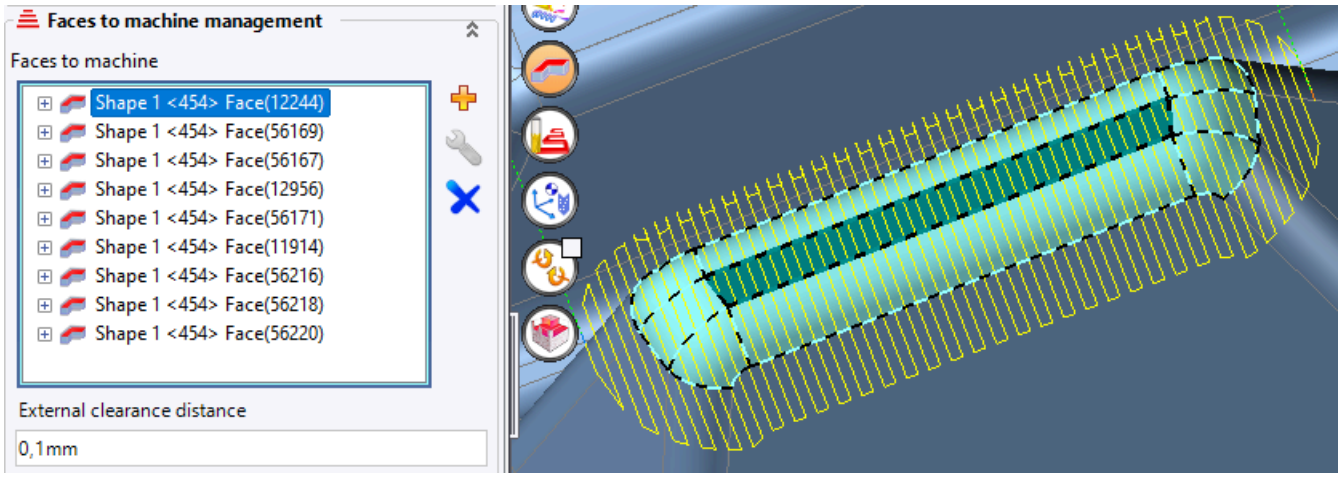
Roughing: Management of high speed milling tools with two tangent radii

For classic roughing, you can now use a high speed mill with a two radii profile.



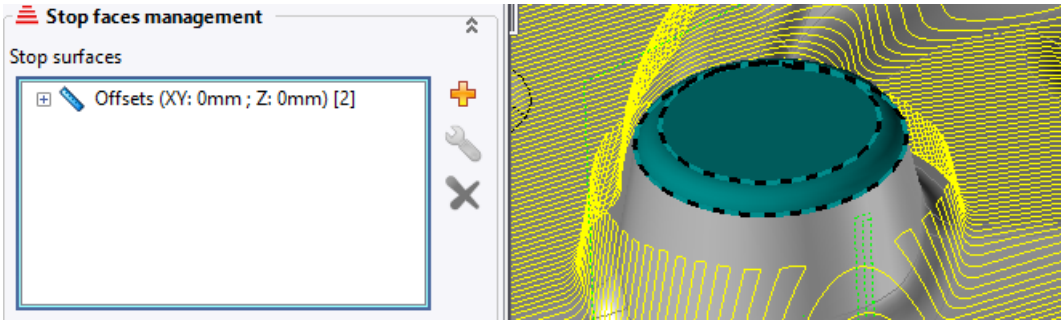
Finishing: Machining only the selected faces on the part

In the **Constant Z**, **Raster passes** and **Constant step-over** finishing operations, you can select faces to be machined in the **Geometry** section and, if required, apply a clearance to extend the machining operation beyond the selected faces.



Finishing: Management of stop surfaces

Stop surfaces can be selected in the **Geometry** section of **Constant Z**, **Raster passes** and **Constant step-over** finishing operations.

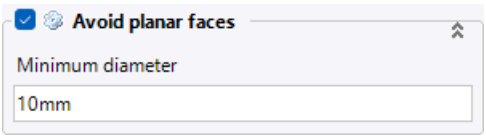


Constant step-over finishing: Contact mode on curves with a radiused mill

The **Contact** bounding mode is now supported in the bounding curves management of a **Constant step-over** finishing operation with a **radiused mill**. The machining area is the same as for a corresponding raster passes or constant Z finishing operation.

Superfinishing: Minimum diameter to avoid planar faces

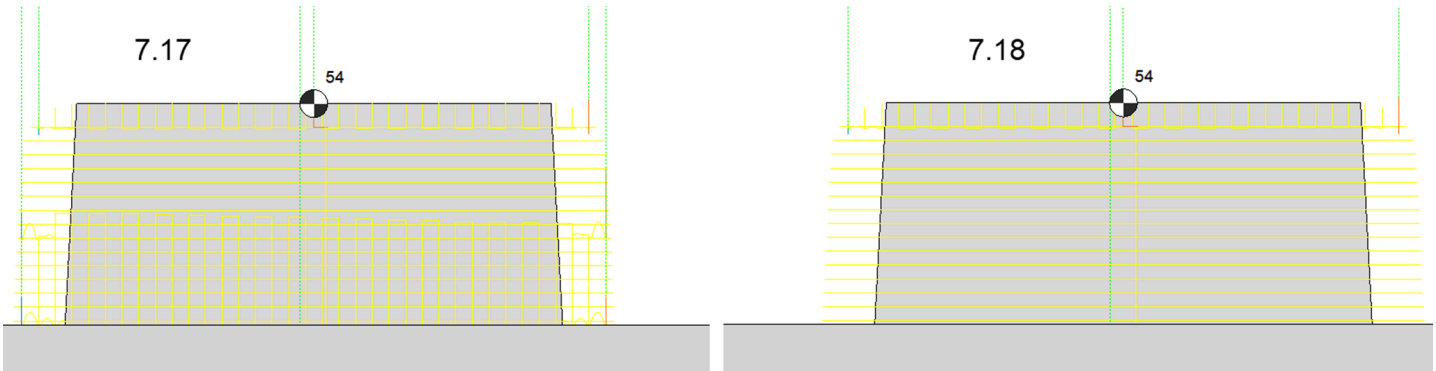
In a superfinishing operation, the new **Minimum diameter** parameter is available when the **Avoid planar faces** option is checked. If the contour of the planar face fits within a circle of minimal diameter, it will not be avoided.



Superfinishing: Improved overlap and Z limitation

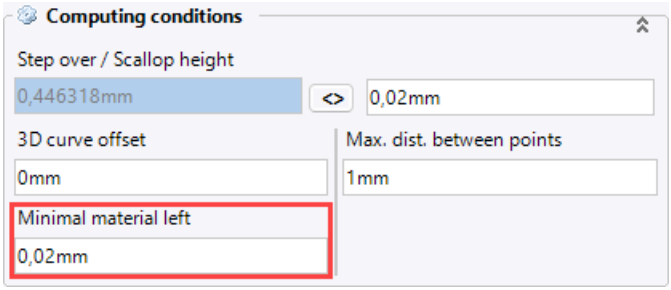
Previously, in a superfinishing operation with a 3D curve offset value greater than 0 and minimum Z-path limitation, there were sometimes unwanted bits of path coming from the horizontal part of the path, going up onto the vertical walls of the part.

The algorithm has been improved so that these paths no longer appear.



Residual machining: Minimum material left

In the **residual machining** operation, the new **Minimum material left** parameter controls the precision of the residual material zones, ensuring to not remachine too small zones.



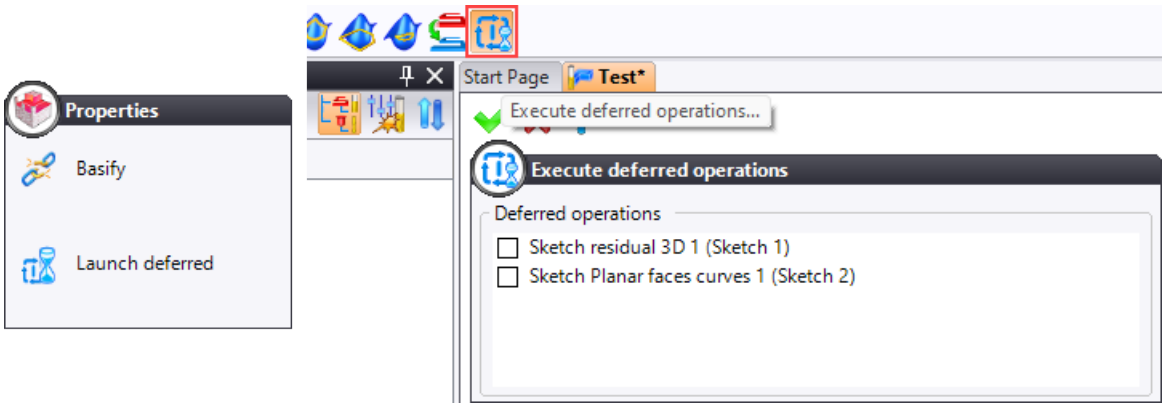
3D contouring: Collision management

The **Collisions** tab has been added to the 3D contouring operation settings. It allows you to avoid collisions between the tool holder and the machine head.

3D sketch: Computing sketches in deferred mode

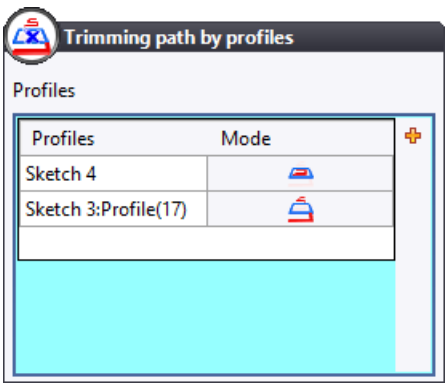
When creating or editing a 3D sketch, the new **Launch Deferred** mode means that the sketch is not computed immediately after the operation has been validated.

In addition, the new **Execute deferred operations** command on the **Additional** tab allows you to select and then execute sketches pending calculation.



3D path trimming: Profile multiple selection

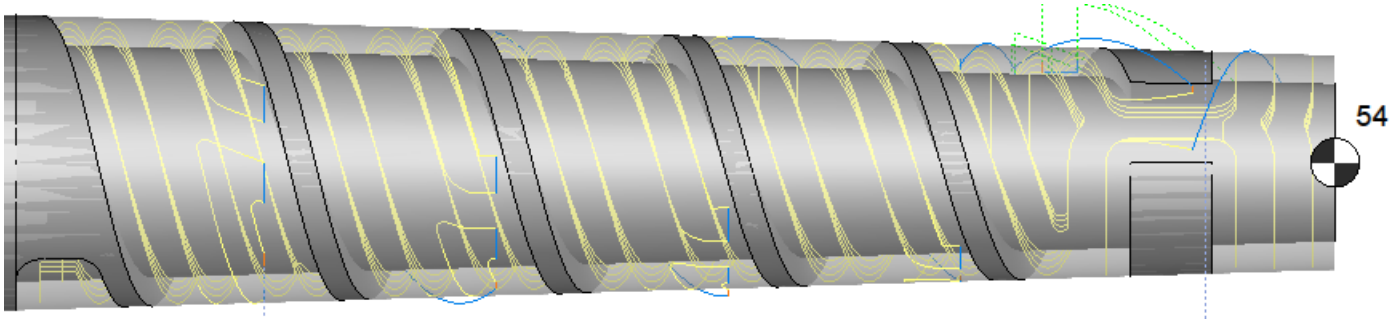
When trimming a 3D path, you can select multiple profiles and choose which side to delete from each profile.



4D milling

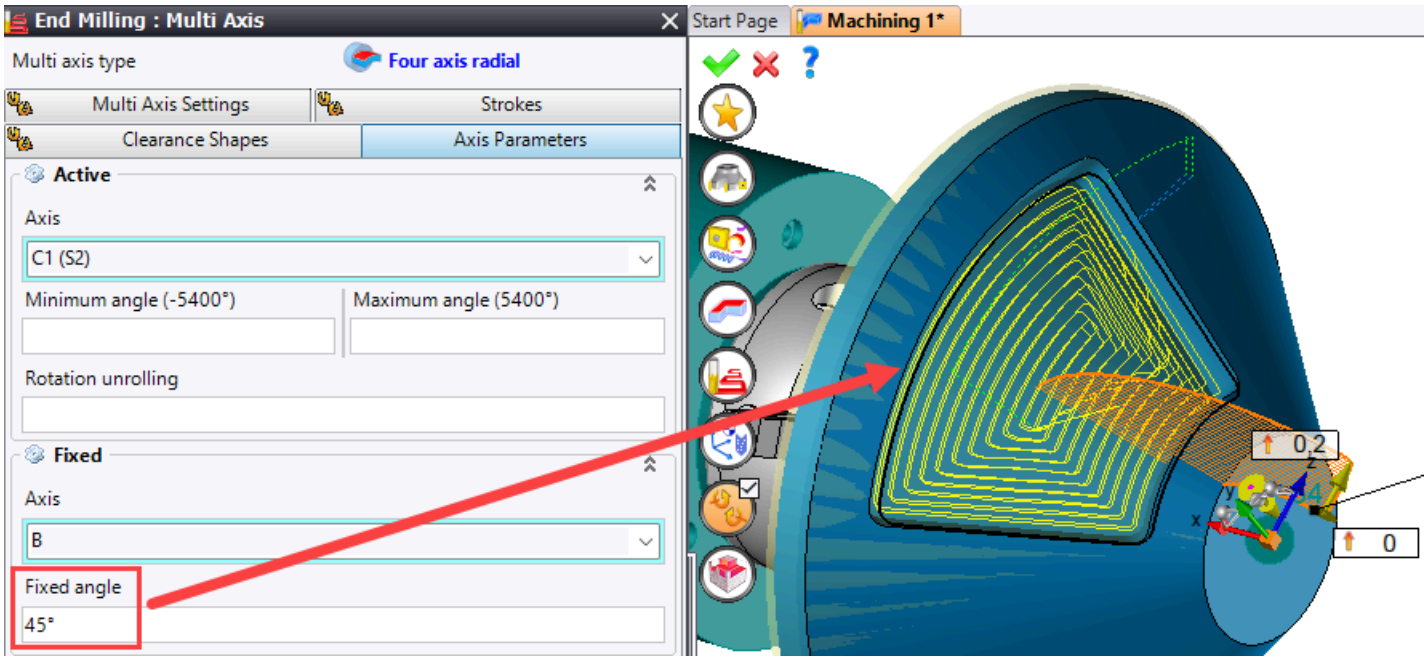
4D milling: New advanced 4-axis radial operation

The new **4X Radial Roughing Advanced** command enables roughing based on a rotating axis.



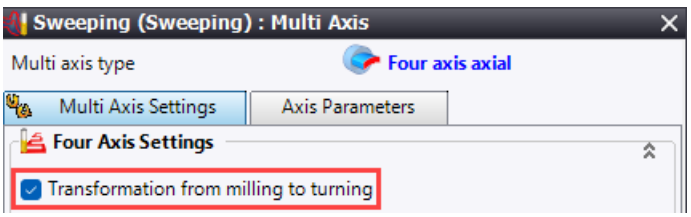
4D milling: Machining on a cone

It is now possible to perform pocket machining on a cone in 4-axis radial mode for end milling and side milling operations.



4-axis axial sweeping: TransMiT mode

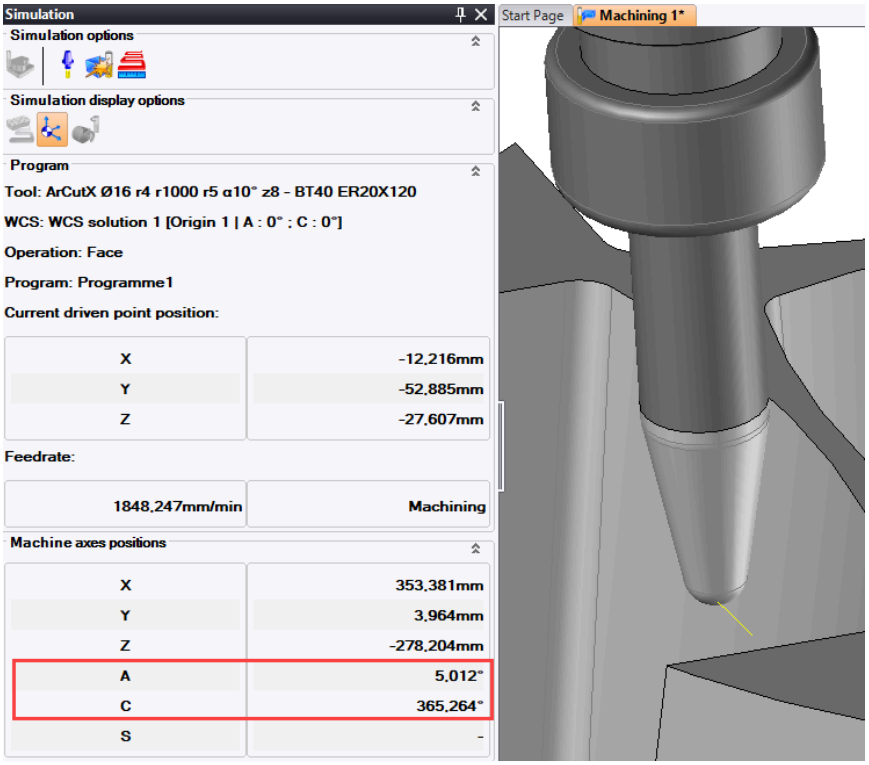
You can activate the **TransMiT** (transformation from milling to turning) mode in a sweeping operation in 4-axis axial mode.



5D milling

Auto 5-axis machining

The new **Auto 5-axis machining** operation facilitates automatic machining of faces with ball nose mill, radiused mill and barrel mill (taper-spherical, oval, etc.) tools .

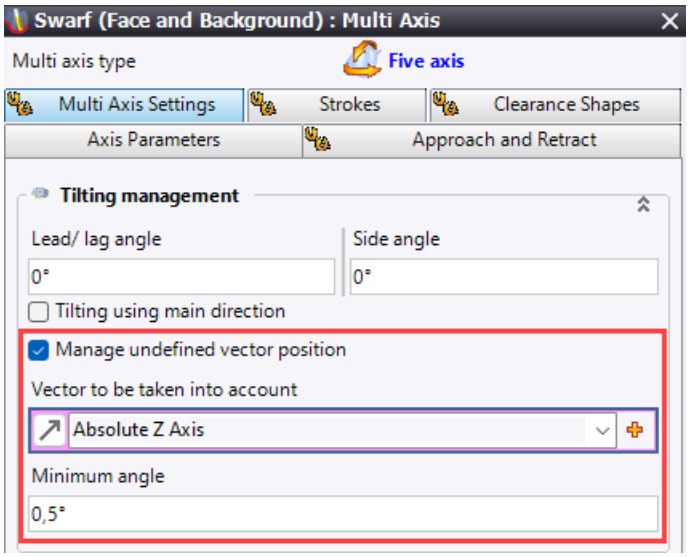


Improved management of undefined normal vector

The **Manage undefined vector position** option has been added to the **Settings** tab of the multi-axis settings of the **Swarf** and **5D Contouring** commands.

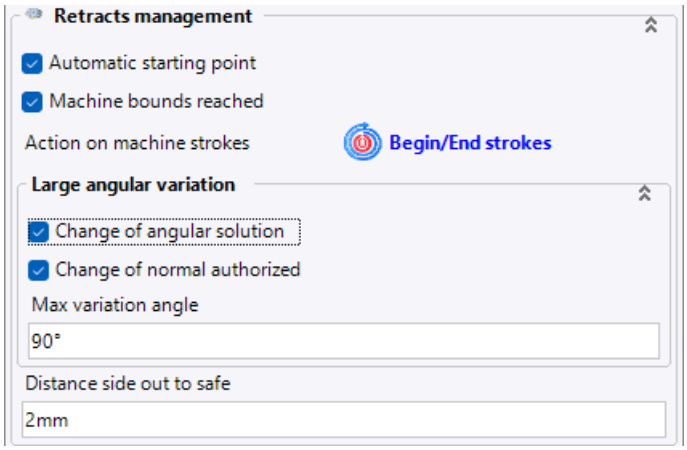
The new **Choose rotation axis** allows you to choose whether solving is executed on only one or on two rotation axes.

It is also possible to specify how to define an undefined normal vector.



Security area retraction in case of strong angular variations

The **Necessary retracts** was renamed **Retracts management** in settings. It allows you to proceed to retraction in the security area as soon as a large angular variation between two consecutive positions (point + normal) is detected.



Asynchronous execution for operations using ModuleWorks

The **ModuleWorks** commands available in **TopSolid'Cam** and listed below can be executed in **asynchronous mode** (available in the command properties):

- **Roughing 3+2**
- **Multi-axis pocketing**
- **4X radial roughing advanced**
- **Multi-blade machining**
- **5X deburring**
- **Port machining**

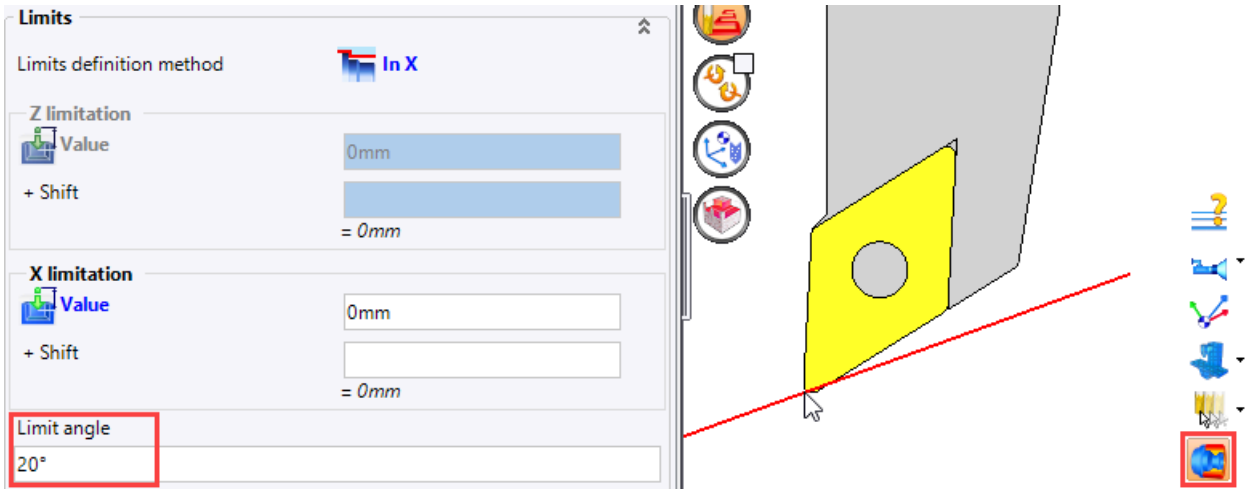
Turning

Turning: Geometry start and end points on the fly

In the turning **Finishing** and **Threading** commands, you can create a point on the fly via the  icon to define the start and end points of a geometry.

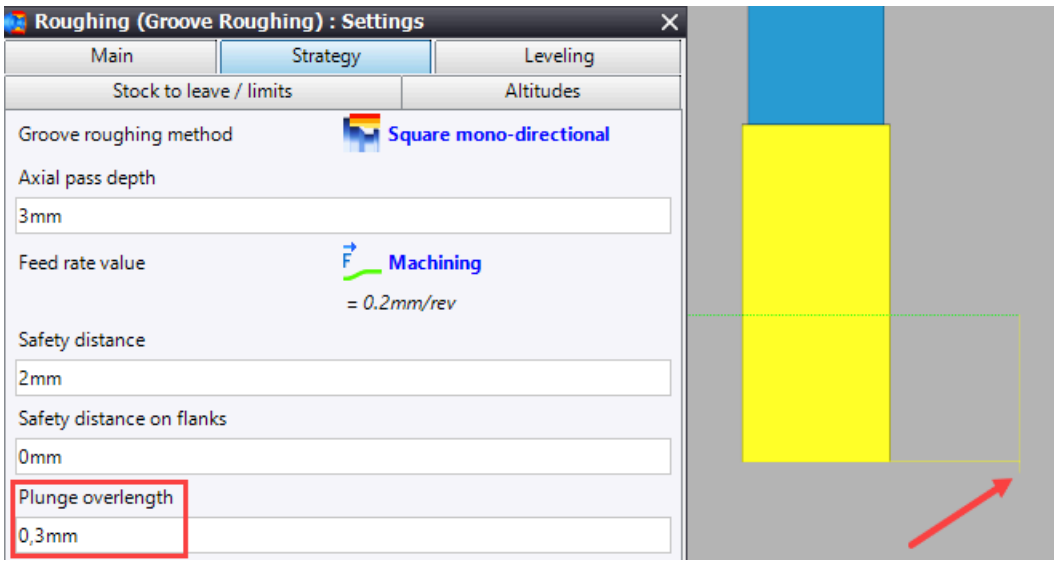
Roughing: Limit angle display on tool

When a machining limit is defined in X or Z, the angle of inclination of the limit is displayed on the tool. A new icon is available in the graphics area display bar to show or hide the limit angle preview.



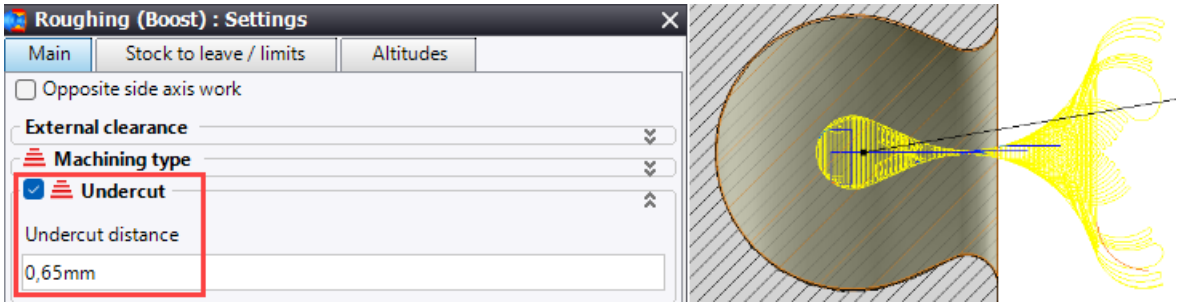
Groove roughing: Plunge overlength

In the groove roughing operation, the new **Plunge overlength** parameter is only available for the **Square mono-directional**, **Square bi-directional** and **Alternate** roughing types.



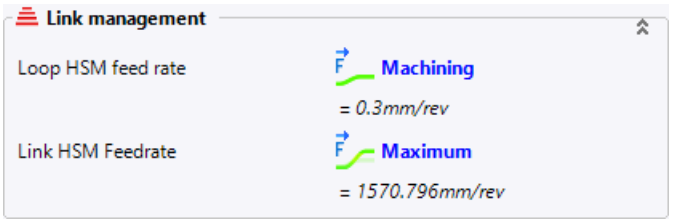
Boost roughing: Undercut management

The turn roughing operation in **Boost** mode now manages undercuts. To do this, check the new **Undercut** option, then enter the draft distance accepted by the tool.



Boost roughing: Loop HSM feed rate and link HSM feed rate

The new **Loop HSM feed rate** and **Link HSM feed rate** parameters are used to define loop HSM feed rate and link HSM feed rate.

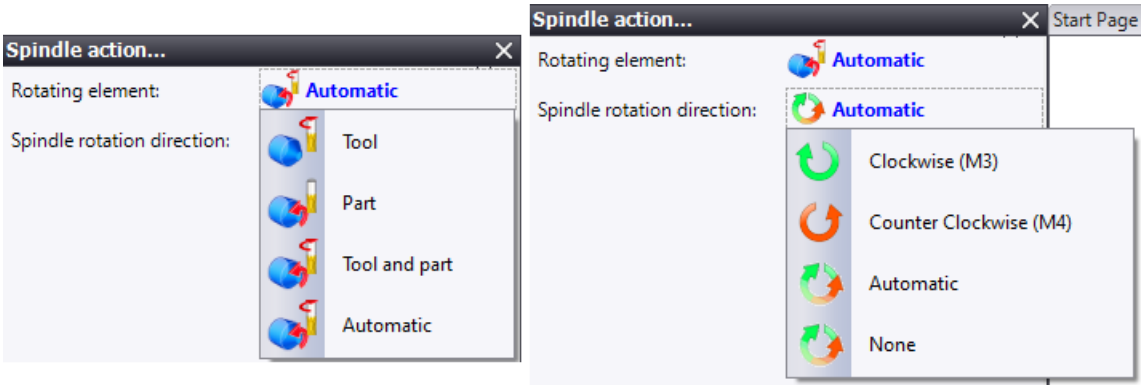


Virtual jog and link movements

Virtual jog: Spindle rotation management

The **Spindle action** function has been added to the virtual jog. It allows you to select the rotating element and the spindle rotation direction.

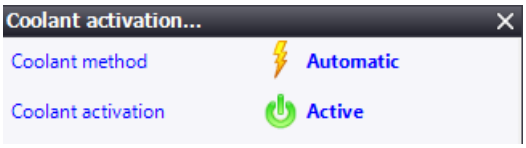
The **Automatic** mode manages the rotation of the spindle selected in the current operation.



Virtual jog: Coolant management

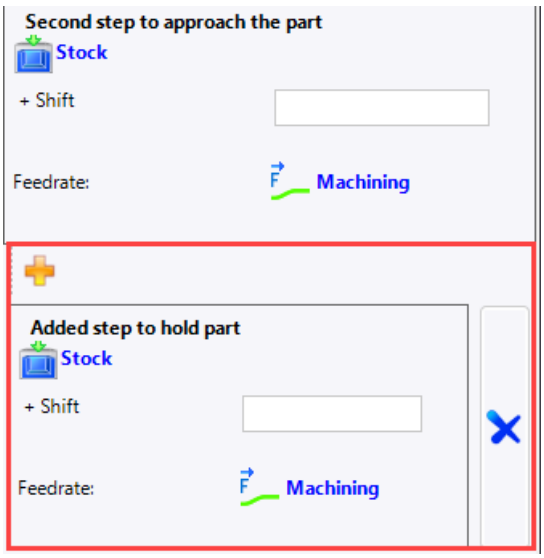
The **Coolant activation** function has been added to the virtual jog. If a coolant document is selected in the machine, it is taken into account.

The **Automatic** mode manages the coolant selected in the current operation, while the **Manual** mode selects all coolants in the coolant document.



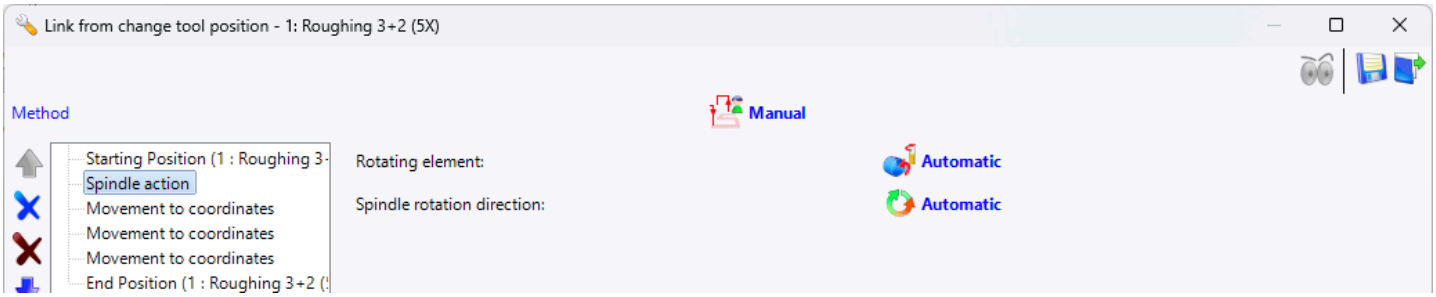
Virtual jog: Additional steps to approach the part

Additional steps can be added to the **Approach to hold part** function.



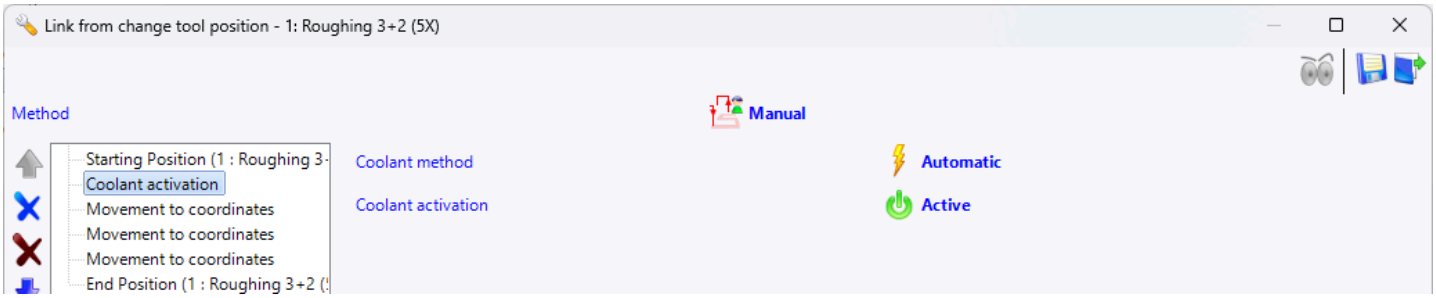
Link movements: Spindle rotation management

Similarly to the virtual jog, the new **Spindle action** function manages/triggers spindle rotation in approach and retract movements.



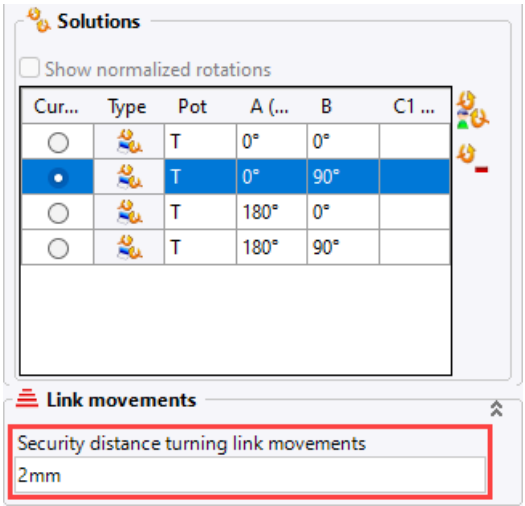
Link movements: Coolant management

Similarly to the virtual jog, the new **Coolant activation** function manages/triggers coolants in approach and retract movements.



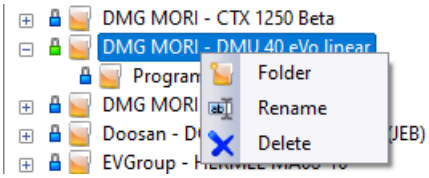
Link movements: Security distance specific to internal turn roughing

The new **Security distance turning link movements** parameter allows you to modify the security distance for the link movement only. It is available on the **ISO** tab of the operation's comments.



Link movements: Subfolders creation

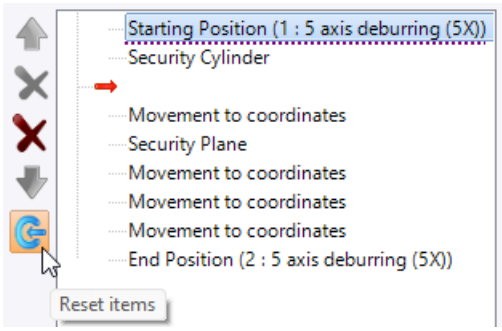
Subfolders can be created and managed when saving or editing link movements.



Link movements: Optimized changeover from manual mode to automatic mode

Modifications made when editing a link movement in manual mode are kept after switching to automatic mode and back to manual mode, as long as the editing window remains open.

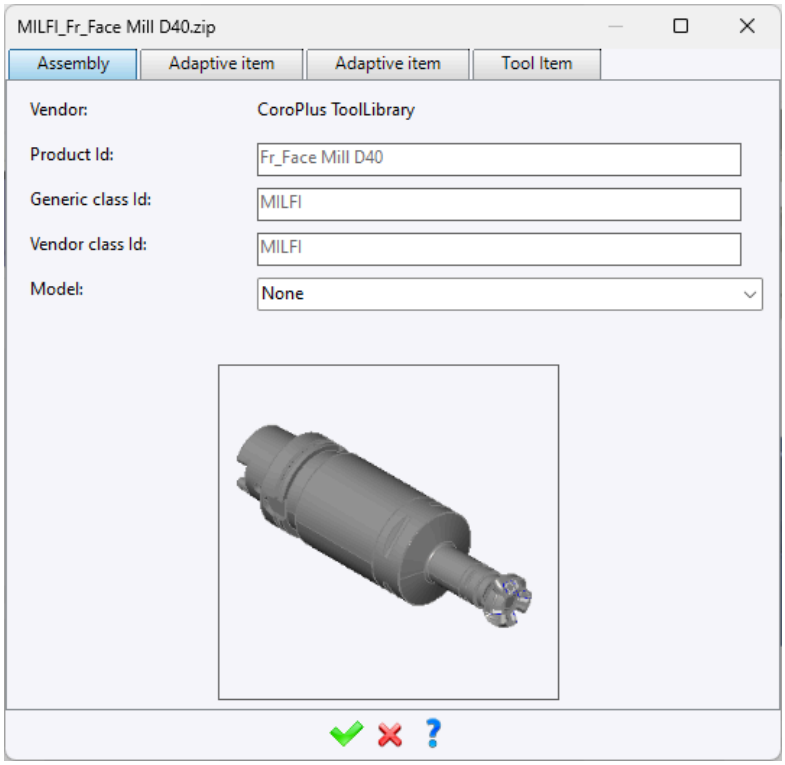
The new **Reset items** option resets the elements in manual mode.



Tools and cutting conditions

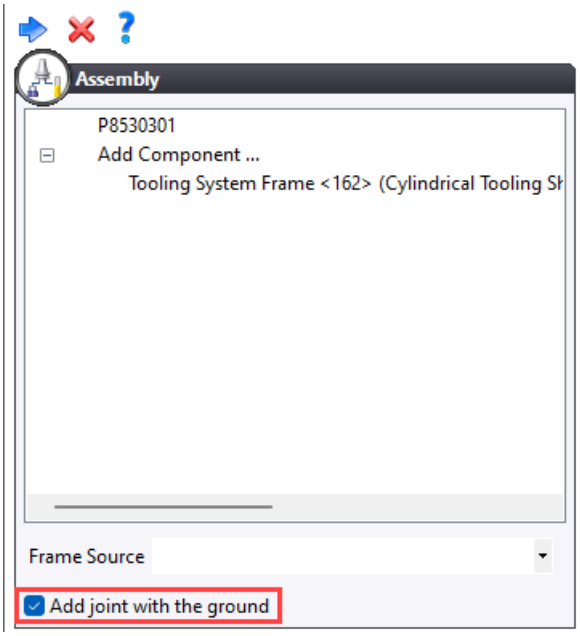
Tools: Importing GTC tools

Tools exported in **13399 (GTC) format** can be imported, either by dragging and dropping the ZIP file, or by using the **Import File with Conversion** command and selecting the **GTC Package (*.zip)** file type. If the ZIP file is usable, the dialog box below appears.



Tools: Adding a joint from the assembly wizard

When using the assembly wizard, a new option allows you to add a joint with the ground, in order to correctly compute the minimum output length of the tool when assembled with a machine component.



Tools: User driven point and tool output distance

You can modify the tool output distance after creating a user driven point. The driven point position will remain consistent.

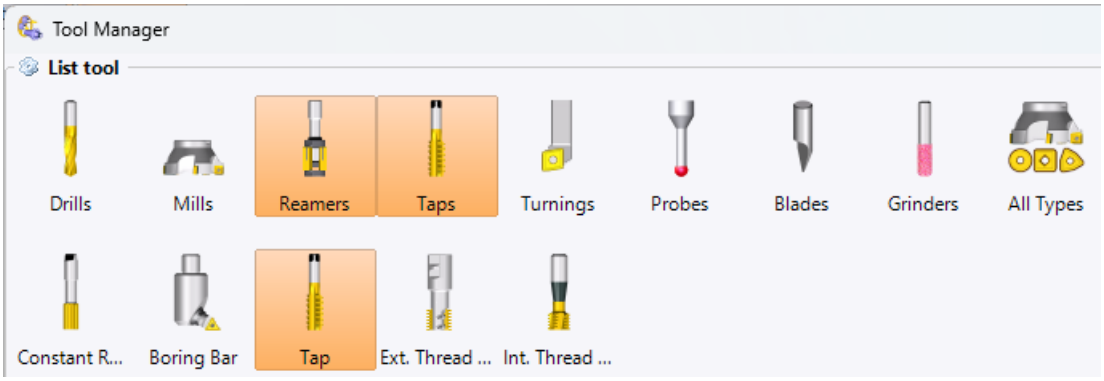
Tools: PP words and notes in workshop documents

PP words and notes added in tools can now be retrieved in workshop documents thanks to two new variables:

- **ToolPPComment** for PP words;
- **ToolWordPPNotes** for notes.

Tool manager: Sorting tool types

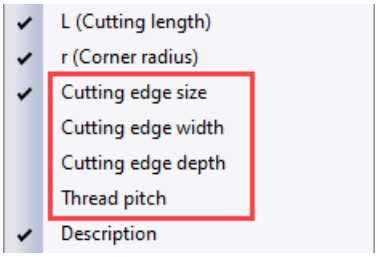
The tool manager now displays a first line listing the tool families, and a second line corresponding to the tool types in the selected family(ies).



Magazine: New variables of tool functions

New variables of tool functions are available:

- **Cutting edge size**
- **Cutting edge width**
- **Cutting edge depth**
- **Thread pitch**

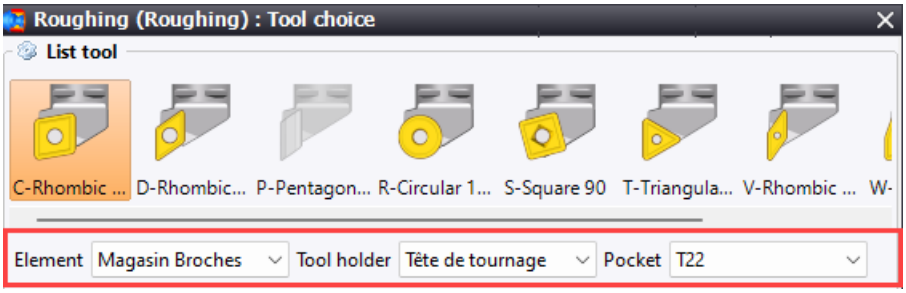


Magazine: Tool selection via mouse wheel

You can either select a tool by checking the corresponding box or use the mouse wheel (central button) to click on the line of the tool you wish to select.

Magazine: Magazine, tool holder and pocket choice lists

The choice lists for magazine, tooling holder and pocket have been grouped together under the tools list. To improve the information visibility, the display is adjusted to the window size.



Magazine and cutting conditions: Number of teeth display

A column can be added to display the number of teeth for each tool in the tool magazine, as well as in the cutting conditions table.

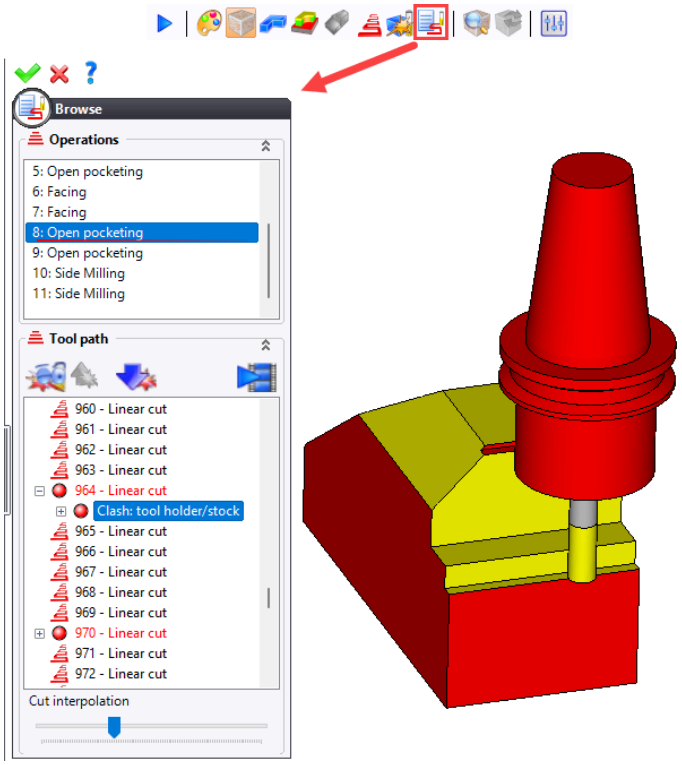
Cutting conditions: Validation of an operation forbidden in invalidity cases

It is no longer possible to validate an operation if one of the cutting conditions is null or invalid. This also applies to all rotation and feed rate speeds in the settings.

Verification and simulation

Verification: Browsing cuts

The new **Browse** command allows you to check machining operations one by one, and to visualize any movements with collisions. This command can be found in the **Verify** tab, as well as in the context menu, and is only active in the verification context in programming simulation mode.

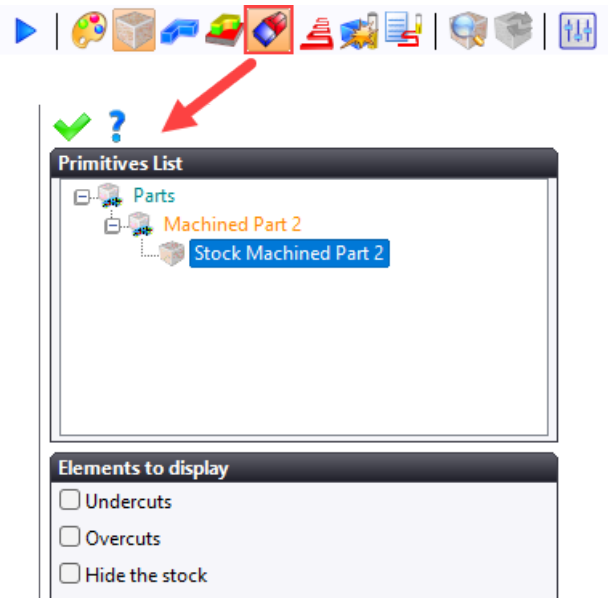


Verification: Display for remaining material

The **Display undercuts/overcuts** command has been added to the **Verify** menu and to the context menu.

It can only be accessed at the end of verification of the selected machining range, with the animation mode (programming or machine) and with the material removal activated.

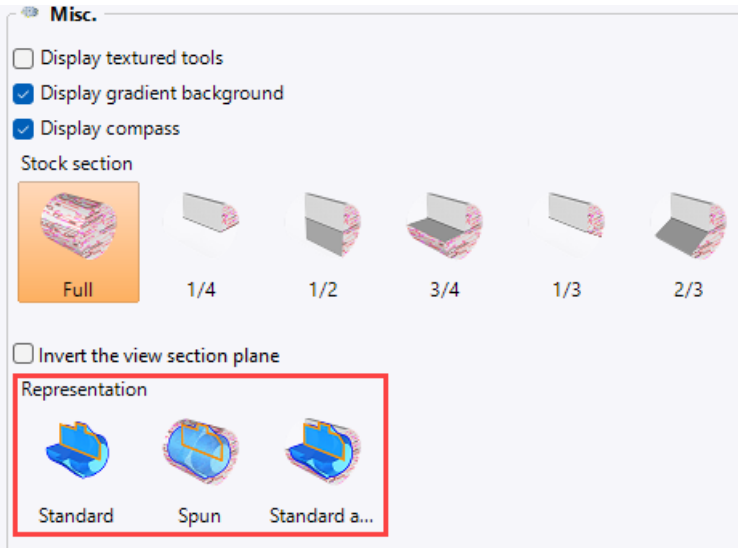
Similarly, at least one stock/finish pair must be present in the verification. Parts without finish and/or no stock are not accepted.



Verification: Simulating rotating elements

In the **Display** tab of the **Settings** command, three new representation modes allow you to adjust the display of rotating elements when checking a turning operation:

- **Standard**: Displays rotating elements in 3D mode (non-rotating).
- **Spun**: Shows elements in rotation mode (silhouette of element rotated).
- **Standard and spun**: Shows rotating elements in 3D mode (opaque display) and elements in rotation mode (transparent display).

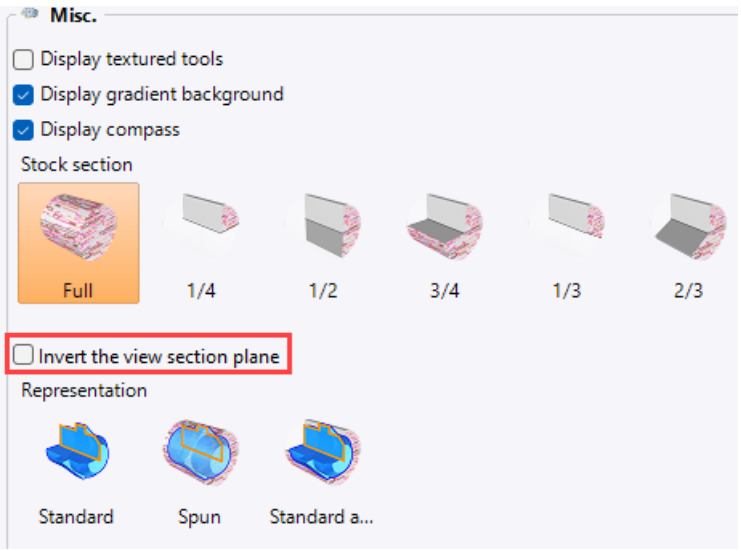


Verification: Cross section view on selected elements

If you have created a graphics cut in the CAD or CAM context, indicating the elements to be taken into account, you can activate it during the verification phase, only on the elements previously selected.

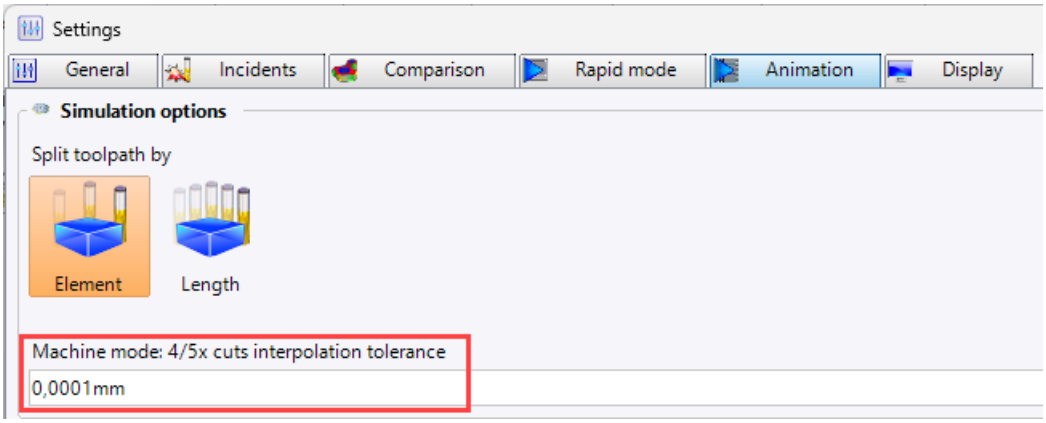
Verification: Inversion of the view section plane

The view section plane can now be inverted in the **Display** tab of the verification settings.



Verification: Toolpath tolerance adjustment in machine mode

A new parameter in the **Animation** tab of the verification settings allows you to adjust the **4/5X cuts interpolation tolerance**, with the aim of improving performance.



Verification: Activation/Deactivation of comparison colors

In the **Comparison** tab of the verification settings, it is now possible to click on a color in the legend to activate or deactivate it globally.

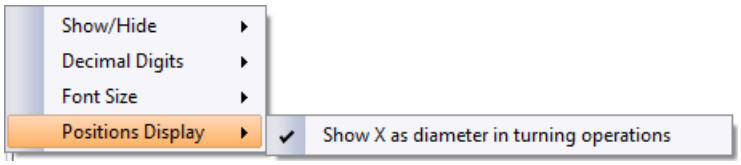
You can also hide a color directly in the comparison context by clicking on the legend; the display is automatically updated.

Verification: Using directional and scroll keys

Using directional arrows and the "Page Up" (previous page) and "Page Down" (next page) keys is now possible while checking to orientate the view.

Verification and simulation: Displaying X as a diameter in turning operations

In simulation and verification, the new **Show X as diameter in turning operations** option displays the X axis coordinate of turning operations as a diameter. It can be accessed via the context menu in the simulation or verification window, and is activated by default.



Verification and simulation: Pause in case of out-of-stroke axis

During simulation or verification, it is now possible to stop at the point where an axis limit is exceeded.

For simulation, check the new **Pause simulation at exceeding machine strokes** option available from **Tools** > **Options** > **Machining** > **Simulation**.

For verification, select the new **Axis limits exceeding** bounding mode available from the **Stop at** section in the **General** tab of the verification settings.

Methods

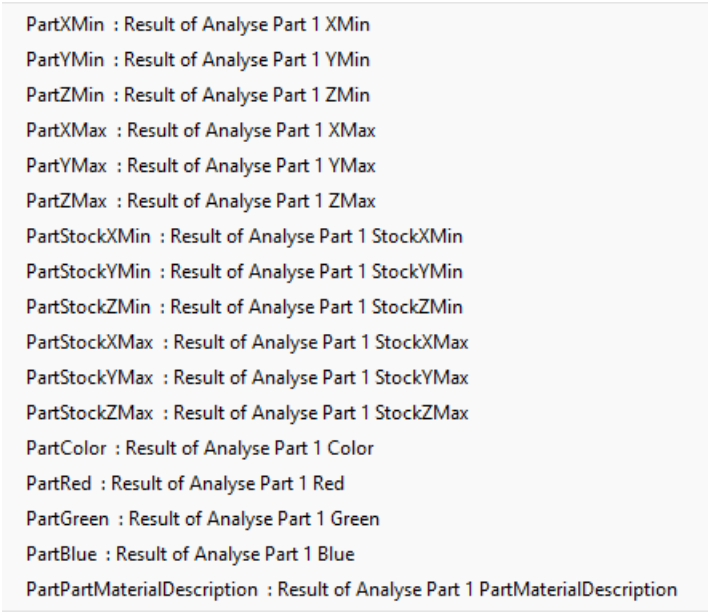
Center window

When creating and/or editing a general milling/turning method or a wire method, the method is now displayed in the central window, rather than in a pop-up window.

Part analysis

The new **Analyze Part** analysis function retrieves the minimum and maximum coordinates of an NC part and its stock, as well as the part's color and material.

WCS are not mandatory, but are essential to get the values for the part or stock dimensions. Coordinates are given in relation to the WCS.



Face area and profile length

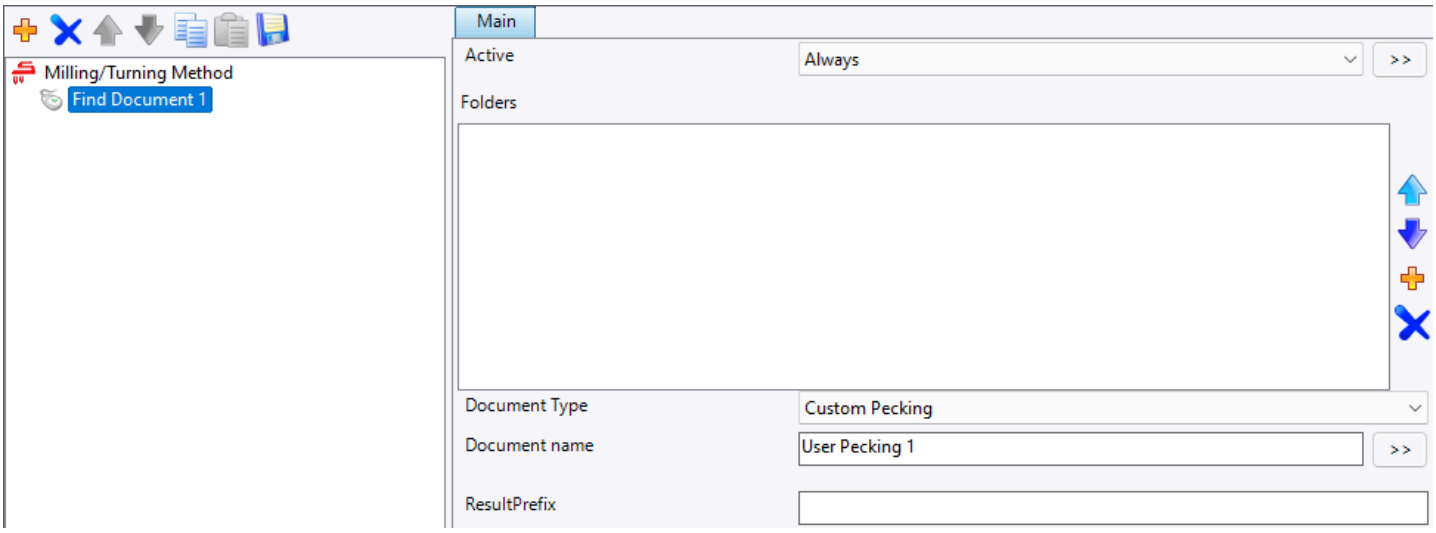
The **Analyze Face** function now retrieves the area of a face and the **Analyze Profile** function retrieves the length of a profile.

Find a document

The new **Find Document** search function allows you to find a document from its name in a specific folder, project or library, and use it like a document variable in methods. If no folder is selected, the method will search in the current project and its references.

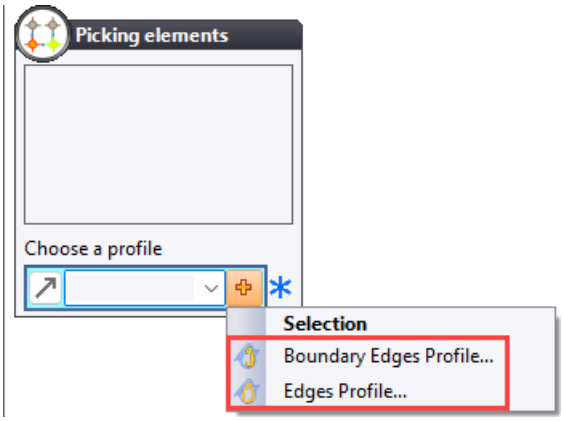
The documents eligible for search are the following:

- Customized lead in/lead out
- Customized drilling
- Link movement



Edge profile in element selection

When executing a general method, you can create a profile on edges directly from the selection of profile-type elements.



Search with special characters

You can search for an element/entity or machine component from part of its name or description by adding a special character. The special character ***** replaces a character or a string of characters and the character **?** replaces a character.

This search mode is compatible with the following actions:

- **Find Elements**
- **Find Machining Feature (MF)**
- **Find Document**
- **Find Machine Axis**
- **Find Machine Element**
- **Find Machine WorkStation**
- **Find WCS from a given geometry**

Material side reversal in 5D contouring operations

Similar to 2D operations, you can now select the material side of the curve to follow in a 5D contouring operation. Three choices are possible:

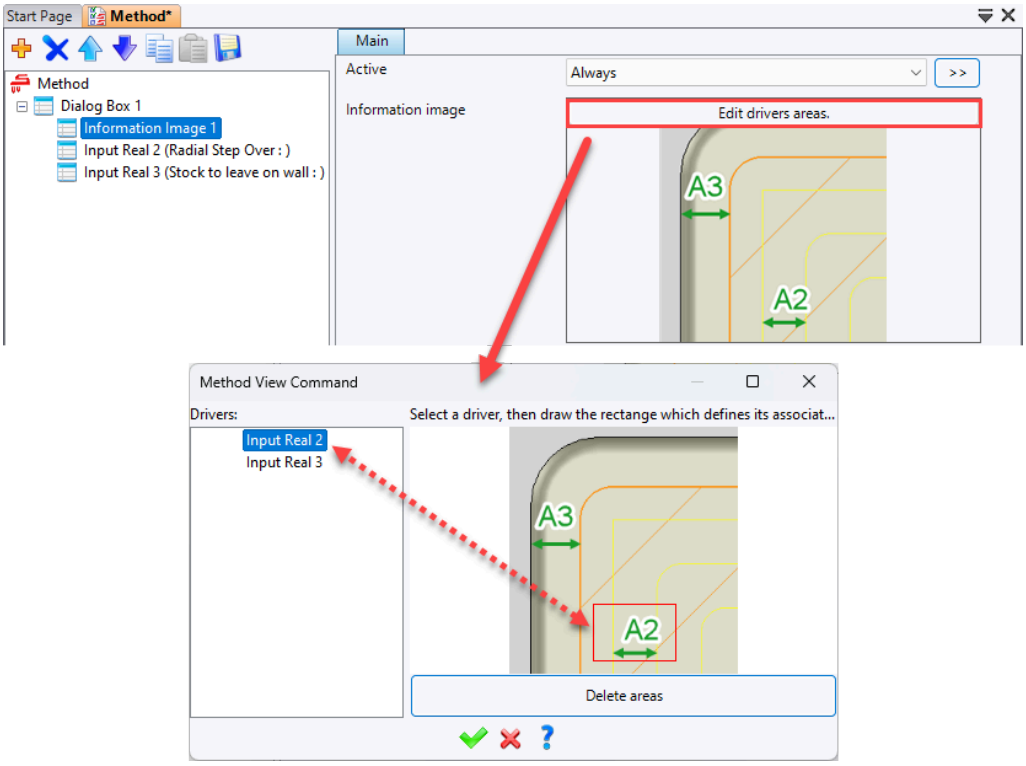
- **Given by profiles**: The direction of the profile gives the material side.
- **Reversed** All profiles are reversed.
- **Result of profile to machine analysis**: Each profile can be reversed or not (the result is provided by the **Analyze Profile** function created before the machining operation).

Resizing of the dialog box and zooming in on images

When executing a method, you can resize the dialog box and zoom in/out on images.

Information image for dialog box

When defining the information image for a dialog box in a general method, the new **Edit drivers areas** button allows you to frame a particular area of the image in red, to make it easier to understand the question when executing the method.



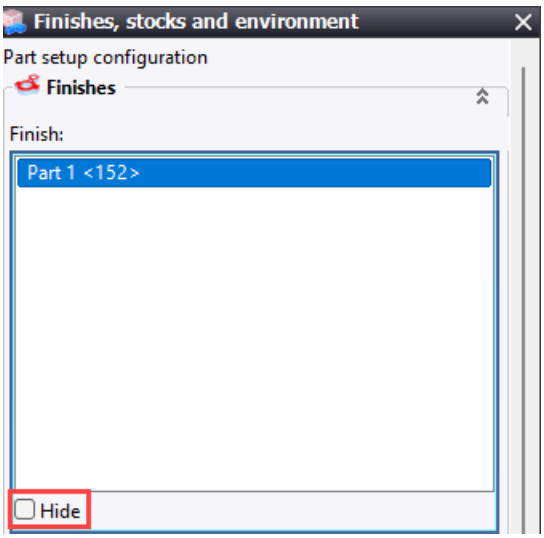
Preselection of the original method

When exchanging a method with another, the original method is now preselected by default if associativity with the method was required.

Miscellaneous

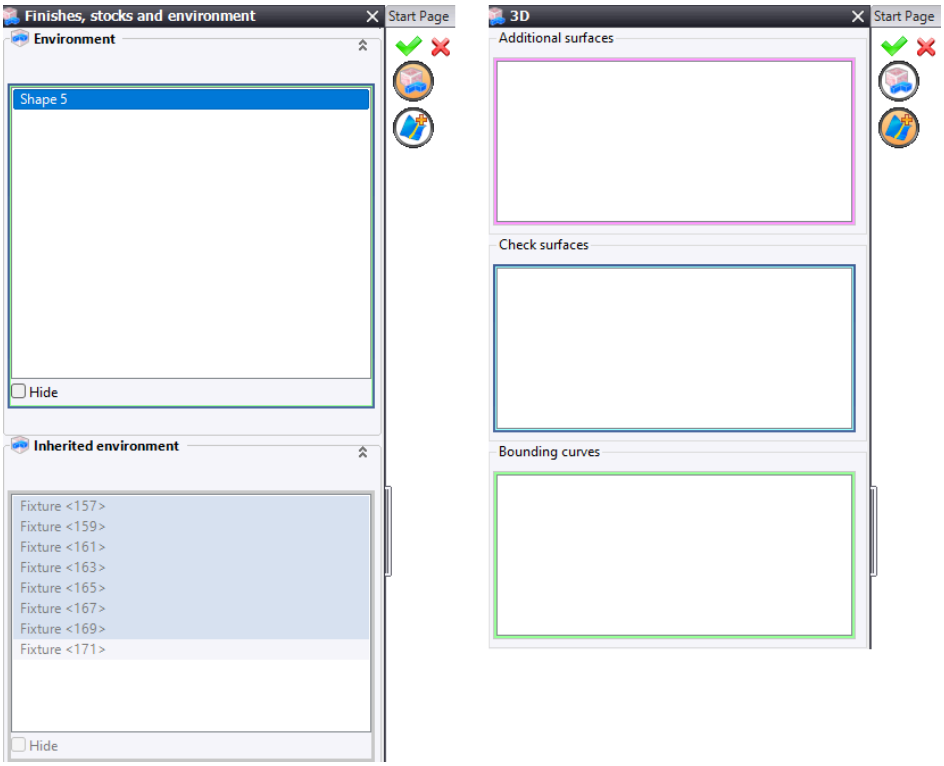
Machined part setup document: Hiding selected entities

When defining the machined part setup document, the new **Hide** option hides the entities already selected in the **Finishes**, **Stocks** and **Environment** fields.



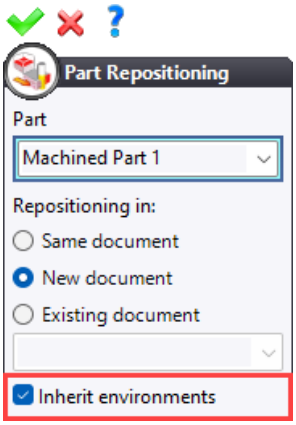
Derived machined part setup document: Derived NC part editing

The new **Edit derived Part NC** command is available in a derived machined part setup document. It allows you to add environment elements, as well as auxiliary elements (additional surfaces, check surfaces and bounding curves).



Part repositioning: Inheriting environments

The repositioning command features the new **Inherit environments** option which allows you to inherit or not from the environment defined in the machined part setup document.



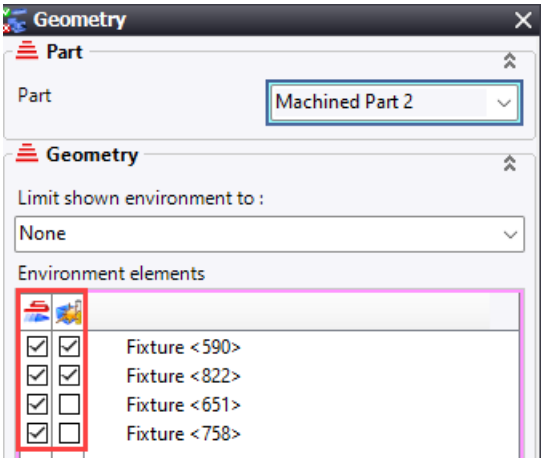
Environment: Creation of environment sets

You can create environment sets to use in environment activation/deactivation. To create a group, right-click on the **Part Environment** folder in the Entities tree and select the **Create environment set** command.

Environment: New environment activation options

The environment activation window displays two new columns of checkboxes:

- The first column is used to show or hide the environment element during simulation/verification.
- The second column is used to take into account or not the environment element in the calculation of machining toolpaths.



Repetition in the operation: Environment management in rapid movements

The environment can now be managed in the link movements of the operation's internal repetitions. If the final altitude defined in the operation is greater than that of the environment, retractions are made to the final altitude.

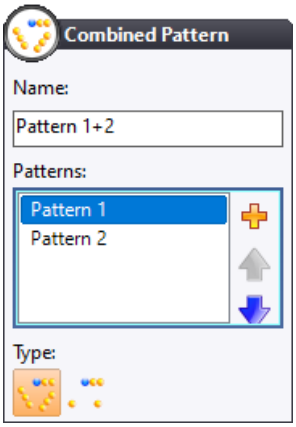
Patterns: Analysis of the number of elements in a pattern

Using the **Analyze** command on a pattern allows you to find out the number of elements it contains.

Patterns: New combined pattern

The new **Combined Pattern** pattern allows you to make the sum or multiplication of multiple patterns.

Patterns are applied in the same order as they appear in the list.



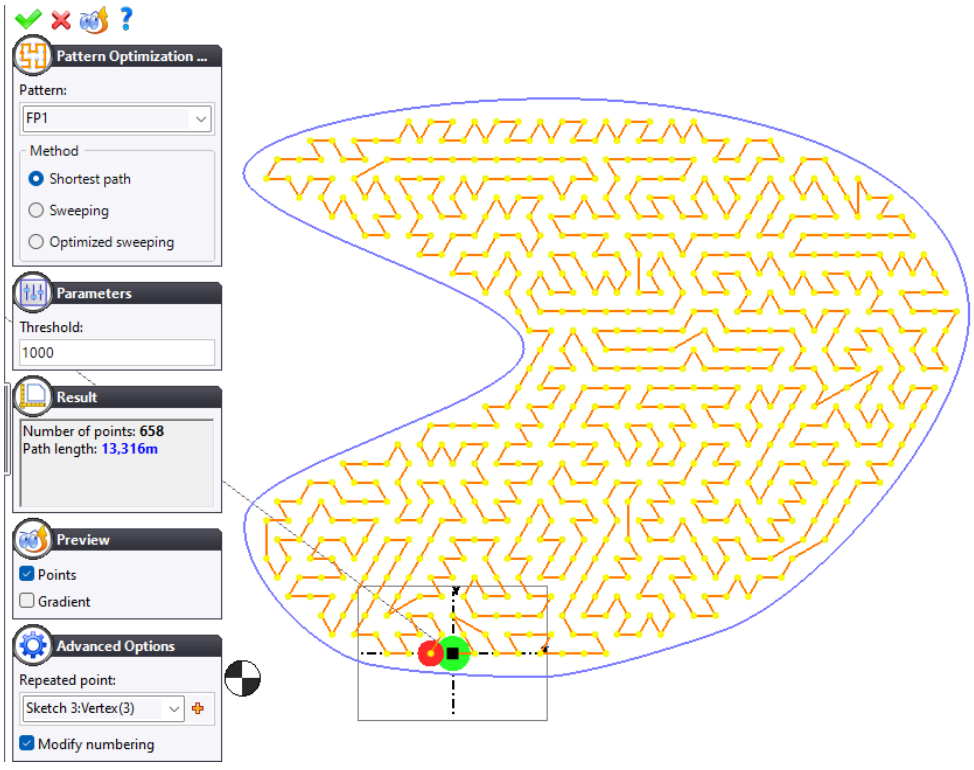
Patterns: Pattern optimization

The new **Pattern Optimization** command offers three modes for optimizing the path through the pattern:

- **2-Opt optimization**
- **Sweeping**
- **Optimized sweeping**

The result displays the number of points, the total path length and the computing time.

You can modify the preview by displaying the points and a gradient of colors from the start (green) to the end (red) of the path.

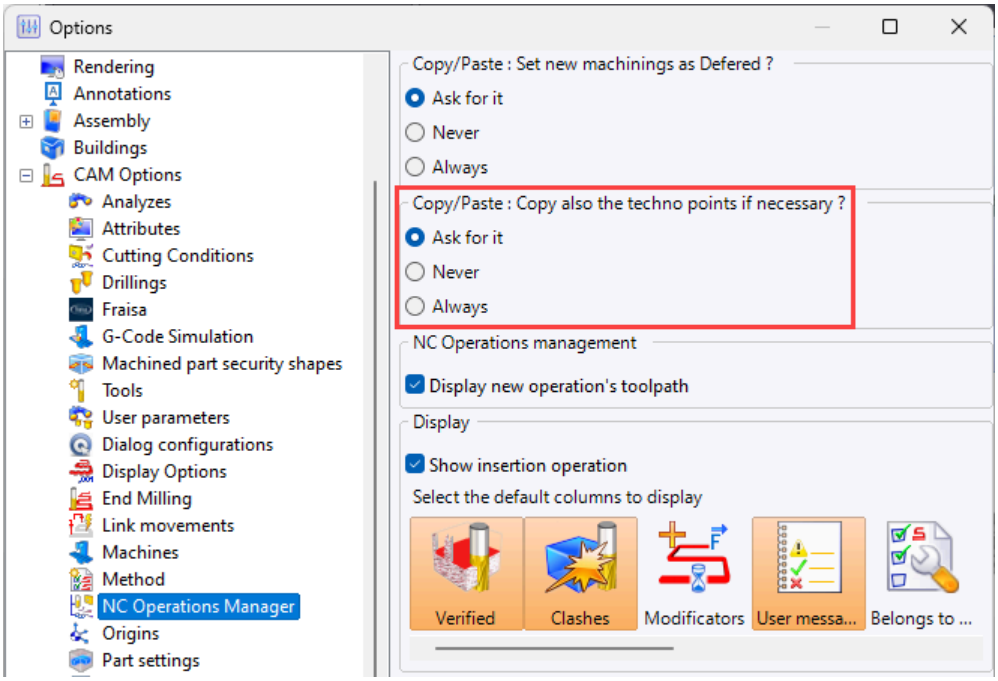


WCS No description for manual WCS

When a WCS (face or frame) is created manually, the **WCS Description** field is no longer filled in automatically with the value of the initial WCS solution.

Copy/paste operations: Behavior of technological points

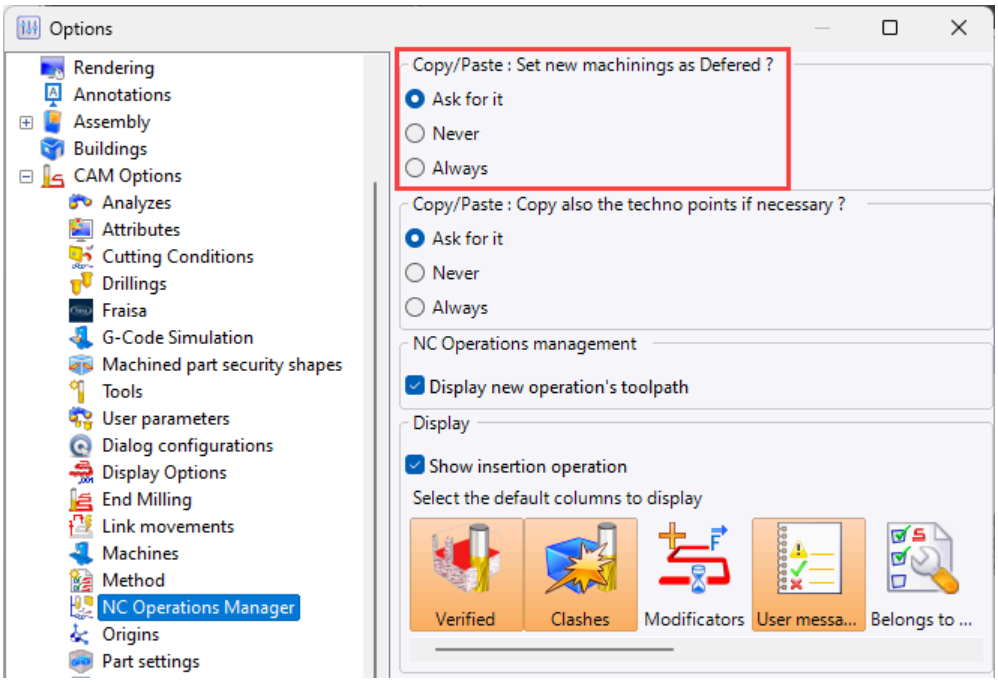
In the **Options** > **CAM Options** > **NC Operations Manager** command, you can choose whether to copy existing technology points when copying/pasting an NC operation.



Copy/paste operations: Creation of machinings in deferred mode

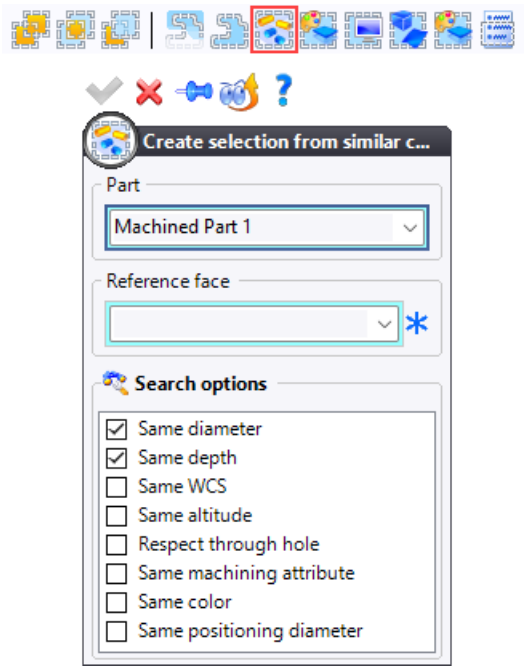
When copying and pasting a machining operation, you can create the operation by activating the **Launch deferred** mode, in order to choose when the operation is to be computed.

This feature can be configured in the **Options** > **CAM Options** > **NC Operations Manager** command.



Selection: Selection of elements with cylinder search

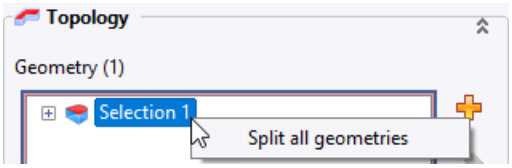
The **Create selection from similar cylinder** selection command has been added to the **Visualization** tab to make it easier to select cylinders with the same criteria (diameter, depth, etc.).



Selection: Associativity of selection with machining operations

In all machining operations that accept a selection of elements, the link is now associative by default. If the selection used is modified (geometries added or deleted), the machining operation must be replayed.

To delete associativity, right-click on the selection and select the command **Split all geometries** command.



Scenario: Simplified return to machining stage

Previously, double-clicking in the scenario window gave access to the final machining stage, but not to go back to the machining stage.

You can now double-click on the desired part origin symbol to return to the corresponding machining stage.

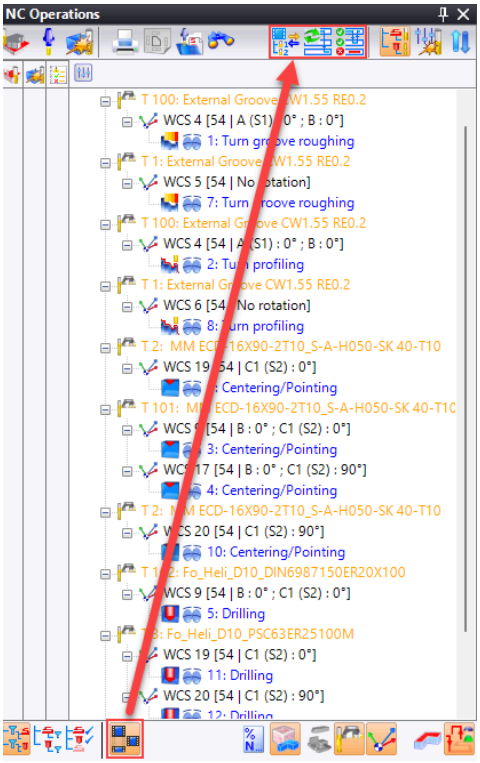
Scenario: Scenario sequence in the operations manager

It is possible to apply the scenario operations order in the operations manager for multichannel machines without requiring part repositioning within the same document (no repositioning step in the machining document).

This feature was already available for kinematics with a single part holder and one channel.

The new **Simplified scenario mode** icon, accessible in the machining final stage, enables the display of operations in the sequence outlined by the scenario. When this mode is activated, three new icons appear on top of the NC Operations tree:

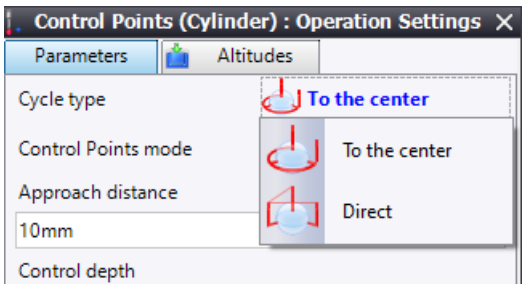
- : Applies the operations sequence from the scenario in the operations manager.
- : Verifies scenario validity.
- : Updates the scenario.



Control points: New cycle types

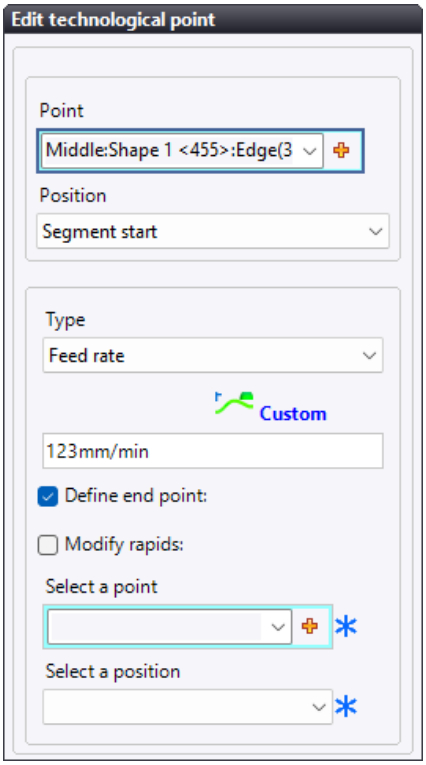
In the **Control Points** command, the **Cylinder**, **Corner**, **Slot/tongue** and **Rectangle** types integrate the new **Cycle type** parameter which proposes the following modes:

- **Direct**: The tool is positioned directly in XY at the first control point.
- **To the center**: The tool is positioned in XY in the middle of the cylinder, slot, rectangle or corner.



Technological points: Definition of an end point for feed rate application

You can define an end point when creating a technological point of **Feed rate** type.



Presets: Filters in the preset selection

You can apply filters when selecting a preset by right-clicking on the desired column name.

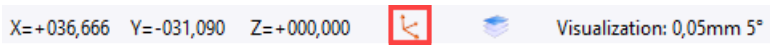
Filtering is about the next columns:

- Preset name
- Tool type
- Tool description
- Machine

Graphics area: Displaying coordinates according to a WCS or an origin

Previously, the display of the mouse's XYZ coordinates in the graphics area (at the bottom of the **TopSolid** window) was depending on the absolute frame.

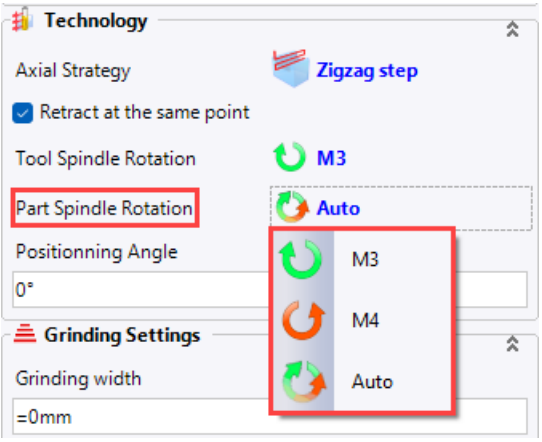
It is now possible to specify another frame to obtain a second display of the mouse's XYZ coordinates.



Grinding

Part rotation direction

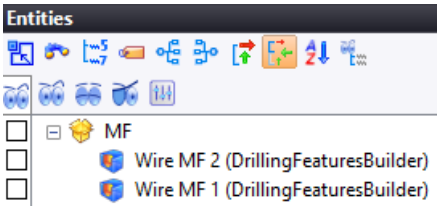
In a grinding operation, the new **Part spindle rotation** parameter allows you to choose the rotation direction of the part. The **Auto** mode matches the behavior found in previous versions.



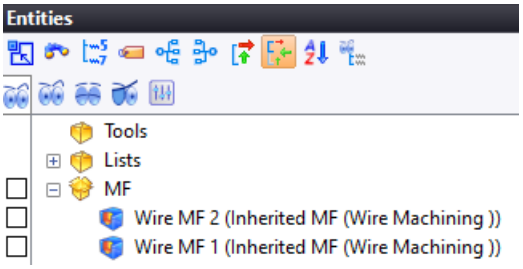
Wire

Wire: Information transmission between the wire document and the machining document

When creating a machining document from a wire document, information can be transmitted via drilling MFs to automatically drill the threading holes.

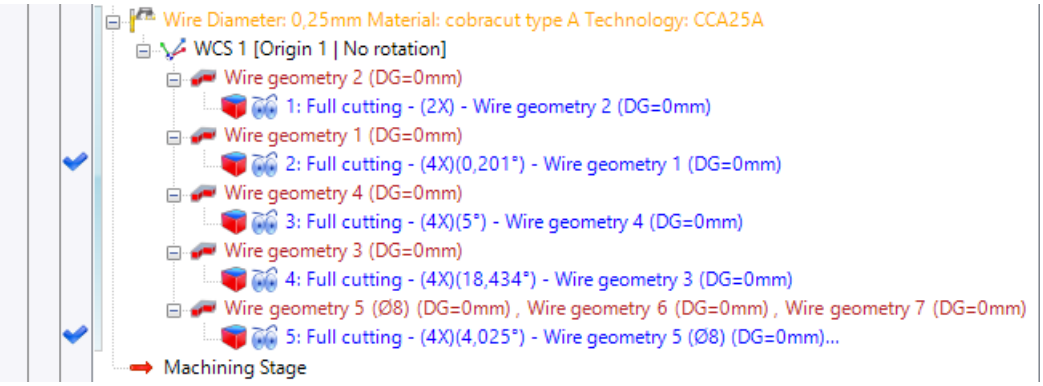


The machining document created from the wire document automatically inherits from the wire document's MFs, enabling the associated drilling operations to be performed.



Wire: Cutting type in the operations name

The wire operation name displays information on the cutting type (**2X** or **4X**) and the maximum angle.

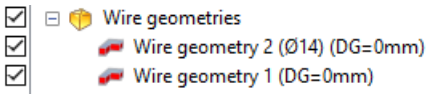


Wire geometry: Wire geometry name

When creating a wire geometry, the **Name** field is now blank by default. If you leave this field blank, the default name will be applied (behavior from previous versions). If you enter a name, this will define the start point of the wire geometry, before downsizing and possibly the diameter.

Wire geometry: Diameter in the wire geometry name

If the geometry is a cylinder or a circle, the wire geometry name contains the diameter value.



Wire geometry: Improved visualization of limited wire geometry

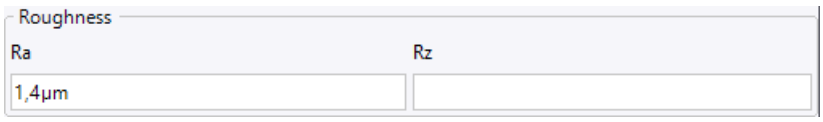
If a limitation is specified in a wire geometry via the **Starting point** and **Ending point** fields, its visualization is now improved in the final preview in the graphics area.

Wire features analysis: Red arrow on wire geometry

When the **Echo Arrow in Dialogs** display command is activated in the wire document, a red arrow is displayed in the graphic area, pointing to the geometry selected in the wire features analysis.

Wire technology: Roughness management

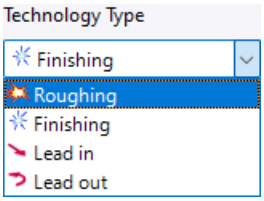
You can enter roughness values from a wire technology document.



Wire technology: Management of lead in and lead out technologies

Lead in and **Lead out** new technology types are available in the wire technology document, as well as in the wire method.

Each type of technology now features a distinct icon, simplifying identification when searching a technology.



When selecting the main technology in the wire technology document, if only one lead in or lead out technology exists, it is now automatically selected in the **Lead in** or **Lead out** tab.

Similarly, when selecting a technology from a method, if a lead in or lead out technology is required and the technology document contains only one lead in or lead out technology, it is automatically selected.

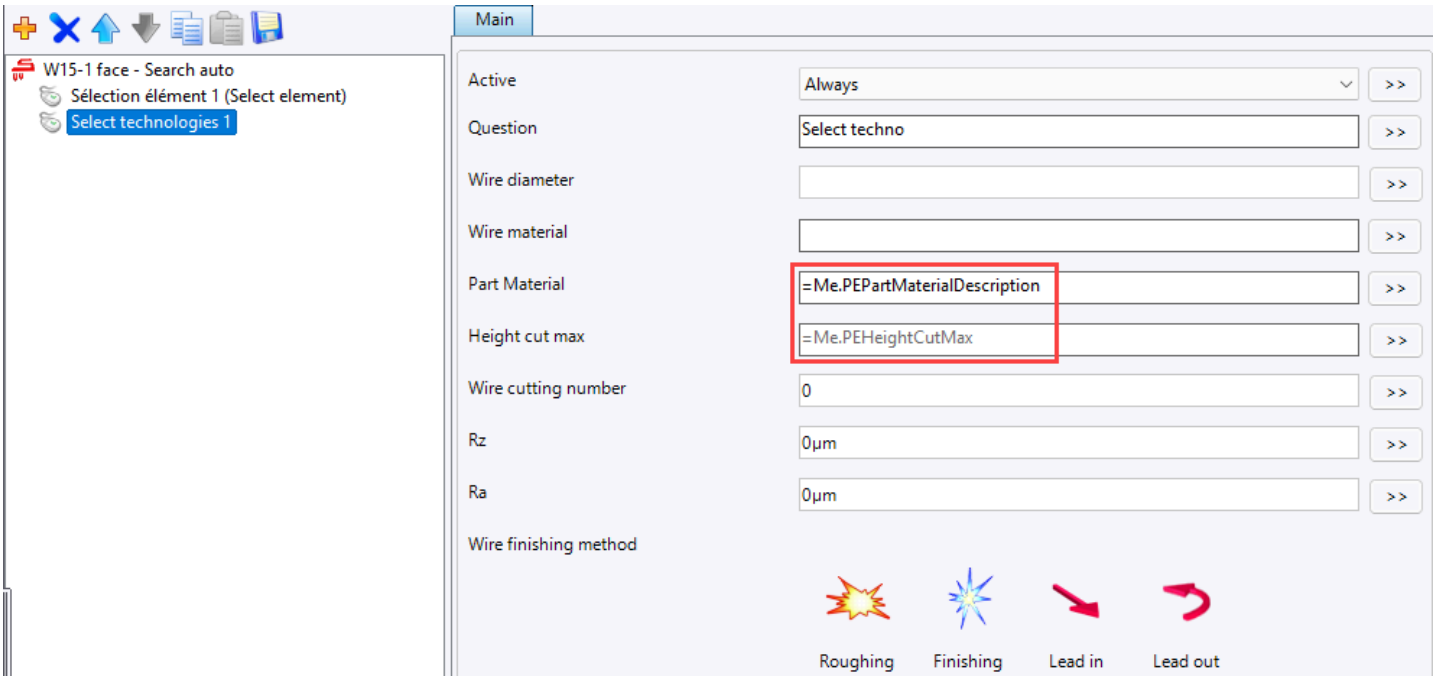
Wire technology: Current technology by default

A wire technology is selected when creating the first cutting operation on a geometry.

In the second cutting operation created on the same geometry, the technology selected by default is now the technology used for the previous operation.

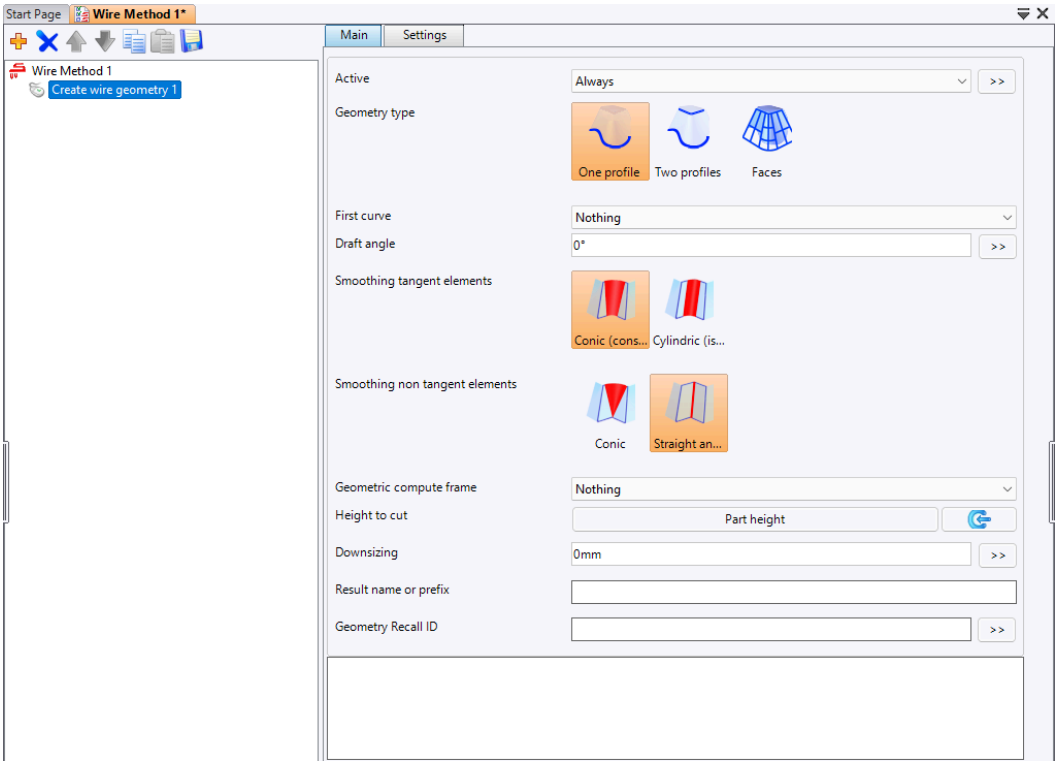
Wire method: Choice of technology according to the part material and thickness

The two new variables **PartMaterialDescription** and **HeightCutMax** are available in the wire method. They can be used in a technology selection action to set the part material and maximum cut height by default.



Wire cutting method: Wire geometry creation

The new **Create wire geometry** function creates wire geometry directly within the method. The same parameters can be defined as those available when manually creating a wire geometry in a wire document.



Workshop document: Falling part

The new **DistanceFallingPartCutting** property is available in the **Standard** > **Machining** >. **Geometry** and **Standard** > **Machining** > **Operations** > **Wire** > **Cutting** sections of workshop documents.

This value matches the falling part specified in the second part of the wire geometry definition.

Workshop document: Wire technology document name

In order to retrieve the name of the wire technology document used in the cutting operation, the following three properties have been added to the **Standard** > **Machining** > **Operations** > **Wire** section of workshop documents:

- **TechnologyDocumentName**
- **TechnologyDocumentNameLeadIn**
- **NameDocTechnoLeadOut**

Workshop document: Wire material

In order to retrieve the wire material name defined in the wire technology, the following three properties have been added to the **Standard** > **Machining** > **Operations** > **Wire** section of workshop documents:

- **WireMaterialTechnoDoc** (wire material name for main cutting)
- **WireMaterialTechnoDocLeadIn** (wire material name for lead in cutting)
- **WireMaterialTechnoDocLeadOut** (wire material name for lead out cutting)

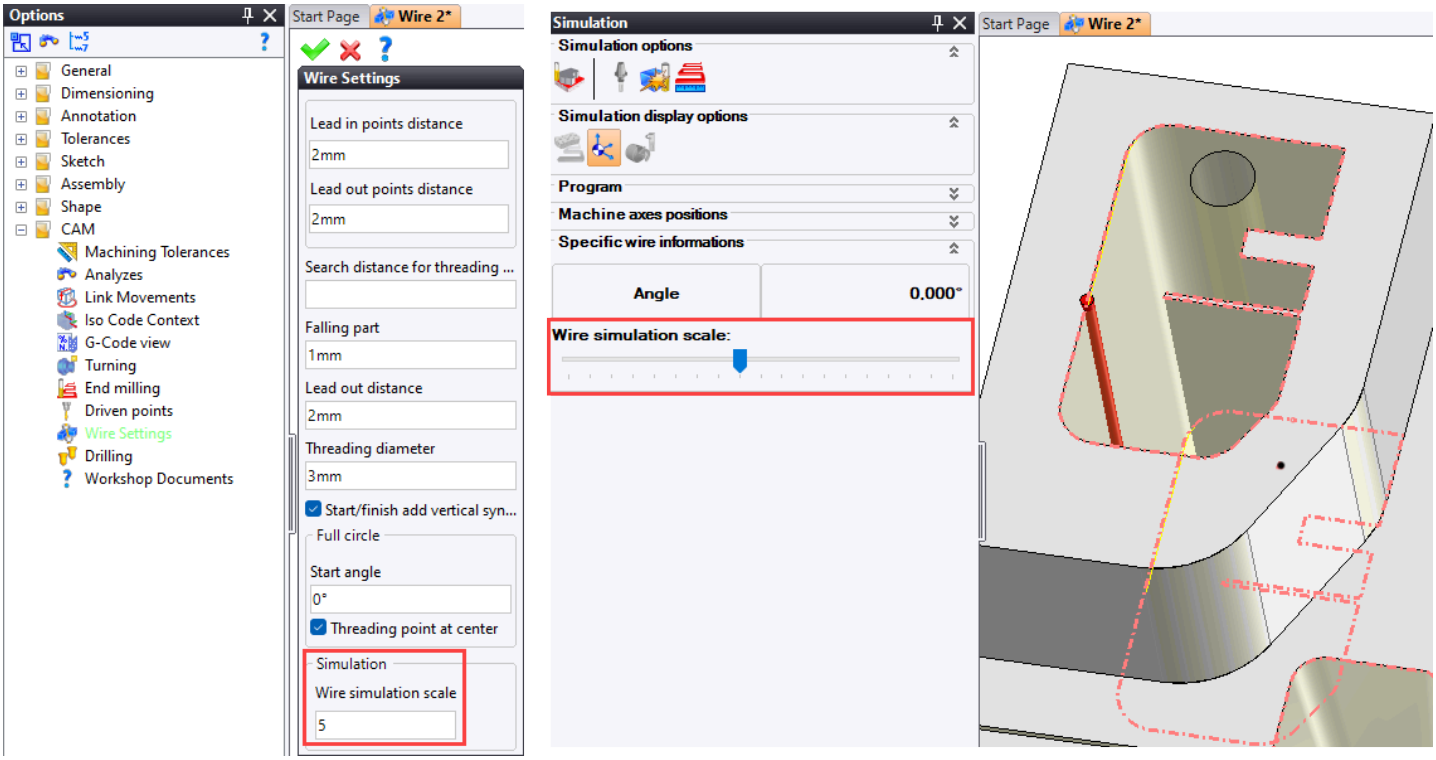
Simulation: Maximum draft angle

The maximum wire draft angle defined in the wire machine is taken into account when simulating a cutting operation. The wire angle text is displayed in red if the maximum value is exceeded.

In the operations manager, the operation is underlined and a user message appears, plus an error message when the operation is executed. Synchronizations outside the limits are shown in red in the graphics area.

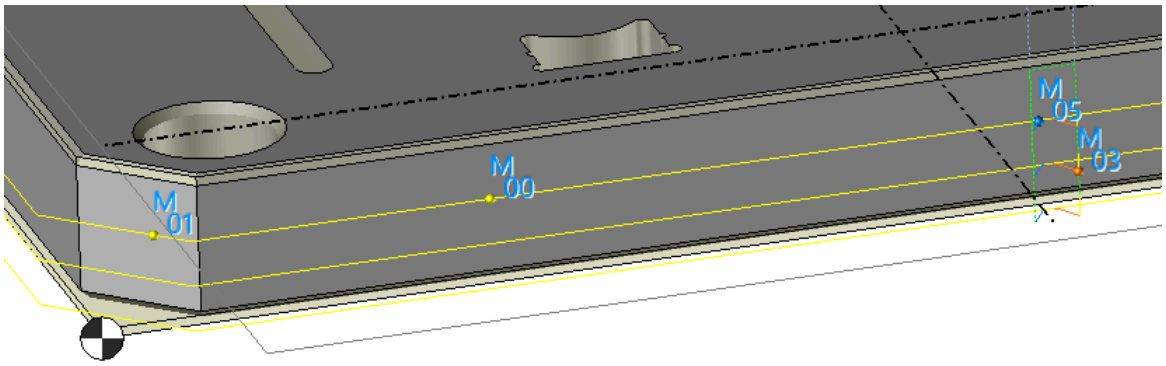
Simulation: Wire simulation scale

You can adjust the scale to the wire diameter during simulation, without modifying the actual wire diameter. This parameter is available in the wire document options, as well as in the simulation window. You can set a scale factor for each wire document.



Technological points: Improved display for stops

The visualization of **Stop** technological points has been improved. The letter M followed by the number is displayed if the value is between 1 and 5; otherwise, only the letter M is displayed.



What's New in TopSolid'Tooling 7.18

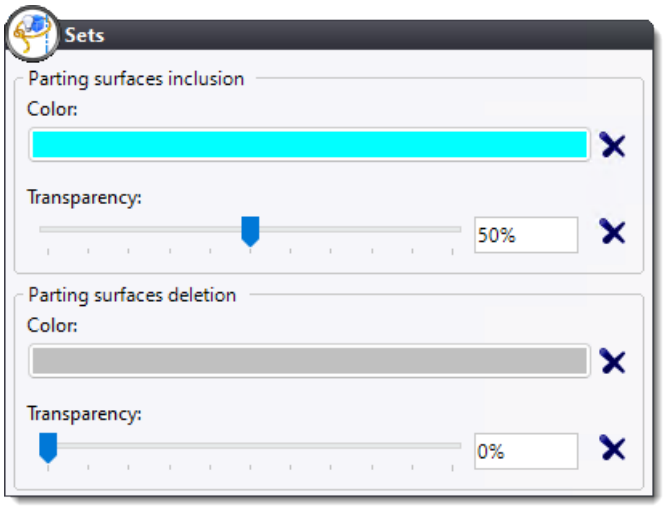
This chapter describes the new features of the **Split**, **Mold**, **Electrode**, **Strip** (Strip design) and **Progress** (Progressive die) applications in in **7.18** version of **TopSolid 7**.

TopSolid'Split

Set attributes

From now on, only surface shapes can be added to the **Parting Surfaces** set and only solid shapes can be added to the **Action Insert Shapes**, **Insert Shapes** and **Core Cavity Block Shapes** sets.

In the **Split Blocks** document's Options tree, the new **Split Blocks** > **Attributes** > **Sets** category allows you to assign color and/or transparency to parting surfaces when they are included in or removed from the set.



You can also predefine attributes for the **Action Insert Shapes**, **Insert Shapes** and **Core Cavity Block Shapes** sets.

The color and transparency of the parting shapes are transferred to the parts generated by the **Split Blocks** document and included in the mold.

Publishing entities

The **Publishings** submenu has been added to the **Tools** menu, which now allows you to publish entities in a **Split Blocks** document. These are then copied to the assembly from the **Split Blocks** document and to the mold.

Layers

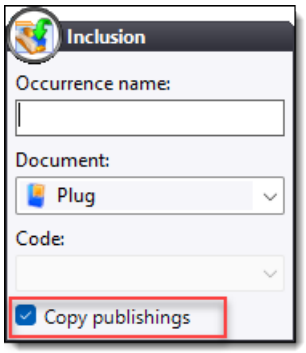
If layers are defined for parting shapes, they are associated to the parts generated by the **Split Blocks** document. When the assembly from the **Split Blocks** document is included in the mold, layers are kept on the part shapes.

Layers must be similar between **Split Blocks** and **Mold** documents. It is recommended to predefine the layer sets in two **Split Blocks** and **Mold** template documents.

TopSolid'Mold

Including core cavity blocks

In the inclusion command, the new **Copy publishings** option allows you to copy the entities that are published in the **Split Blocks** document.



Heel

Initialization of settings

To facilitate and speed up the creation of similar heels, you can initialize the parameters and settings of the **Heel** command, based on those used to create the first heel.

Simply select a face from the first heel before launching the command.

Pin

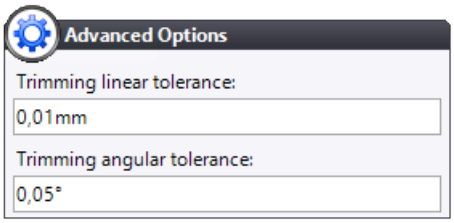
Initialization of settings

To facilitate and speed up the creation of similar pins, you can now initialize the parameters and settings of the **Pin** command, based on those used to create the first pin.

To do this, simply select a face from the first pin before launching the command.

Trimming tolerances

The trimming created by the **Pin** command is based on a linear tolerance of 0.01mm and an angular tolerance of 0.05°. To modify these values, it was previously necessary to use a context-sensitive command on the trimming operation performed in the pin document. Trimming modeling tolerances can now be adjusted directly in the **Advanced Options** section of the command, thus avoiding in-place editing of the pin.

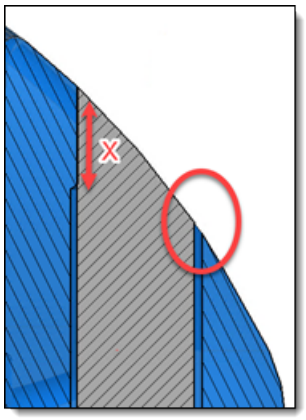


Pin processes

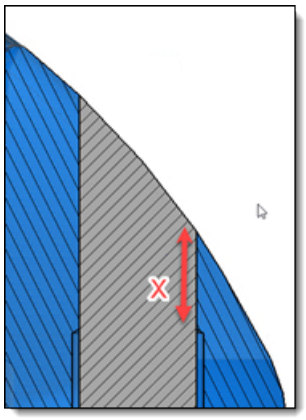
Boring length

The boring length was previously computed from the machined length of the pin.

In version **7.18**, this length is computed from the minimum machined length, thus avoiding machining problems in case of pins with a large amplitude trimming.



Boring length computed from the machined length.



Boring length computed from the minimum machined length.

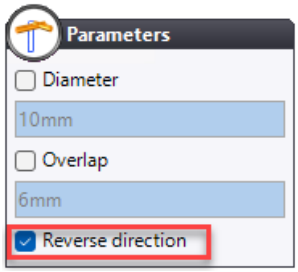
Cooling

Symmetrical pattern repetition

It is now possible to repeat the components automatically included in a cooling using a symmetrical pattern, which automatically generates a **Local mirror** operation.

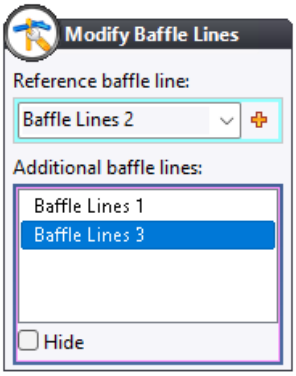
Modify drillings

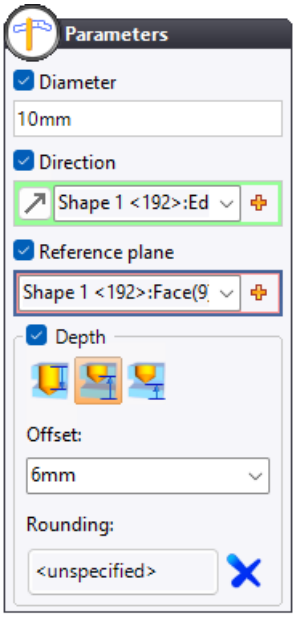
The **Modify Drillings** command features the new **Reverse direction** option, which makes it easy to reverse the directions of several cooling drillings simultaneously.



Modify baffles lines

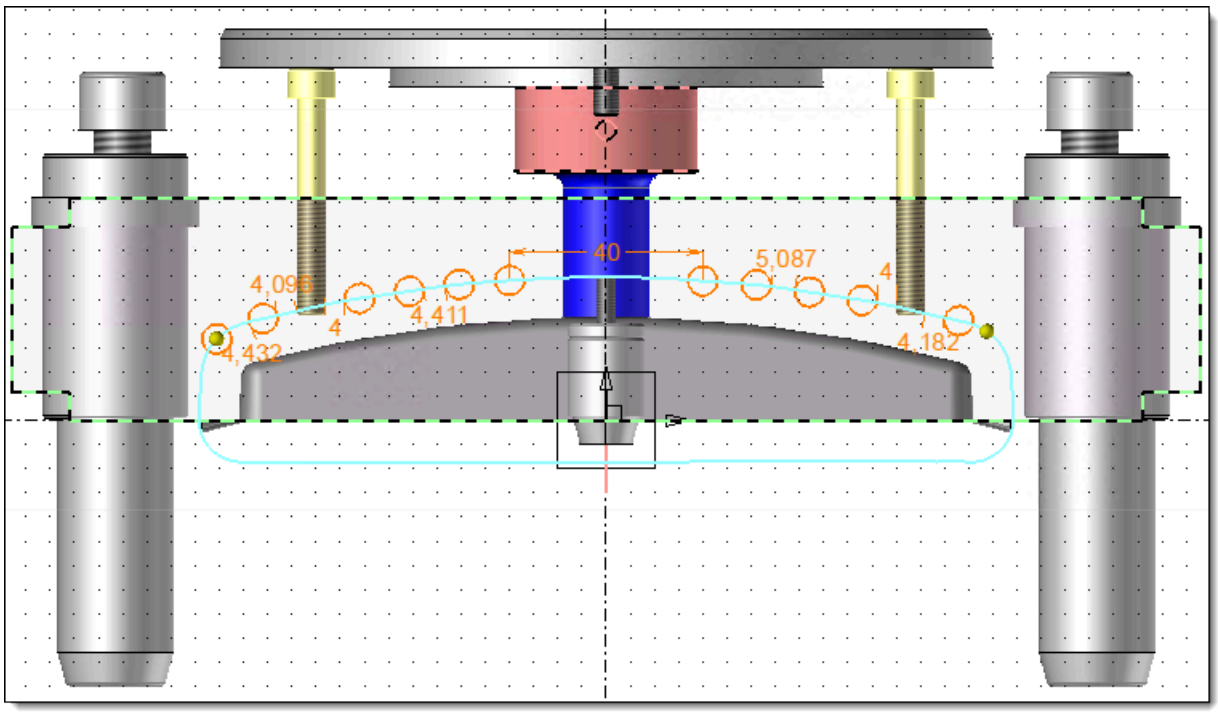
The new **Modify Baffle Lines** command is available when editing a cooling sketch. It allows you to easily modify the diameter, direction, reference plane and depth calculation mode for a batch of selected baffle lines.



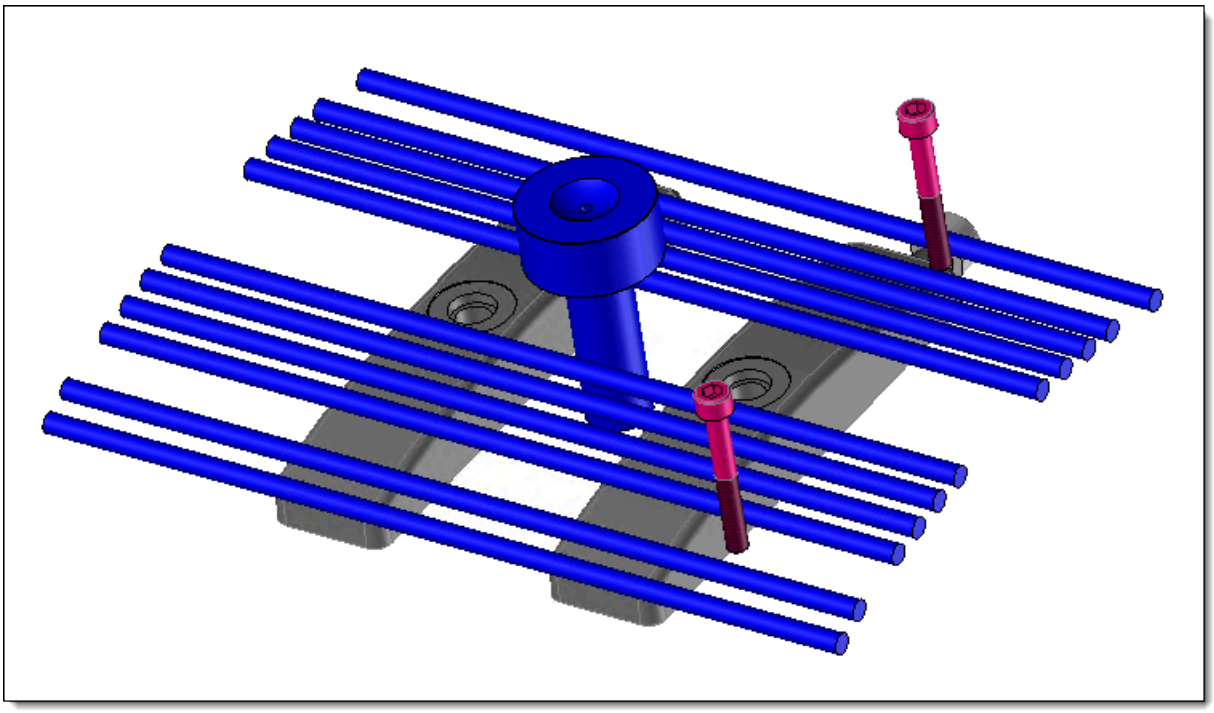


Drilling distribution wizard

The new **Drillings Distribution Wizard** command is available in a cooling sketch. It distributes drillings over a profile enclosing the parts to be cooled. Drillings are distributed with a minimum distance between each one, while framing the elements to be cooled (in pink) and avoiding, with a margin, the elements selected in yellow.

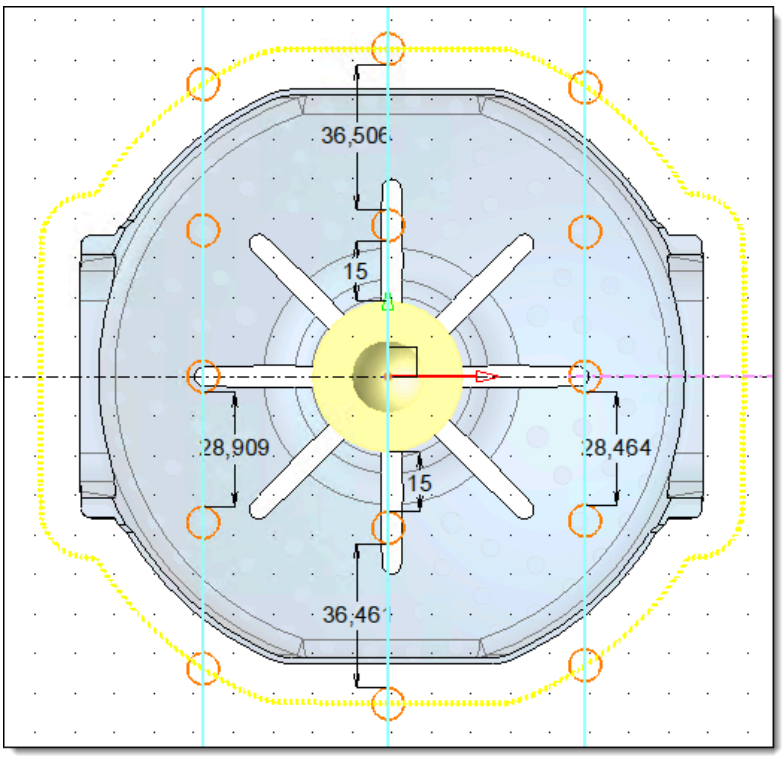


The wizard does not create any commands in the cooling sketch that can be modified later. This generates a sketch that can be modified in the cooling sketch solving, by modifying and/or adding constraints.

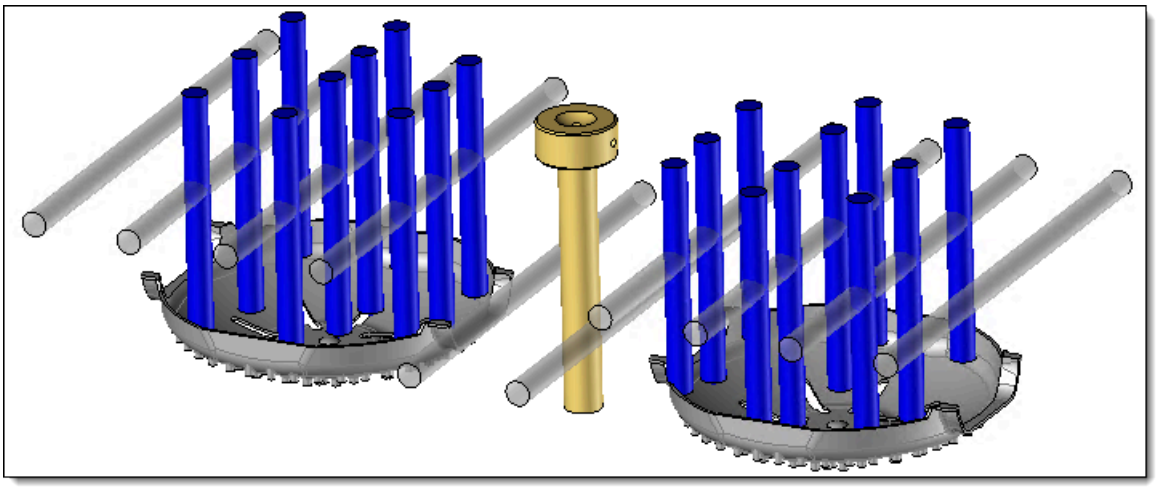


Baffle line distribution wizard

The new **Baffle Lines Distribution Wizard** command is available in a cooling sketch. In the same way as the drilling distribution wizard, it allows you to distribute baffle lines on profiles. The baffle line distribution area is defined by a silhouette with margin of the parts in the molded set.



Drillings are distributed with a minimum distance between each, while framing the elements to be cooled and avoiding yellow elements with a margin.

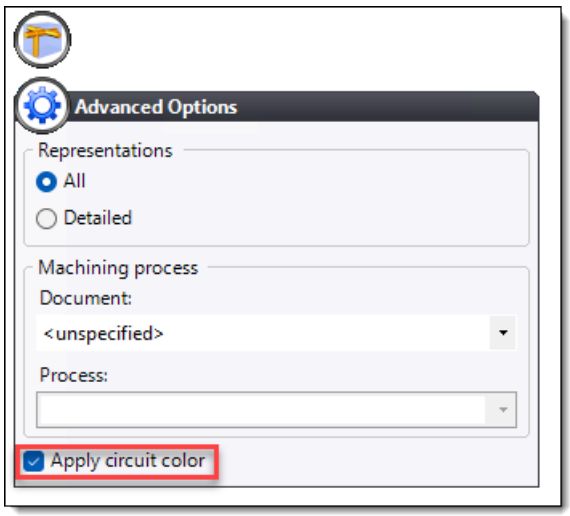


The wizard does not create any commands in the cooling sketch that can be modified later. This generates baffle line points that can be modified in the cooling sketch solving.

Cooling circuit process

Previously, the color of the cooling circuit was transferred to the faces resulting from the cooling process.

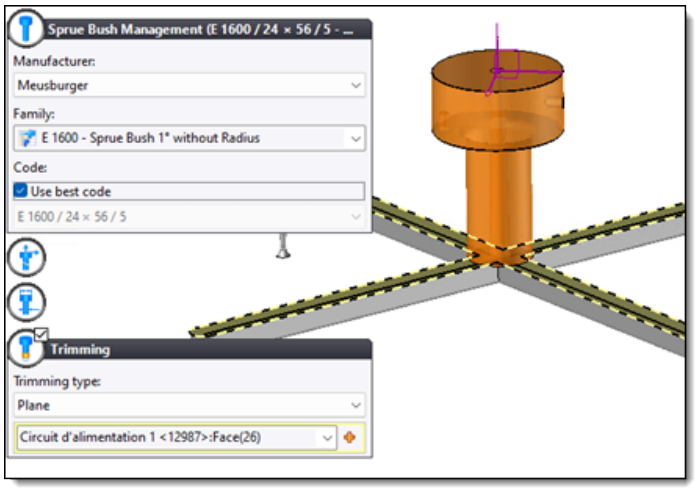
In version **7.18**, the new **Apply circuit color** option, available in the advanced options, allows the color of the cooling circuit to be applied to the faces resulting from the process.



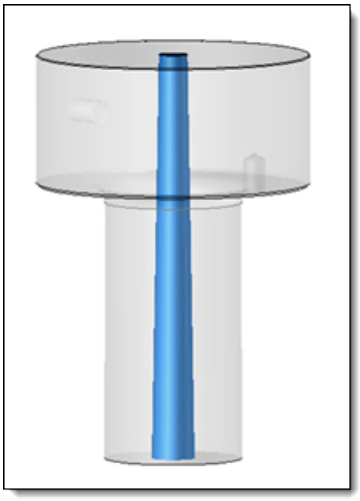
Sprue bush

The new **Sprue Bush** command allows you to include a sprue bush type component. The sprue bush is sized according to the mold environment and automatically limited to the desired length.

In addition, a runner shape matching the plastic material in the sprue bush has been defined in the sprue bush components. It is automatically taken into account when creating the plastic sprue.



*Sprue bush sizing and limitation with the new **Sprue Bush** command.*

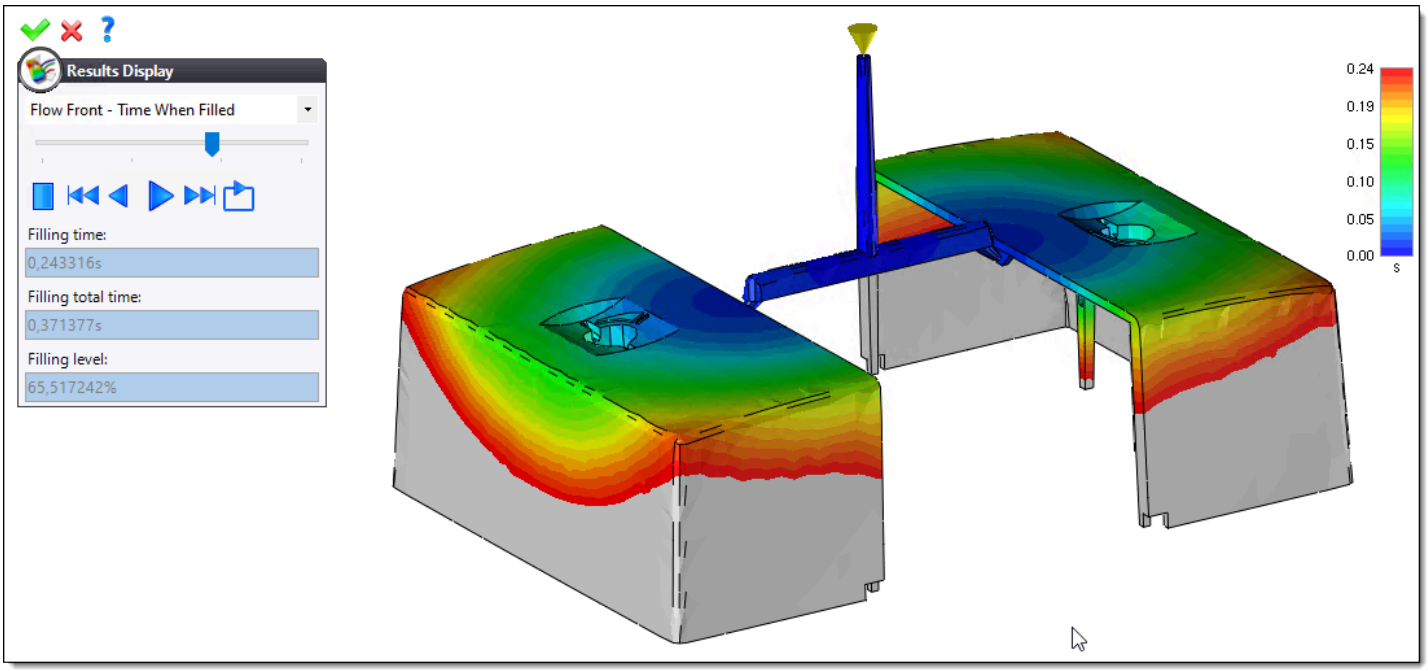


The runner shape is automatically limited with the sprue bush.

Plastic sprue

The new **Plastic Sprue** command automatically creates the mold's plastic sprue.

A part document is created, containing the union of molded set parts and runner, as well as runner shapes declared in sprue bush components. This allows you to create more precise plastic flow analysis of plastic injection.



Gate

Initialization of settings

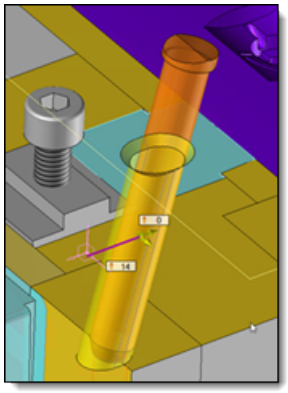
To facilitate and speed up the creation of similar gates, you can initialize the parameters and settings of the **Gate** command, based on those used to create the first gate.

Simply select a face from the first gate before launching the command.

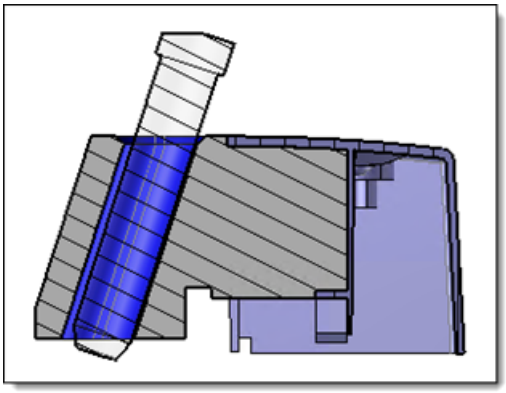
Angle pin

Process on slide

The machining created by including the angle pin in a slide can now be performed with a chamfer-type blend. It is also possible to apply a machining process to the faces machined in the slide.



Angle pin process with chamfer-type blend.



Machining process on the angle pin process in the slide.

Initialization of settings

To facilitate and speed up the creation of similar angle pins, you can initialize the parameters and settings of the **Angle pin** command, based on those used to create the first angle pin.

To do this, simply select a face from the first angle pin before launching the command.

Springs

Initialization of settings

To facilitate and speed up the creation of similar springs, you can initialize the parameters and settings of the **Springs** command, based on those used to create the first spring.

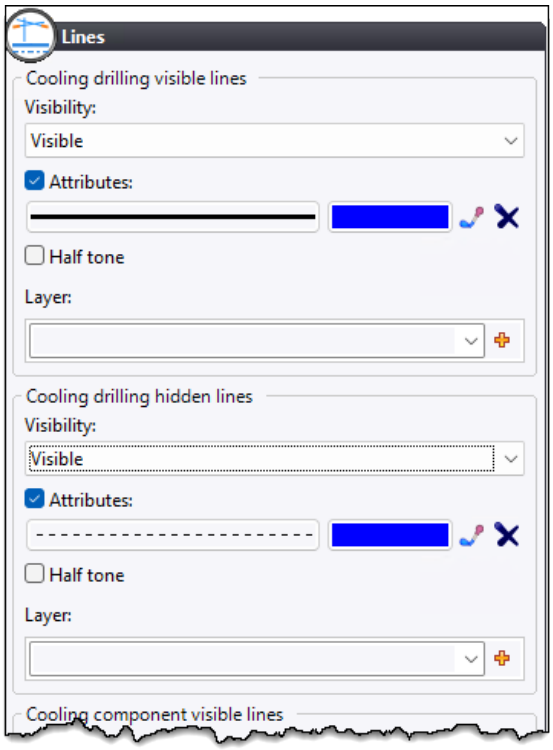
Simply select a face from the first spring before launching the command.

Drafting

Cooling circuit style

In the drafting context, the cooling circuit style offers new categories for fine-tuning the attributes of different cooling circuit lines.

For cooling drilling lines, visible and hidden lines are differentiated. The same applies to cooling component lines.

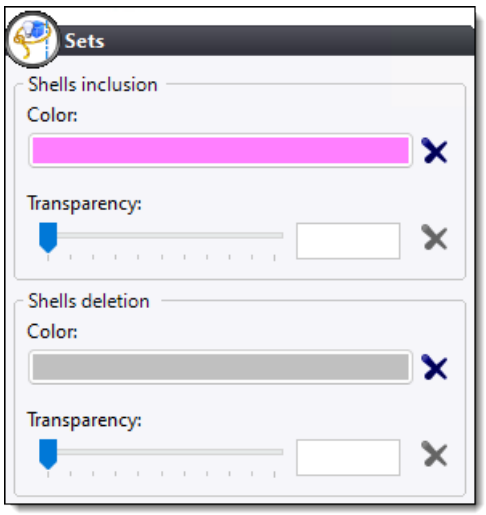


TopSolid'Electrode

Set attributes

From now on, only surface shapes can be added to the **Shells** set, only solid shapes can be added to the **Eroding Shapes** set and only parts can be added to the **Electrodes** set.

In the **Electrodes** document's Options tree, the new **Electrodes** > **Attributes** > **Sets** category allows you to assign a color and/or transparency to shells when they are included in or removed from the set.



You can also predefine attributes for the **Eroding Shapes** and **Electrodes** sets.

Eroding shape

Initialization of settings

To facilitate and speed up the creation of similar eroding shapes, you can initialize the parameters and settings of the **Eroding Shape** command, based on those used to create the first eroding shape.

Simply select a face from the first eroding shape before launching the command.

Electrode

Initialization of settings

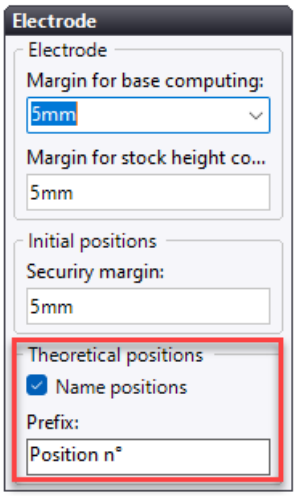
To facilitate and speed up the creation of similar electrodes, you can initialize the parameters and settings of the **Electrode** command, based on those used to create the first electrode.

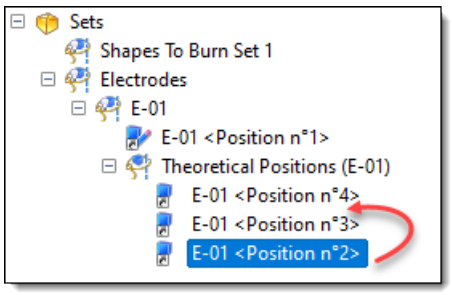
To do this, simply select a face from the first electrode before launching the command.

Theoretical positions

In the **Electrodes** document's Options tree, the new **Name positions** option allows you to name the different theoretical positions of an electrode by defining a prefix.

Positions are named in the set of theoretical positions related to the electrode. You can modify their order by moving positions within the set. The electrode position number depends on its position in the list.





The new **Construction** > **Patterns** > **Pattern Optimization** command modifies the order of positions in a pattern to optimize the path between the points of this pattern. The positions of the theoretical electrodes created with an optimized pattern will then naturally be ordinated in this order.

Electrode wizard

Initialization of settings

To facilitate and speed up the creation of electrodes of the same type, you can initialize the parameters and settings of the **Eroding Shape** and **Electrode** commands launched by the wizard.

Simply select a face from the first electrode before launching the command.

Electrode/mandrel assembly process

The **Work** document features the new **Electrode/mandrel assemblies** process that automates the generation of assembly documents. This automated system creates an assembly document for each electrode featured in the Electrodes document and incorporates both the electrode and its corresponding mandrel.

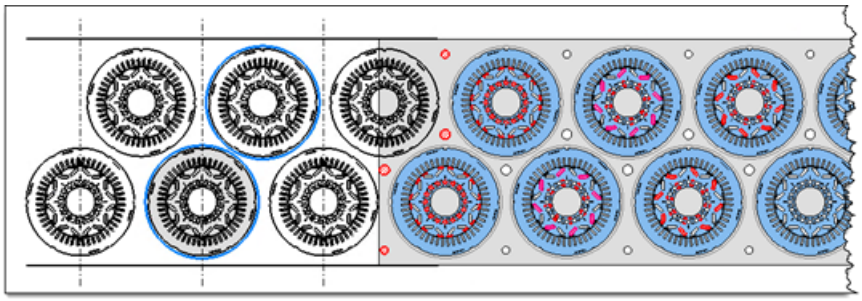
TopSolid'Strip (Strip design)

Strip

Imprint section

In the advanced options of the **Strip** command, the new **Imprint section** option allows you to imprint a section around each station during strip creation, reducing computing times when creating punches.

The section must be drawn around the shapes to be striped.



When the **Imprint section** and **With simplified skeleton** advanced options are coupled, it is possible to improve the overall creation time for the strip and its punchings by around 50%.

Without thickness

In the advanced options of the **Strip** command, the new **Without thickness** option creates a surface strip. This allows you to create strips from surface station shapes for quotations, for example.

TopSolid'Progress (Progressive die)

Head

Initialization of settings

To facilitate and speed up the creation of similar punch heads, you can initialize the parameters and settings of the **Head** command, based on those used to create the first head.

Simply select a face from the first head before launching the command.

Springs

Initialization of settings

To facilitate and speed up the creation of similar springs, you can initialize the parameters and settings of the **Springs** command, based on those used to create the first spring.

Simply select a face from the first spring before launching the command.

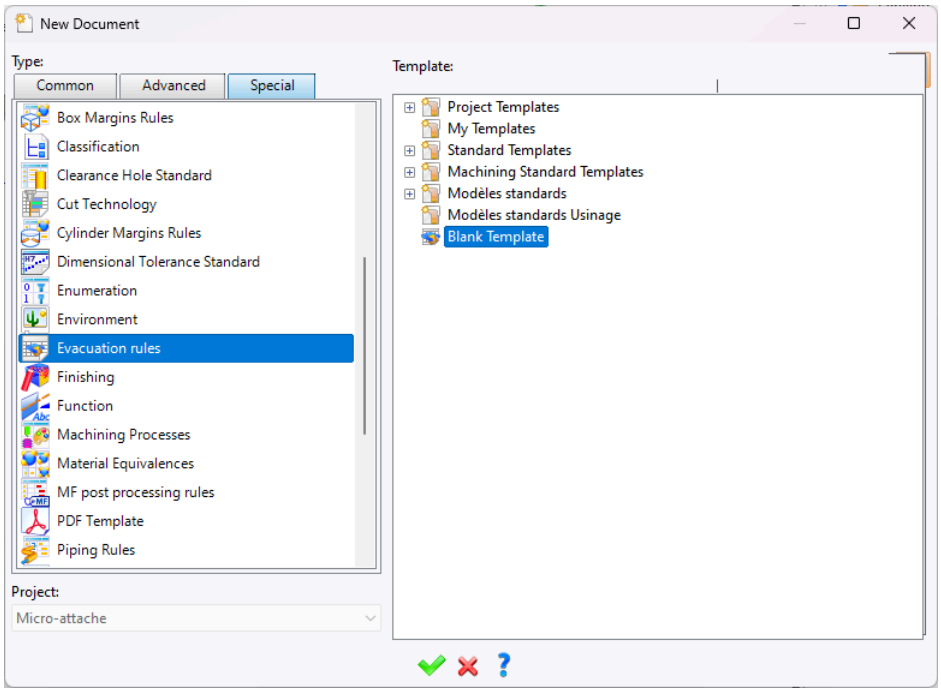
What's New in TopSolid'Cut 7.18

This chapter describes what's new in the **sheet metal machining** application of **TopSolid 7** software in version **7.18**.

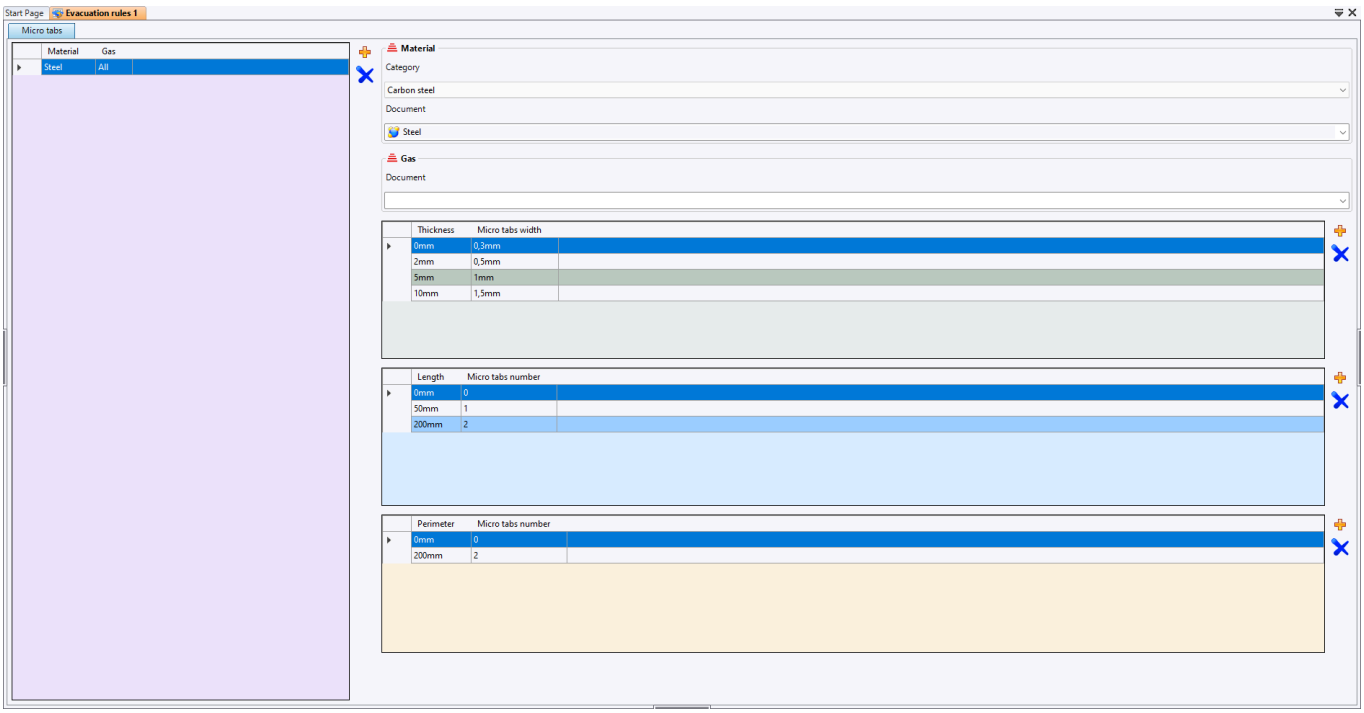
Microtabs

Evacuation rules document

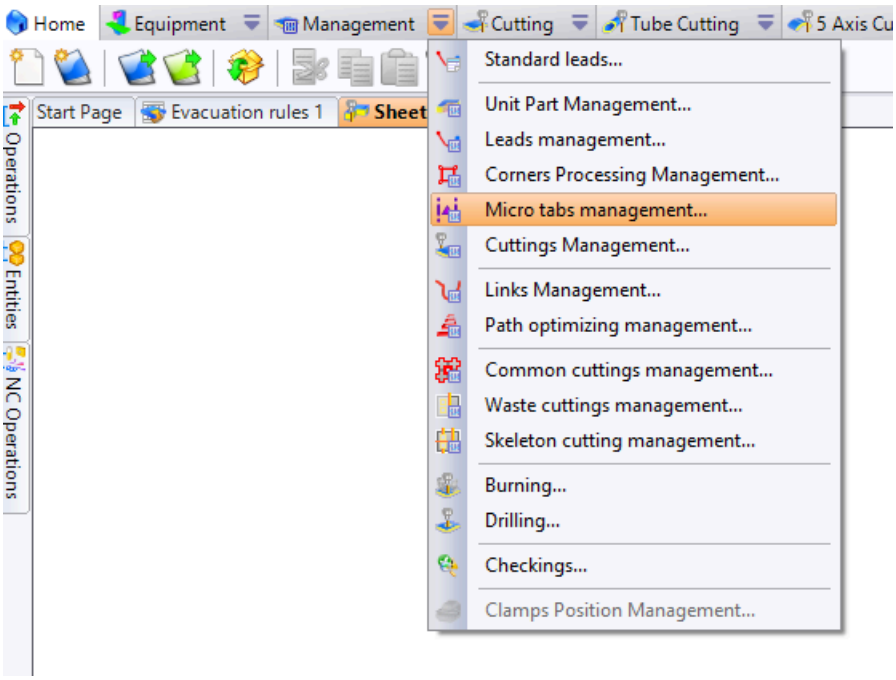
Important changes have been made to the microtabs, including the introduction of the new **Evacuation Rules** document. This document allows you to define a microtab width based on a range of thicknesses. Additionally, it specifies the number of microtabs considering both the size of the selected element and the perimeter of the chosen contour.



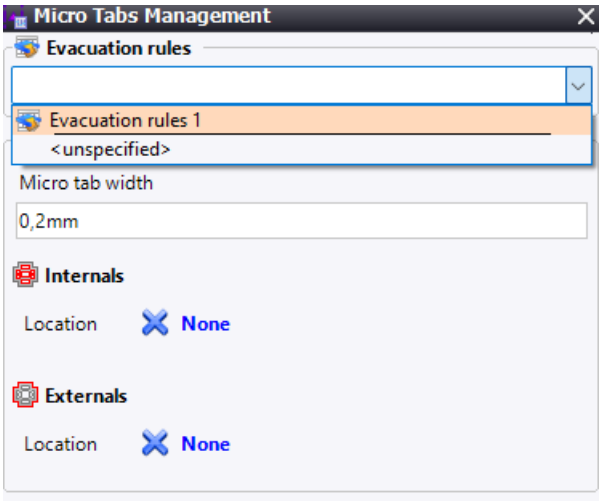
The **Evacuation rules** document is presented as follows and can be completed according to your preferences.



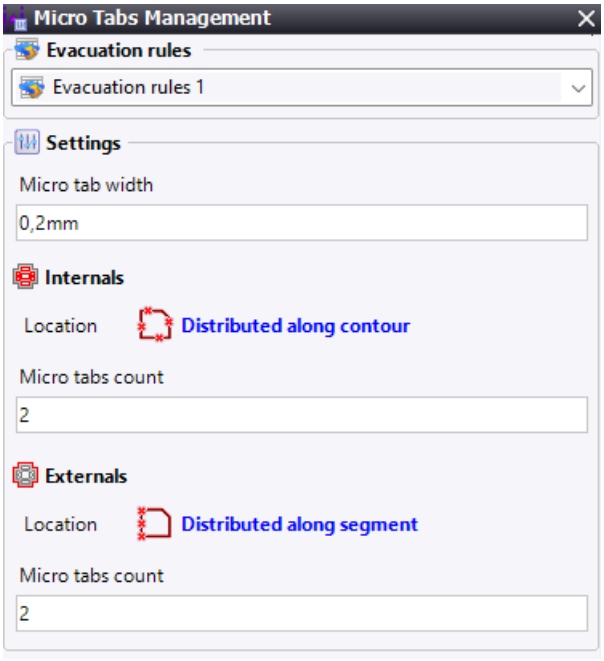
Once the document has been completed and saved, it can be selected in a sheet metal CAM document, via the **Management** > **Micro Tabs Management** command.



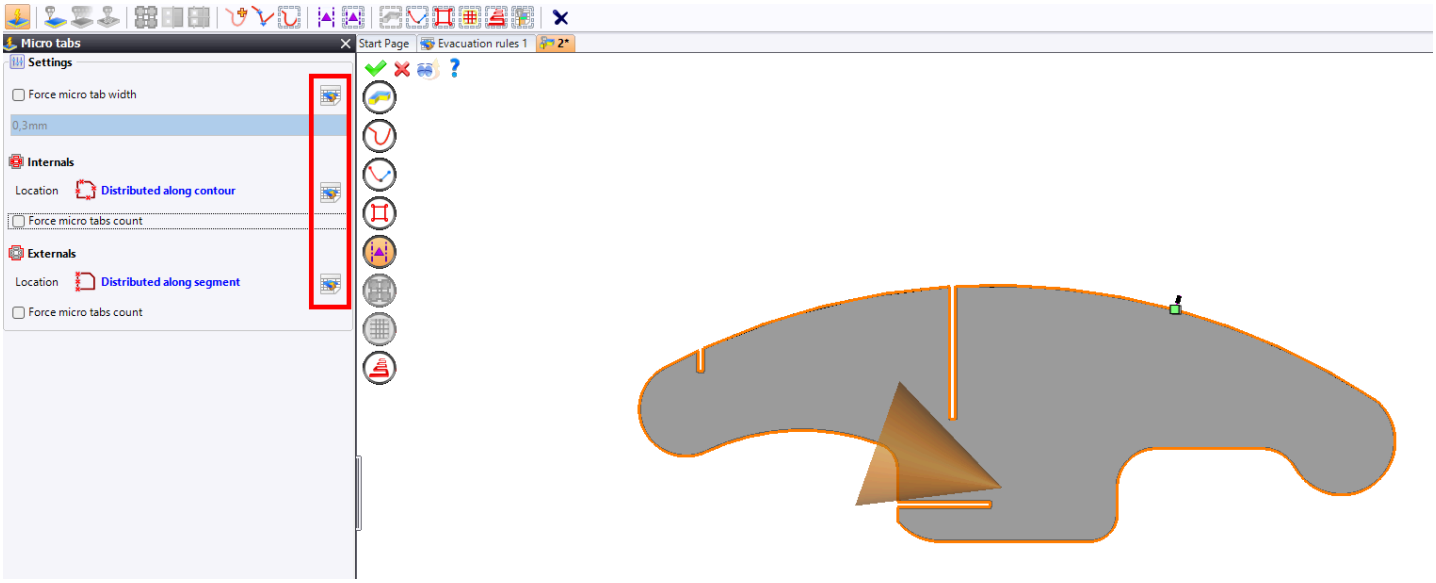
For the evacuation rules document to be available for selection, its location must be referenced in the current project.



In the evacuation rules document, length and perimeter rules are used by choosing to apply microtabs distributed over each segment for length and distributed over the entire contour for perimeter.

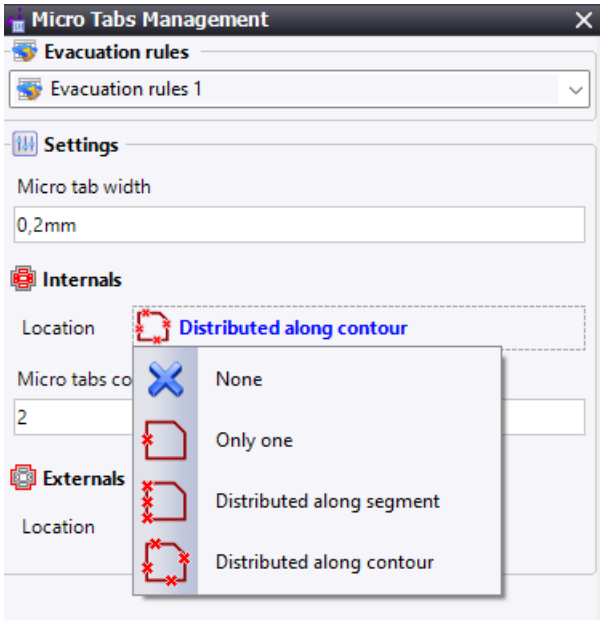


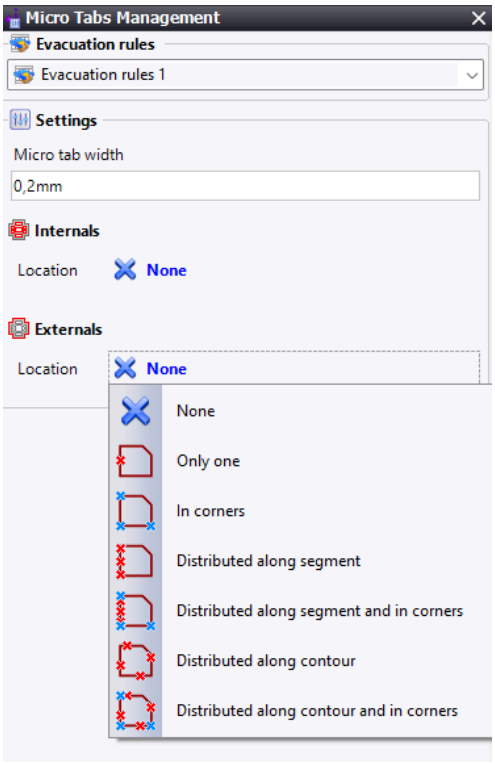
An icon matching the evacuation rules is displayed in the **Micro Tabs** option from the **Automatic Machinings** command or in the **Micro Tabs** command. It indicates that microtabs will be allocated according to the parameters defined in the evacuation rules document. If the icon does not appear, the value displayed is the one set in the **Micro Tabs Management** command.



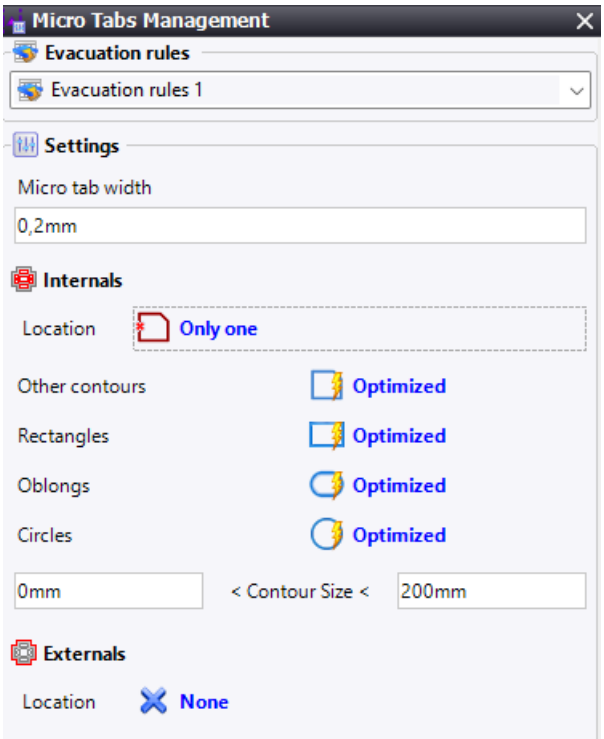
Positioning microtabs

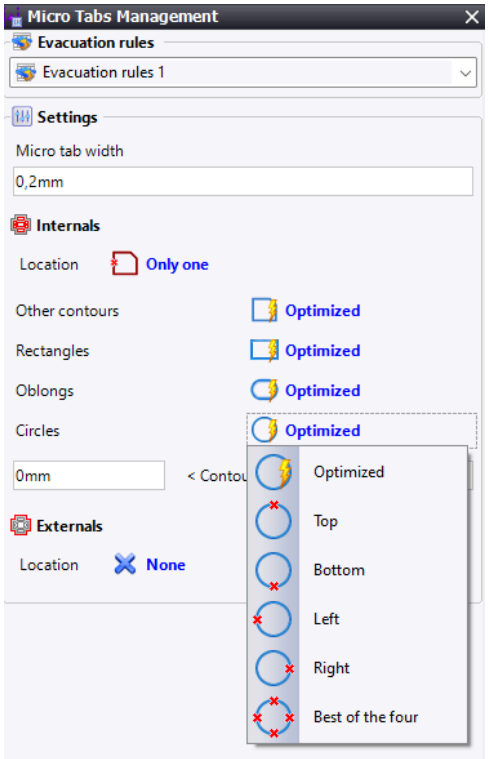
New modes for better positioning of microtabs are available in the **Micro Tabs Management** and **Automatic Machinings** commands.





New configurations are also available, giving you more possibilities for positioning the microtabs.

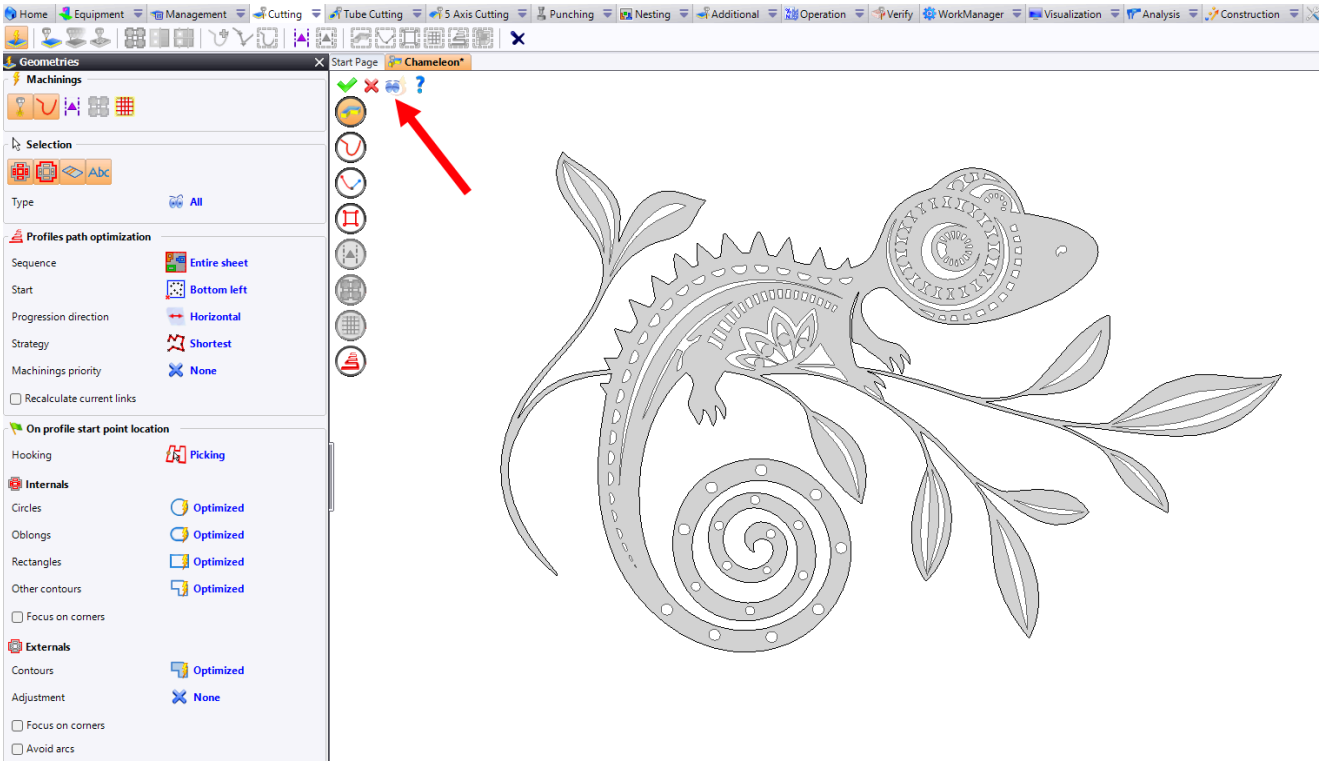




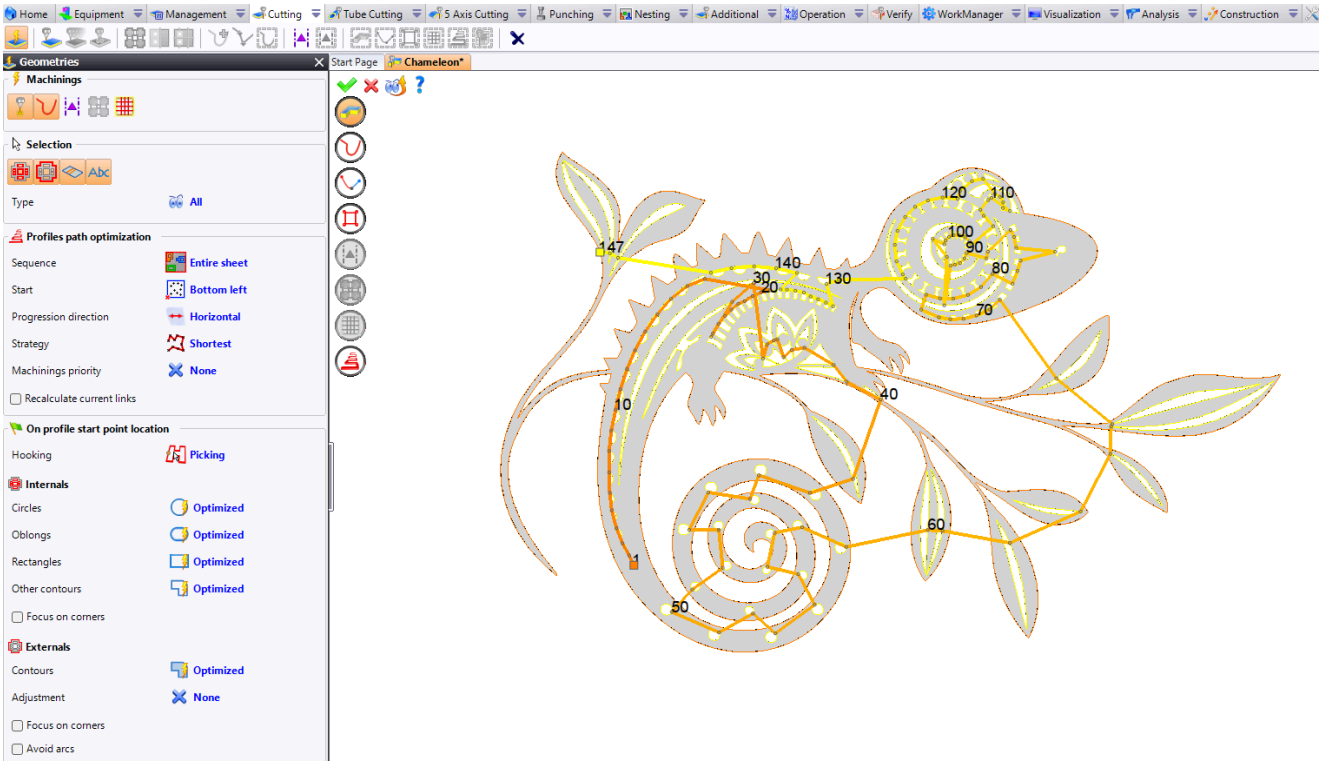
Automatic machining

Cut preview

A cut preview is now available in the **Automatic Machinings** command, allowing you to preview the future cut and see the result of the operation in advance.



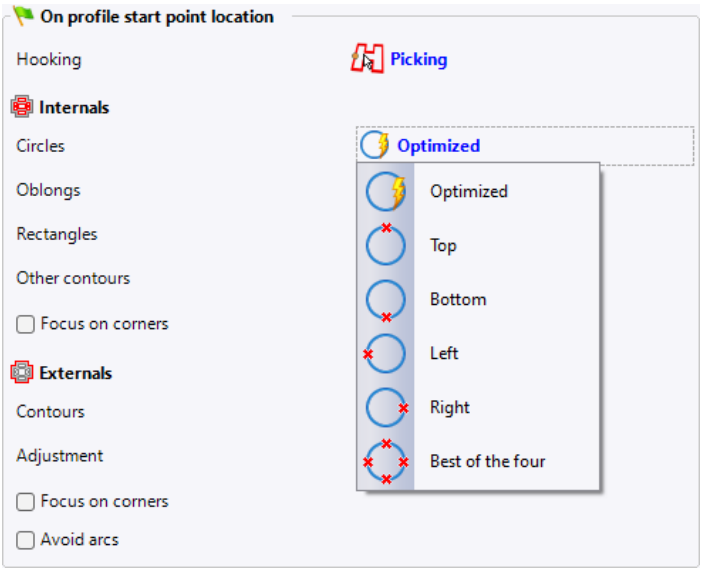
The different numbers represent the machining toolpath, with 1 corresponding to the starting point.

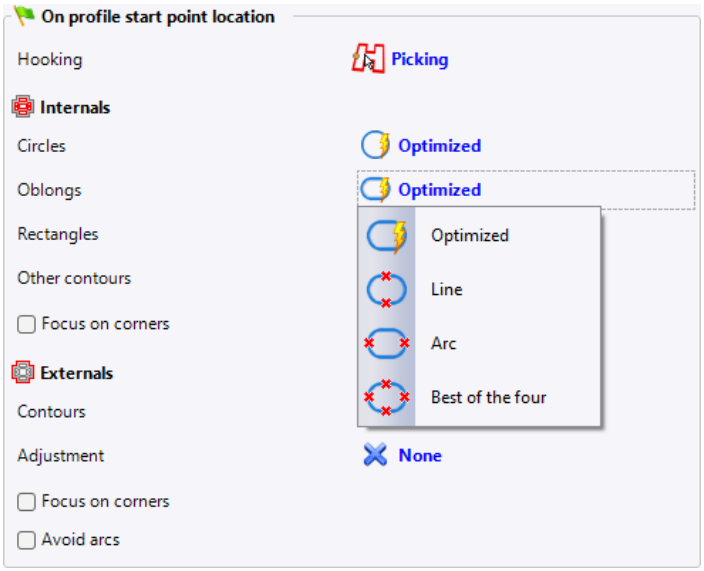


Start point

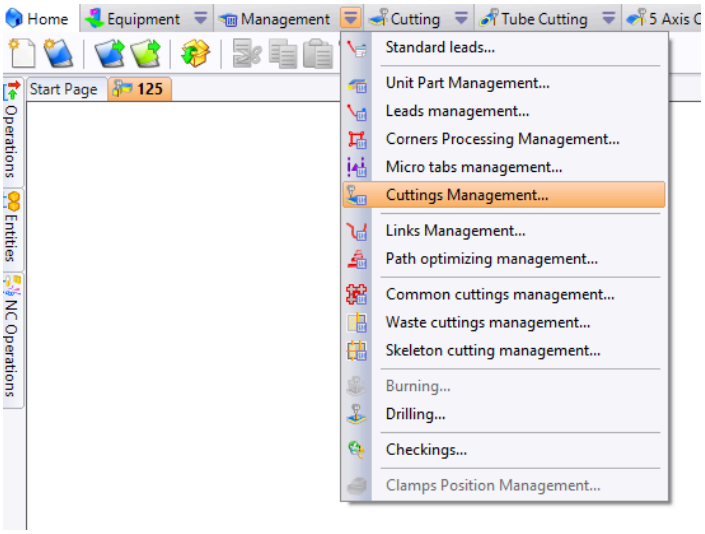
Starting point configuration is now possible for both internal and external contours.

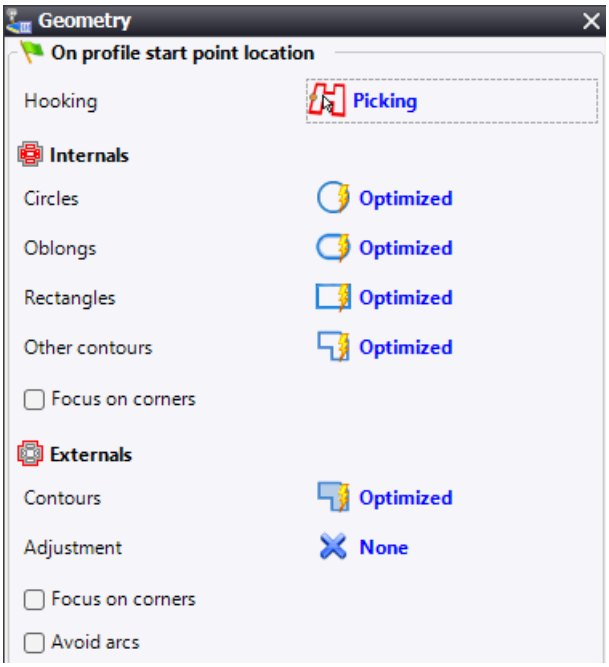
You can customize the starting point from different shapes. These new options also work with leads.





It is also possible to configure these options in the **Cuttings Management** command.

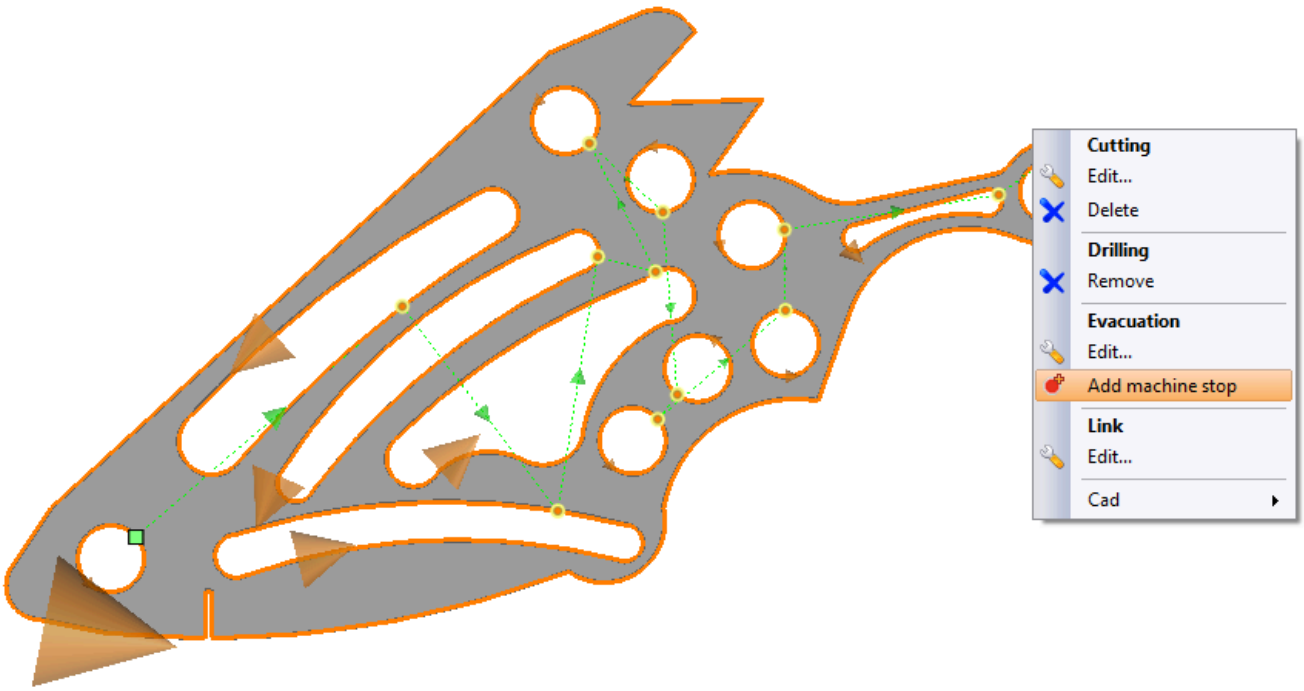




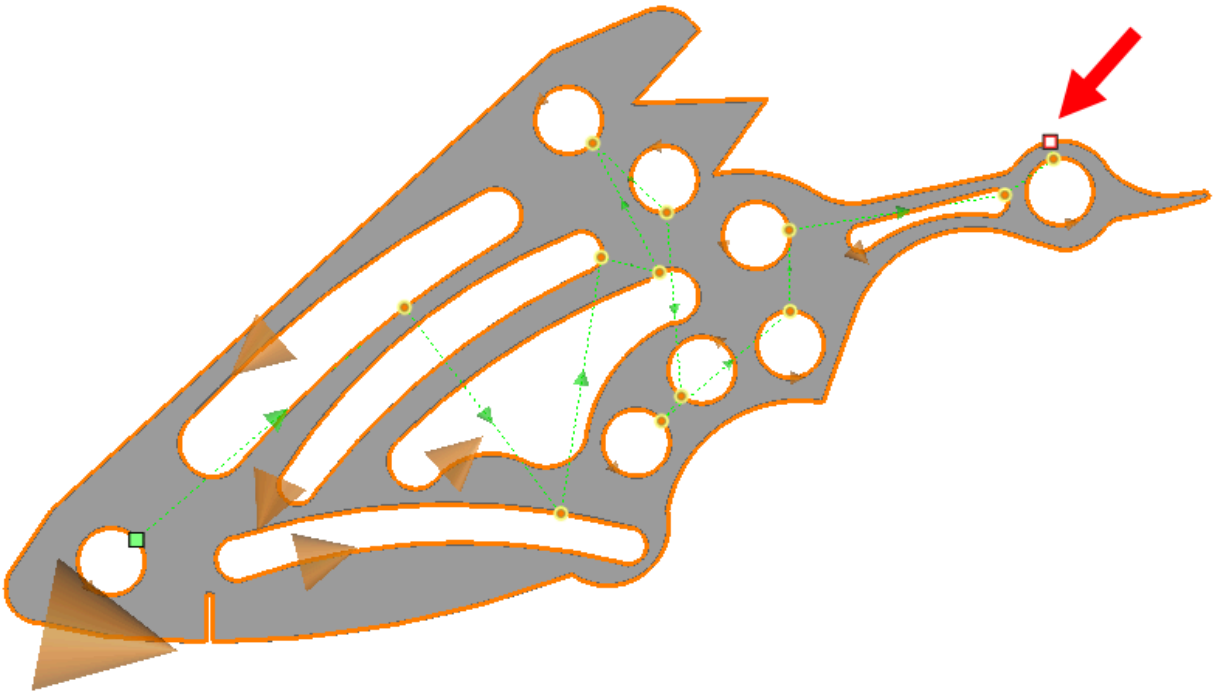
Note that in all cases, concave corners are systematically avoided. If the point selected by **TopSolid** is a concave corner, the nearest key point is automatically preferred.

Machine stop

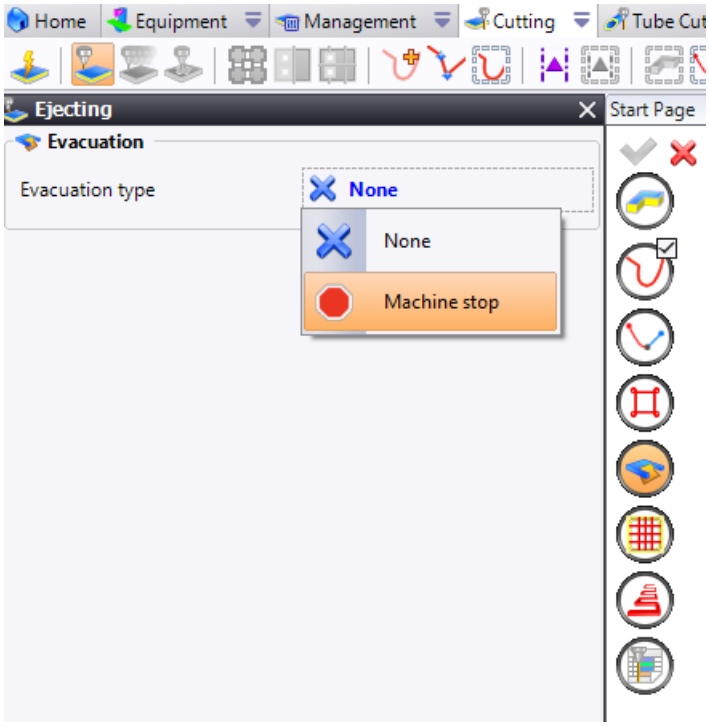
You can now add machine stops to almost all 2D cutting elements. To do this, right-click on a cutting or other 2D cutting element and select the **Add Machine Stop** command from the context menu. The machine stop is then positioned on the starting point for cutting.



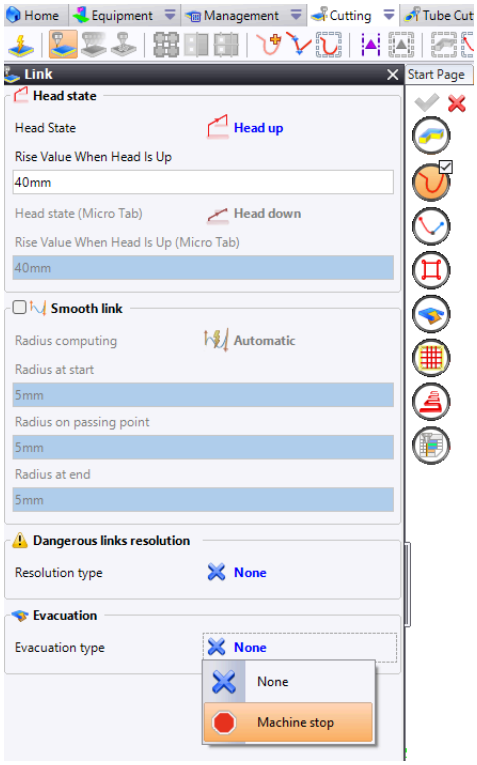
The machine stop is represented by a red square. It can be deleted via the context menu.



You can also add a machine stop manually, directly from the **Evacuation** option.



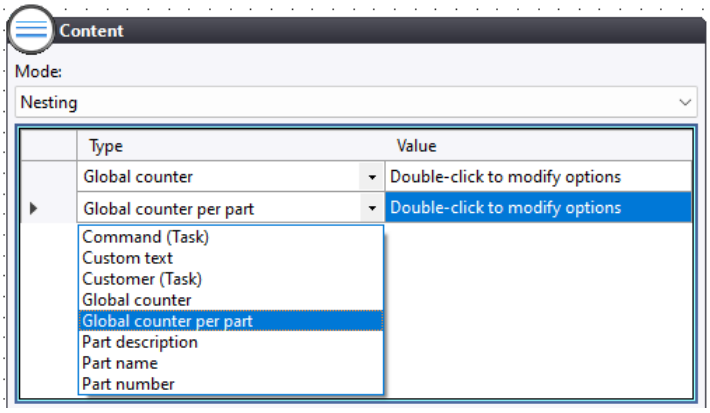
Machine stops can also be added to links.



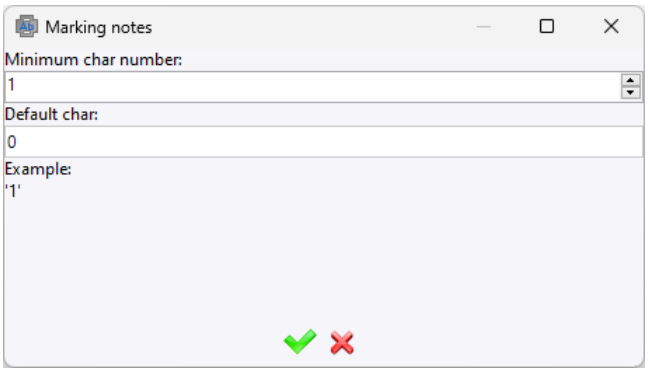
Marking

Counters

Marking has been improved and offers two new options: **Global counter** and **Global counter per part**.



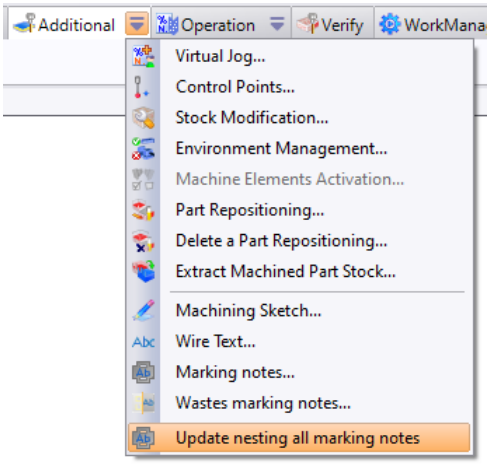
To set the counter format, simply double-click in the cell. It is then possible to define the minimum number of characters and the completion character.

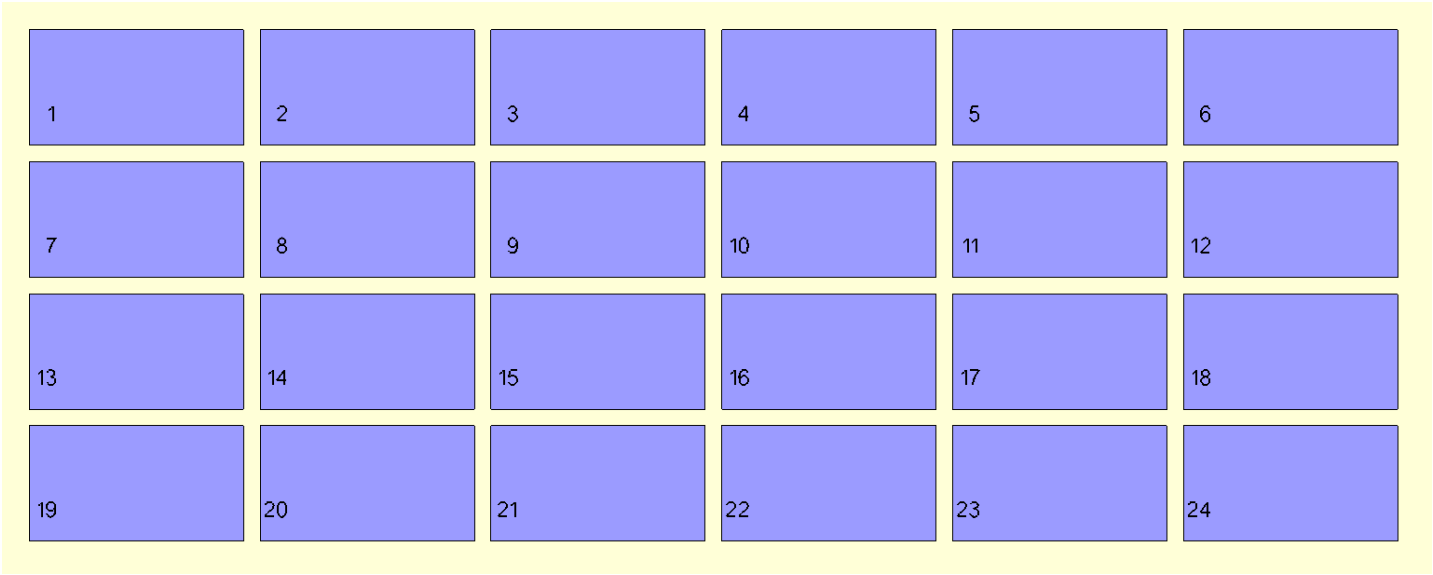


These options are only available in a nesting context, i.e. in the document template in nesting mode or in a document resulting from nesting.

The global counter increments parts without distinguishing them, while the global per part counter simply numbers each part type from 1 to N. The counters induce a dependency of each sheet metal part from those preceding it in the nesting.

In addition, the new **Update Nesting all Marking Notes** command forces the recalculation of all nesting marking operations. It is only available in the **Additional** menu of the machining document.



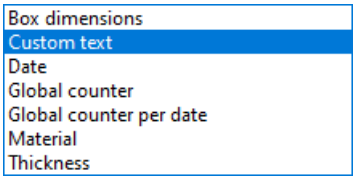


Marked nesting with the global counter.

Waste marking notes

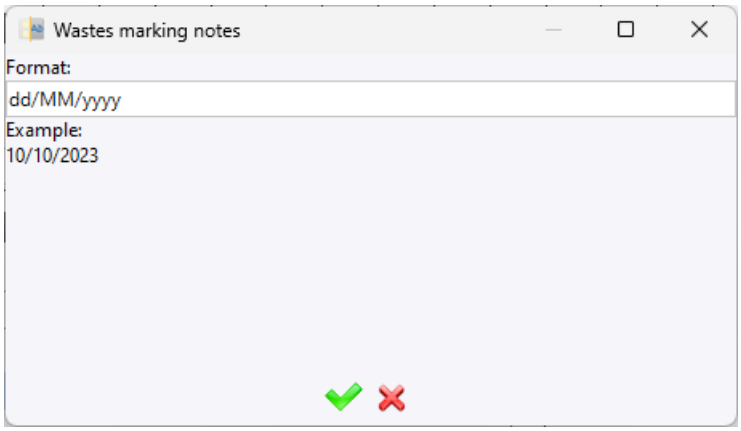
The new **Wastes Marking Notes** command is available in the **Additional** menu of a template document or a sheet metal cam document coming from a nesting, if a waste cutting operation has been performed.

Several options are available for writing.

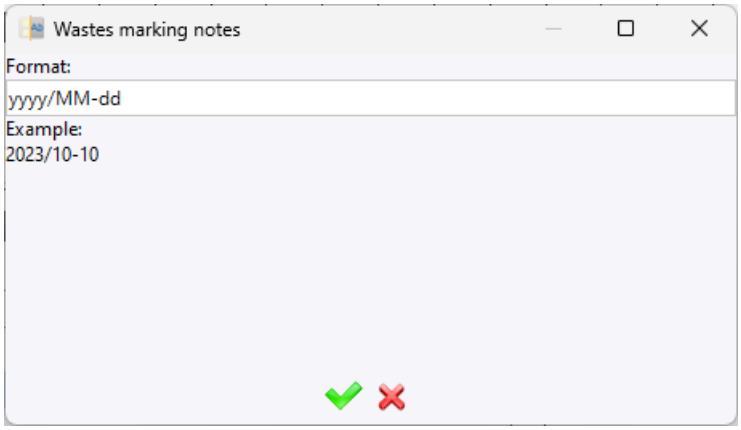


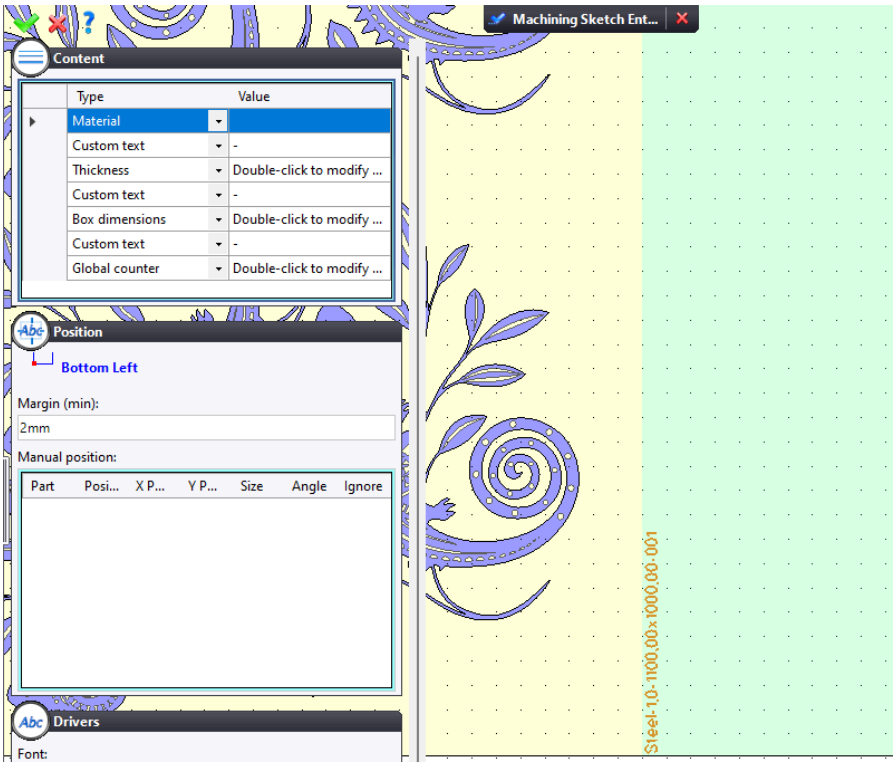
Here is the format for the elements used to indicate the date:

- **dd** matches the day;
- **mm** matches the month;
- **yyyy** matches the year.



The order of the elements can be customized from your requirements.



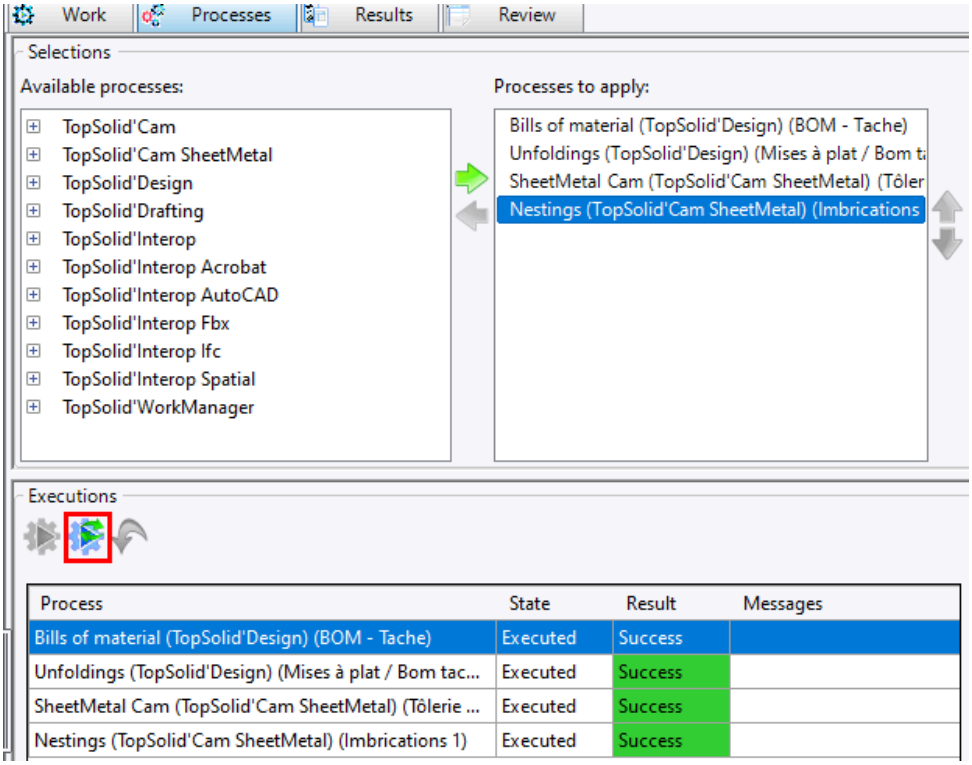


Example of waste marking.

Work document

Reexecution

The new **Reexecute** option is available in the **Work** document.

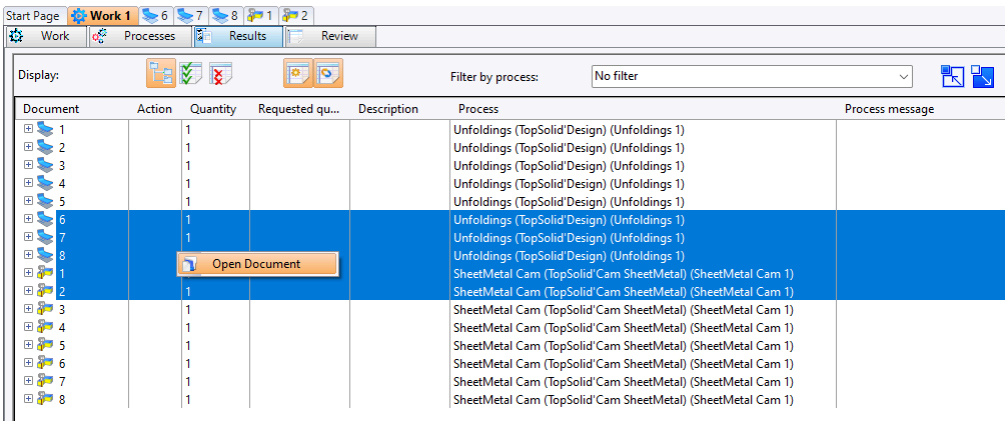


It operates on the following principles:

- If a document has already been created by the previous execution, it will not be recreated, but simply updated. Updating may involve a simple replay of operations, or the launch of additional operations.
- If a document is missing, it will be created and processed like a classic execution. For example, if you generate a drafting from an assembly BOM, and you have added parts to the assembly, new draftings will be created for the new parts.
- If a document must be deleted, it will be. For example, if you generate a drafting from the BOM of an assembly, and you have deleted parts from the assembly, the draftings for these parts will be destroyed.

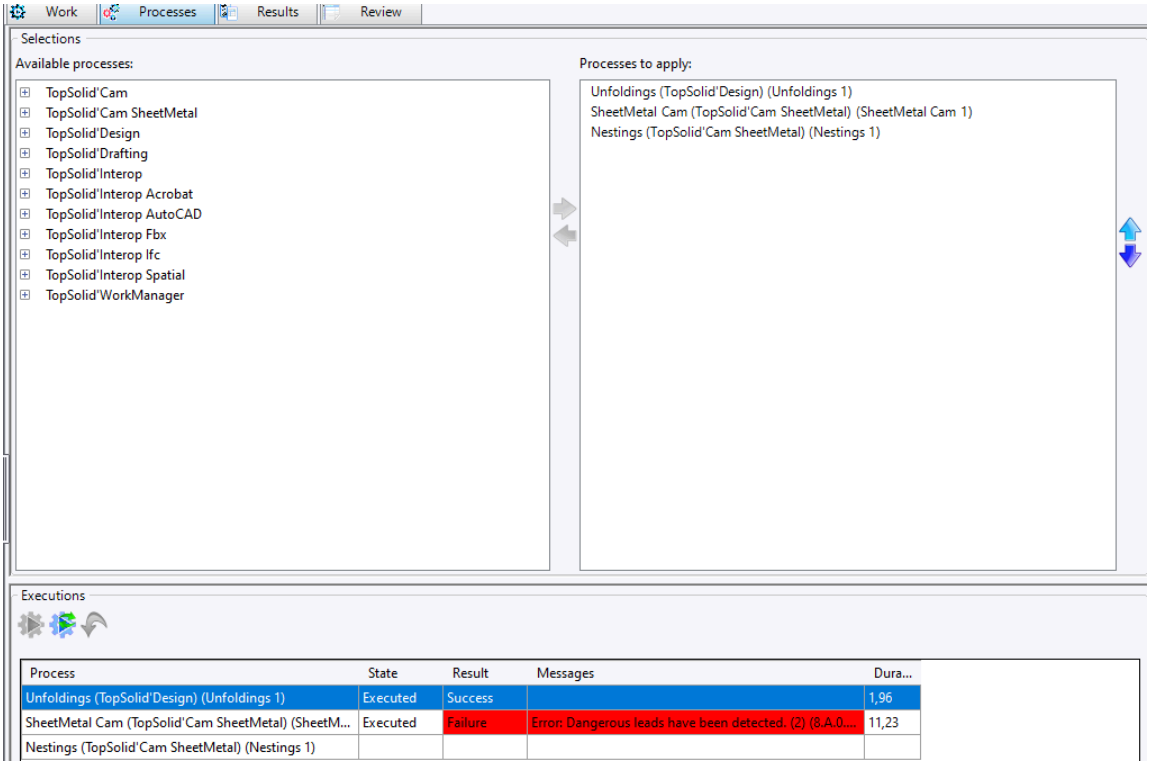
Multiple selection

Multiple selection is now possible in the **Results** tab.

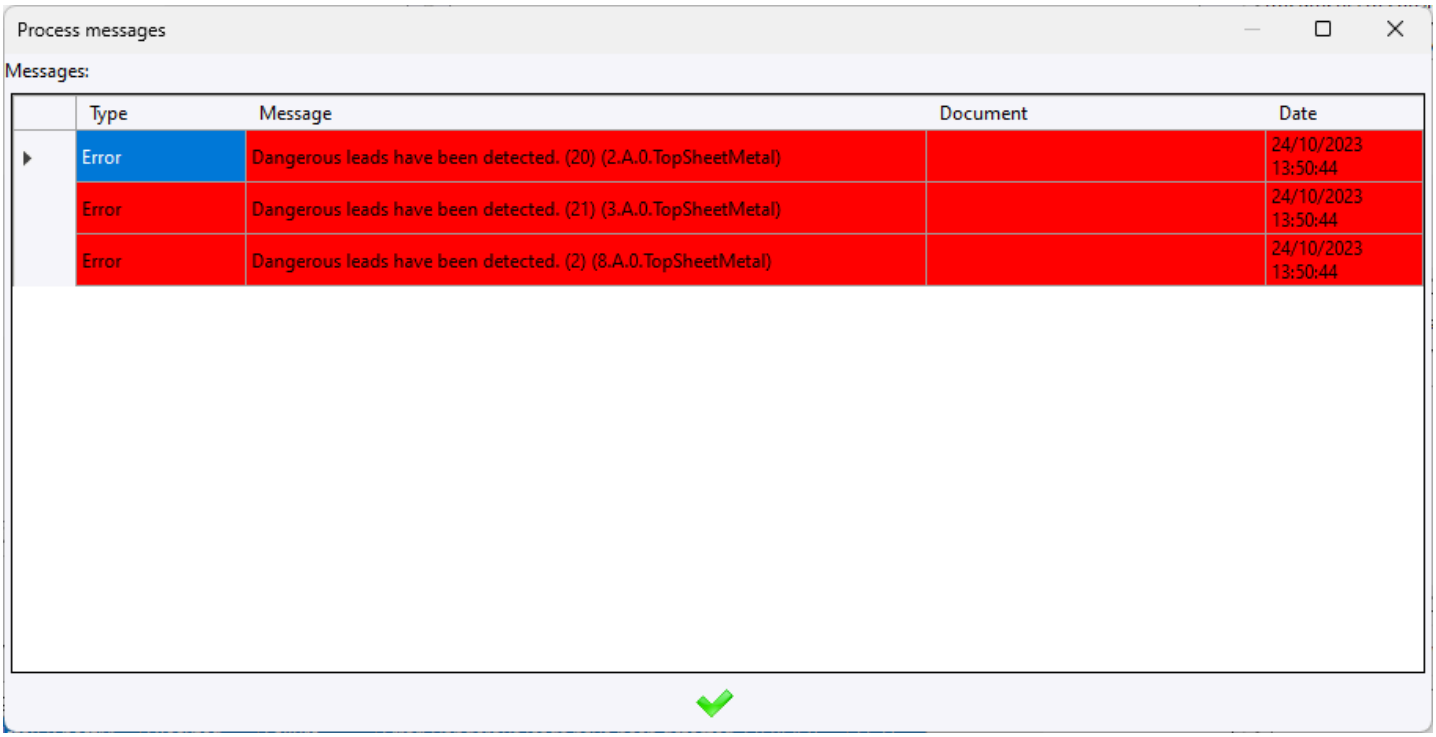


Displaying invalidities

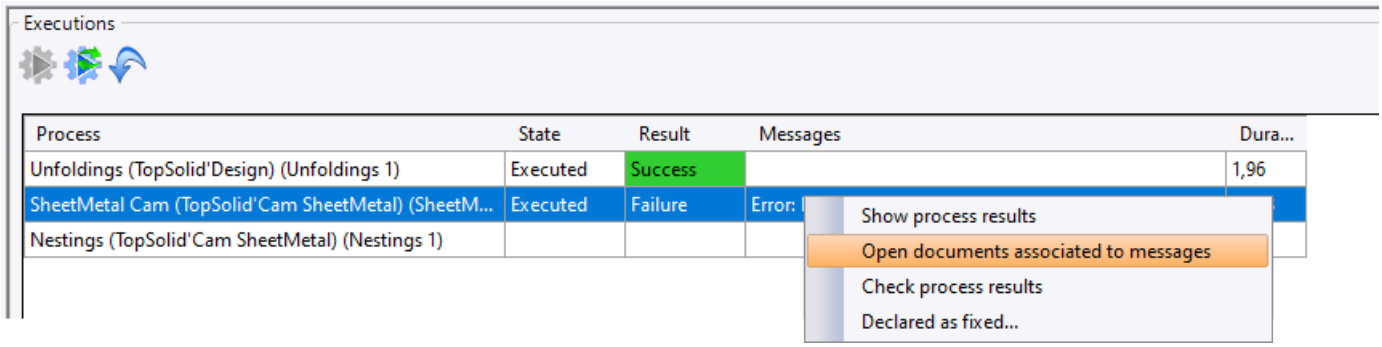
Error messages associated with processes now pinpoint the problematic document(s) for clearer identification.



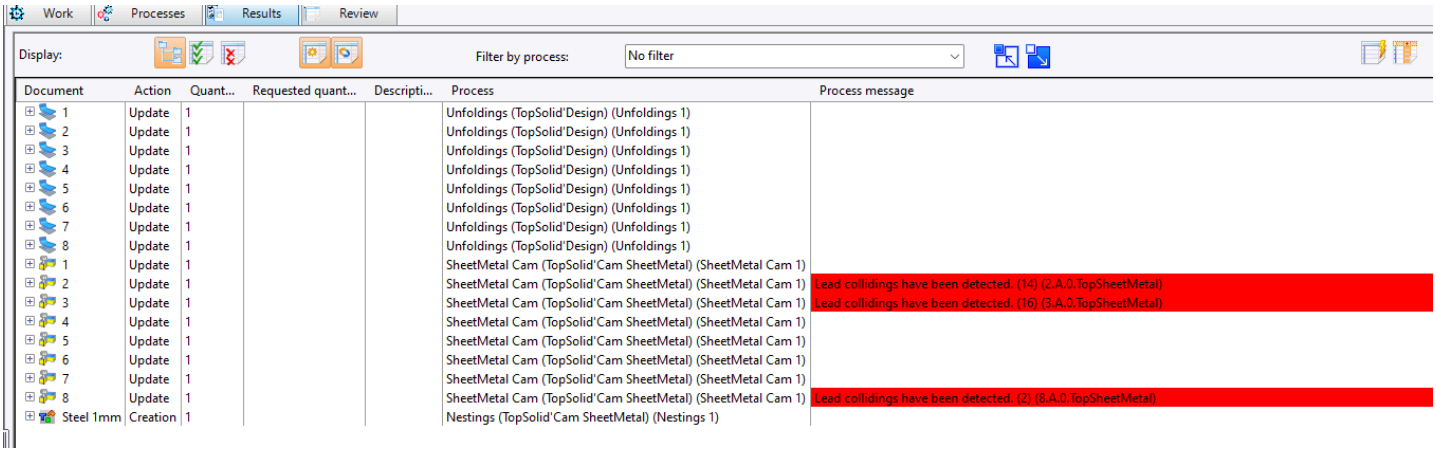
By clicking on the error message (the red line), the window below appears and displays the list of incorrect documents.



You can open documents via the **Open documents associated to messages** context command.



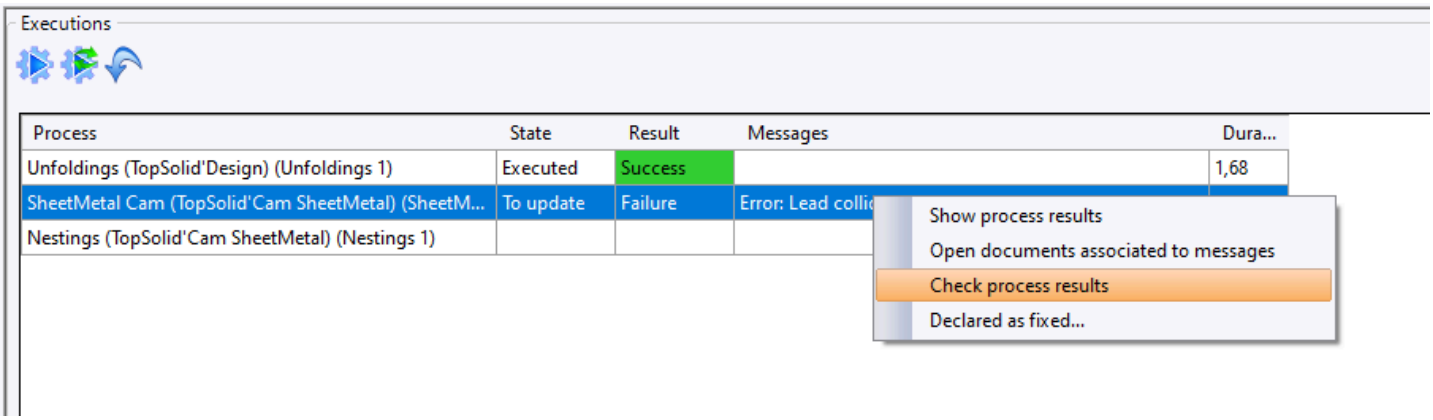
Error messages are also present in the new **Process message** column from the **Results** tab.



You can choose to display or hide this column, and define its position via the configuration of processing result columns command.

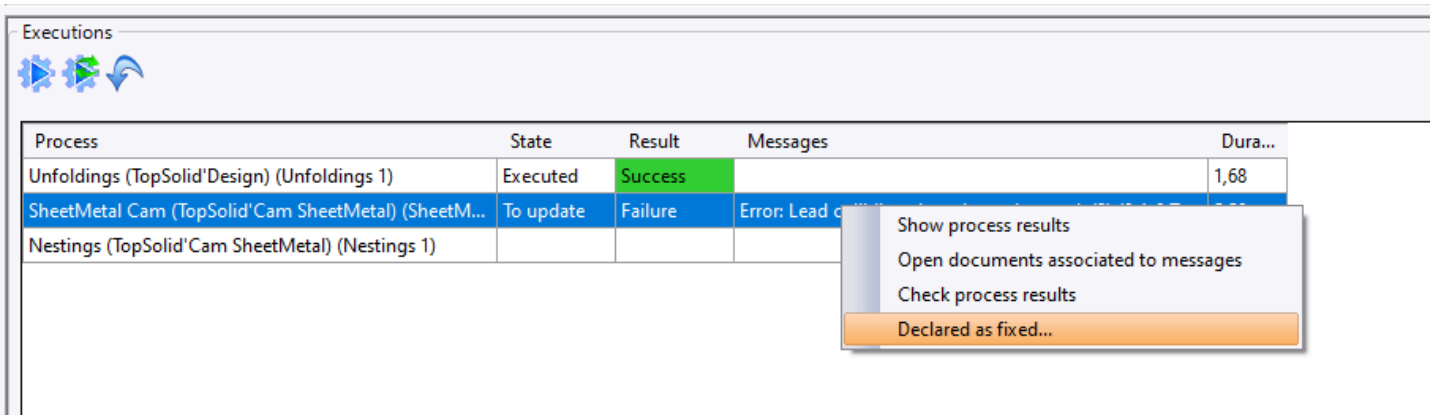
Error correction management

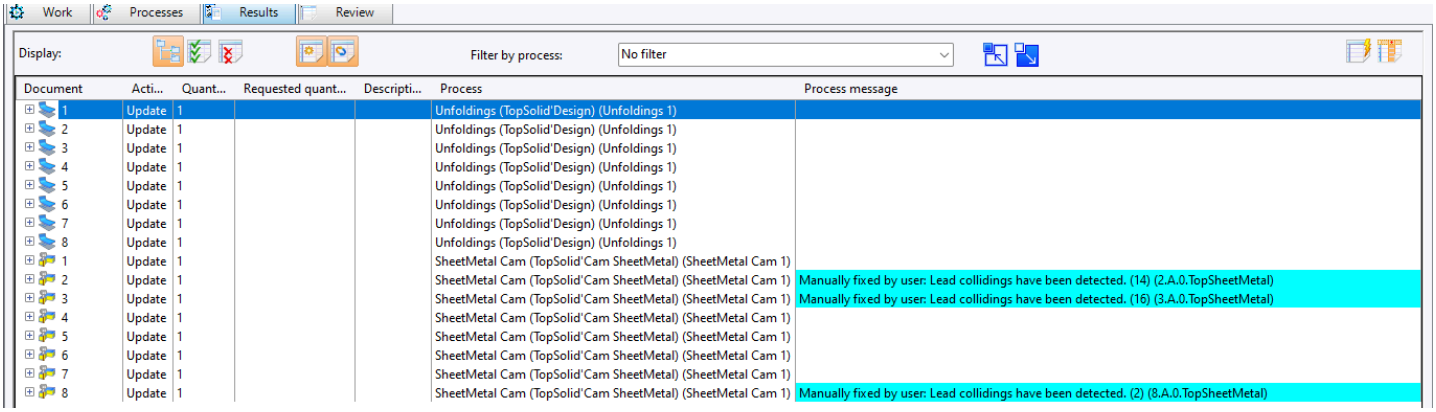
The new **Check process results** command enables users to check a process after implementing corrections. It can be accessed from the context menu on an invalidity.



If all errors are corrected, the process returns to **Success** state and execution can continue.

In other cases, or when the error correction cannot be verified, it is always possible to declare the process as corrected.


Executions
Process
State
Result
Messages
Dura...
Unfoldings (TopSolid'Design) (Unfoldings 1)
Executed
Success
1,68
SheetMetal Cam (TopSolid'Cam SheetMetal) (SheetM...
To update
Failure
Error: Lead c
Nestings (TopSolid'Cam SheetMetal) (Nestings 1)
Show process results
Open documents associated to messages
Check process results
Declared as fixed...


Work
Processes
Results
Review
Display:
Filter by process:
No filter
Document
Acti...
Quant...
Requested quant...
Descripti...
Process
Process message
1 Update 1 Unfoldings (TopSolid'Design) (Unfoldings 1)
2 Update 1 Unfoldings (TopSolid'Design) (Unfoldings 1)
3 Update 1 Unfoldings (TopSolid'Design) (Unfoldings 1)
4 Update 1 Unfoldings (TopSolid'Design) (Unfoldings 1)
5 Update 1 Unfoldings (TopSolid'Design) (Unfoldings 1)
6 Update 1 Unfoldings (TopSolid'Design) (Unfoldings 1)
7 Update 1 Unfoldings (TopSolid'Design) (Unfoldings 1)
8 Update 1 Unfoldings (TopSolid'Design) (Unfoldings 1)
1 Update 1 SheetMetal Cam (TopSolid'Cam SheetMetal) (SheetMetal Cam 1)
2 Update 1 SheetMetal Cam (TopSolid'Cam SheetMetal) (SheetMetal Cam 1) Manually fixed by user: Lead collidings have been detected. (14) (2.A.0.TopSheetMetal)
3 Update 1 SheetMetal Cam (TopSolid'Cam SheetMetal) (SheetMetal Cam 1) Manually fixed by user: Lead collidings have been detected. (16) (3.A.0.TopSheetMetal)
4 Update 1 SheetMetal Cam (TopSolid'Cam SheetMetal) (SheetMetal Cam 1)
5 Update 1 SheetMetal Cam (TopSolid'Cam SheetMetal) (SheetMetal Cam 1)
6 Update 1 SheetMetal Cam (TopSolid'Cam SheetMetal) (SheetMetal Cam 1)
7 Update 1 SheetMetal Cam (TopSolid'Cam SheetMetal) (SheetMetal Cam 1)
8 Update 1 SheetMetal Cam (TopSolid'Cam SheetMetal) (SheetMetal Cam 1) Manually fixed by user: Lead collidings have been detected. (2) (8.A.0.TopSheetMetal)

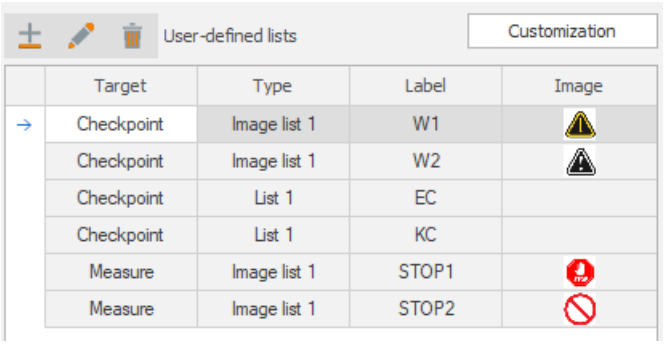
What's New in TopSolid'Inspection 7.18

This chapter describes what's new in **Process**, **Ergonomics**, **Indicators** and **Interface** for **TopSolid'Inspection Creator**, **Controller** and **Analyst** applications in version **7.18** of **TopSolid 7**.

Process

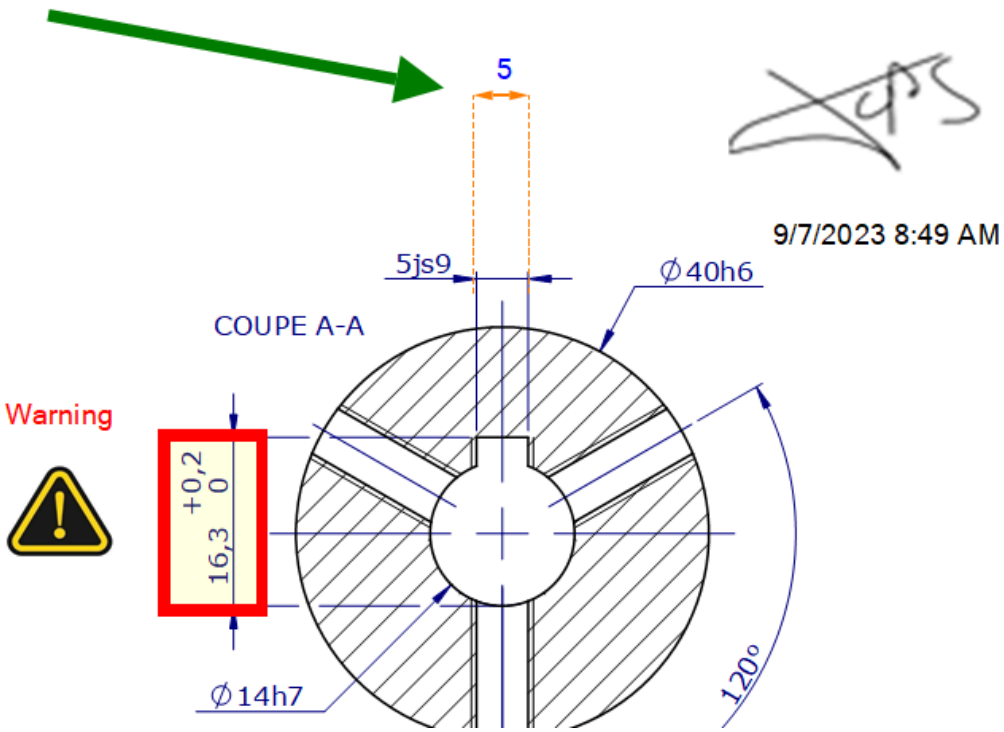
Customizable additional data

Additional data such as free text, choice lists and image lists can be **customized** and complete the control plan at the **checkpoints** and **measures** levels, to take into account the specific features of your quality processes.



PDF annotations

Annotations such as text, images, shapes, signatures and dimension lines can now be added to PDF drawings, providing the workshop with **added value drawings** containing all the information required for the checkings to run smoothly.



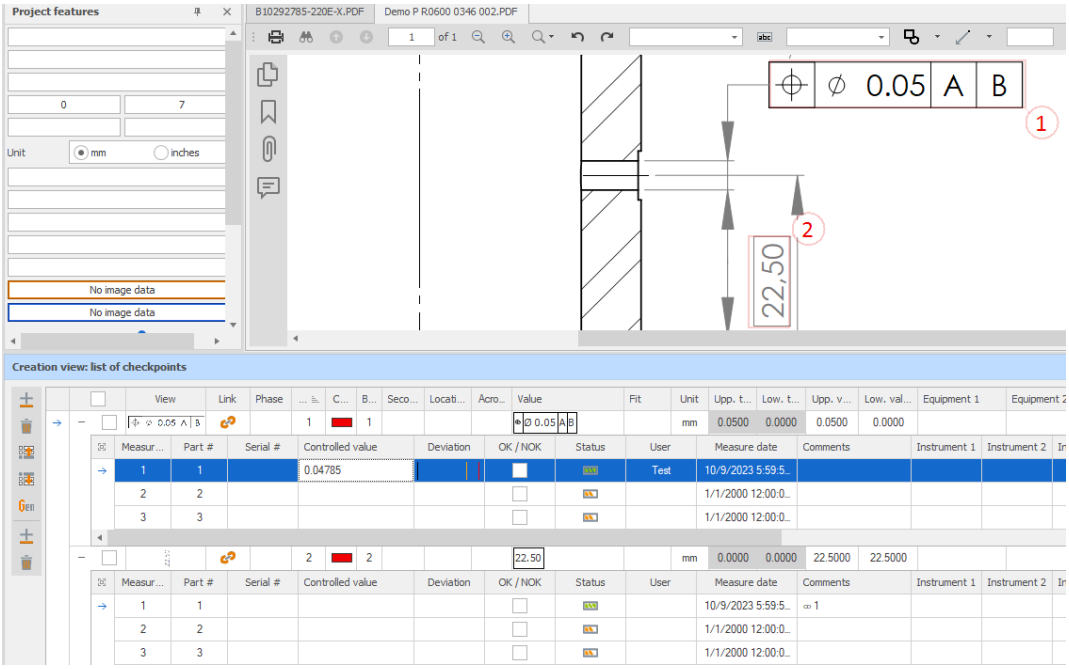
Linked and theoretical exact dimensions

Improvements have been made in order to **avoid unnecessary checkings**.

The principle of linked dimensions allows you to link several dimensions together to minimize the number of checkings and manage theoretical exact dimensions.

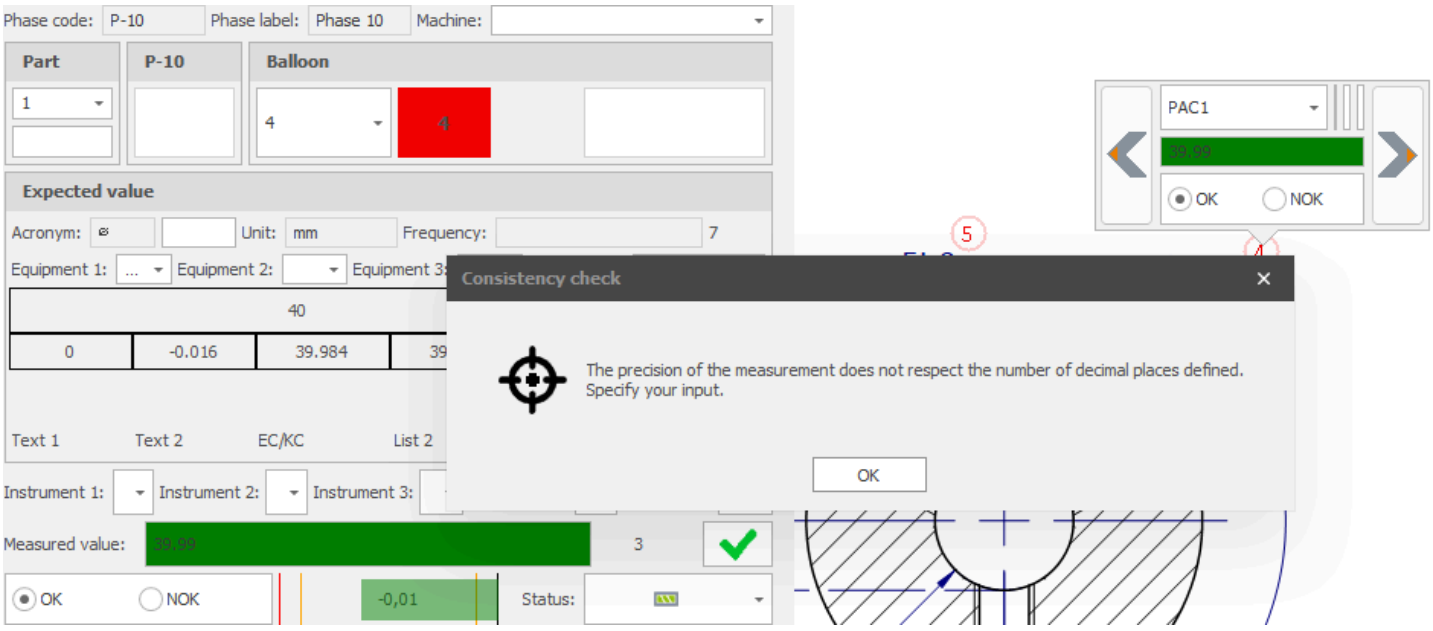
Measuring a non-ISO GPS linked dimension applies the same conformity status to associated dimensions with an equal or greater tolerance deviation.

Measuring an ISO GPS geometric tolerance applies the same conformity status to the associated theoretical dimensions.



Measurement accuracy

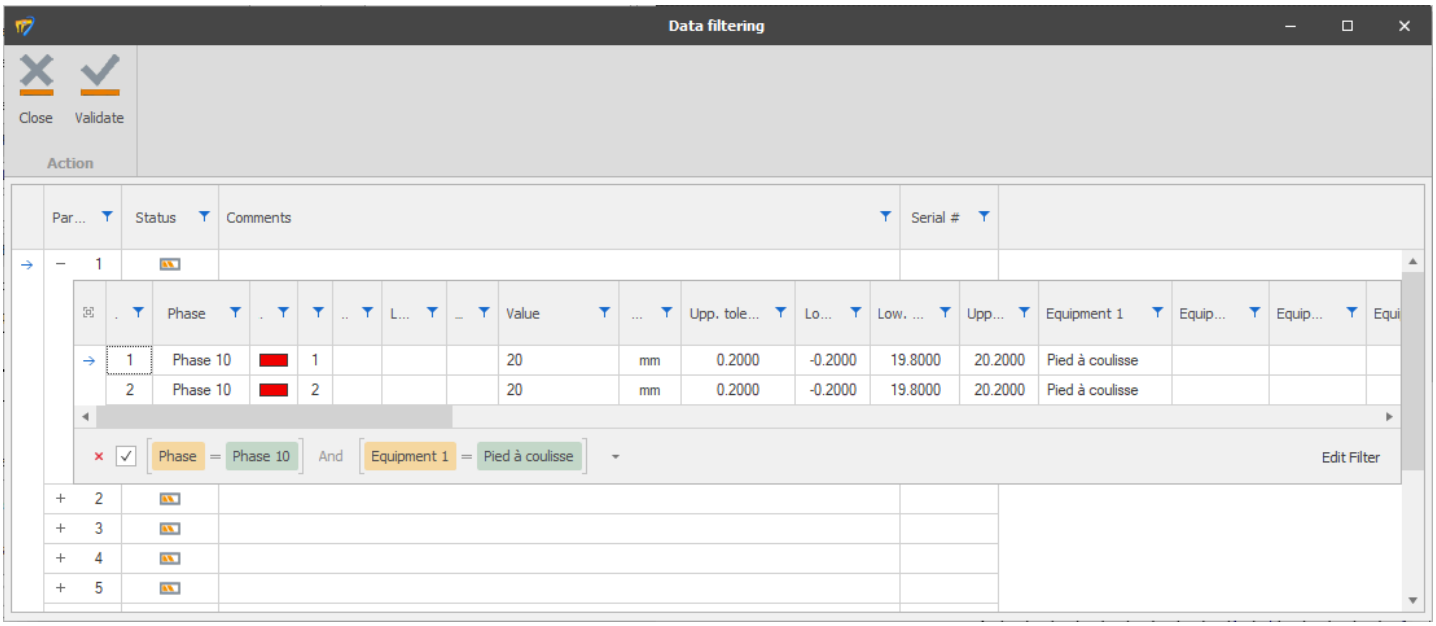
Constraining a measurement accuracy on a dimension prevents lower accuracy inputs. The control means can be parameterized to enforce a specific degree of accuracy and **adhere to quality requirements**.



Ergonomics

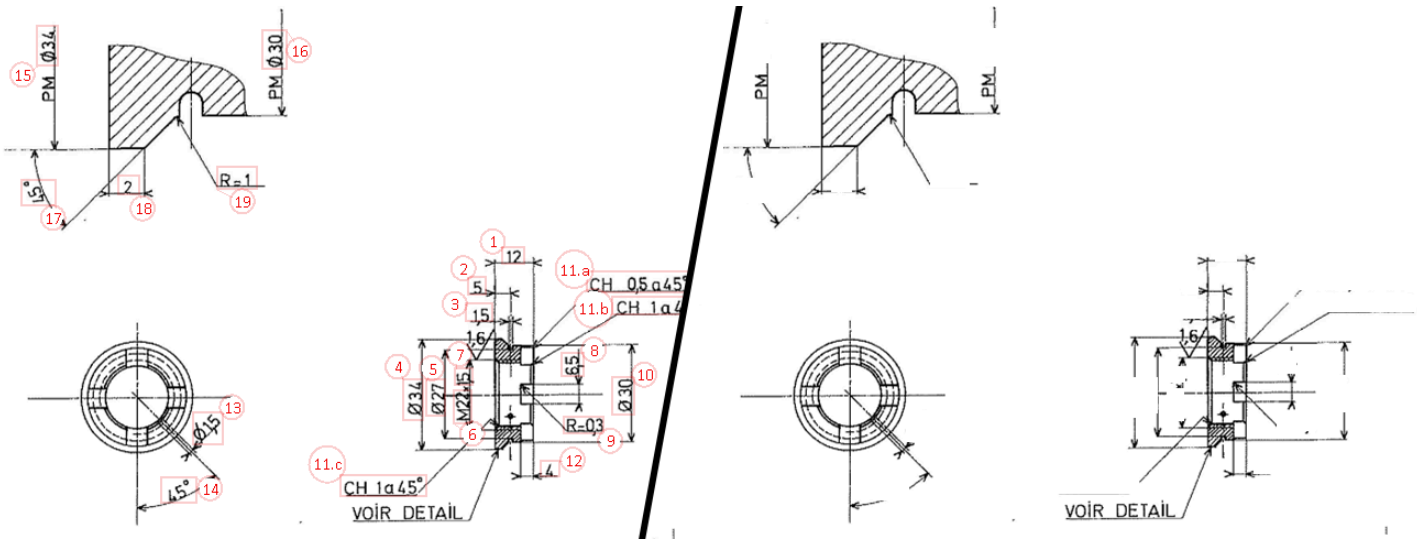
Filtering checkpoints

In the main program and in the **Controller** interface, a filtering system has been implemented for the dimensions to be checked. This facilitates reading, selection from grids and **measurement session organization**.



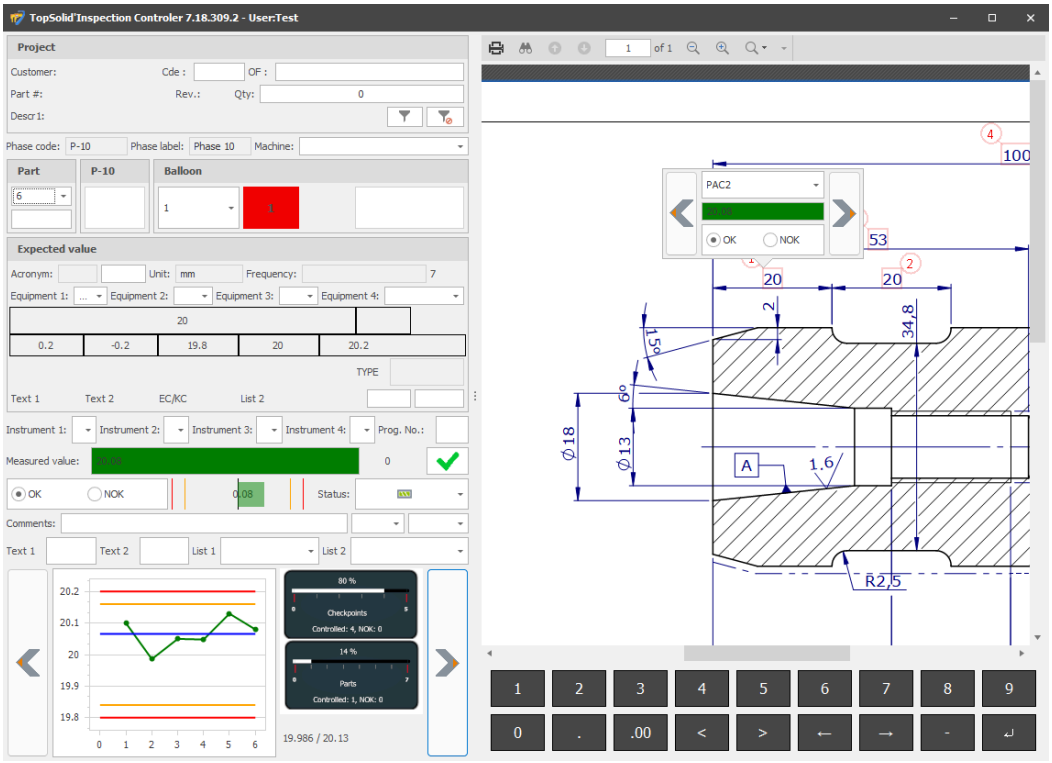
Masking balloons and drawing dimensions

A new feature allows you to "clean" the part drawing with a single click, making balloons and dimensions invisible so that you can more easily assess the remaining items for ballooning ensuring that nothing is overlooked.



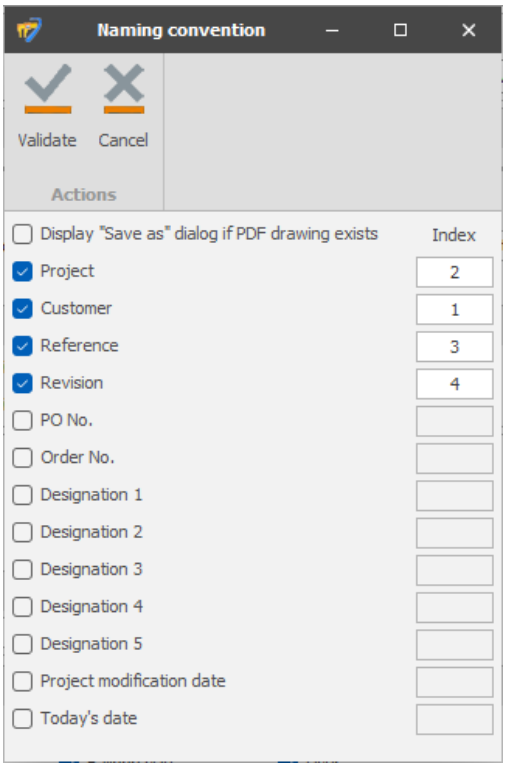
Virtual keyboard

A virtual keyboard for numeric input is now available in **Controller**, **facilitating numerical dimensions inputs through the touch screen** without using the Windows keyboard.



Naming convention

To **make it easier to save projects and exported documents**, a naming convention can be activated and parameterized to define automatic rules for ITL project, ballooned drawing, report export and Excel export file names.

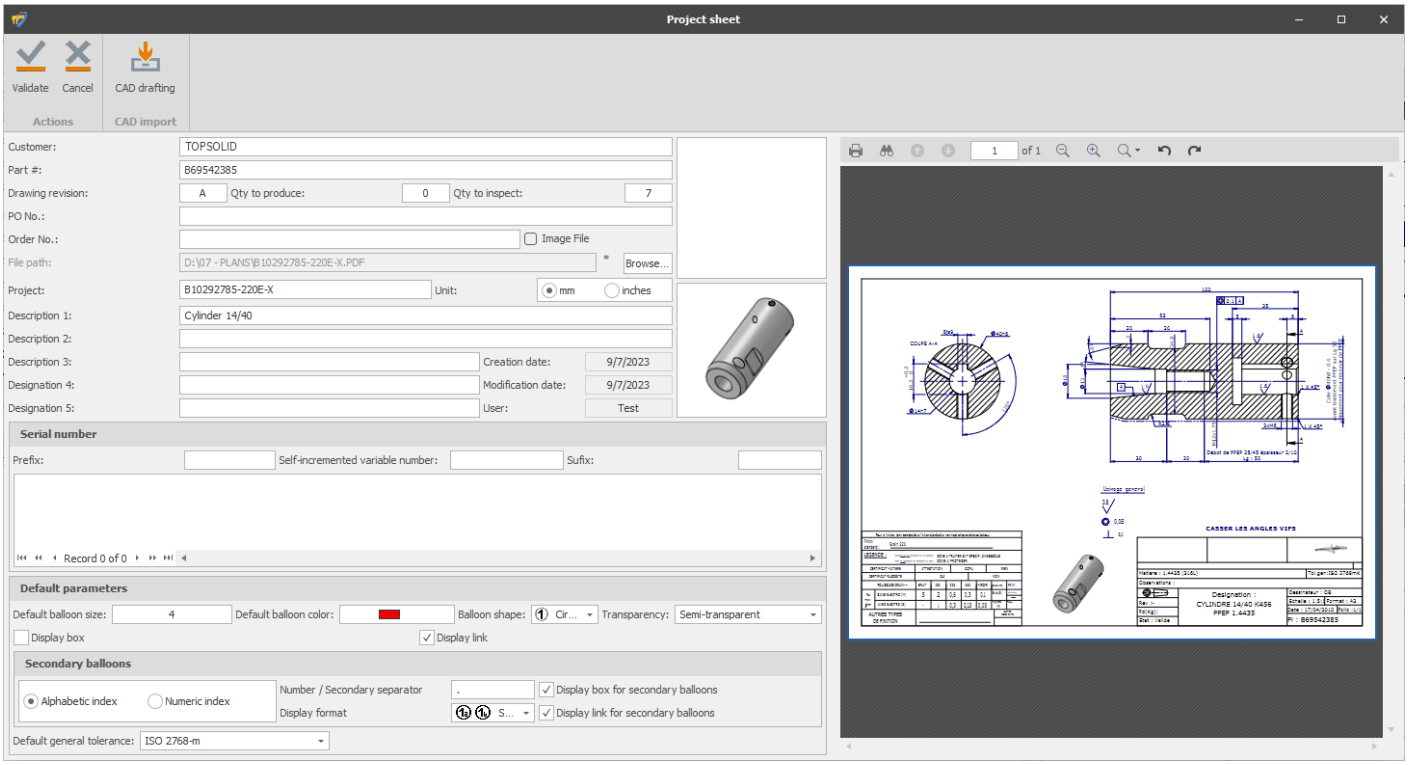


Mass modifications of checkpoints

The properties of a **checkpoint** selection can be modified all at once.

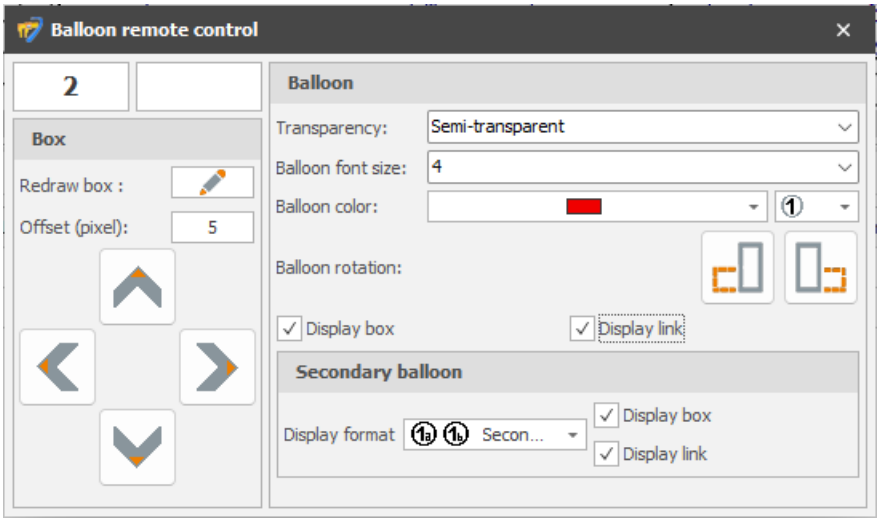
Drawing display in project creation

The selected drawing is displayed in the project creation, **making it easier to acquire information from the drawing** at the image and text levels.



Individualizing balloons components

To further **customize the layout of the ballooned drawing**, you can update, for each balloon, the display of the border, link and label.

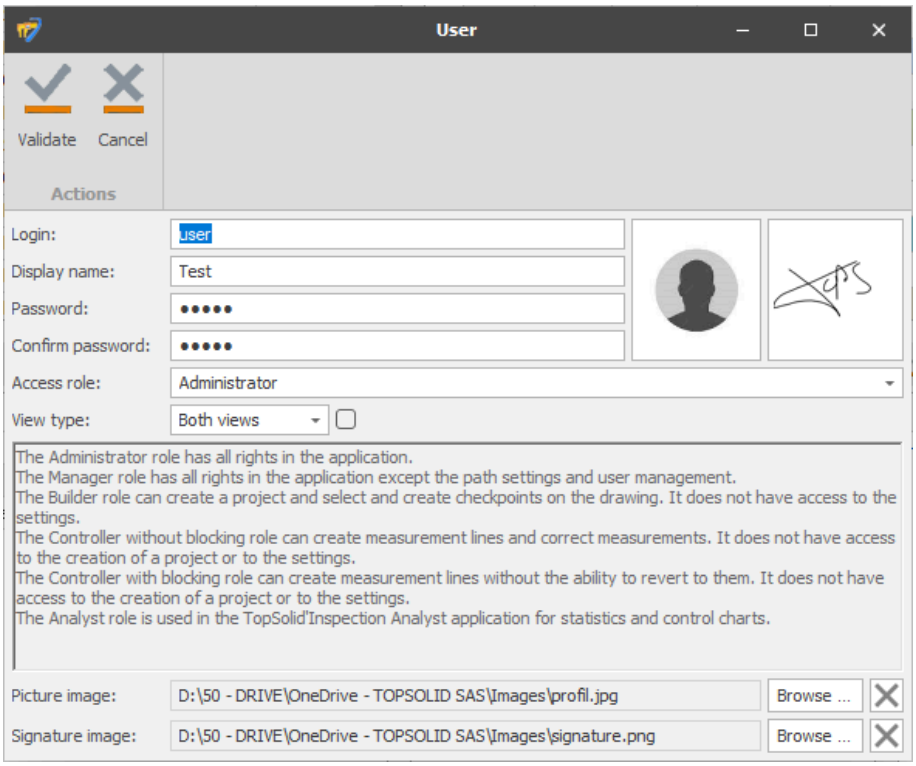


User, password and signature

Username input assistance is available in connection patterns of the different programs, to **simplify connection process**.

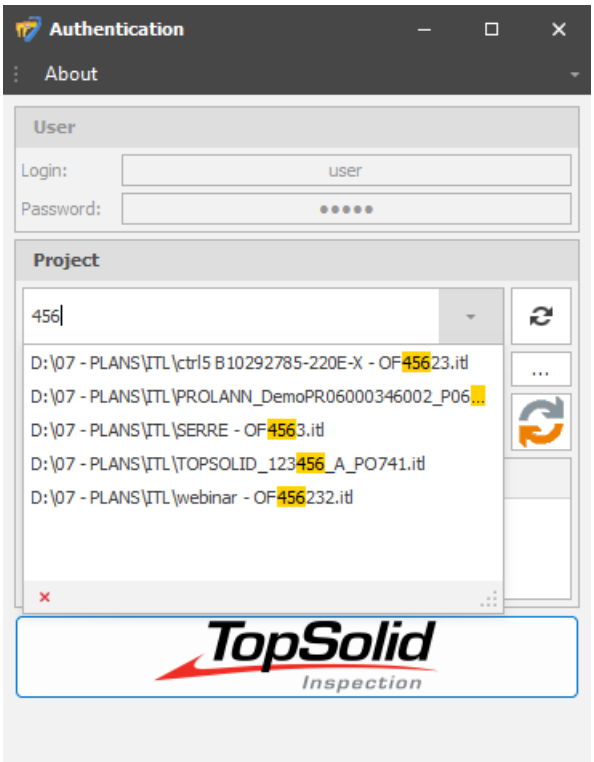
Users can associate two signature images with their profiles. These are available in the editions as well as in the annotations of PDF files to **sign the drawing and the reports**.

To enhance security, users are required to input their password twice in the user management section.



Project search system in Controller

The **Controller** interface integrates a project search system to **facilitate the availability of projects** for workshop inspection.



Managing the display of report templates

When **TopSolid'Inspection** is opened, the **REPX** files from the **\Template** directory are copied to the param files path. This displays a large number of templates in the printing windows. In order to **display only the templates you need**, you can rename the **REPX** files you don't need in the **\Template** directory by adding the "_" character (underscore).

Mass deletion of checkpoints

In order to **speed up the creation of control plans**, the deletion can involve several selected checkpoints.

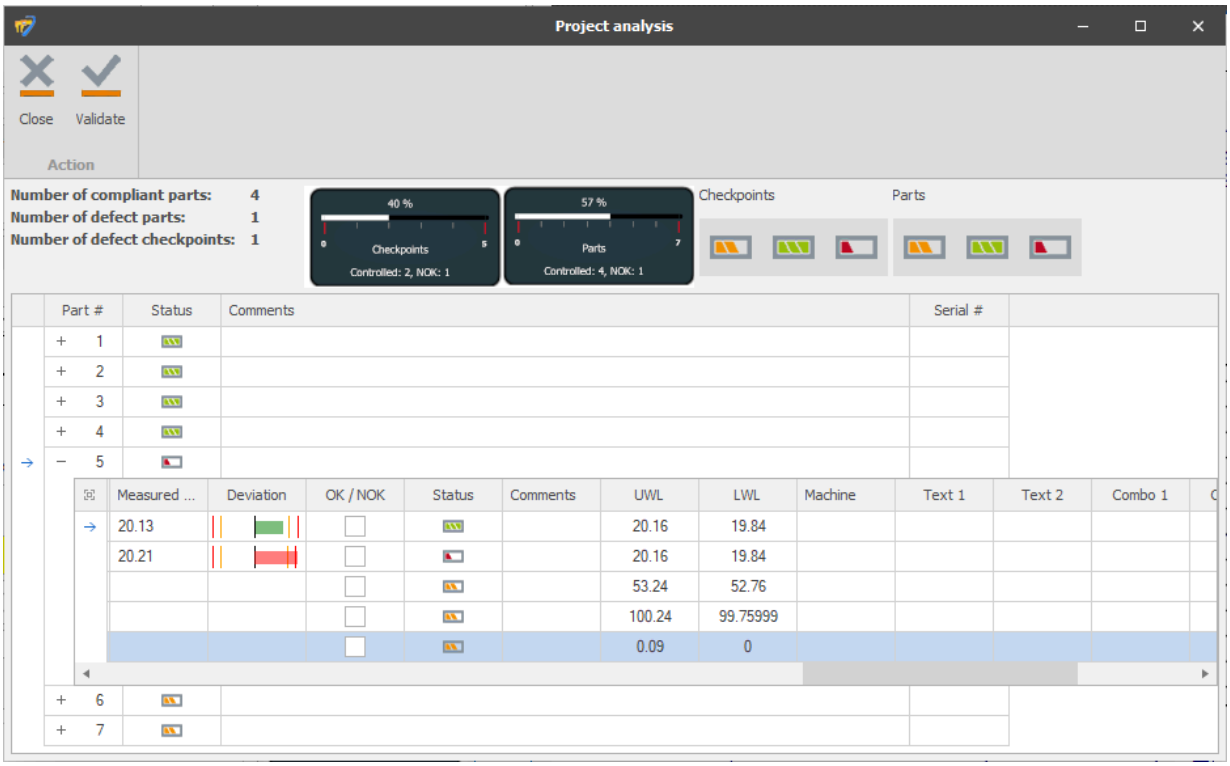
Modification of measurement data

A user with the **Administrator** role can modify the controller name and the date of a measure in the main **Creator** program, allowing **the measurement session to be administrated retrospectively**.

Indicators

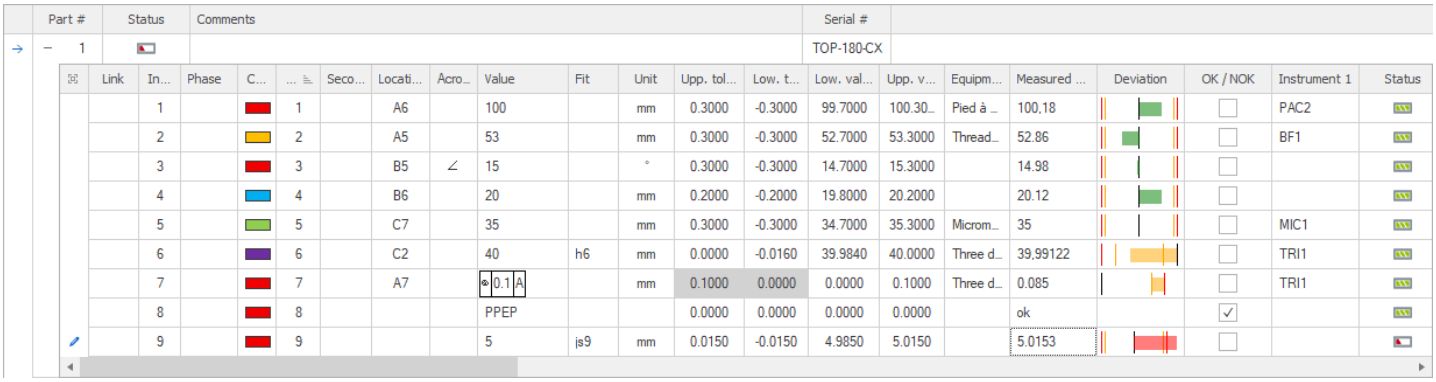
State of progress and summary

The **Controller** workshop interface presents two graphics providing dynamic information about the **state of progress** of the checkings. At the end of each measurement session, **summary indicators** are displayed for analysis and verification of checkings.



Measurement deviation and min/max

New **compliance indicators** of measures are available in the measure tables of **Creator** and **Controller**. Minimum and maximum measured values are displayed dynamically, together with a graphic showing the deviation between dimensions and tolerances.



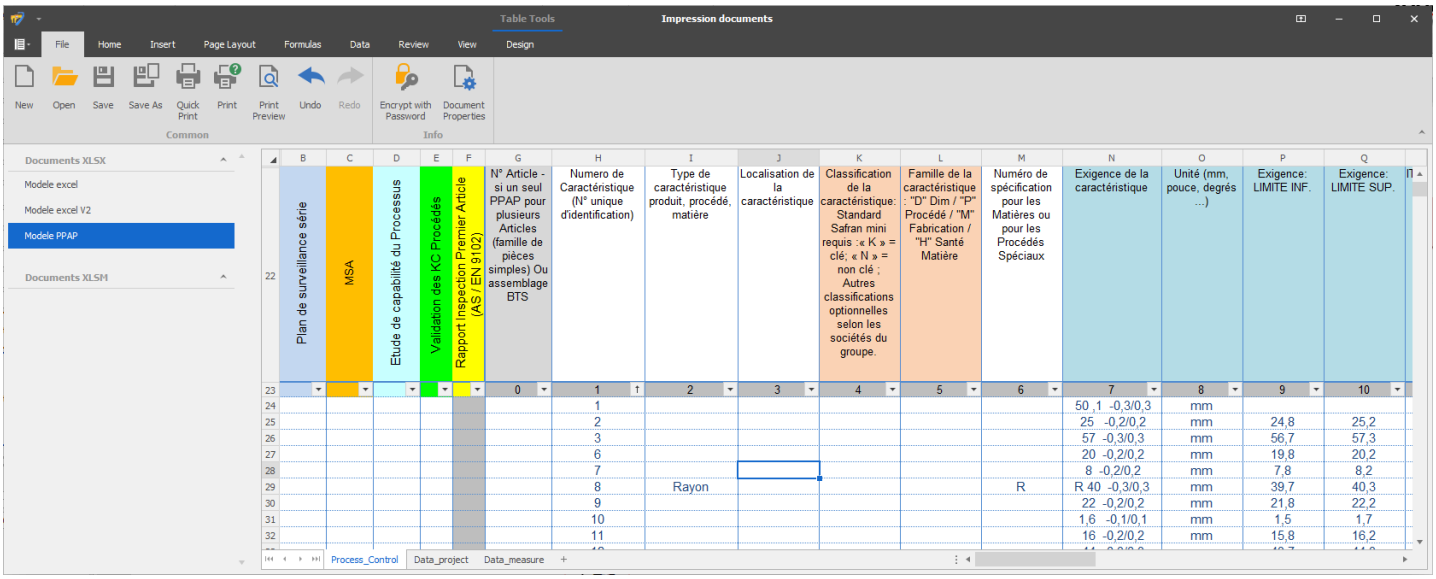
Reports from the Analyst program multi-criteria search

The results of a multi-criteria search in the **Search** tab of the **Analyst** program feed the data source for printing reports. This makes it possible to **exploit all data** in customized reports.

Interface

PPAP/DVI Excel export

A **PPAP/DVI** Excel template is available for exporting the control plan in the expected format of the industrial substantiation file (DVI).



New Excel export format

A new direct export format to Excel is available to facilitate communication with third-party software. It exports all the properties of checkpoints and associated measures.

New Excel import format

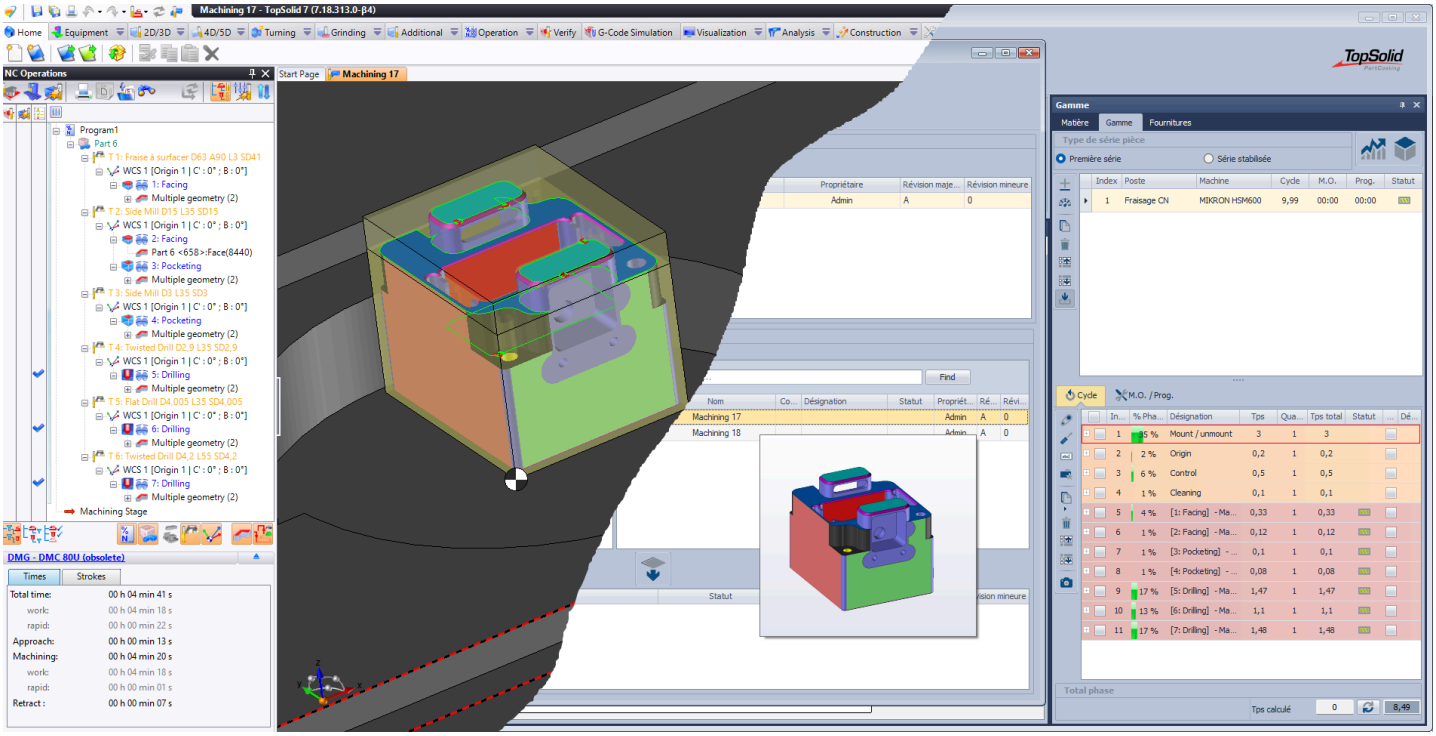
In order to simplify communication with third-party software, CMM data can now be imported from CSV and TXT file formats, in addition to the existing Excel format.

What's New in TopSolid'PartCosting 7.18

This chapter describes the new features of the **TopSolid'PartCosting** application in version **7.18** of **TopSolid 7**.

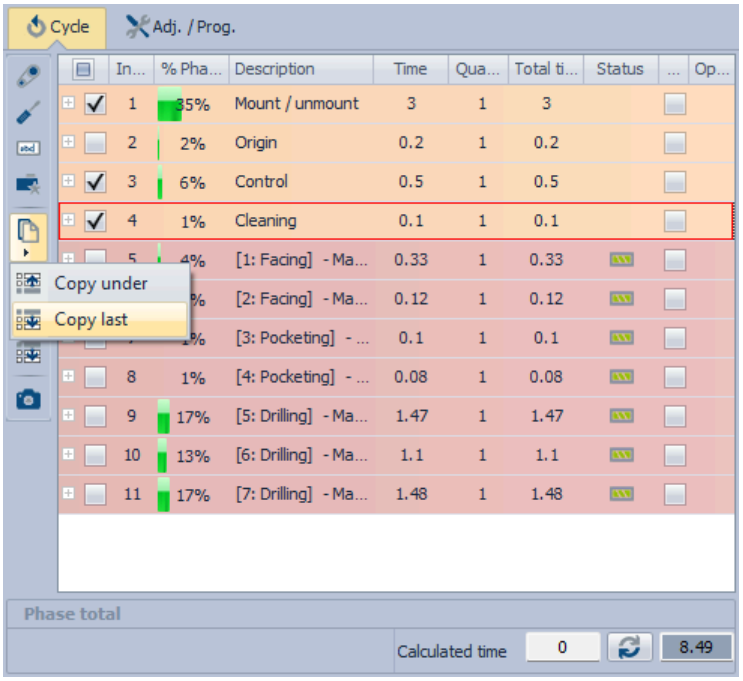
TopSolid'Cam program time recovering

TopSolid'Cam and **TopSolid'PartCosting** communicate via Automation to **use the precise times of an NC program** for quotation. At the **TopSolid'PartCosting** operation plan level, it is possible to link one or more machining documents to retrieve operations, their precise times and the tools used.



Copy of machining cycle elements

To facilitate management of the machining cycle, the **Copy cycle element** button allows selected elements to be copied either from below each element (as before), or at the end of the cycle.



Ergonomics add-ins

Version **7.18** offers a set of new features to enhance the user experience in terms of software navigation and tools:

- total display in action windows;
- hiding unnecessary **Material** groups;
- display of a **Total** line in the list of profiles.

Software performance and fluidity improvement

A work about the **loading time improvement** of lists has been carried out and optimized regarding the **Study** (**parts** list loading time) and **Administrative** (**cases** and **estimates** list loading time) module levels.