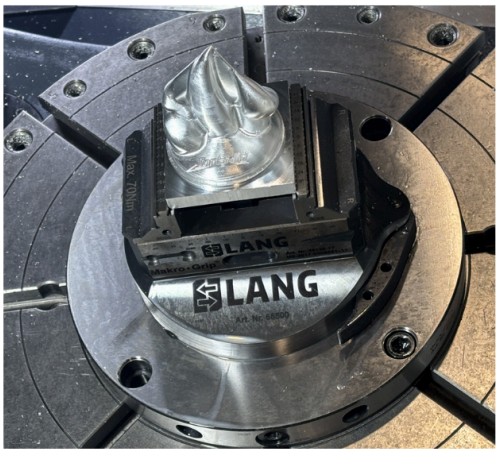
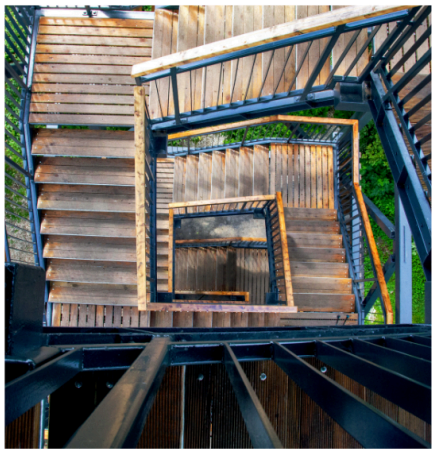
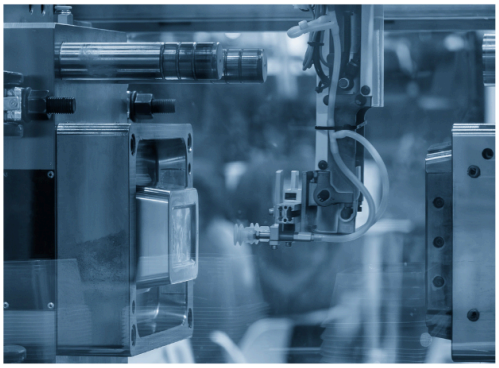
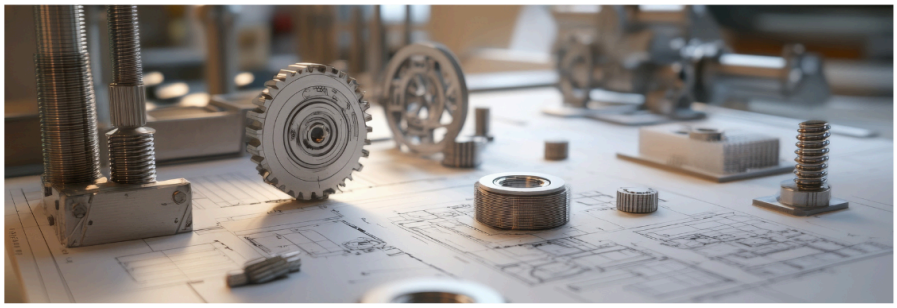




TopSolid
2025





What's New
TopSolid 7.19

TopSolid
INTEGRATED
DIGITAL
FACTORY

7, Rue du Bois Sauvage
F-91055 Évry, France
Web: www.topsolid.com

Version 7.19

Rev. 02

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

> **Tip:**
> **Any questions about TopSolid?** Find the answers you need in our comprehensive TopSolid'Faq! It's your go-to resource for solutions to the most common queries.
>
> **Elevate your skills with TopSolid'Learning**. Ready to take your skills to the next level? Whether you're a beginner or looking to sharpen your expertise, our online platform, TopSolid'Learning is designed to help you grow. Learn at your own pace and unlock the full potential of our software, making it an essential part of your projects.

Contents

Welcome to TopSolid 2025!

Discover the innovative brand-new features and major enhancements of **TopSolid 7.19**, designed to elevate your user experience and maximize your daily productivity.

Dive deeper into these updates by visiting our e-learning platform or contact your local branch for expert guidance tailored to your needs.

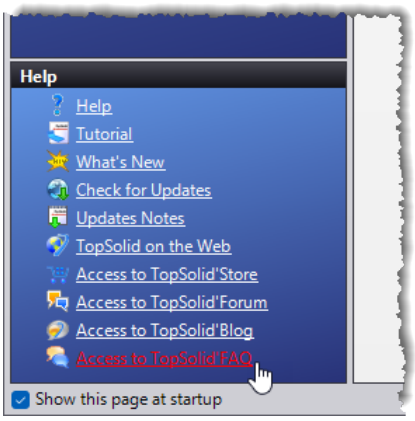
What's New in TopSolid'Design 7.19

Explore the brand-new features of the **CAD** applications of **TopSolid 7**!

User interface

Start page

Access to the **FAQ** of **TopSolid** has been added to the **Help** section, located at the bottom left of the start page.

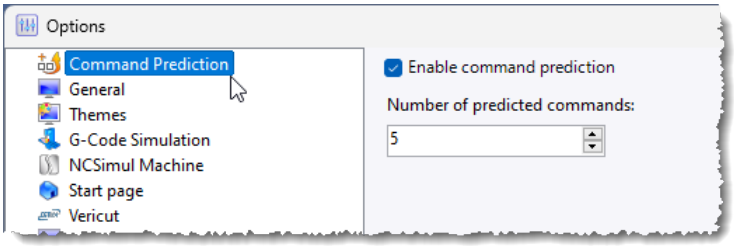


Command prediction

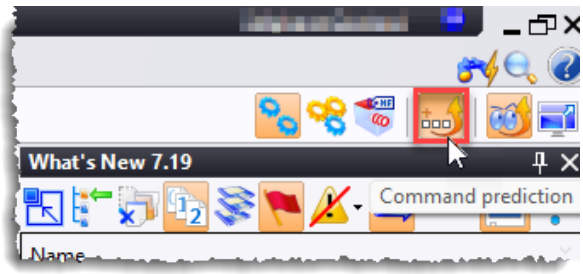
In the graphics area, an icon bar containing the most relevant commands is displayed either at the bottom of the screen, above the context menu, or when an entity is selected.

You can enable the command prediction, and the number of icons predicted, in the **Tools** > **Options** > **Command Prediction** command.

The display of the command prediction bar can be adjusted via the **Command Prediction** icon in the **Home** tab.



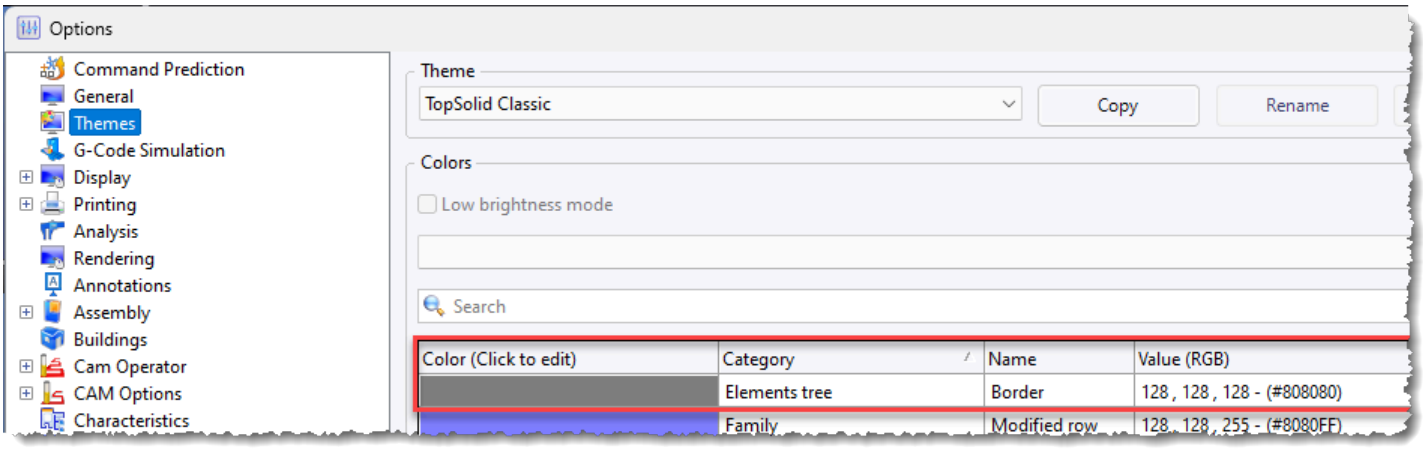
Enabling the command prediction.



Display of the command prediction icon bar.

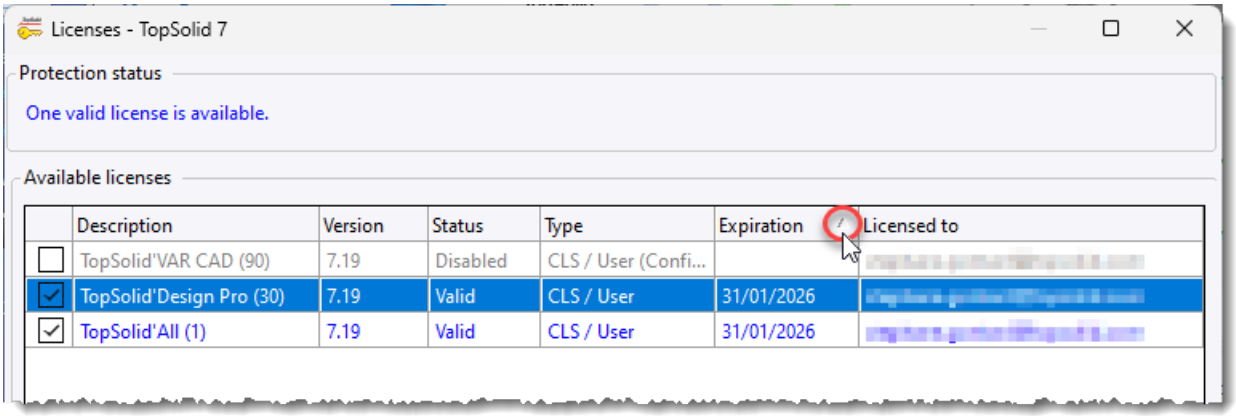
Themes

It is possible to configure the color of the tree borders.



Licenses

In the **Licenses** window, table columns can now be sorted by clicking on the column title.

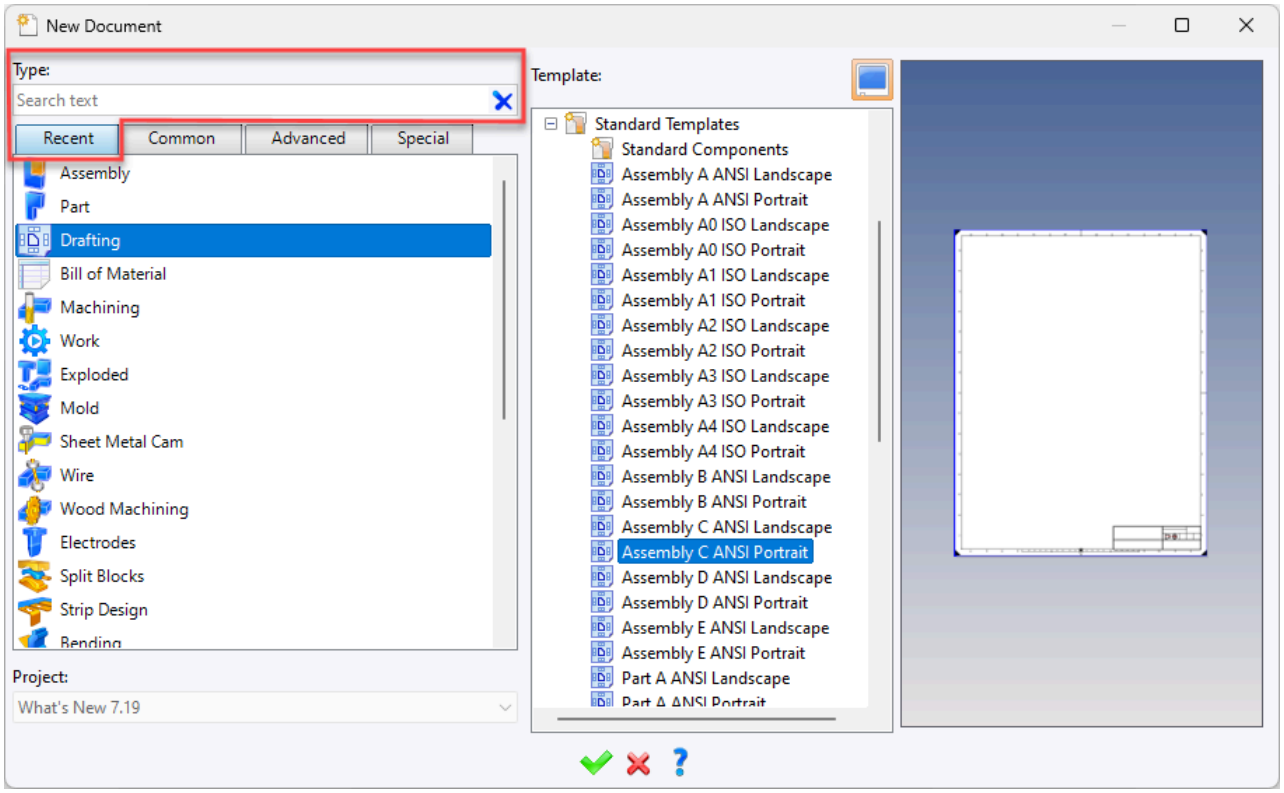


Applications can only use compatible licenses, meaning the **TopSolid'Pdm Server** application cannot use a **TopSolid'Pro** license or a **TopSolid'Steel** license.

PDM

New document

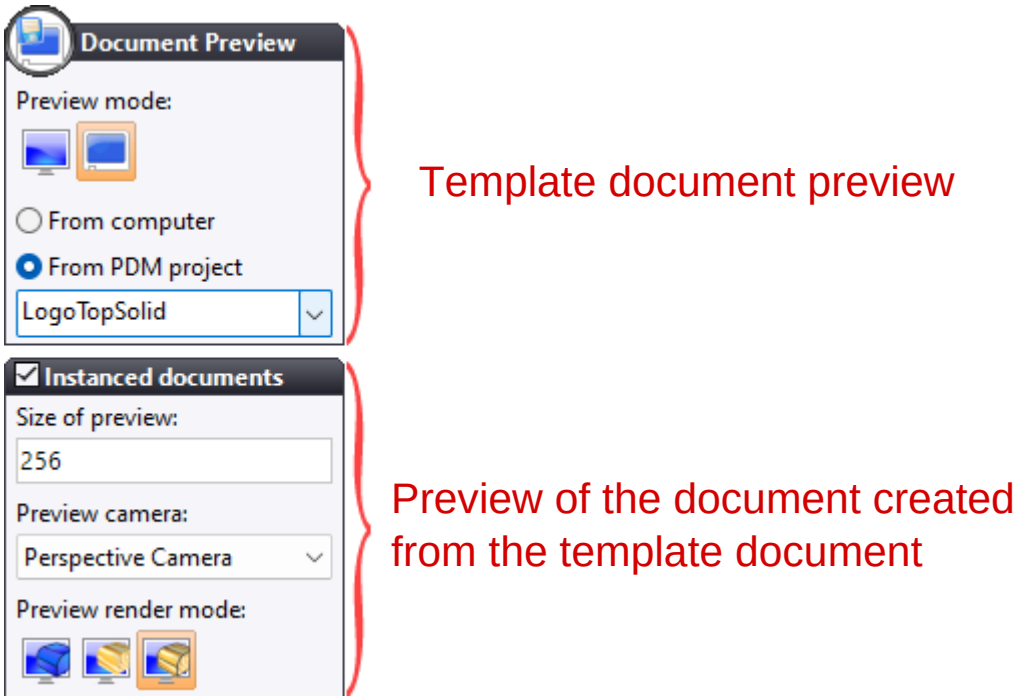
In the document creation window, a new search field for document types has been added, along with a **Recent** tab that displays the most recently created document types.



Document preview

The **Instanced documents** section has been added to the menu of the **File** > **Document Preview** command.

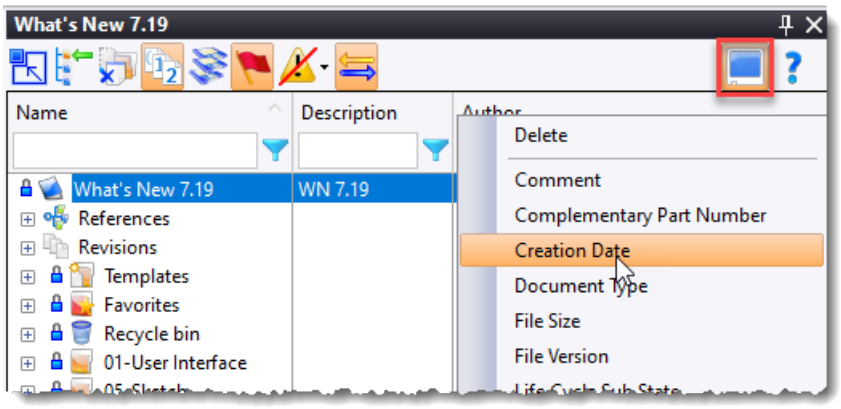
It defines the preview of the document created from a template.



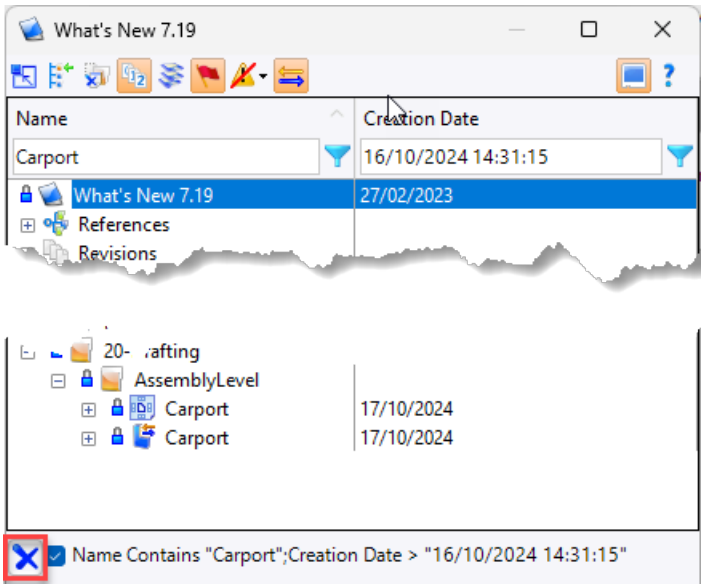
Project tree

The Project tree can be displayed with different columns (Description, Part Number, Author, etc.). Each column can be sorted alphabetically and/or filtered.

The tree banner features a new icon that allows you to display or not the preview of the selected document.



Additionally, a button to remove all active filters has been added at the bottom of the tree. This button is also present in the project manager window and in BOM of the Parts tree.



Properties

The document size analysis window is now modeless, meaning that it is possible to start another command while waiting for the document size analysis to finish.

This window can also be accessed using the **Tools** > **File size** command.

Outlook link

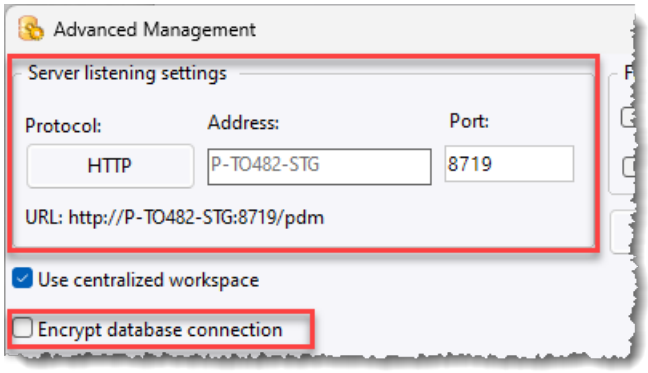
A document contained in an **Outlook** message can now be dragged directly into the Project tree. Depending on the document type, **TopSolid** will offer an import with or without conversion.

Advanced management

HTTP or HTTPS protocols are used for communication between the client and the server.

The HTTPS protocol enables data encryption between the client and the server.

In addition, the **Encrypt database connection** option encrypts the connection between the PDM server and the SQL instance.



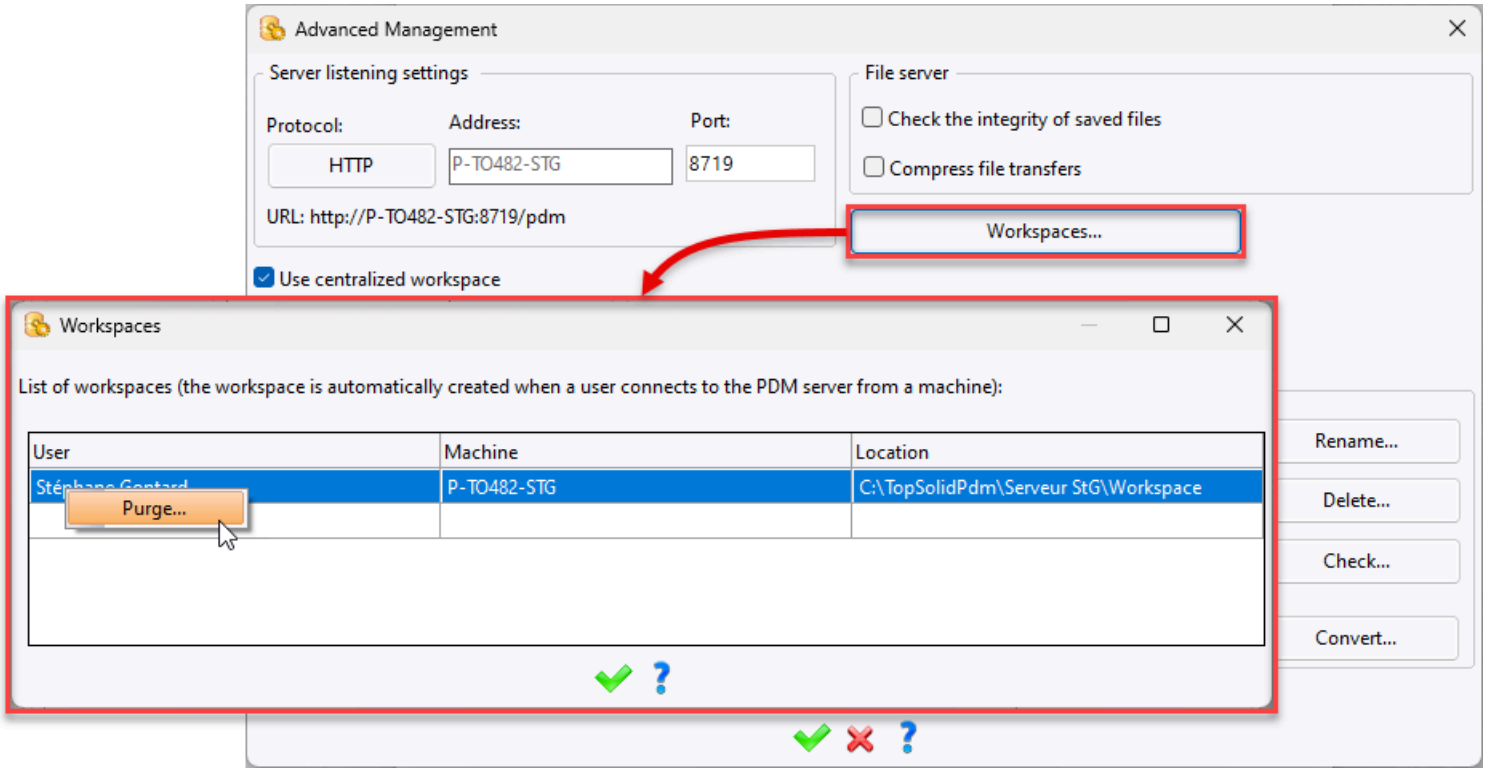
Asynchronous check-in

In client/server mode, during a backup or check-in operation, files are no longer sent directly to the server; they are sent in the background, so that you can check in immediately without having to wait for the end of the transfer.

Workspaces

In client/server mode, the **Workspaces** button has been added to the advanced server management window.

It allows listing the locations of the workspaces of users who have worked on the server, and the **Purge** context menu command allows deleting unused files.



Security

In client/server mode, security can be defined on a folder containing a project template as well as on a project template, allowing, for example, only the templates of a specific department or site to be made available.

Server stopping

The server can now be restarted when users are connected. Their session is no longer deleted as in previous versions. This can be used, for example, to manage network micro-outages.

Archiving

Archived projects can now be purged, and since this deletion is permanent and irreversible, a confirmation message is then displayed.

Missing files searching

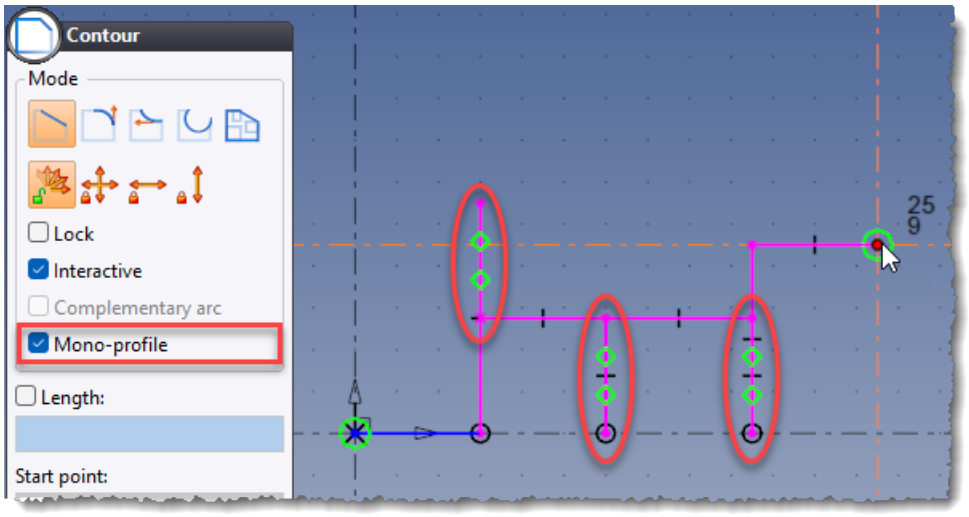
The **-SearchMissingFiles** argument, followed by the destination path, can be added to the shortcut for launching the **TopSolid'Pdm Server** application.

It provides a list of missing files.

Sketch

Contour

The **Mono-profile** option has been added in the dialog box of the **Contour** command. When the sketch is in **Automatic Profiles** mode, only one profile is created, even if it contains self-intersecting areas.



Text

The value of the spacing for a text or a wire text can now be negative. This modification also applies to the detailing texts in a drafting document.

Spacing = + 0.1

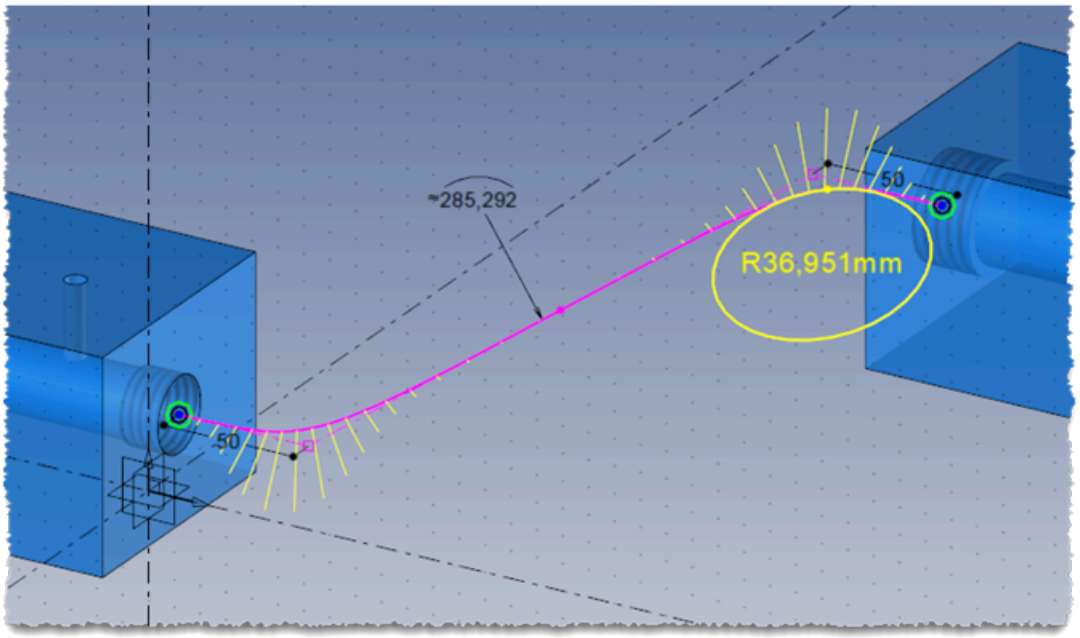
Spacing =-0.1

3D sketch

The **Curvature** command has been added to the context menu on a spline curve. It draws a circle with the radius of curvature on the selected point.

In addition, you can identify the position of the curve's minimum curvature using the new **Add Passing Point to Minimum Radius** context-sensitive command.

In addition, the new **Length Dimension** command allows you to constrain the length of a curve to impose its length.

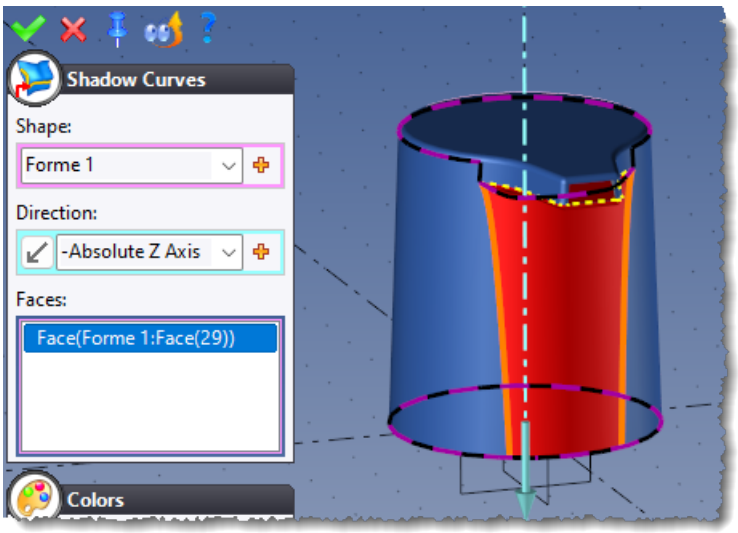


Chamfer

The **chamfer** command is available from the sketch operations menu.

Shadow curves

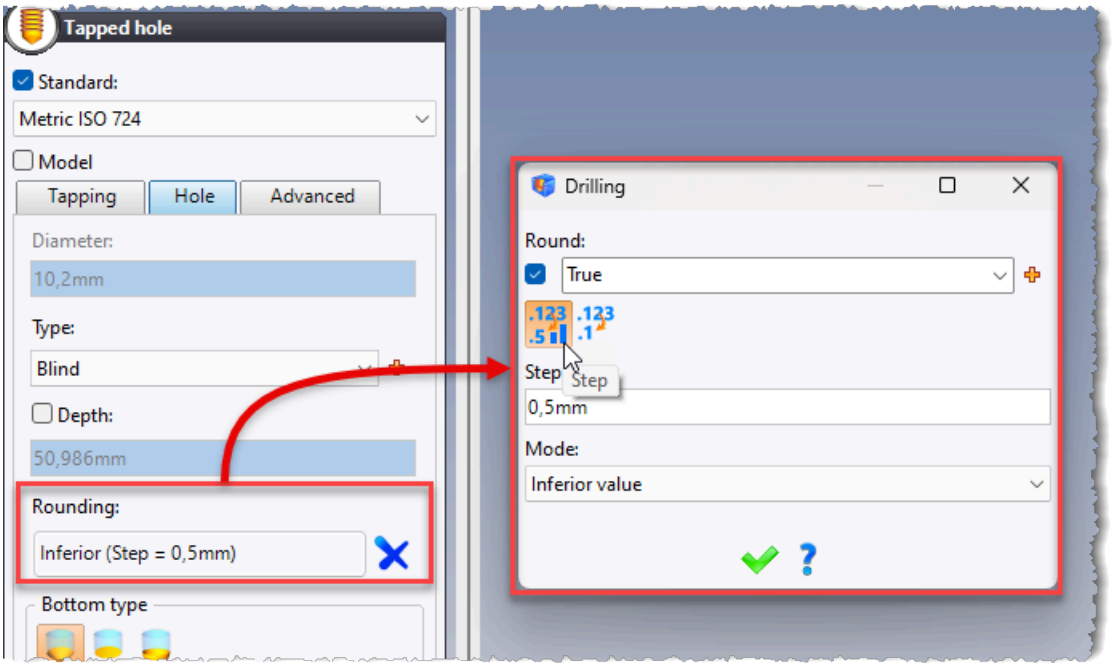
The new **Shadow Curves** sketch operation allows you to trace the curves delimiting the shadow zones of a part from a direction.



Shape

Drilling

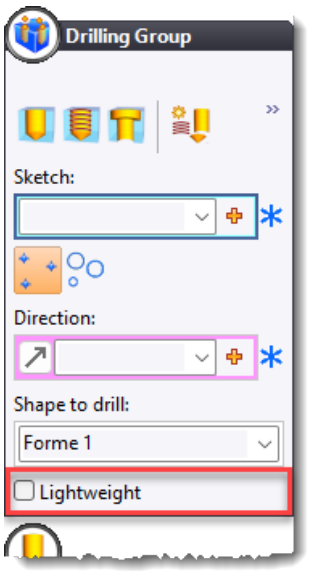
When creating a not-through tapped hole, a new button allows rounding the depth using a step instead of a precision. This option is particularly useful for computing the depth of cooling circuit baffles.



Drilling group

The **Lightweight** mode has been added to the **Shape** > **Other Operations** > **Drilling Group** command.

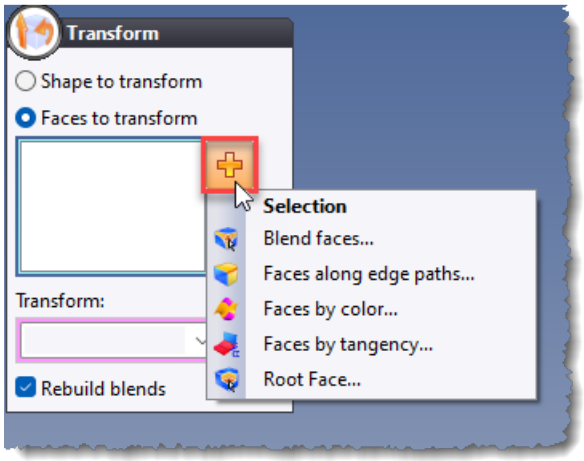
As for the **Drilling** command, it allows you to create drilling operations without creating the corresponding Boolean operations.



Picking faces

The special input button for selecting multiple faces (root face, blend faces, faces by color, etc.) has been added to the following commands:

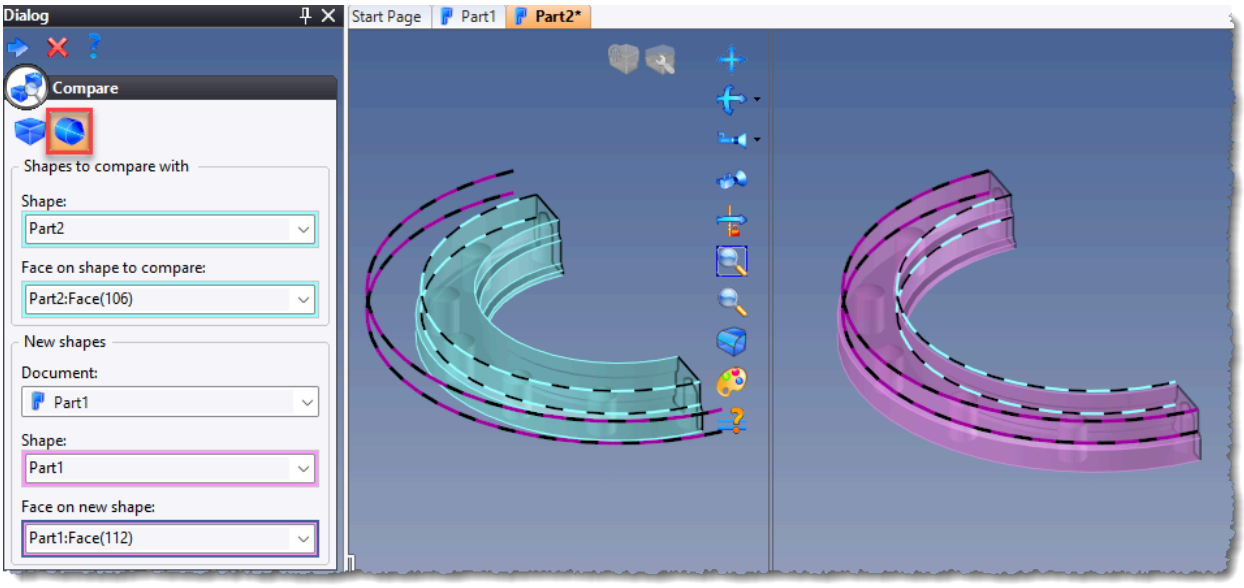
- **Shape** > **Other Operations** > **Hollow**
- **Shape** > **Other Operations** > **Transformation**
- **Shape** > **Other Operations** > **Deformation**

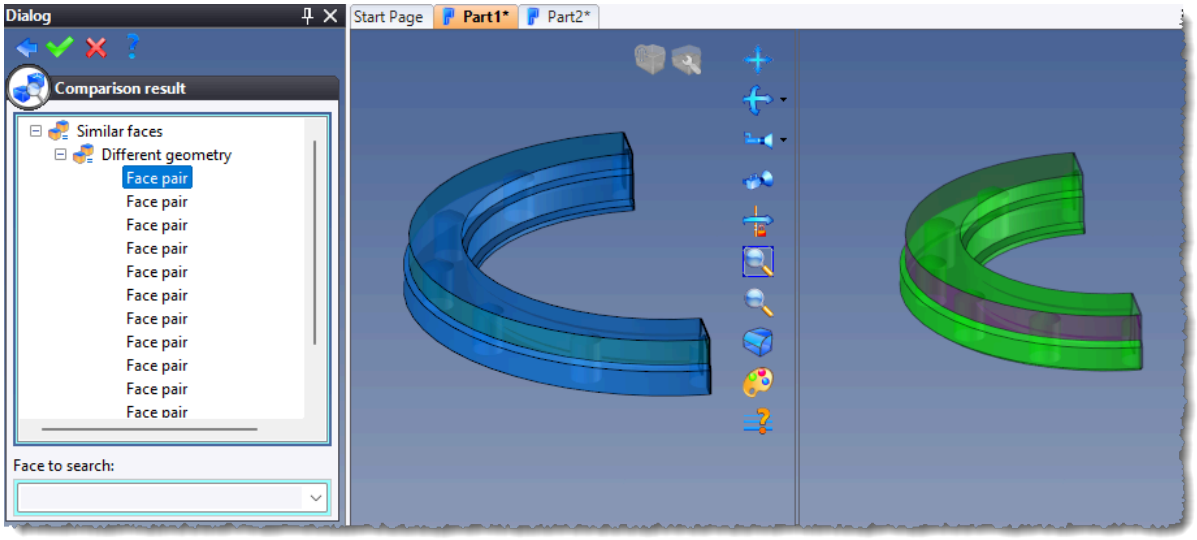


Comparing and replacing

A new comparison mode specific to revolved shapes has been added to the **Shape** > **Comparison** > **Compare**, **Shape** > **Comparison** > **Replace** and **Replacement wizard** commands from an assembly document.

It is particularly useful for comparing parts with similar topologies but of different sizes.

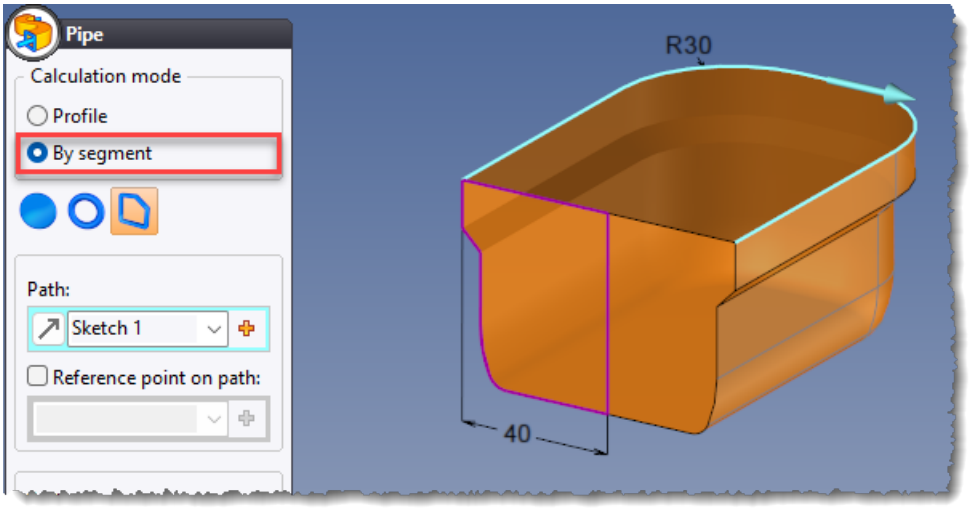




Surface

Pipe

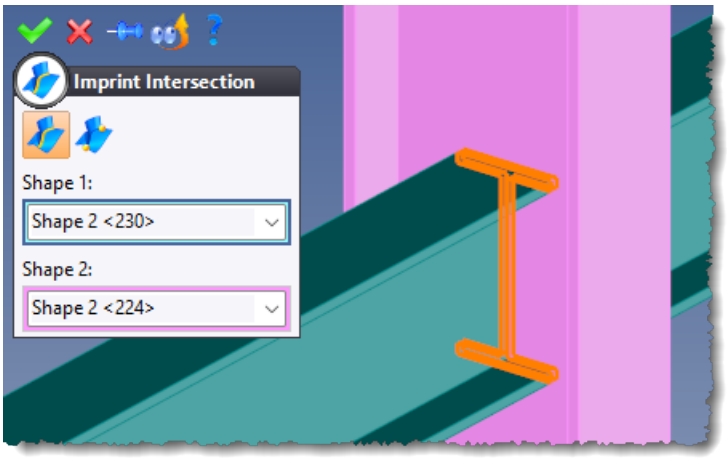
The new **By segment** calculation mode corresponds to the extruded bar shape calculation algorithm used in **version 6** and allows for managing self-intersections of the section.



Imprint intersection

The new **Imprint intersection** command has been added to the **Surface** menu. It is mainly used in the **FEA** plugin to prepare shapes and features two modes:

- **Edge to face**: computes the intersection curve(s) between two shapes and imprints the result on each shape.
- **Edge to edge**: identifies common edges and their limits (points), and imprints them on each shape.



Sheetmetal

Lateral faces straightening

The new **All straightenable edges** option automatically selects all the edges to be straightened on the part.

In this case, if no edge needs to be straightened, the operation does not become invalid; it simply "does nothing". This new mode ensures the replaying of a sheet metal part, whatever its geometry, and is very useful for defining generics for sheet metal components.

Sheet metals recognition

Operated property

The **Operated = False** property is created in parts converted as extruded bars. If the extruded bar is operated in the assembly, this property can be modified to **True**, as for standard extruded bars.

Deep analysis

The new **Deep analysis** option detects extruded bars according to the three axes of the enclosing box. It can be used to identify "short" extruded bars whose section is greater than their length.

This option can be time-consuming if there are many parts to analyze.

Calculations x 3: The calculation time can be up to three times longer. Therefore, it is recommended to activate this option only when certain parts have not been recognized.

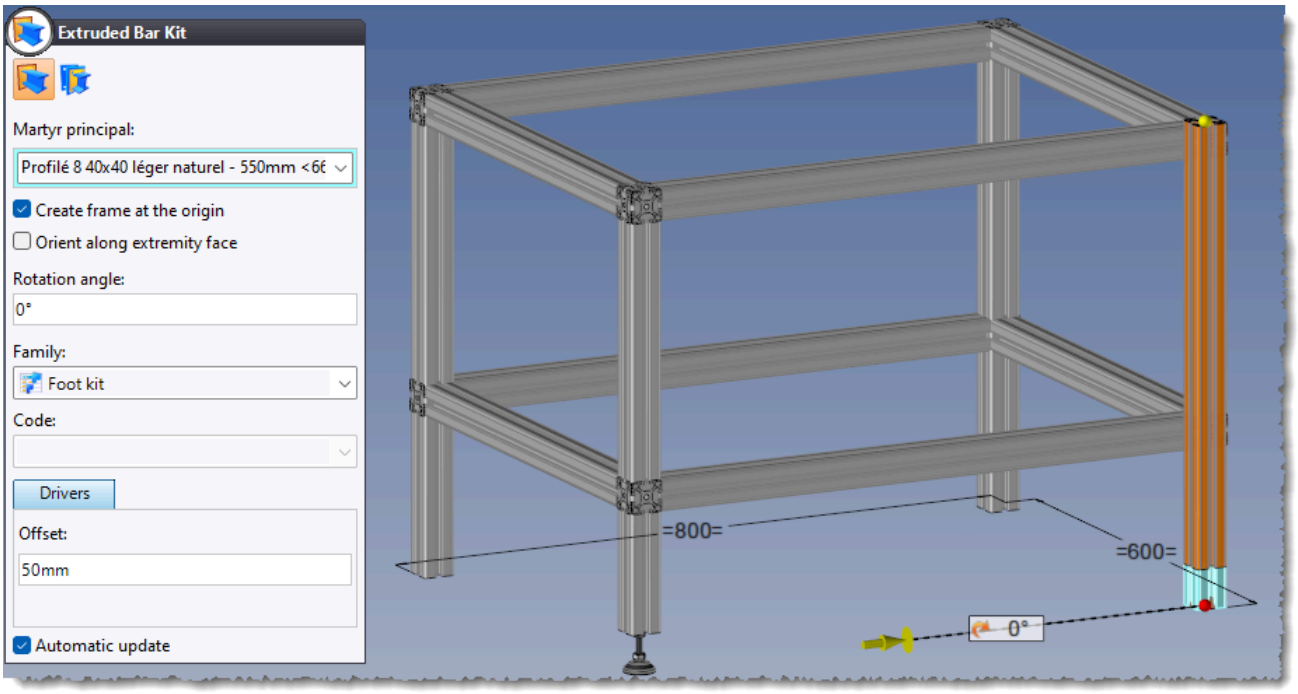
Assembly

Extruded bars

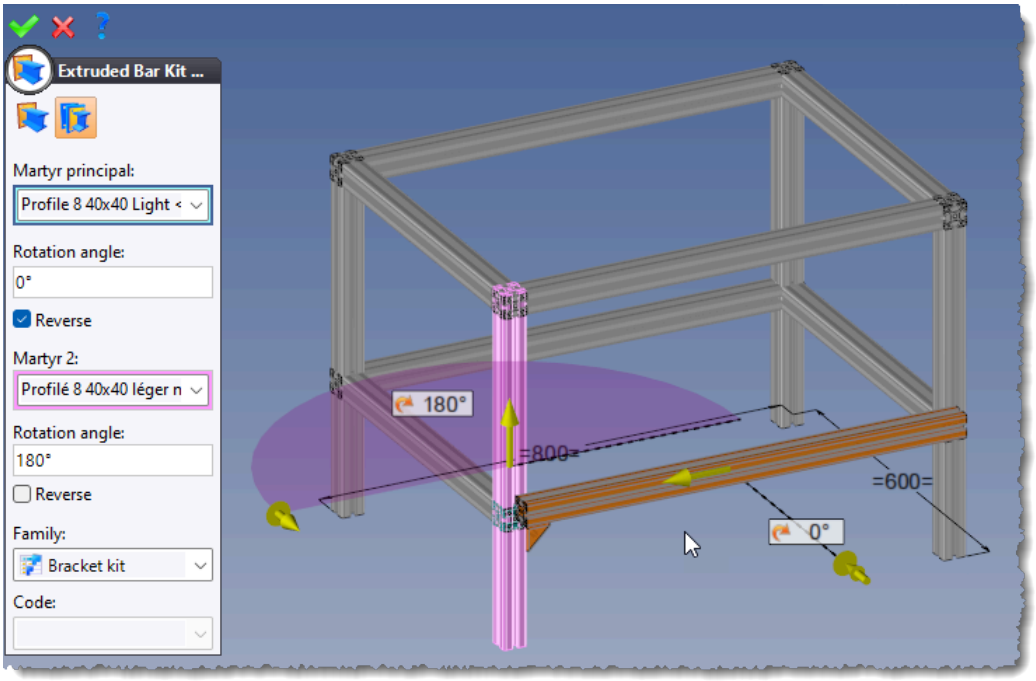
Extruded bar kit

The new **Modeling** > **Extruded Bar Kit** command allows you to quickly position a component either at the end of an extruded bar, or at the intersection of two extruded bars. This command starts automatically when a family, whose generic document provides the **Extruded Bar Kit** function, is included.

The new **Modeling** > **Extruded Bar Kit Creation** command allows you to define the components of the extruded bar kit and its inclusion method (at the end of an extruded bar or at the intersection of two extruded bars).



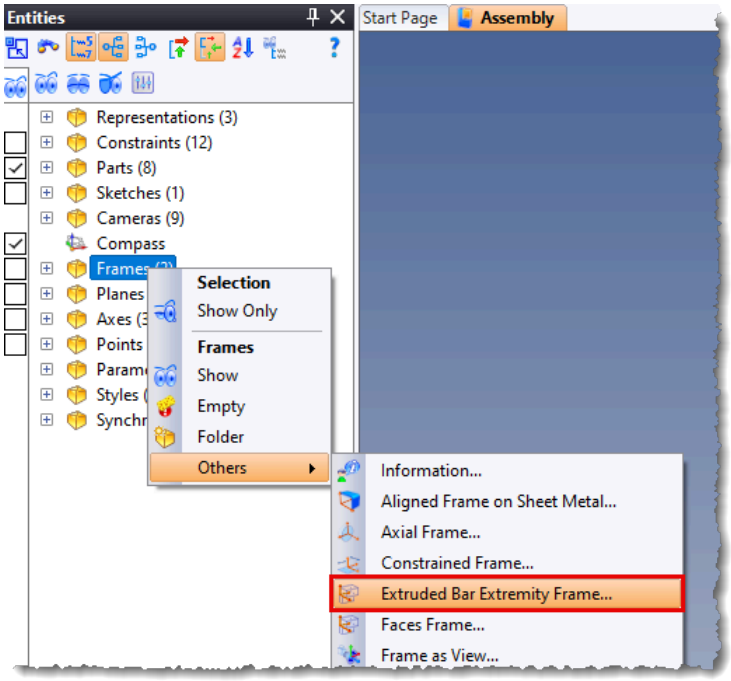
Example of positioning an extruded bar kit at the end of an extruded bar.



Example of positioning an extruded bar kit at the intersection of two extruded bars.

Extremity frame

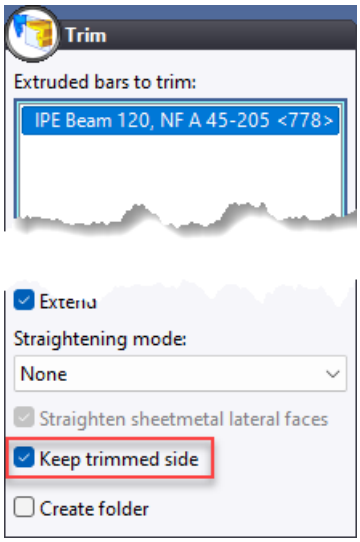
The new **Others** > **Extruded Bar Extremity Frame** command is accessible by right-clicking on the **Frames** folder from the Entities tree. This frame element can be used to create an in-place part, or to position another part.

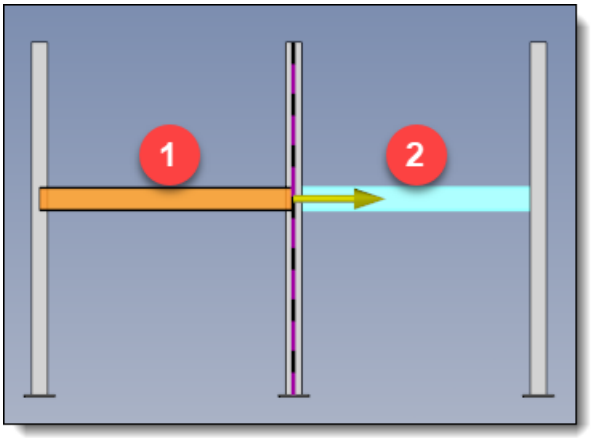


These commands are also available by clicking on the **Special Inputs** icon in the **Construction** > **Frames** > **Frame** command.

Keep trimmed side

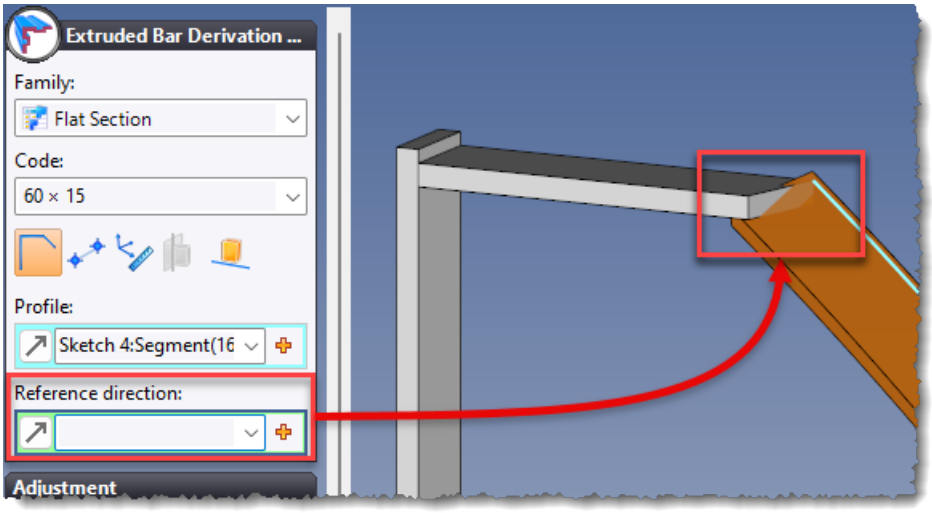
The **Keep trimmed side** option has been added to the dialog box of the **Modeling** > **Trim** command.



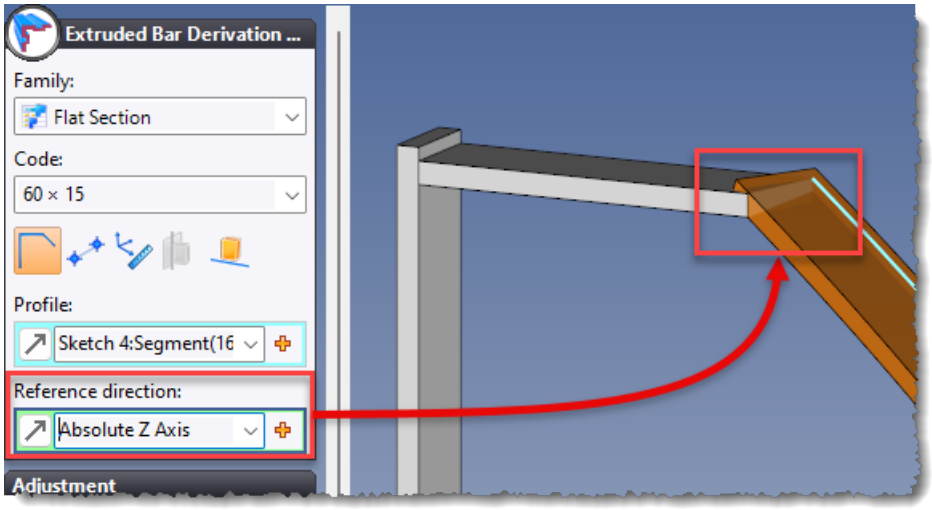


Vertical direction

When creating a curved extruded bar, the reference direction is now used like a vertical direction to maintain the verticality of the extruded bar.



Without vertical direction



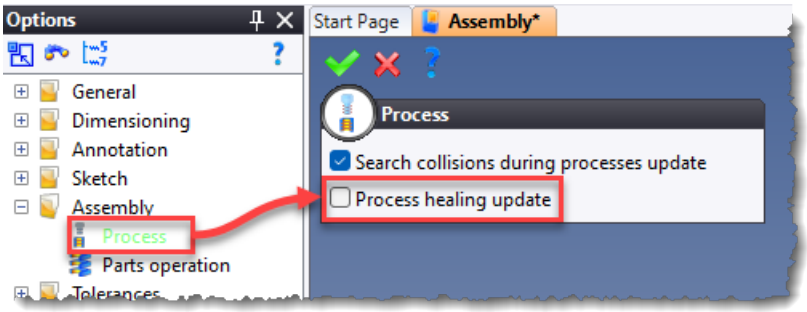
With vertical direction

Healing

The new **Process healing update** option, available in the **Assembly** section of the Options tree, allows you to control the presence of the process when modifying the component's position.

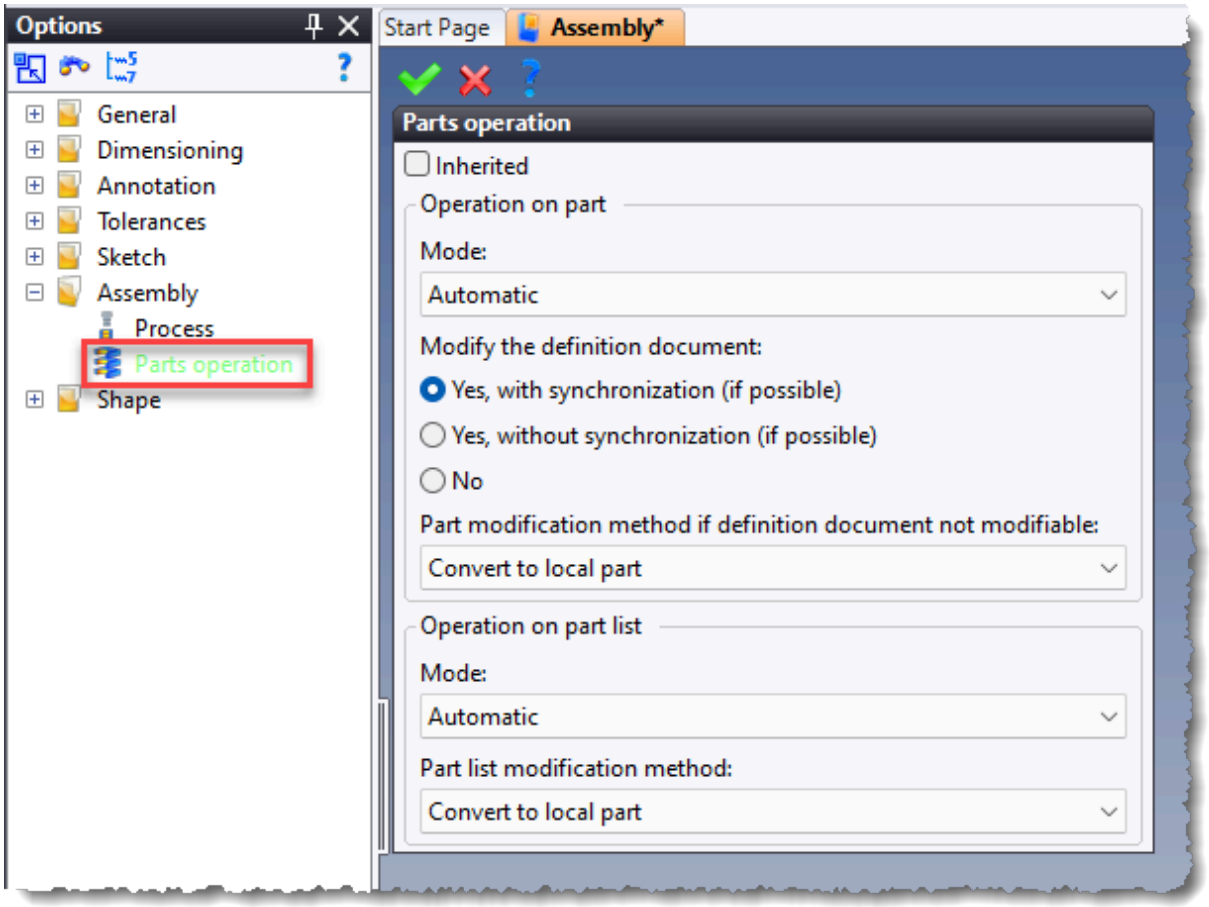
Indeed, when the component is moved outside the part it is operating on, its process can either be deleted (healing option checked) or kept (healing option unchecked).

If the option is unchecked, the process will be automatically replayed when the component collides again with the operated part.



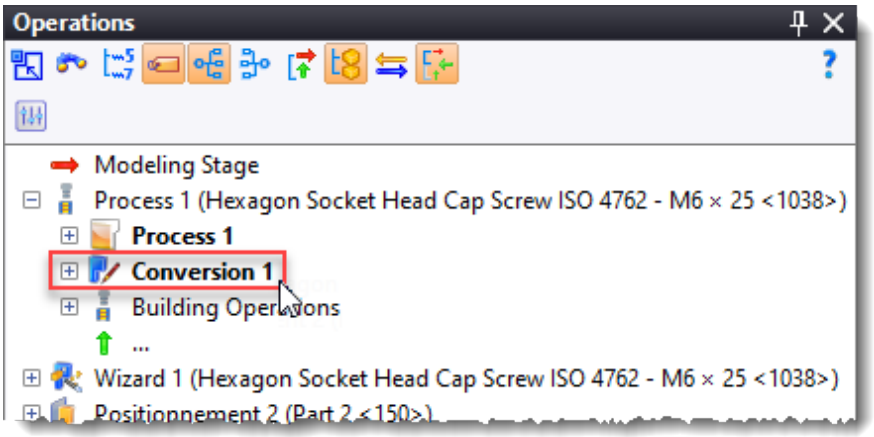
Conversion to local part

In the assembly document, you can define options for part modifications in the document options. By default, these options are inherited from the application options defined in **Tools** > **Options** > **Assembly** > **Parts Operation**.



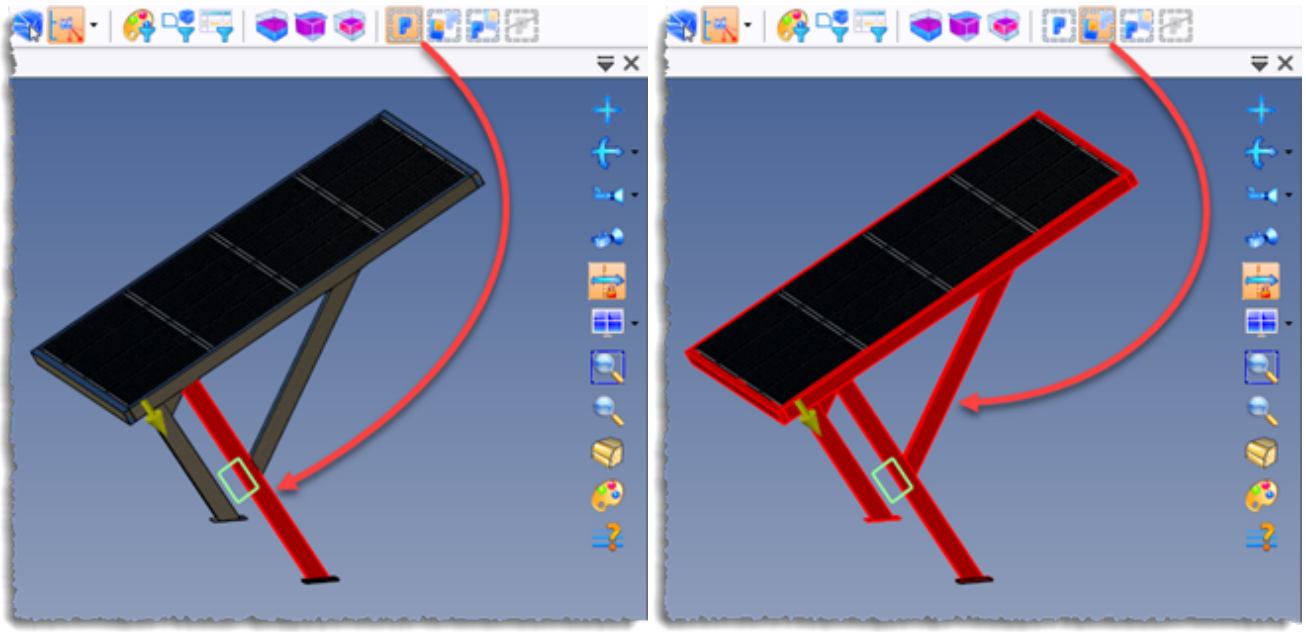
In addition, the conversion operation to local part is stored under the **Process** operation. As a result, when a component that has performed its processes on a part is moved to a new part, it is automatically converted to a local part.

This allows the processes of the moved component to be performed, provided that the **Search collisions during processes update** document option is enabled.



Assembly operations

The **Visualization** > **Part** and **Visualization** > **Assembly -> Part** modes are now taken into account during assembly operations (trim, pocket, etc.).



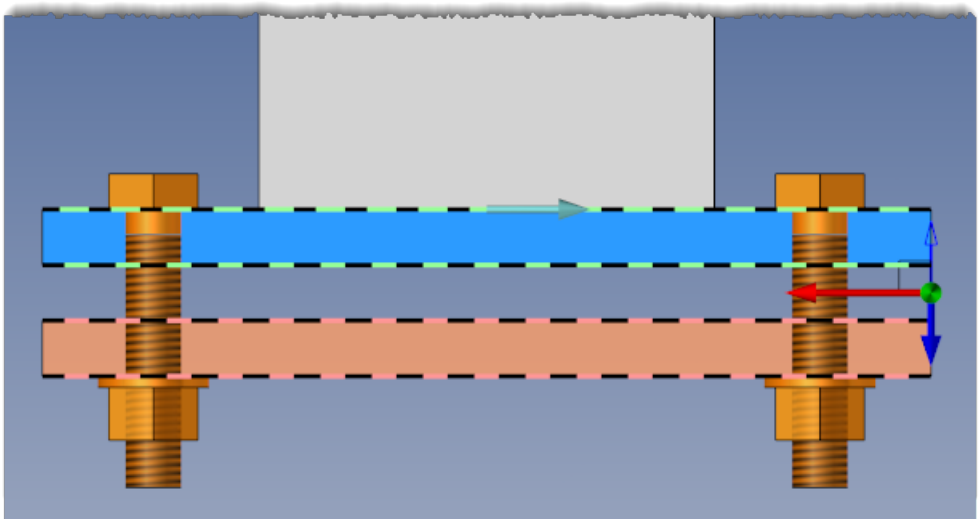
Analysis

The new **Analyze Process** command, accessible via the context menu on a process, from the Operations tree, allows you to copy the process with the driver values into a new temporary document that cannot be saved. When creating a component, it will be very useful for understanding and resolving any potential issues.

Assembly kit

The **Assembly Kit** command was modified and you can:

- place an assembly kit between two parts with clearance;
- manage a third thickness.



Creating automatic processes

The new **Modeling** > **Processes** > **Create Automatic Processes** command is used to create an operation that groups together all the processes at the top of the Operations tree. It is unique for each document.

It is useful when the **Automatic Process** mode is disabled.

It allows you to replay all processes in a single operation.

Envelope block with wizard

When a component containing an envelope block was included with a wizard, a zone for entering the envelope block dimensions has been added to the dialog box.

Division

The operation performances have been optimized. From now on, only the parts affected by the operation are converted to local parts.

Driver consistency

The **Check drivers consistency** option has been added to the **Advanced options** section of the **Modeling** > **Assembly Kit** and **Modeling** > **Distribution** commands.

When replacing a kit or a distribution component, it can be used to verify that the number of drivers is similar.

Management of error documents

When a document is in error, it is possible to create an in-place part, an in-place assembly, a local part or a local assembly. This allows, for example, to repair it without basifying all operations in error. You will simply have to reconnect them to the new part or assembly you have created.

Building

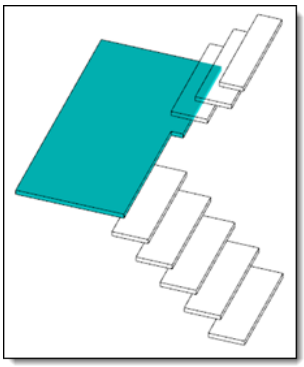
Staircase layout

Classic landing

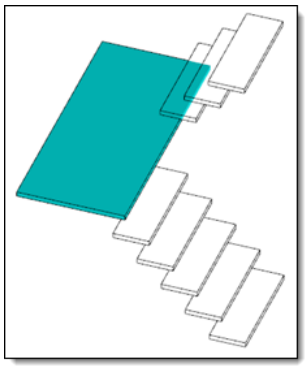
It is now possible to define the application mode for nosing after a flat landing.

In previous versions, the landing behaved like a tread, and the nosing was modifying its geometry.

Thanks to the new **Nosing on landing** option, the next flight is shifted from the nosing to keep the landing's theoretical geometry.



Flat landing with "classic" nosing management



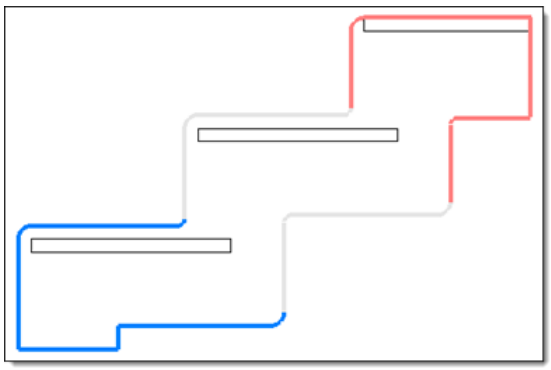
*Flat landing with **Nosing on landing** option*

Moreover, as for the **Winder** and **Spiral** landing types, it is possible to adjust the dimension of a **Classic** landing type by increasing its area of influence on one or both flights adjacent to the landing.

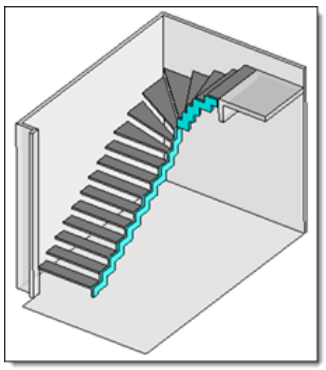
Stringer document templates

You can create stringers using a template document. This document defines the stringer shape to be kept from one flight to the next.

Stringer example templates are available in the **TopSolid Building** library.



Sawtooth stringer type document template It is defined by three sketches representing each part of the stair flight (start, intermediate and end of the flight).



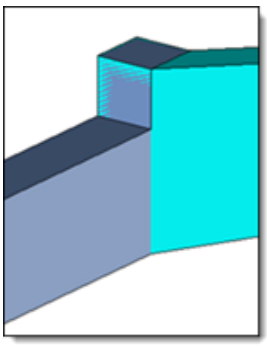
Sawtooth stringer created from document template.

Junctions of stringers

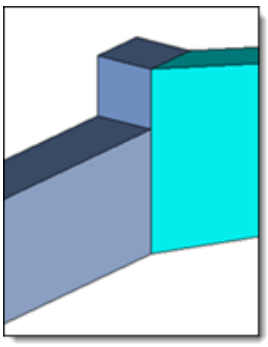
This new option makes it possible to adjust two consecutive stringers, one by the other.

A table appears, automatically listing, for each pair of consecutive flights, the stringer of the lower flight and the stringer of the upper flight. The following two junction modes are available:

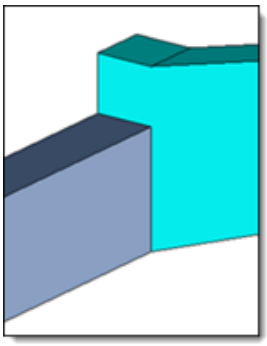
- **Adjusted** The two stringer sketches are adjusted to coincide with each other. If stringers are of the **Shape** type, an additional option is provided, allowing the selection of a trimming mode.



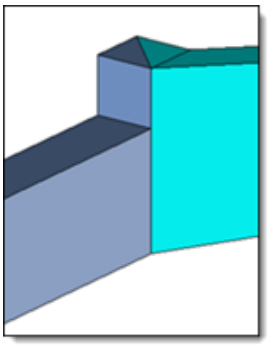
No cut: *covering bottom (gray) stringer/ covering top (blue) stringer.*



Covering bottom stringer: *covering bottom (gray) stringer/ covered top (blue) stringer.*

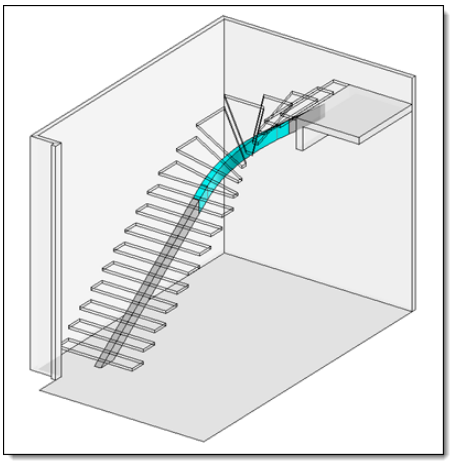


Covered bottom stringer: *covered bottom (gray) stringer/ covering top (blue) stringer.*



Miter trim: *a miter trim is created between the two stringers.*

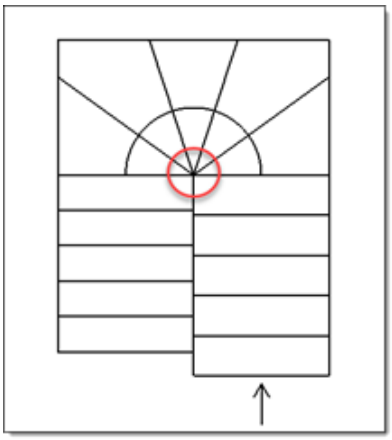
- **Fitting**: This option is only available for **Under tread** type stringers. It is used to create a curved stringer between two straight stringers.

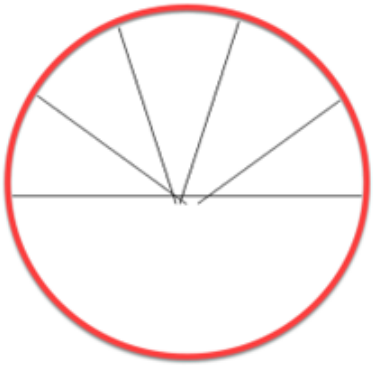


Advanced options

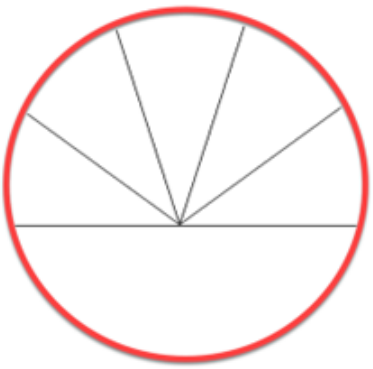
In the **Advanced options** section of the **Staircase Layout** command, two new options are available to simplify the layout sketch:

- **Simplification tolerance of tread fronts**: In some cases, the edges of the treads may not be joined. This option allows you to adapt the sketch geometry to ensure coincidences between each tread. It only modifies the 2D sketch.





The sketch produced with a simplification tolerance finer than that of the modeling is not joined.



The sketch produced with a simplification tolerance similar to that of the modeling is not joined.

- **Simplification tolerance of tread backs**: This tolerance allows for the simplification of 3D tread sketches by removing the very small segments and simplifying the tread type.

Modeling

Marking

The **Marking** command (for assemblies or parts) now includes the new **Color** advanced option, which allows assigning a color to the edges marked by the marking operation or to the sketch of the lightweight marking.

Methods

The following operations are now managed by the method document:

- Sketch modification operations
- Construction: transformations and transformation patterns
- In a part document: thickening, pattern union
- Processes: repetition generation
- Cut (extruded bars)
- The operations of the analysis stage are repeated for in-place parts created by a method

Moreover, you can add a 3D sketch as a driver for a method.

Bill of materials

Grouping options

The **Group by property** option has been renamed to **Group by property column** which groups parts when the properties, visible or hidden, in the bill of materials, are identical.

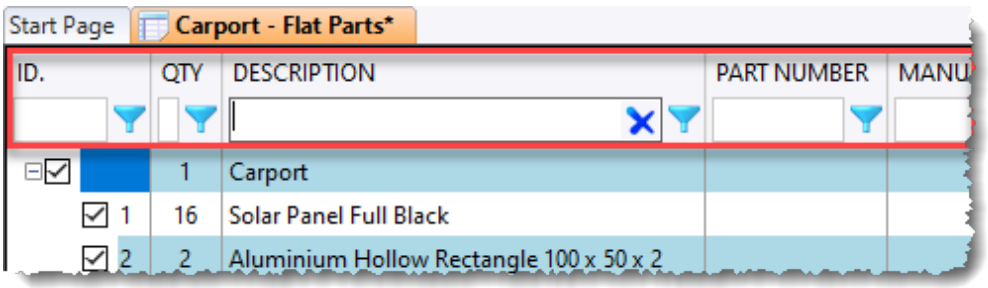
The **Compare shapes geometry** option has been replaced by the **Ungroup different geometries** option.

When two parts are strictly identical, the shape comparison (**Ungroup different geometries** option) only compares the topology of the shapes and does not take other properties into account.

Additionally, the **Consider local modifications** option now only applies to parts that have been locally modified (via the **Modeling** > **Shape** > **Locally Modifiable Parts** command).

Search

Search fields have been added to the header of each column.



Preview of local assemblies and parts

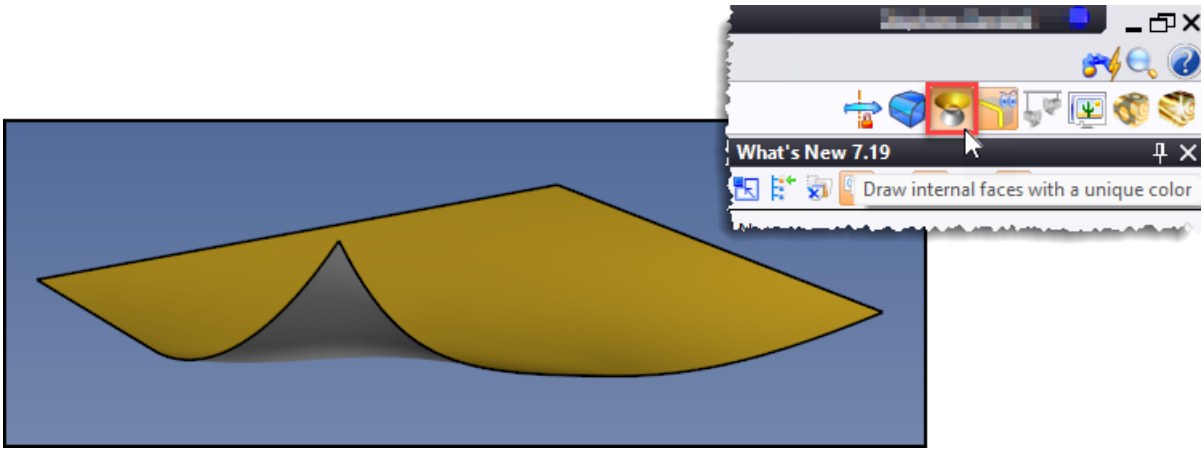
A specific preview is now calculated for local parts and assemblies.

Visualization

Material side

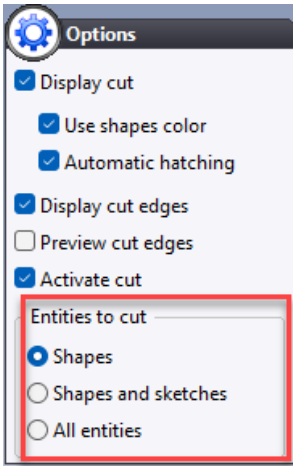
The new **Draw internal faces with a unique color** mode enables fast identification of the material side of the surface.

The color of the internal faces can be set in the **Tools** > **Options** > **Themes** section.



Graphic cut

When creating a graphic cut (by plane, by profile or by box), you can choose to cut only the shapes, the shapes and sketches, or all entities.

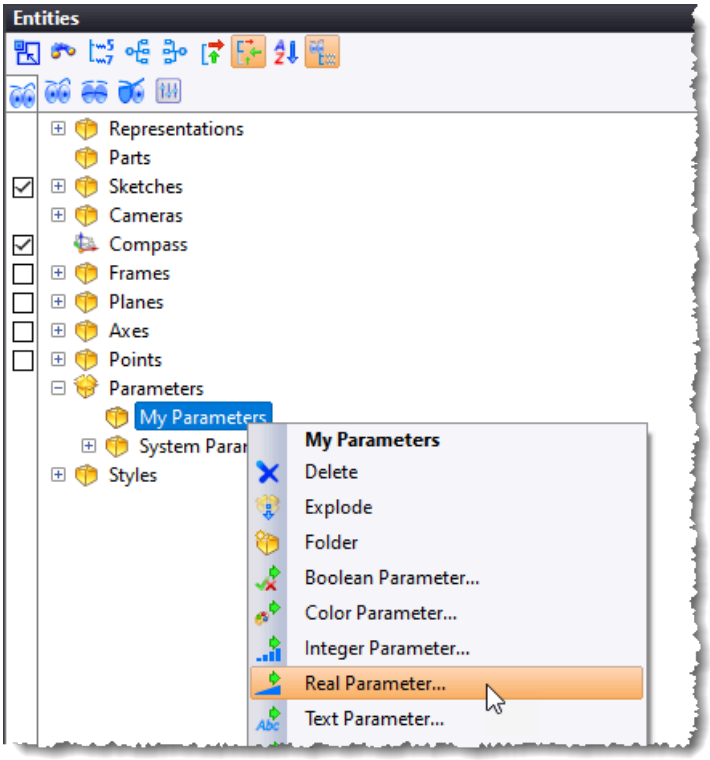


Construction

Parameters

Entities tree

It is possible to create a parameter from the context menu of a subfolder of the **Parameters** folder.



New functions

Two new functions are available in the parameter expressions:

- choose(val; test1; val1; test2; val2; ...; valDefault)
  Returns val1 if val < test1, then val2 if val < test2, etc. otherwise returns valDefault
- switch(cond1; val1; cond2; val2; ...; valDefault)
  Returns val1 if condition cond1 is true, otherwise val2 if condition cond2 is true, otherwise returns valDefault

Text parameter

The **Text** parameters include new functions (upper, lower, fillLeft, fillRight and replace), allowing you to modify the character strings.

- **Upper**: converts all characters to uppercase
- **Lower**: converts all characters to lowercase
- **fillLeft**: adds characters on the left of the character string.
- **fillRight**: adds characters on the right of the character string.
- **Replace**: replaces one character string by another

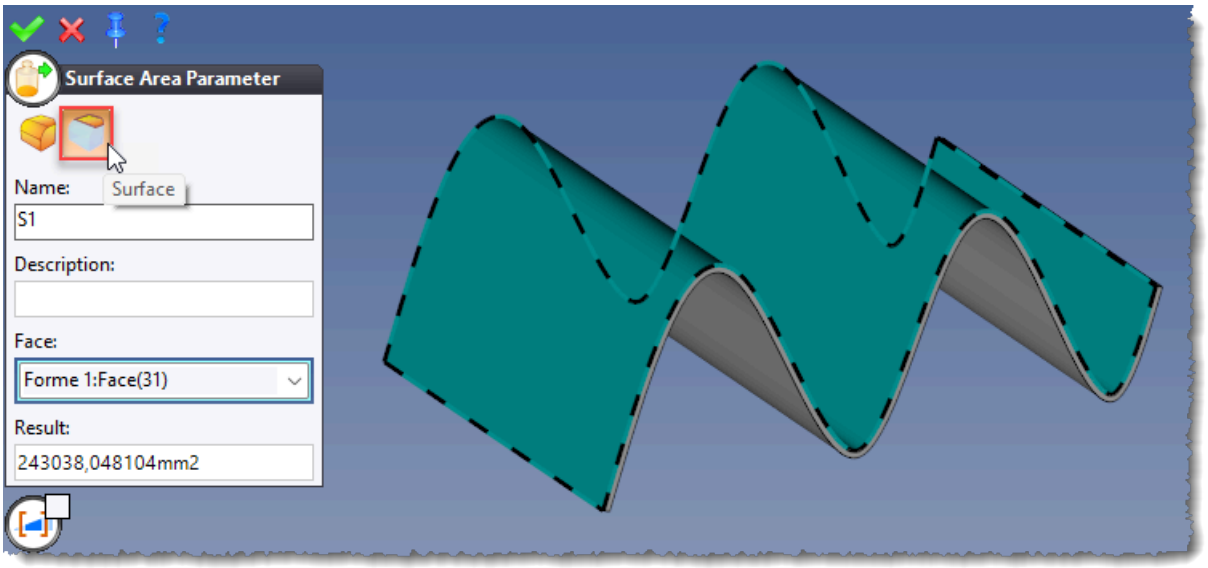
  Example:

  Let two text parameters be **A = "Design"** and **B = "7.19"**

Parameter	Formula	Result
C	=upper(A)	DESIGN
D	=lower(A)	design
E	=fillLeft(A; "TopSolid'";15)	TopSolid'Design
F	=fillRight(E;B;19)	TopSolid'Design7.19
G	=replace(F;B; "2025")	TopSolid'Design2025

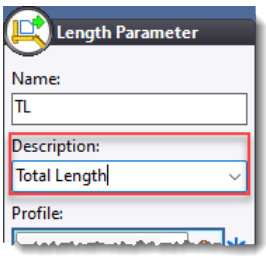
Surface area parameter

The **Surface** mode has been added to the **Surface Area Parameter** command. It allows the selection of a face on a shape.



Description

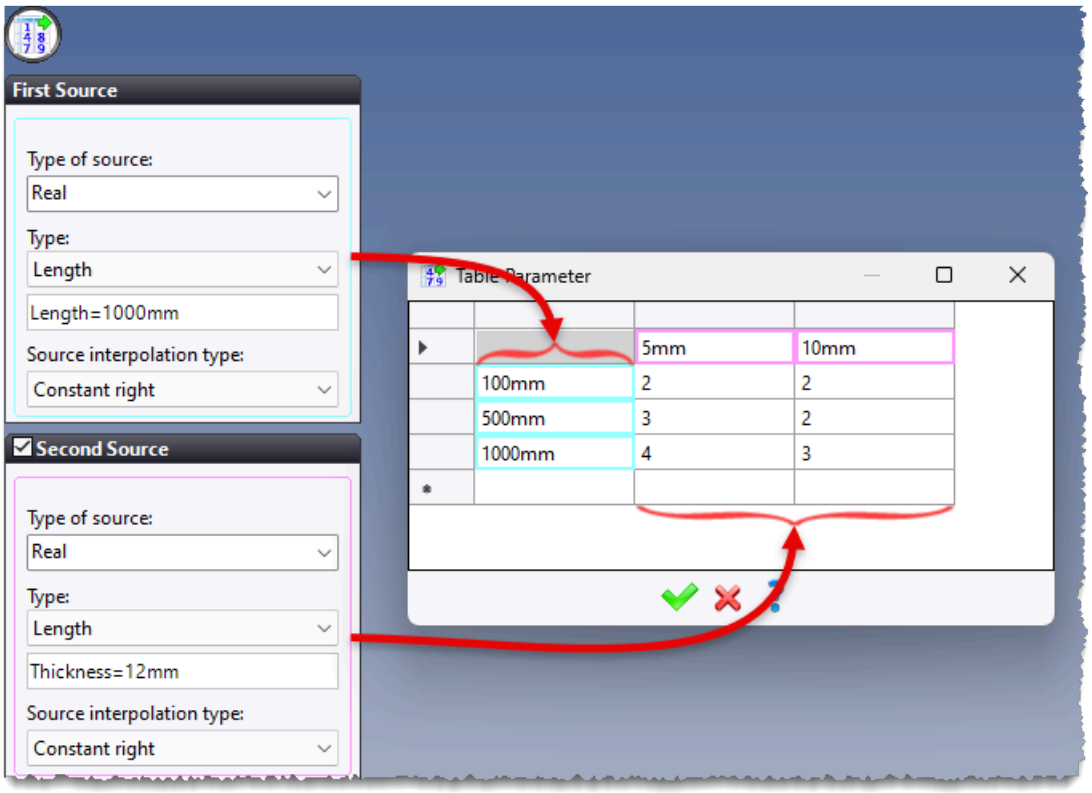
The **Description** field has been added to all parameter creation commands.



The following table parameters have been added and include two data sources:

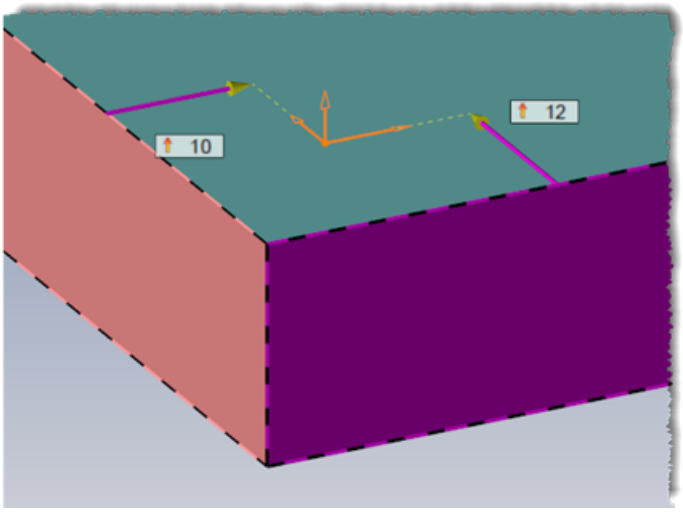
- Table Parameter
- Tabulated Integer Parameter
- Table Enumeration Parameter
- Table User Enumeration Parameter
- Table Text Parameter
- Table Material Parameter
- Table Coating Parameter
- Table Finishing Parameter

The **Table Family Parameter** and **Table Code Parameter** commands also feature two data sources.



Constrained frame

When creating a constrained frame, the positioning arrows are shifted to improve the frame's visibility.



Frame list

The **Frame List** command has been added to the **Construction** > **Frames** menu. It allows defining a set of frames consisting of a main frame and auxiliary frames.

This new entity can be used as a driver in a **Family** document.

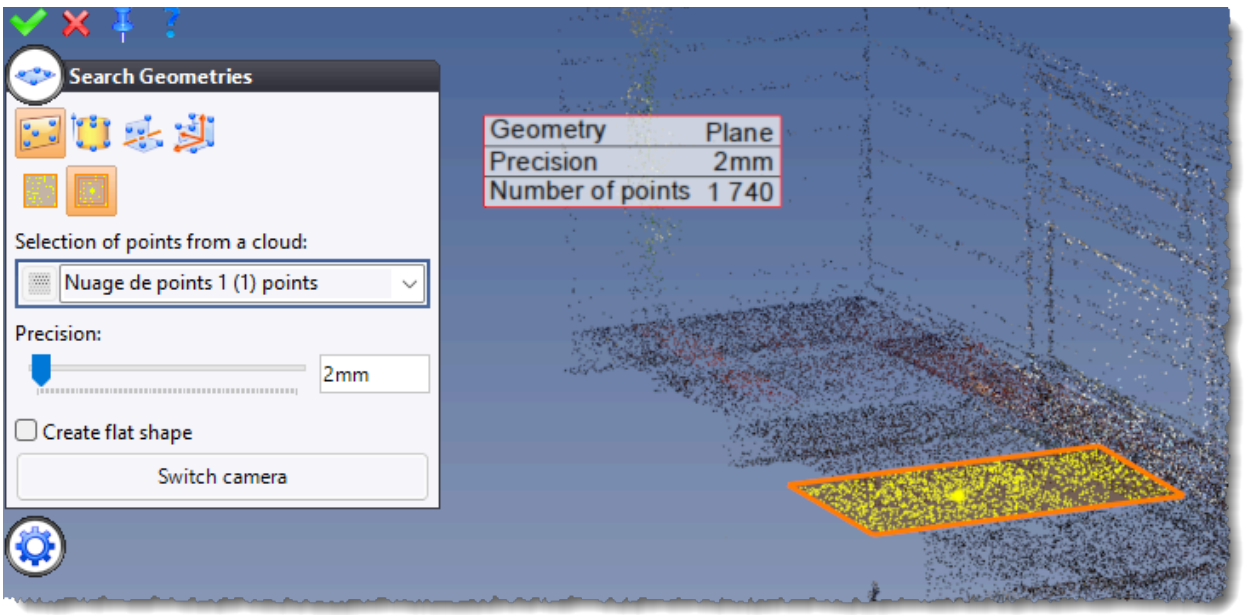
Point cloud

The geometry search command has been enhanced with several improvements:

Plane detection

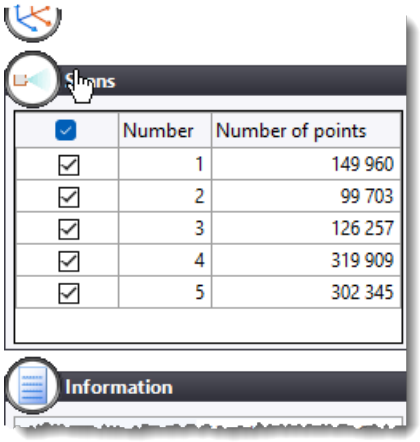
Two new detection modes are now available:

- **By sampling**: detection by selecting a list of points and specifying the plane's normal.
- **By propagation**: detection by selecting a list of points. This option also allows you to create a flat shape.



e57 file import

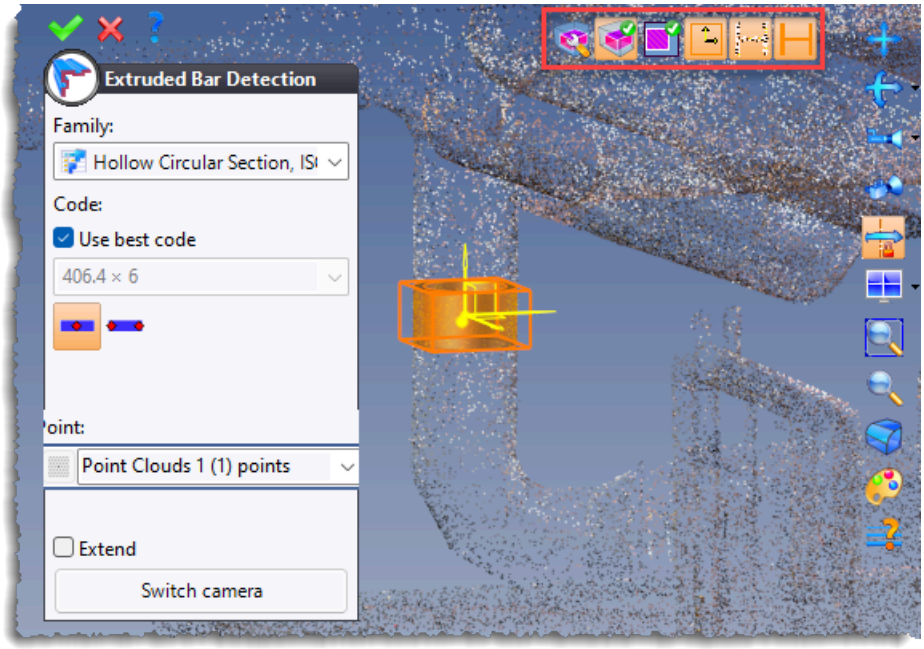
When an e57 file contains several point clouds, the dialog box for the **Construction** > **Point Cloud** command now features a panel for selecting the clouds to be imported.



Extruded bar detection

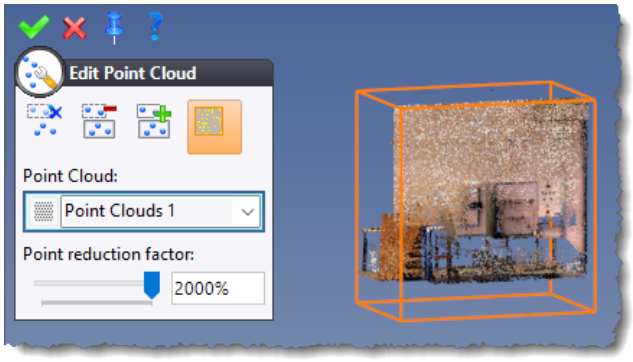
The new **Extruded Bar Detection** context command allows you to recreate extruded bars by specifying one or two points.

An icon bar to display the enclosing box, the section and extruded bar points is provided at the top right of the graphics area.

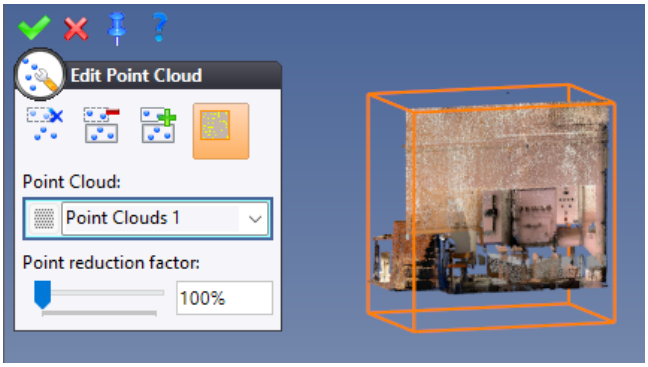


Refining

When editing a point cloud, the new **Refining** option increases the level of detail in certain areas.



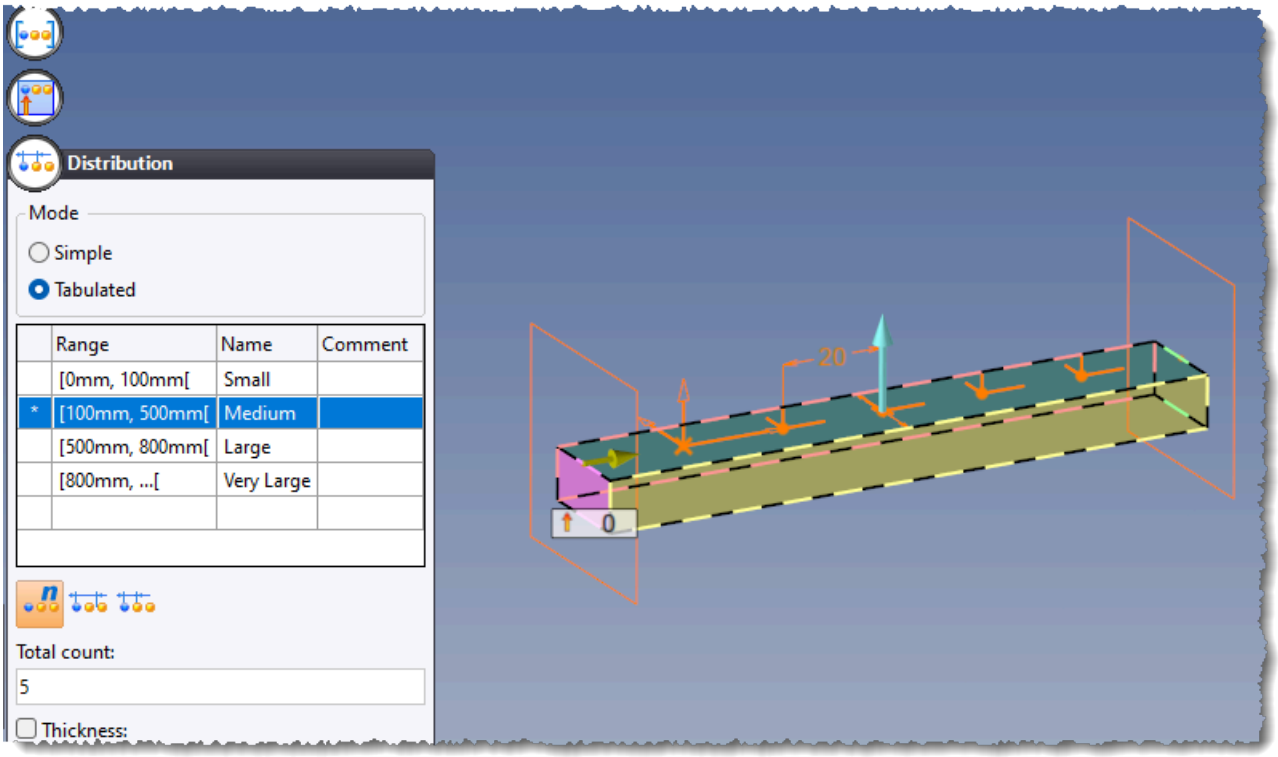
Example of 2000% refining



Example of 100% refining

Constrained linear pattern

The new **Tabulated** mode allows you to define different values (mode, number, etc.) for each interval.



Envelope block

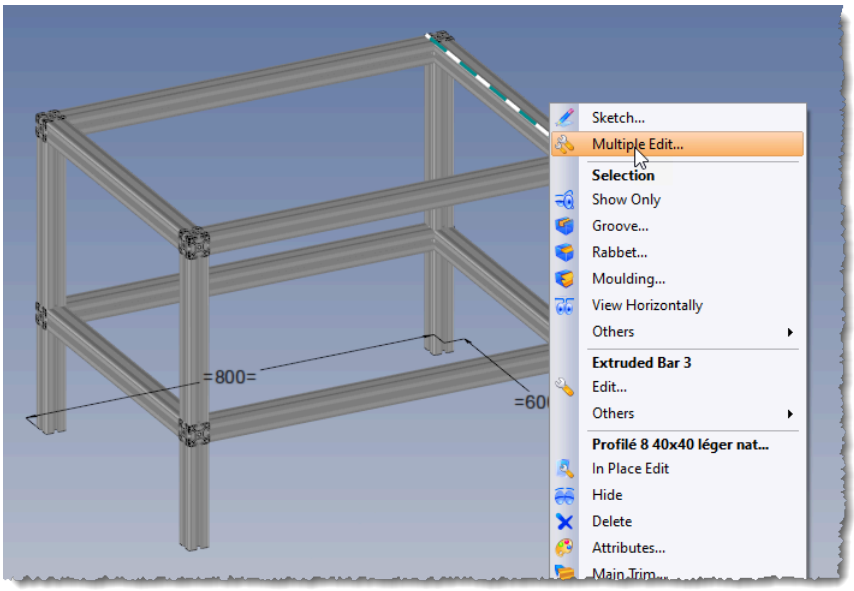
You can serialize a constrained envelope block.

Tools

Batch modification

The **Tools** > **Multiple Edit** command has been added to the menu.

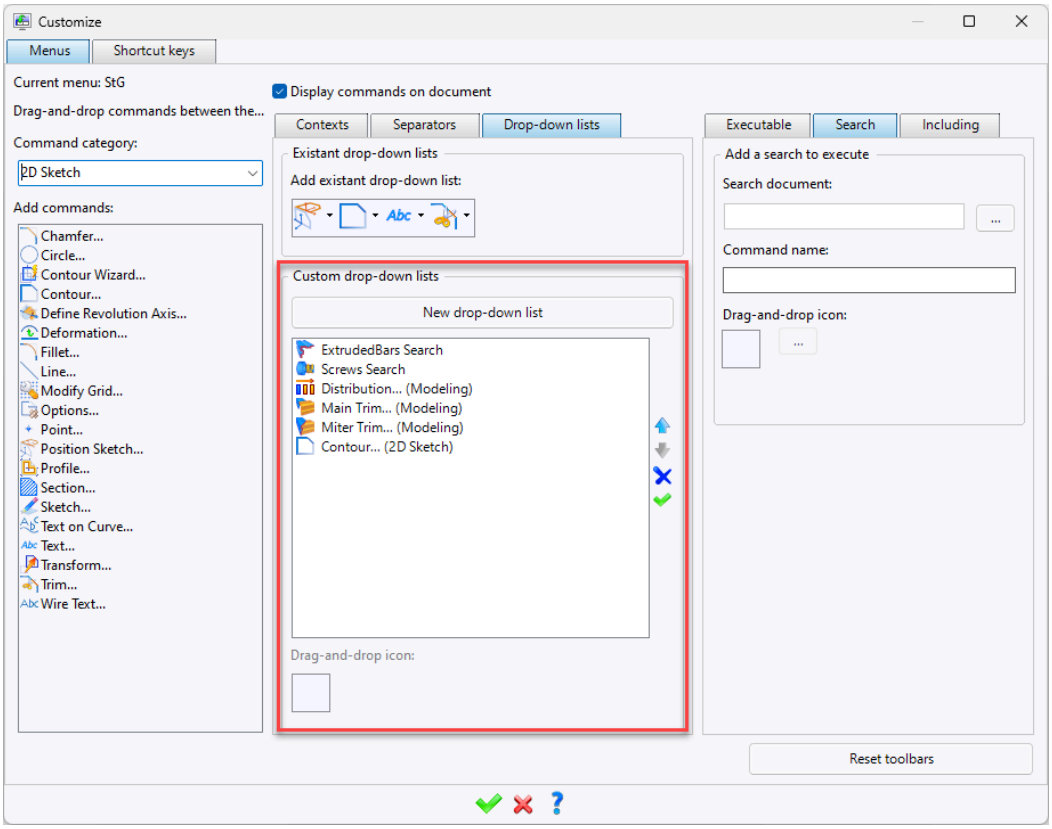
It allows you to modify several extruded bars or component inclusions at the same time. It can also be accessed from the top of the context menu of an extruded bar or an inclusion.



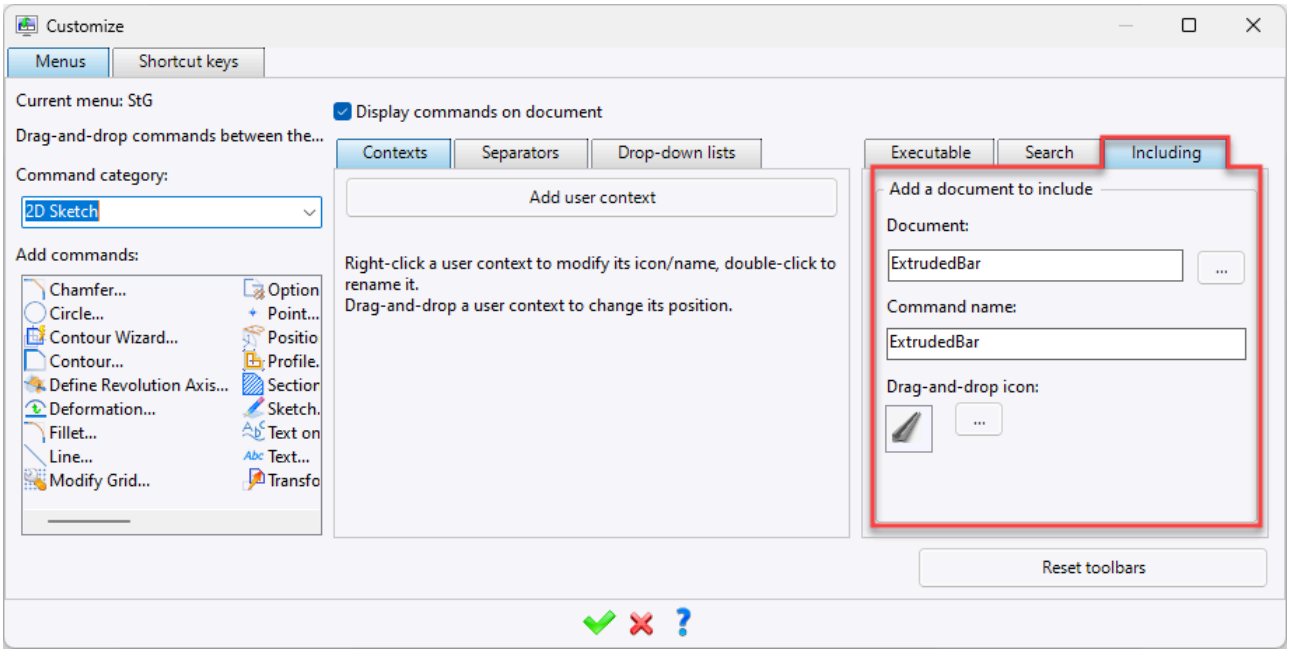
Drop-down list customization

The **drop-down lists** tab has been added to the window of the **Tools** > **Customize** command.

It allows you to create your own drop-down lists of icons and to modify existing ones.

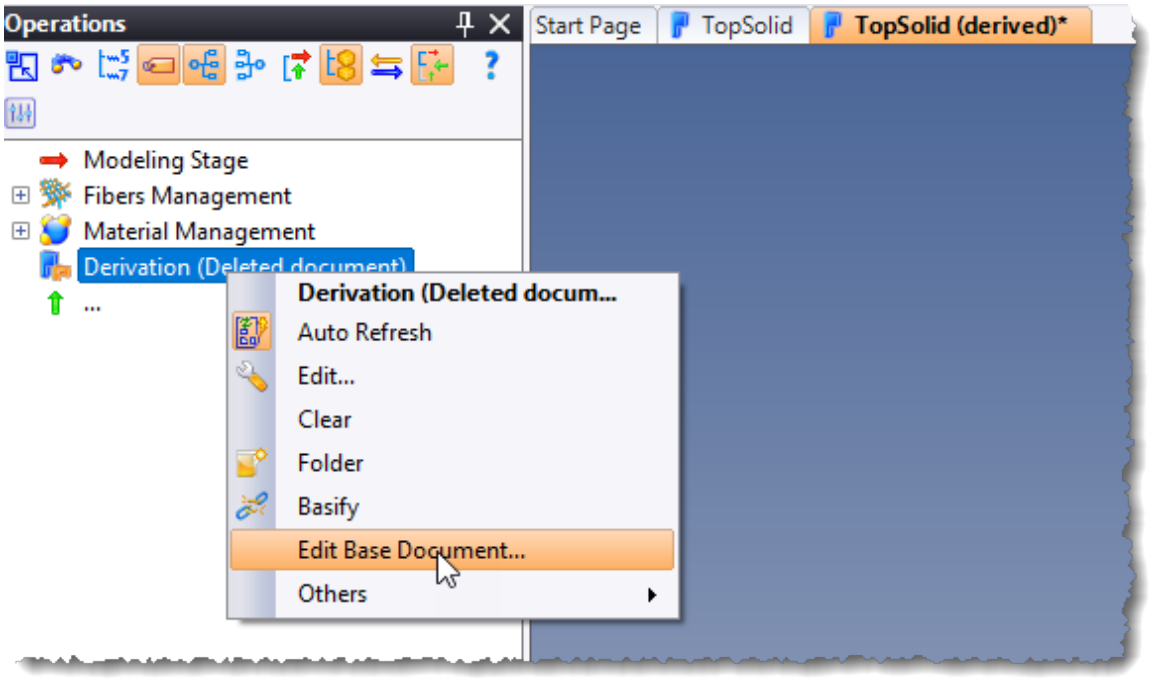


Moreover, when a document is frequently used (inclusion of a family, a method, etc.), the new **Including** tab allows you create an icon to include it.



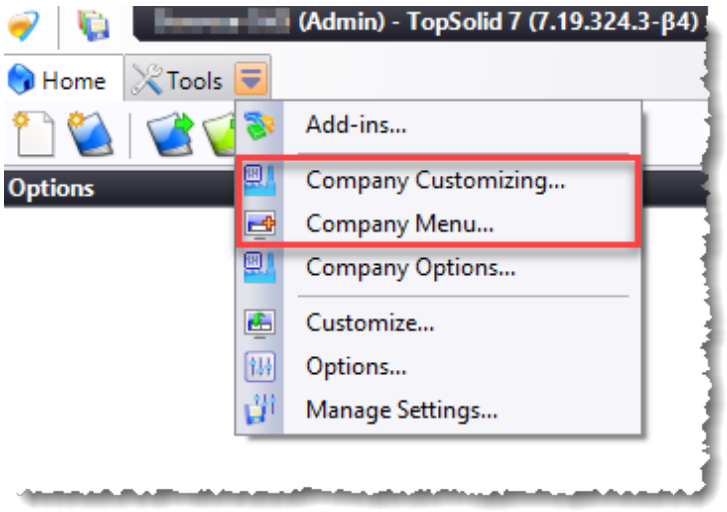
Derivation

From the **Derivation** operation in the Operations tree, you can change the derivation template. To do this, you must clear the document and then use the new **Edit Base Document** context command to select the new derivation template.

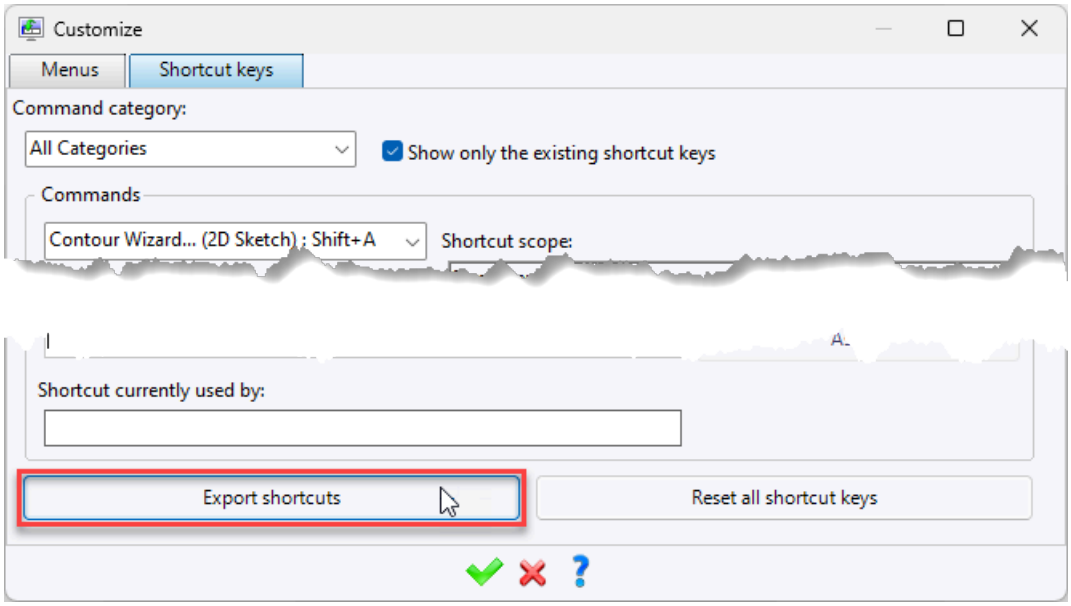


Menus and keyboard shortcuts

In client/server mode, menus and shortcuts can now be managed by the administrator in the same way as company options.

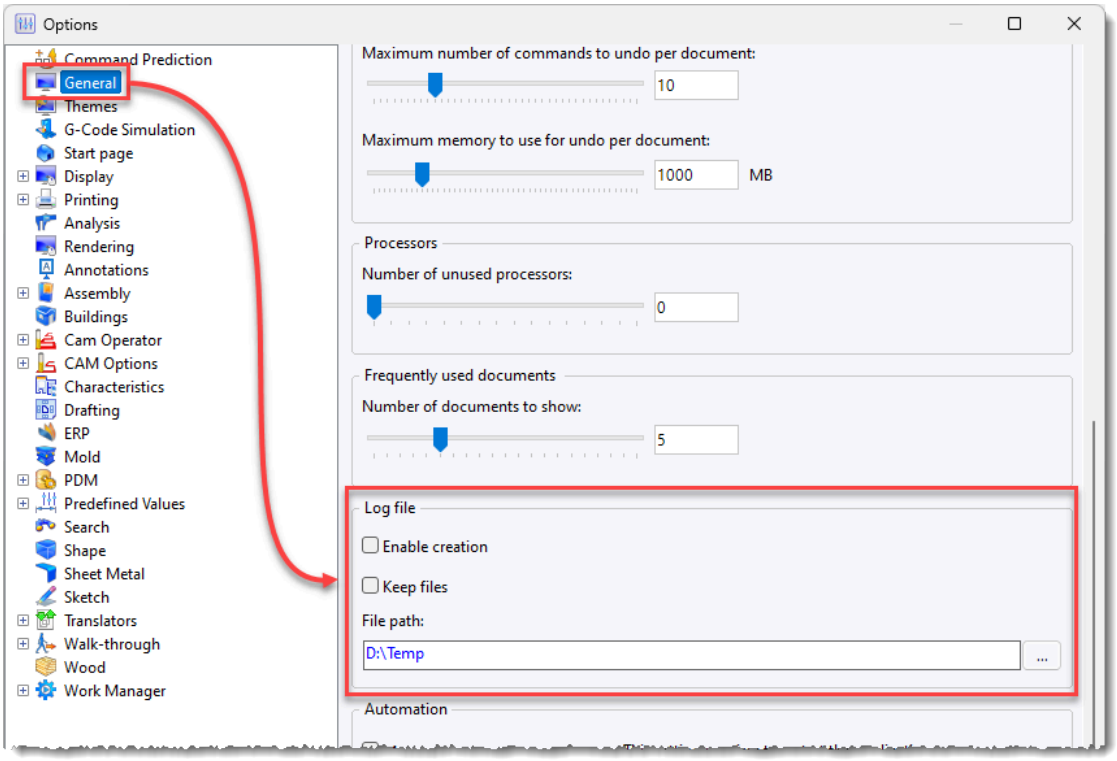


Moreover, the **Shortcut Keys** tab of the **Tools** > **Customize** command now features a new button for exporting the list of keyboard shortcuts in Excel (.CSV) format for printing.



Log file

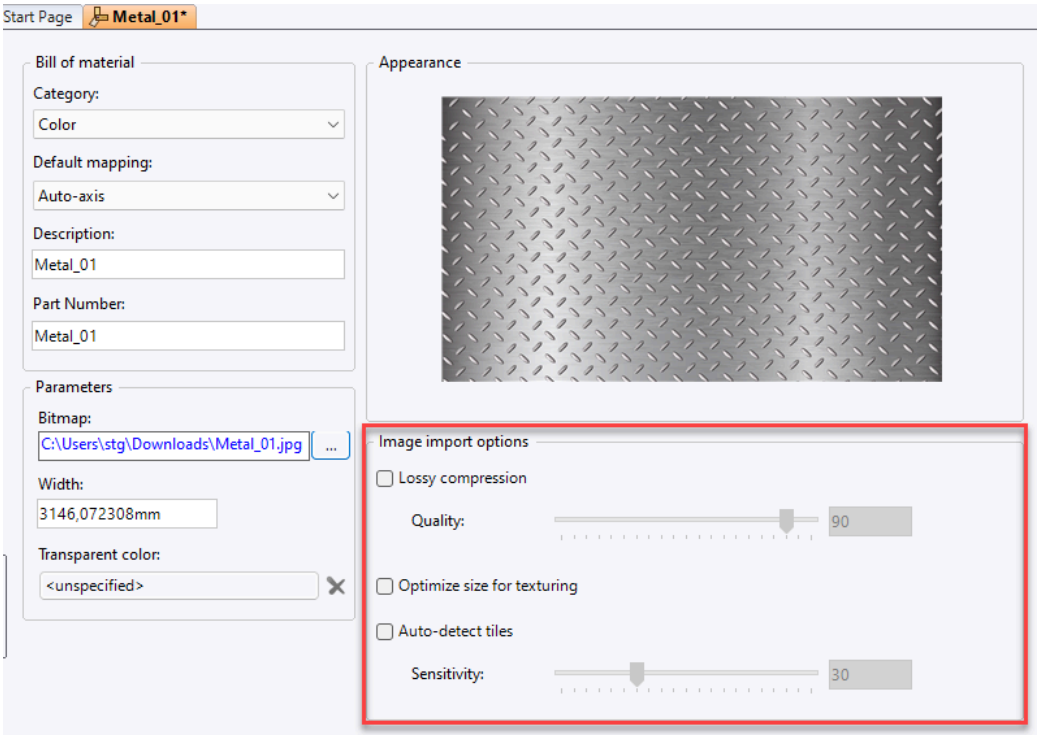
The creation of a log file to trace events, and the choice of its location can be defined from the **General** section of the **Tools** > **Options** command.



Materials and Textures

Optimization

When creating a texture, new options allow optimizing the size of the created documents.



Nesting (optional module)

Created document naming

You can now set the naming of documents created by nesting, according to the following two options:

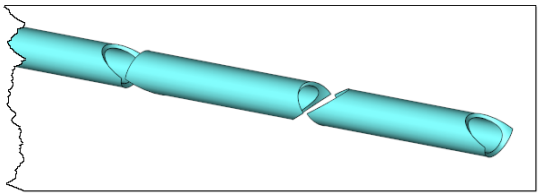
- A root name, followed by a counter with a prefix (for example, IMB_n°1, IMB_n°2, IMB_n°3).
- Document names are created based on the root name set in the template document. This allows, for example, documents to be named according to the machining program used.

Bar nesting

Non-planar-cut straight sections

Previously, straight sections with non-planar cuts were not correctly processed by bar nesting.

In this new version, the lengths of these sections are automatically computed in the overall length mode rather than in the extended length mode. In this particular case, the length displayed in the nesting will be 0mm.



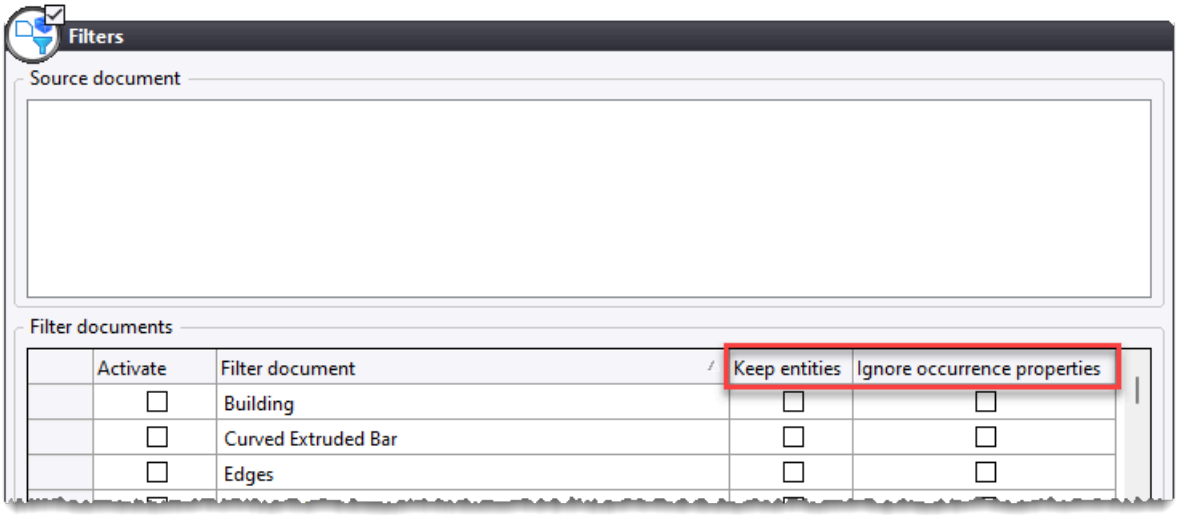
Created document naming

The bar nesting module now includes the same improvement for document naming as the nesting module (see chapter *Created document naming* (on page 34)).

Drafting

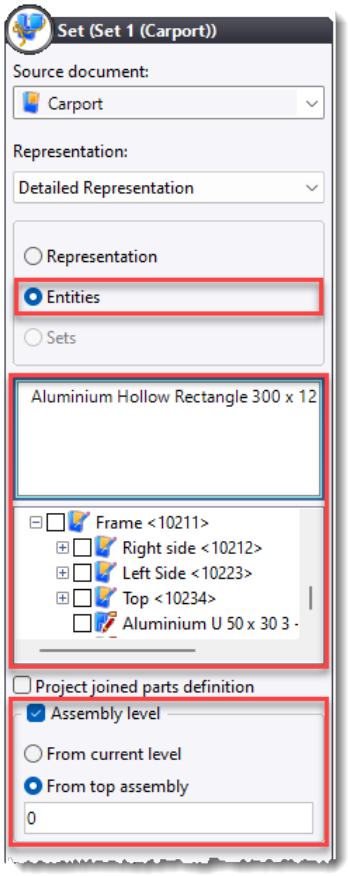
Projection set

The dialog box of the **Filters** section has been modified. It now allows you to keep entities and ignore occurrence properties.

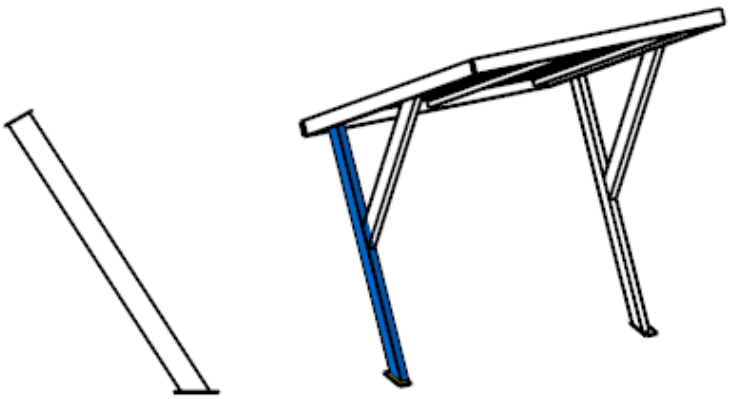


Assembly level choice

The **Entities** mode from the **View** > **Set** command features a new section for selecting the assembly level.

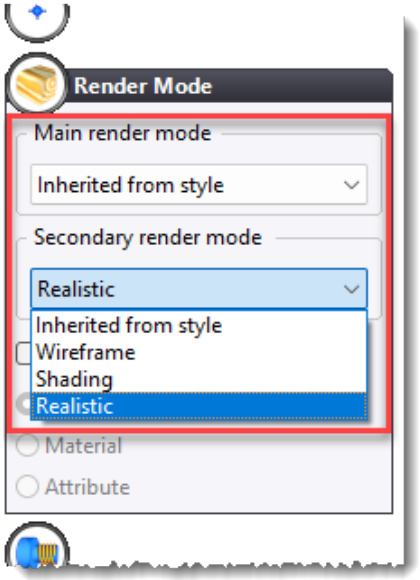


This feature can be used with an exception style, as shown below, and with the new **Automatic Exception** context command in order to automatically localize a part in the assembly view.

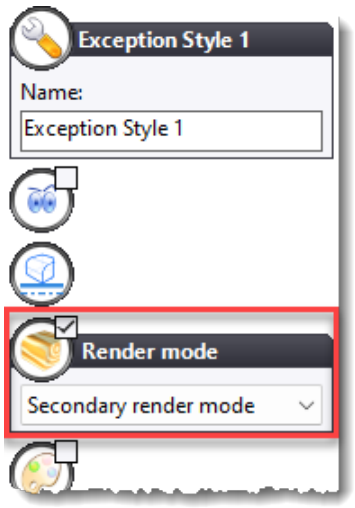


Exception style

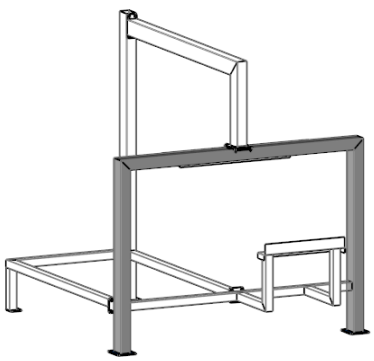
In the view creation or editing dialog box, the **Render Mode** section now includes two drop-down lists that allow you to define a primary render mode and a secondary render mode.



The **Render Mode** section has also been added to the **Exception Style** command. It allows you to choose between the primary and secondary render modes for the selected parts.

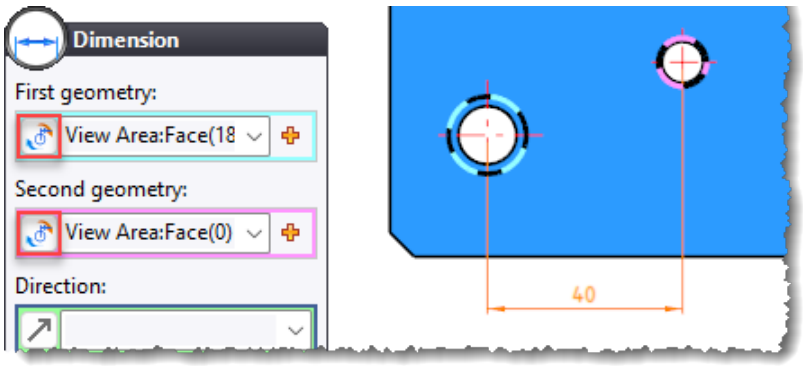


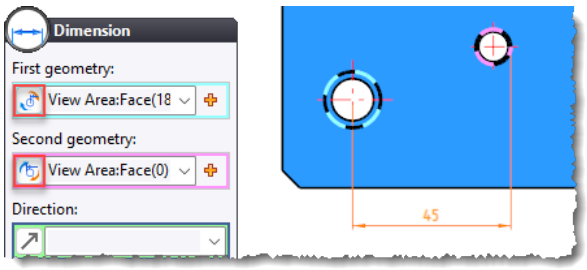
You will get the view as shown below:

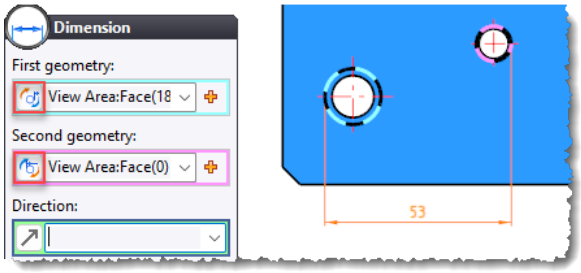


Circle dimensioning

When selecting a circle, a three-state button allows you to hook either from the center, or from one side or the other of the circle.

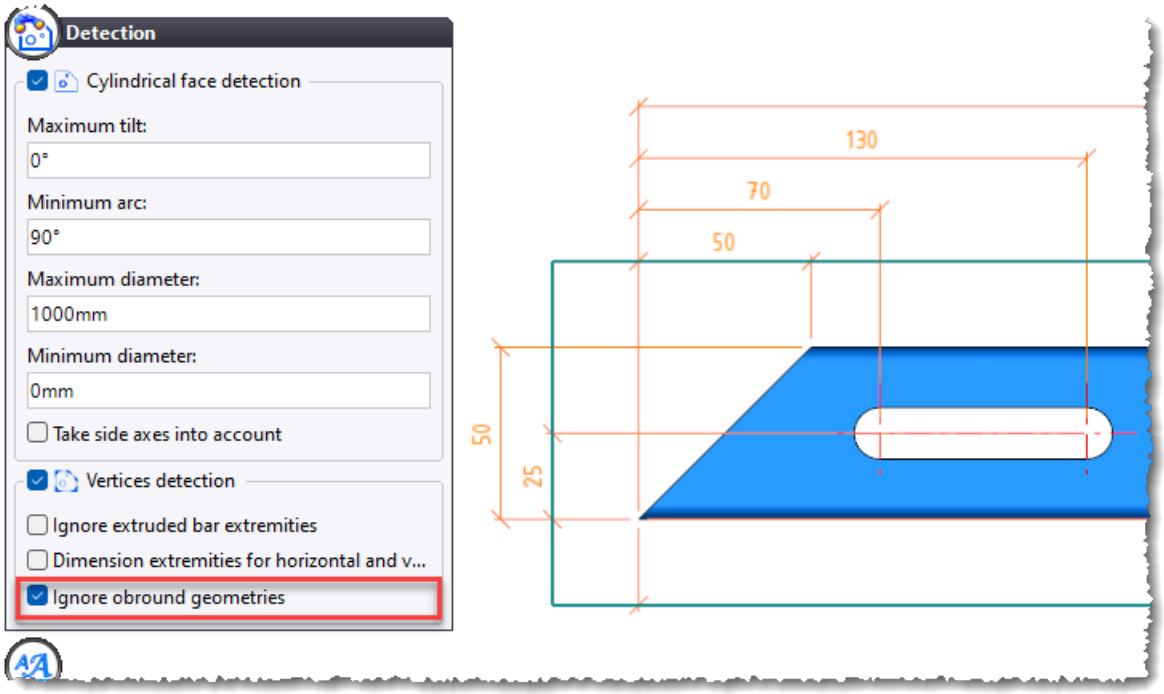


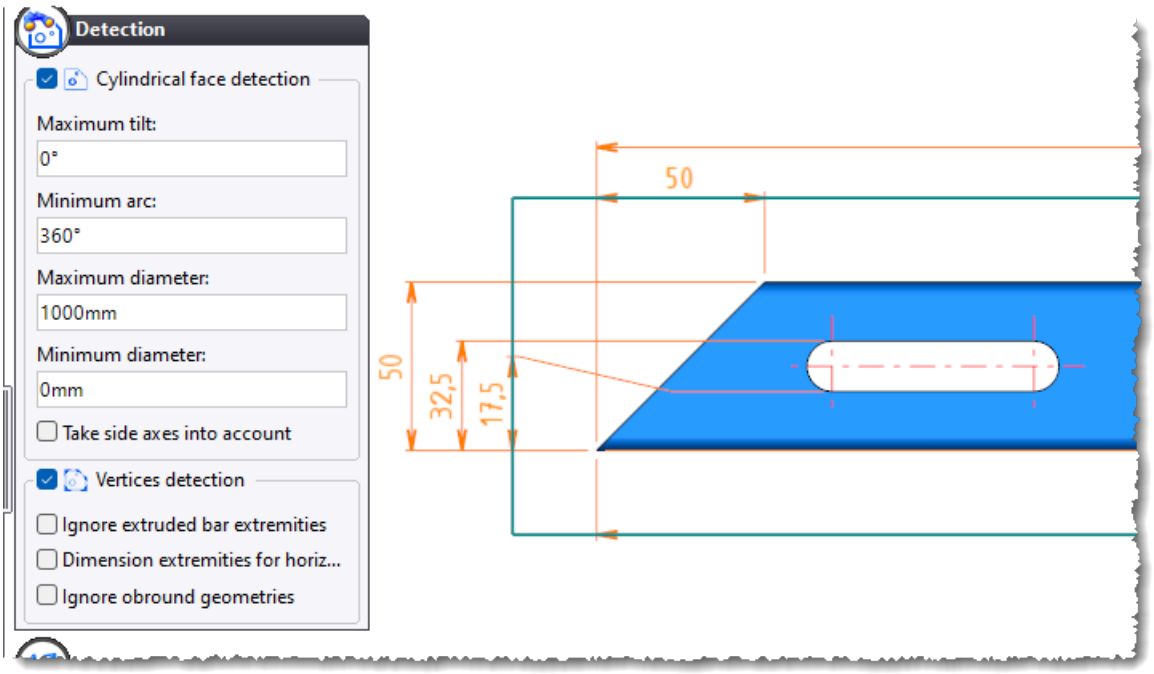


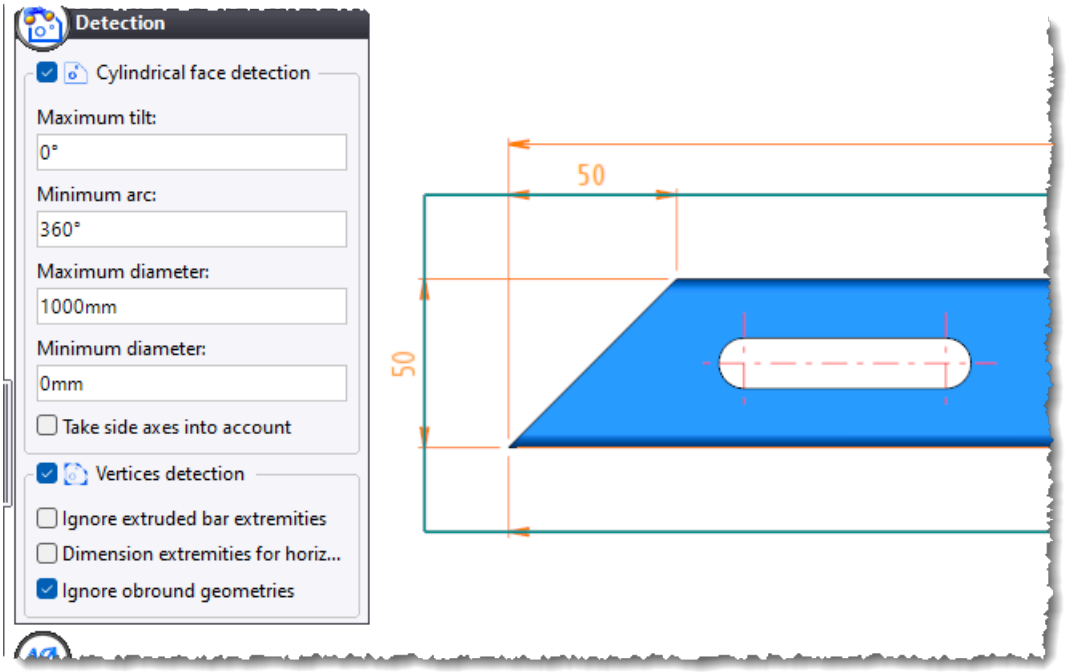


Automatic composite dimensions

The **Ignore obround geometries** option allows dimensioning only the centers of the circular arcs of obround shapes, which helps simplify the dimensioning.

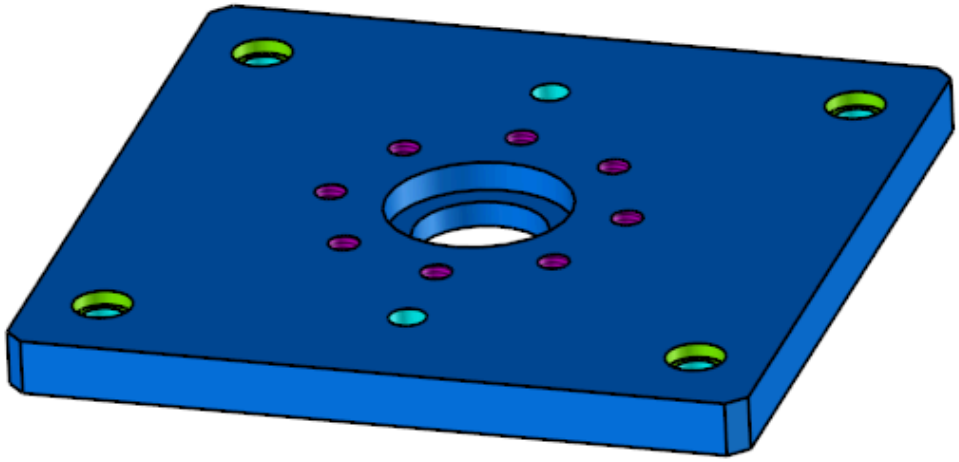



Detection
Cylindrical face detection
Maximum tilt:
0°
Minimum arc:
360°
Maximum diameter:
1000mm
Minimum diameter:
0mm
Take side axes into account
Vertices detection
Ignore extruded bar extremities
Dimension extremities for horiz...
Ignore obround geometries
50
50
32,5
17,5


Detection
Cylindrical face detection
Maximum tilt:
0°
Minimum arc:
360°
Maximum diameter:
1000mm
Minimum diameter:
0mm
Take side axes into account
Vertices detection
Ignore extruded bar extremities
Dimension extremities for horiz...
Ignore obround geometries
50
50

Machining process

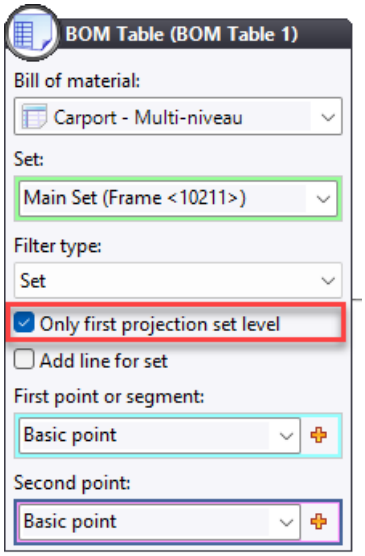
Information about a machining process (document name and major revision, process name, and RGB color values) can be retrieved in a note and/or a drilling table. These properties can be defined in a dimension style or a table style.

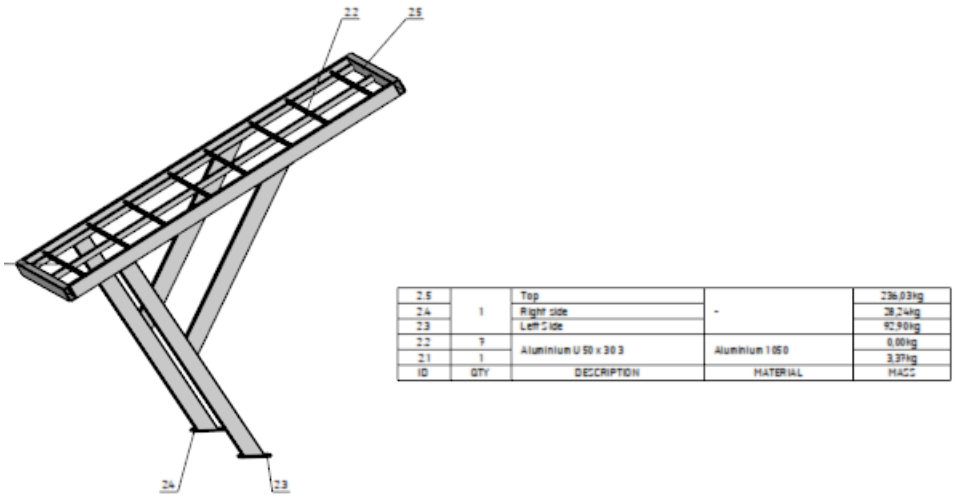


Name	Type	Symbol	Diameter	Coordonnées	Altitude	Work depth	Document name	Machining process	RGB
Tapped hole 18	Tapped hole		M10	128,28mm x 128,28mm	0mm	Through	MachiningProcess1 - A	Process 2	R=255 G=0 B=255 (Magenta)
Tapped hole 17	Tapped hole		M10	100mm x 140mm	0mm	Through			R=255 G=0 B=255 (Magenta)
Tapped hole 16	Tapped hole		M10	140mm x 100mm	0mm	Through			R=255 G=0 B=255 (Magenta)
Tapped hole 15	Tapped hole		M10	71,72mm x 128,28mm	0mm	Through			R=255 G=0 B=255 (Magenta)
Tapped hole 14	Tapped hole		M10	128,28mm x 71,72mm	0mm	Through			R=255 G=0 B=255 (Magenta)
Tapped hole 13	Tapped hole		M10	60mm x 100mm	0mm	Through			R=255 G=0 B=255 (Magenta)
Tapped hole 12	Tapped hole		M10	100mm x 60mm	0mm	Through			R=255 G=0 B=255 (Magenta)
Tapped hole 11	Tapped hole		M10	71,72mm x 71,72mm	0mm	Through			R=255 G=0 B=255 (Magenta)
Spot facing 10	Spotfacing		16mm	180mm x 175mm	0mm	4mm		Process 1	R=128 G=255 B=0
Spot facing 9	Spotfacing		16mm	180mm x 25mm	0mm	4mm			R=128 G=255 B=0
Spot facing 8	Spotfacing		16mm	20mm x 175mm	0mm	4mm			R=128 G=255 B=0
Spot facing 7	Spotfacing		16mm	20mm x 25mm	0mm	4mm			R=128 G=255 B=0
Hole 6	Hole		12mm	180mm x 175mm	0mm	Through		Process 3	R=0 G=255 B=255 (Cyan)
Hole 5	Hole		10,2mm	100mm x 167mm	0mm	Through			R=0 G=255 B=255 (Cyan)
Hole 4	Hole		12mm	180mm x 25mm	0mm	Through			R=0 G=255 B=255 (Cyan)
Hole 3	Hole		12mm	20mm x 175mm	0mm	Through			R=0 G=255 B=255 (Cyan)
Hole 2	Hole		10,2mm	100mm x 33mm	0mm	Through			R=0 G=255 B=255 (Cyan)
Hole 1	Hole		12mm	20mm x 25mm	0mm	Through			R=0 G=255 B=255 (Cyan)

Automatic indexing

The **Only first projection set level** option has been added to the **BOM Table** and **Automatic Indexing** commands. It allows you to not index all the subassemblies and parts located in the different levels of the assembly.

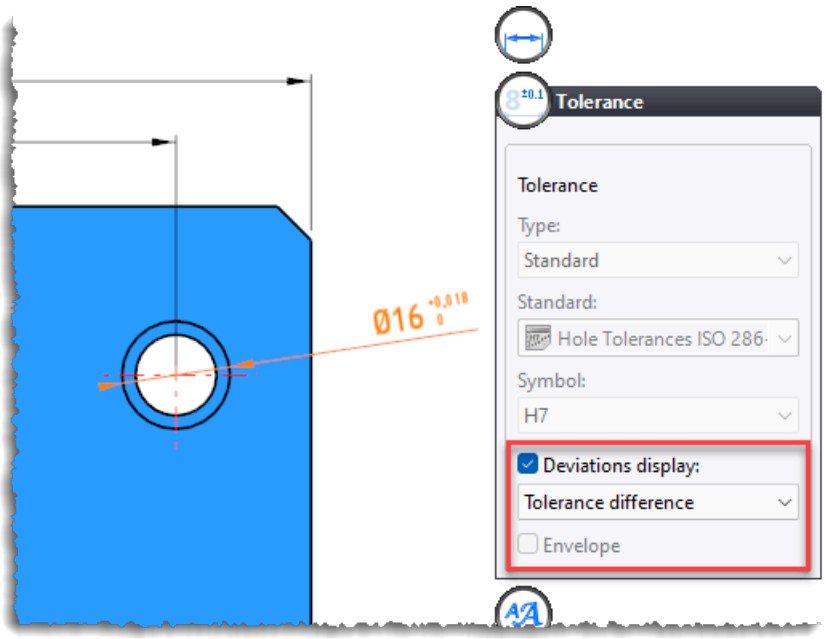




Projected annotations

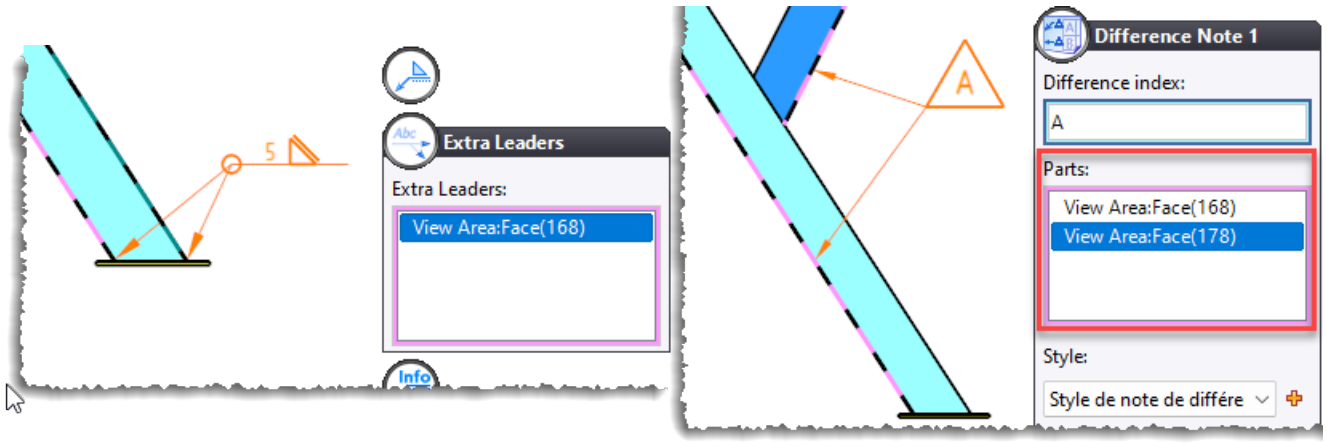
The **Tolerance** section has been added to the dialog box for editing a projected annotation.

It modifies the tolerance display format for the 3D document.



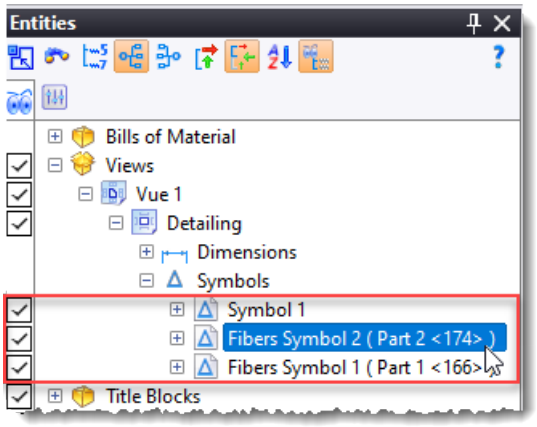
Weld symbol and difference note

The **Extra Leaders** section has been added to the dialog box of the **Detailing** > **Weld Symbol** command. The dialog box of the **Detailing** > **Difference Note** command has been modified to allow multiple parts to be selected.



Symbols

The visibility of symbols of a view is managed by a checkbox in the Entities tree. For fiber symbols, the part name and occurrence are displayed after the symbol name.



Unfolding

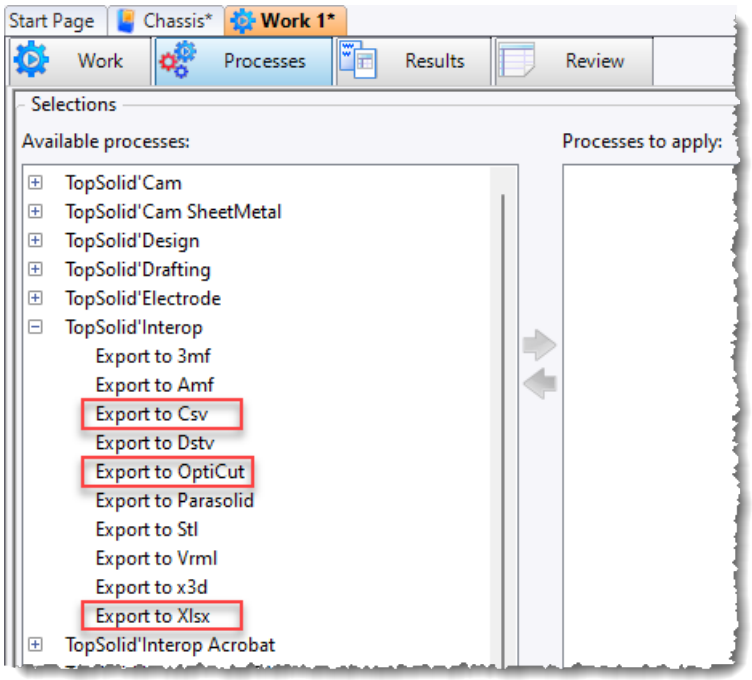
Open document

The new **Open Document** context command allows directly opening the document included in the unfolding document without having to select the part in the graphics area.

Work document and Work Manager

Export

Two new processes for exporting a bill of materials in **CSV**, **Xlsx**, or **OptiCoupe** format have been added under the **TopSolid'Interop** node.



2D modeling

Symbol

The **Profiles** option has been added to the **Symbol** command. As a reminder, this option transforms the symbol's entities into many profiles, and then allows the use of the **Trim** command.

FEA

Static analysis (cm² solver)

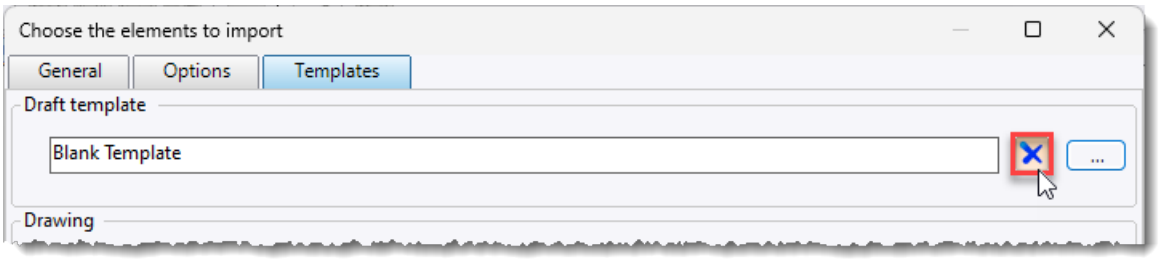
When creating a new analysis, the thermal analysis option was deleted.

The commands for defining thermal loads are now available in the **Analysis** menu during a linear static analysis.

Translators

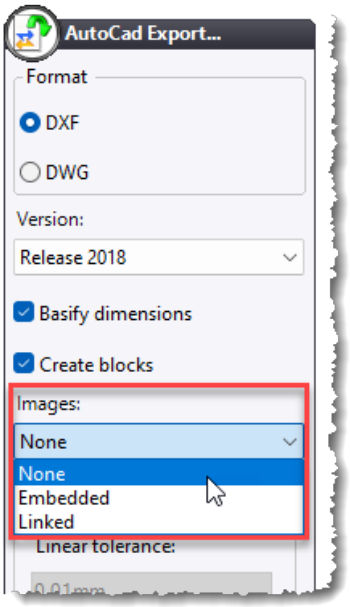
Selection of templates

When importing a document, the new **Delete** button allows you not to use templates.



AutoCAD

When exporting a drafting document in **AutoCAD DXF/DWG** format, the **None** option allows you to exclude images and shaded views from the export.

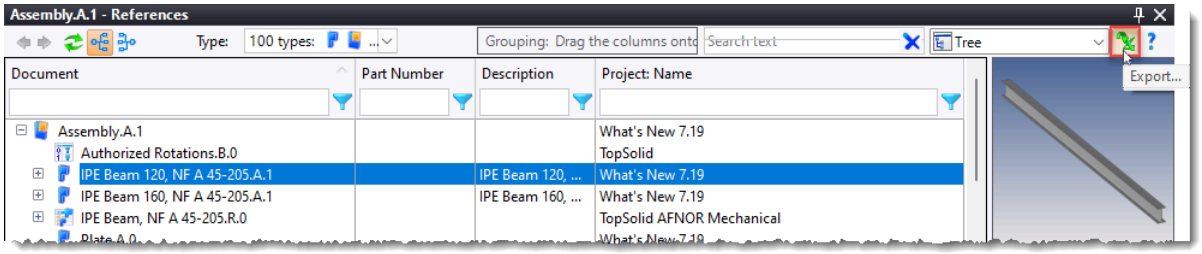


STEP import

When importing an assembly, the new **Import as local parts** option groups all parts in the assembly document and avoids creating multiple documents.

Table export

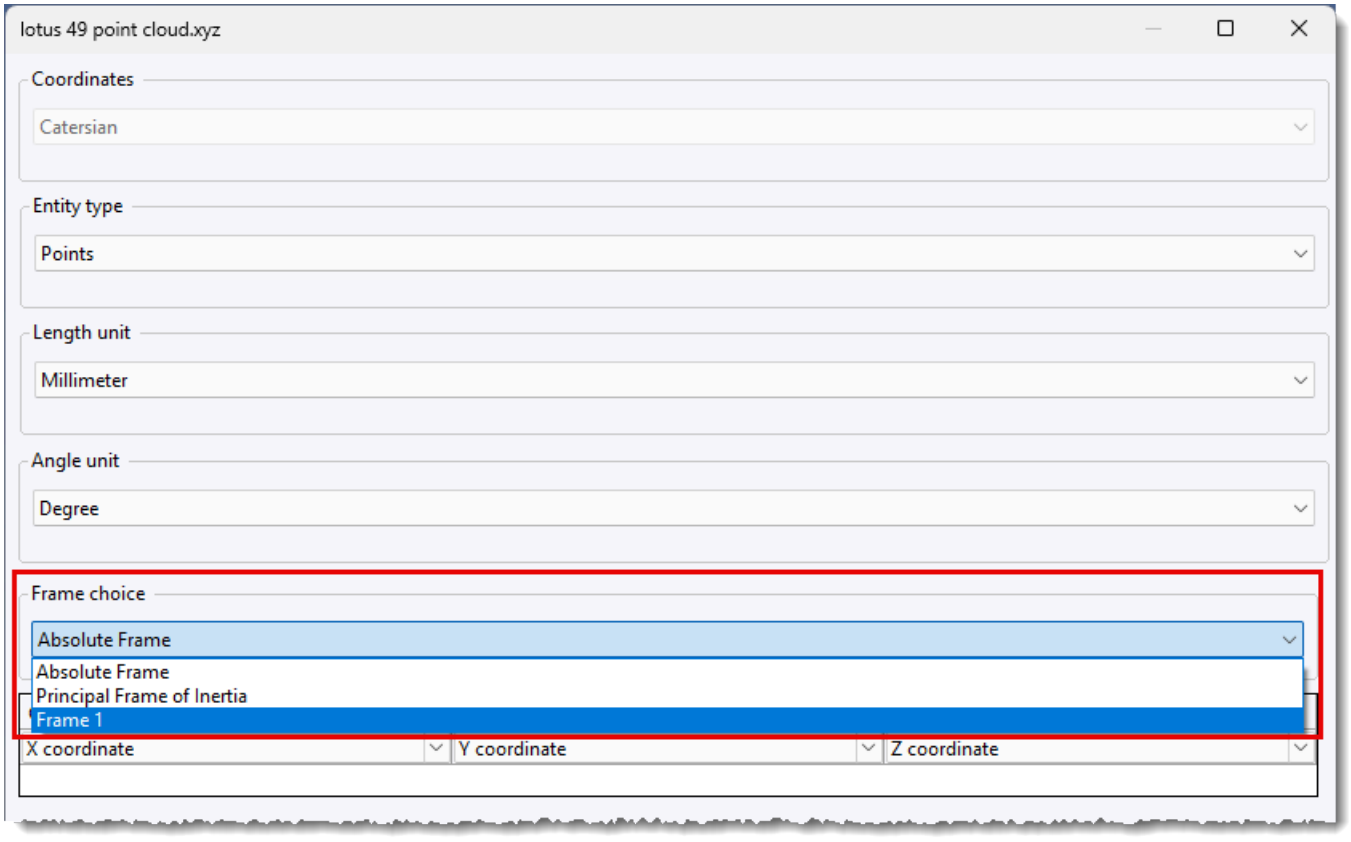
A button for exporting tables in CSV format has been added to the reference and back references windows, as well as the Parts tree of an assembly.



Point file import

When importing a point file (.xyz file) using the **TopSolid7** > **File** > **Translators** > **Import File into Document** command, it is possible to select the point insertion frame.

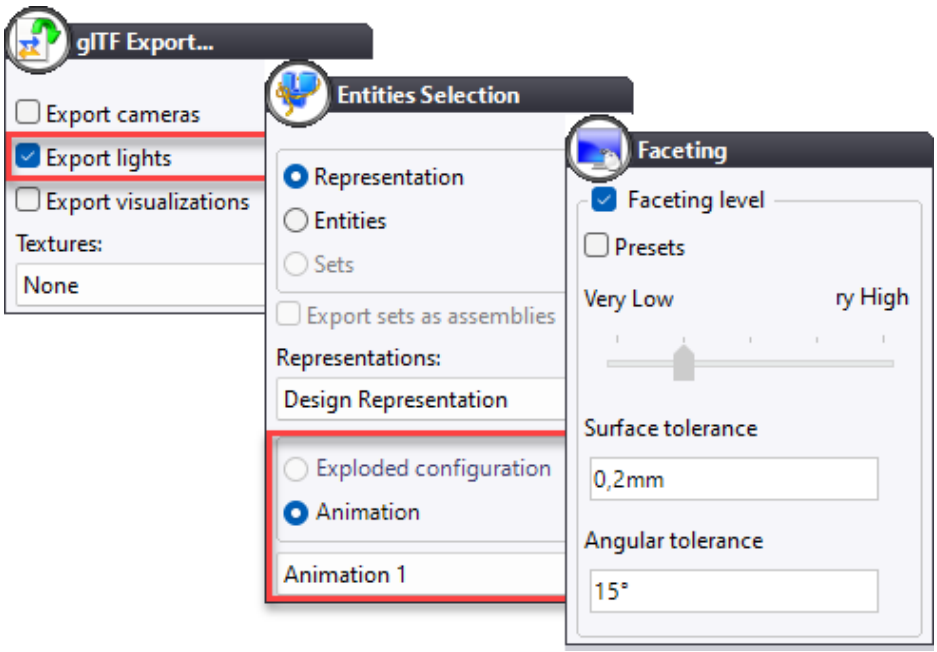
In addition, the start of the point file is displayed at the bottom of the window, and you can choose the meaning of each column (X coordinate, Y coordinate, Radius, Angle, etc.) or ignore it.



GITF export

In the **Tools** > **Options** > **Translators** > **gITF** command, the new **Maximum Texture Size** section in the **gITF** node allows you to define the size of exported textures. This helps reducing the generated file size.

In addition, options for exporting lights, managing exploded animations and managing faceting have been added to the dialog box.



PDF export

The **Export document with conversion** context command now supports the **Drafting Bundle** documents. All the plans in the bundle are exported as pages in the PDF document.

JT importing

PMIs contained in the documents are supported.

Obj import

The **Import textures** option has been added. It automatically creates **Texture** and **Coating** documents and assigns the coating to the imported part.

IFC Interface

Piping lines export

The size of the IFC export of a piping line with tubes and elbows has been greatly reduced. Elbows and tubes are now transformed into polyhedrons which significantly reduces the file size.

Extruded bars export

In the IFC export settings, the new **Create explicit geometry for profiles** option converts profiles into arcs and lines.

IfcElementAssembly import

When the **Create local parts** option is checked, **IfcElementAssembly** entities are imported as local assemblies.

File size

During an IFC export, not meaningful zeros are now deleted, significantly reducing file size.

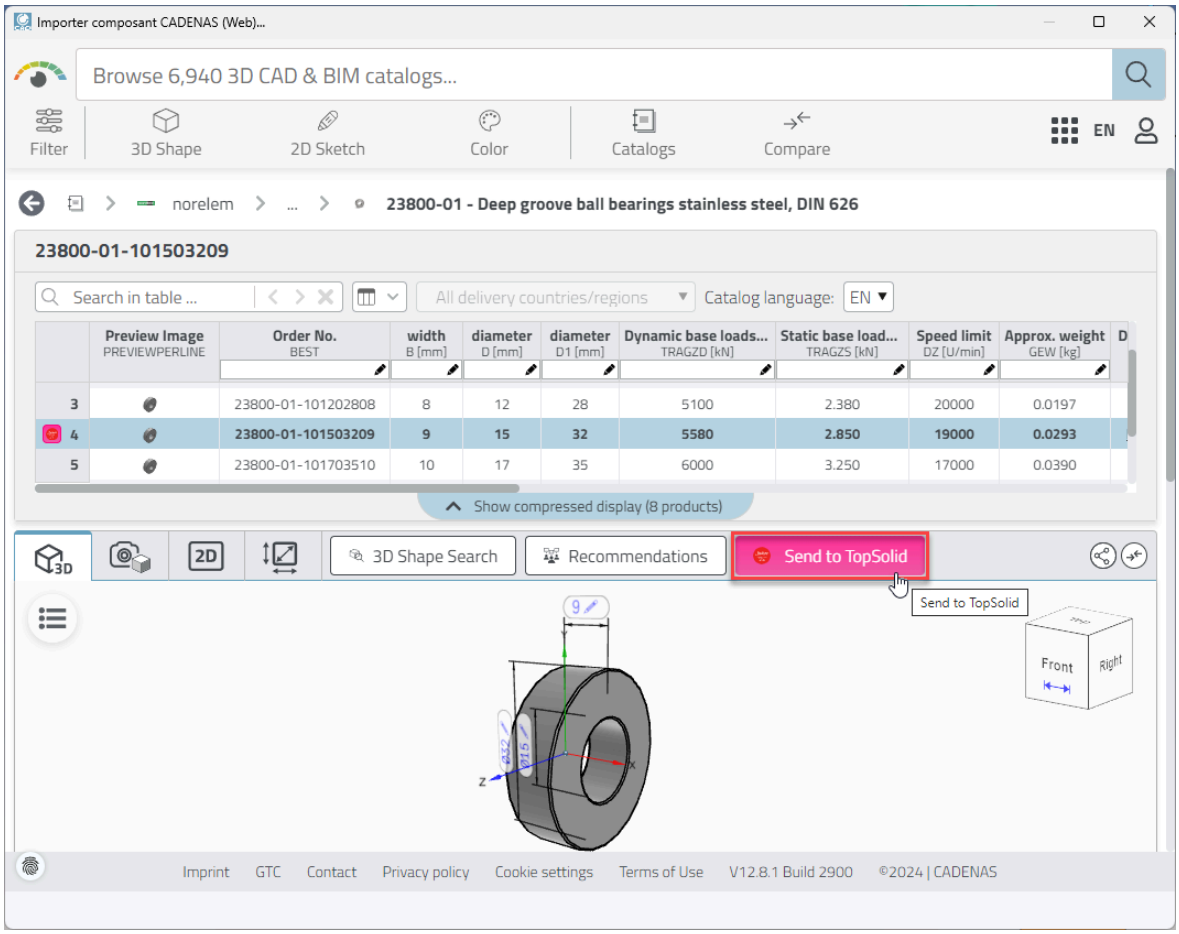
Moreover, similar geometries are also factorized via the IFCSHAPEREPRESENTATION entity.

DSTV Interface

When the **Tools** > **Options** > **Translators** > **General** > **Display output results** option is checked, an export result file (HTML file) of several documents is created in the same folder as the DSTV files.

CADENAS

The link with the Cadenas website has been modified. The **File** > **Translators** > **Import CADENAS (Web) component** command directly references the CADENAS search website and allows you to directly include the component in **TopSolid**.



What's New in TopSolid'Virtual 7.19

Discover the latest features of **TopSolid'Virtual**, **TopSolid 7**'s 3D visualization and virtual reality software!

Navigation

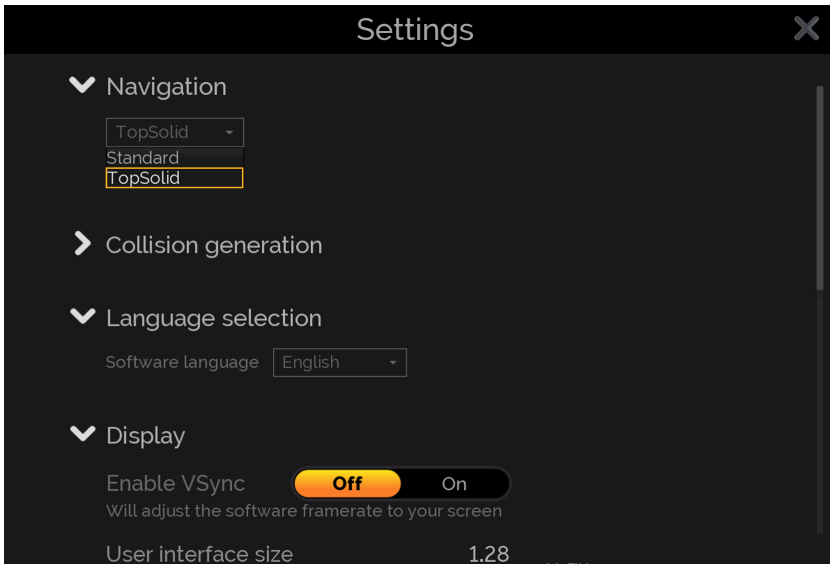
Focus, Drone and Walk modes

The navigation system has been completely redesigned to simplify your exploration of architectural and layout projects. You can now choose from three modes tailored to your needs:

- **Focus** mode: Ideal for rotating around objects.
- **Drone** mode: Provides complete freedom of movement in all directions.
- **Walk** mode: Offers a human-height immersion for a realistic exploration of your project.

TopSolid user settings

A **TopSolid** user preset has been integrated to offer a seamless transition for users already familiar with the **TopSolid** interface. This option ensures the mouse management is identical to that of **TopSolid**, providing a smooth and intuitive user experience.



Virtual reality surface detection

The teleportation tool now detects accessible surfaces, allowing you to move over them instantly. With a single click, you can ascend to another floor, access a rooftop, or dive to the bottom of a pool, offering fast and immersive navigation.

Environment

New physical outdoor environment

In addition to the existing HDR lighting system, a new physical environment featuring sky, sun and clouds has been integrated. You can now precisely adjust the position, time and orientation of your project, simulating the real impact of sunlight. This feature not only enhances rendering quality, but also allows designers to visualize the actual effects of shadows and sunlight on their projects.

3D object library

Customizable 3D elements

The 3D object library is now fully customizable. You can select any 3D element from your scene and convert it into a library item.

This makes it easy to reuse the most common 3D objects in your projects with a simple drag-and-drop operation, without having to retexture them.

Intelligent object placement

The new intelligent placement tool automatically detects the surface on which you want to place an object. You can now drag and drop library elements directly onto your project surfaces, with automatic physical geometry detection for precise, effortless placement.

What's New in TopSolid'Cam 7.19

Explore the brand-new features of the **CAM** applications of **TopSolid 7**!

Hole machining

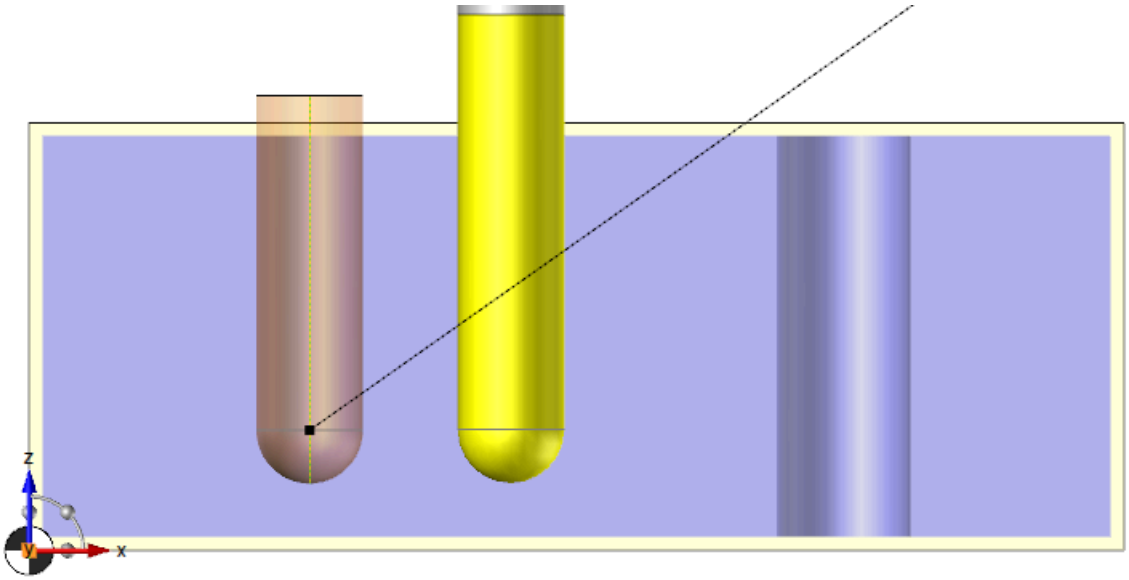
Drilling - Additional fixtures

The **Additional fixtures** section has been added to the hole machining operation. All drillings located "under" the fixtures will be considered unreachable. Its operation is now identical to that of other operations, similarly to end milling.

Drilling - Machining ball end holes

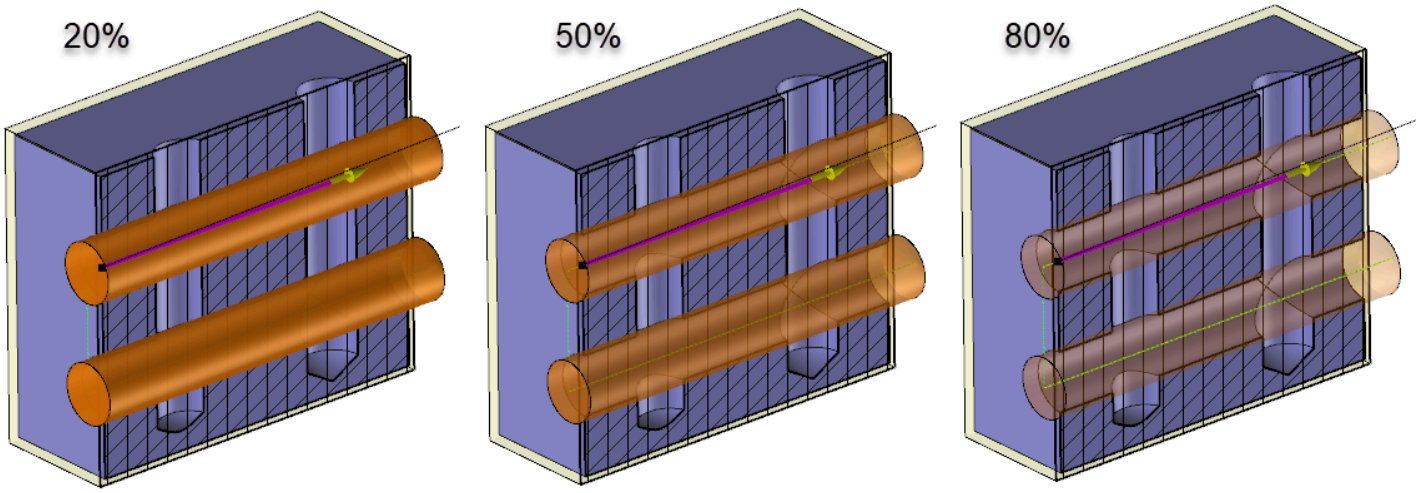
It is now possible to machine ball end holes in the machining document. A new **Ball end drill** tool category is available, with the associated generic tool.

In the methods, you can also define ball end drillings.



Drilling - Preview transparency

In the general options of **TopSolid**, as well as in the machining document options, you can now adjust the transparency when previewing drilling geometries.



Drilling - Geometry preview using the label

The **Geometries Preview** parameter is now available in the drilling operations label. Double-click on the field to display or not the geometric preview of the operation.

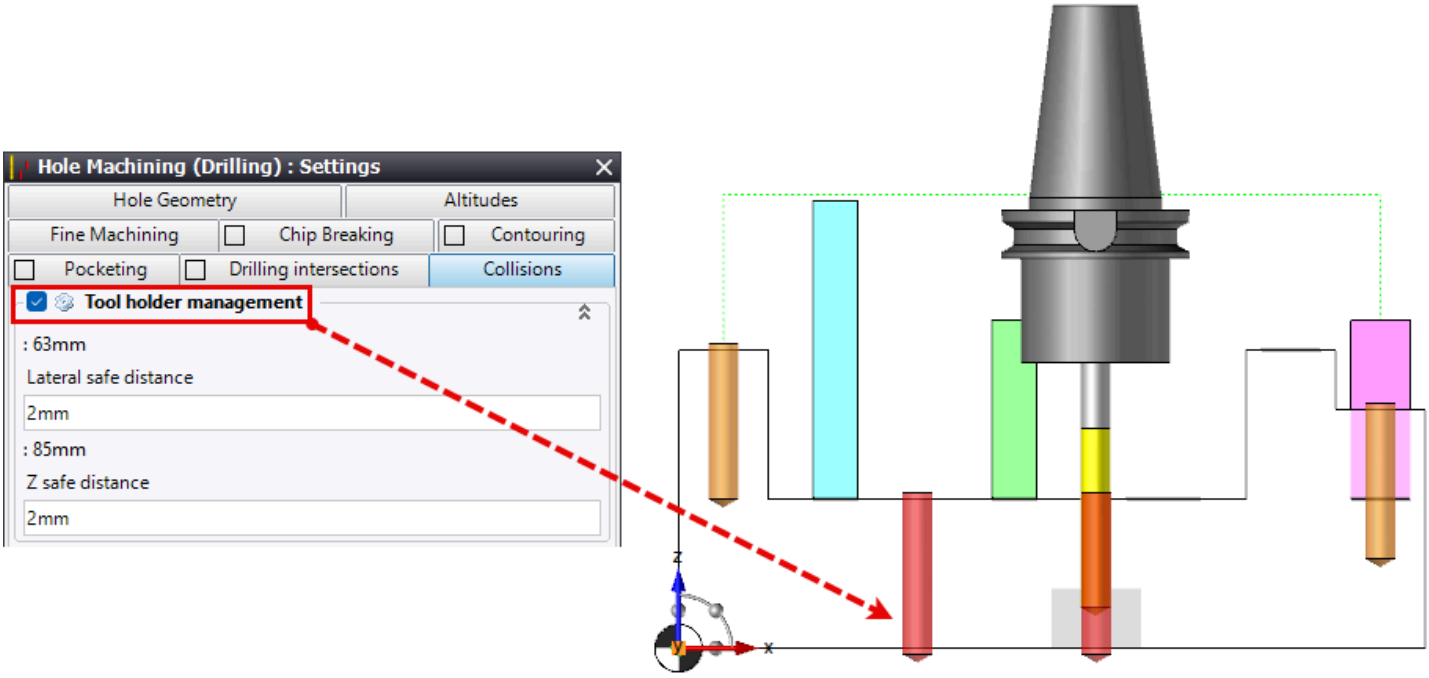
Drilling - Generation of machine cycles in the drilling cycles

The new **Machine cycle** option forces the separation of the drilling cycle.

Drilling - Collision management with the tool holder

A new **Collisions** tab has been added to the drilling operation, similar to other operations managing collisions with the tool holder.

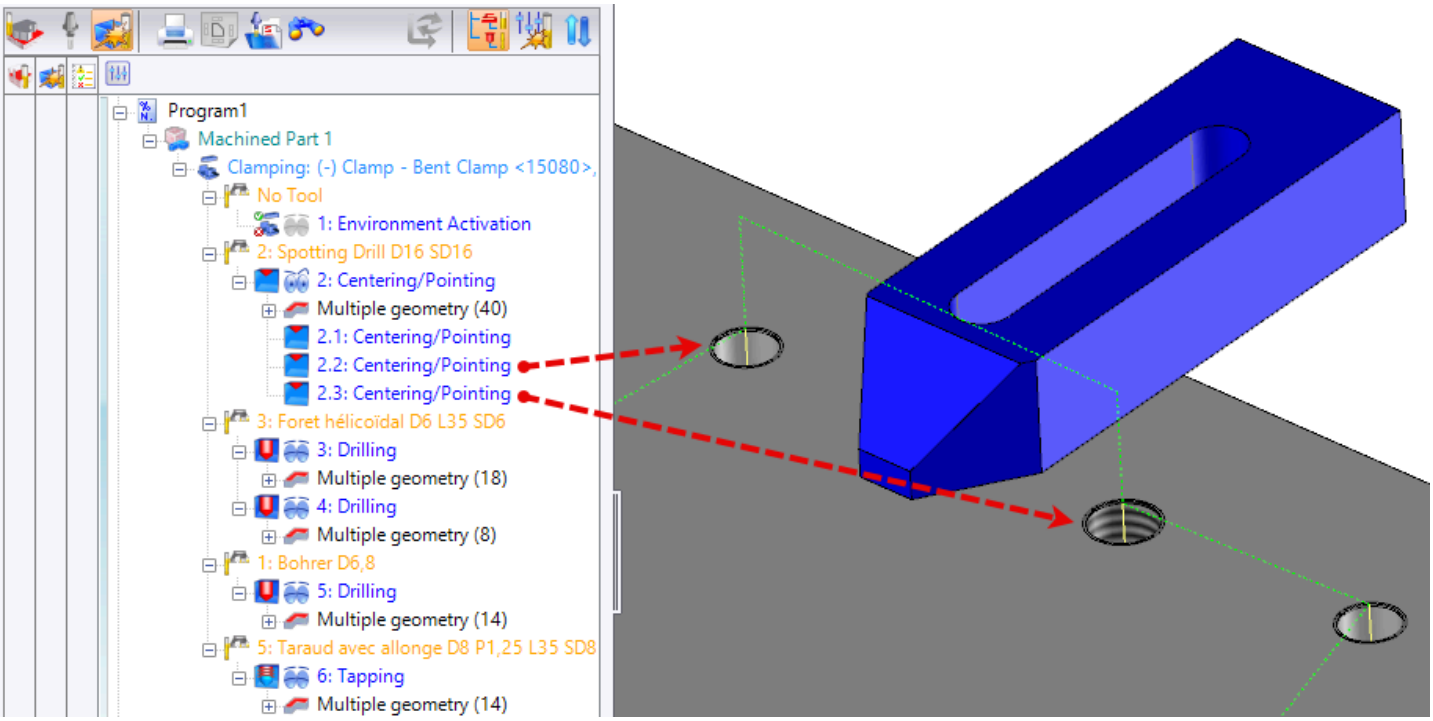
In this tab, the new **Tool holder management** section allows you to ignore geometries that would collide with the tool holder.



Drilling optimization - Environment management

The drilling optimization now integrates environment management.

It takes into account the environment defined beforehand, as well as the additional fixtures of the first operation in the list.



The link between two holes from different operations correctly avoids the fixture.

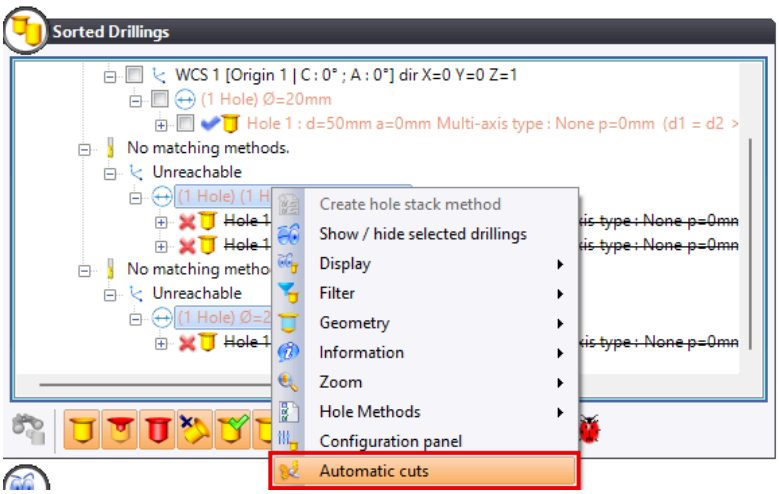
Customized drilling - Absolute pass depth

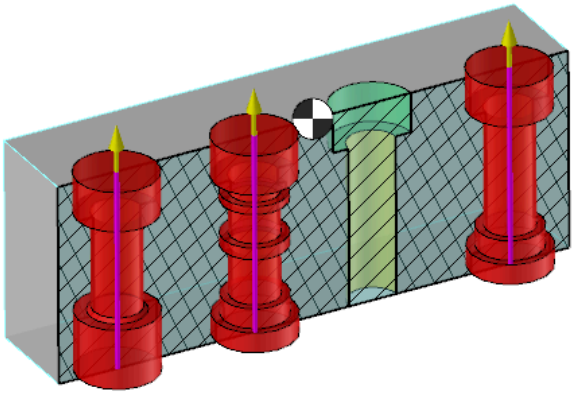
The **Customized drilling** document's interface has been optimized to simplify its use.

For the **Descent** and **Machining** actions, a new **Altitude hole top** option is available in the **Distance type** field. It indicates the pass depth as an absolute value.

Cylinder and drilling analysis - Automatic cutting of undercuts

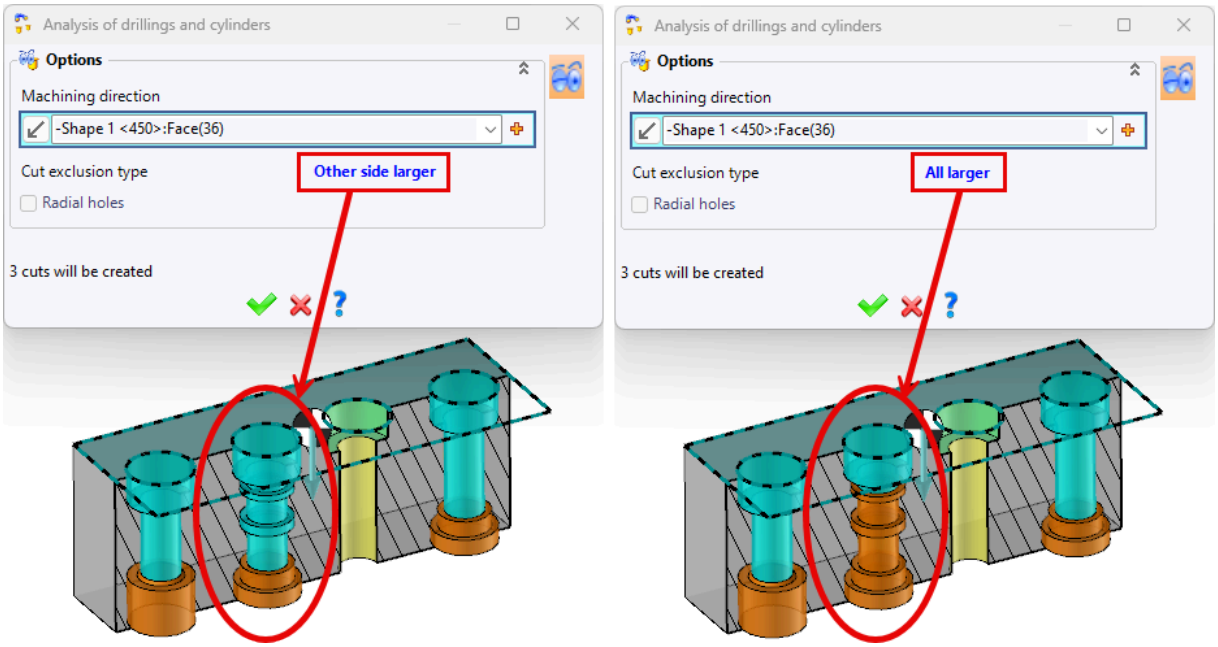
In the cylinder analysis, the new **Automatic cuts** command quickly cuts in half drillings of which one part cannot be machined in the chosen direction (undercut area).





Two cutting types are possible:

- **All larger**
- **Other side larger**



Cylinder and drilling analysis - Information on entities

The display of entity types in the cylinder analysis is now more precise.

MF Analysis - Highlighting the selected element

In the machining features analysis, the selected element is highlighted in the graphics area with a red arrow.

MF Analysis - Sort criterion for hole tolerances

In the machining features analysis, the new **Tolerance/Norm** grouping criterion allows tapping or tolerancing attributes to be taken into account in the sorting.

The **Description** column has also been integrated into the display table. It provides the hole diameter and depth, plus additional information from the tapping or tolerancing attribute.

MF Analysis - Grouping axial and radial holes

In the machining features analysis, the new **Multi axis type** and **Positioning diameter** grouping criteria enable axial and radial holes to be grouped together.

Its operation is similar to cylinder analysis.

2D milling

2D milling - Toolpath limitation to machine limits (linear axes)

In end and side milling operations, the **Machine limits** option has been added to the **Bounds** tab of the **Geometry** dialog box. This is a limitation in the current WCS in XYZ, collinear axes included.

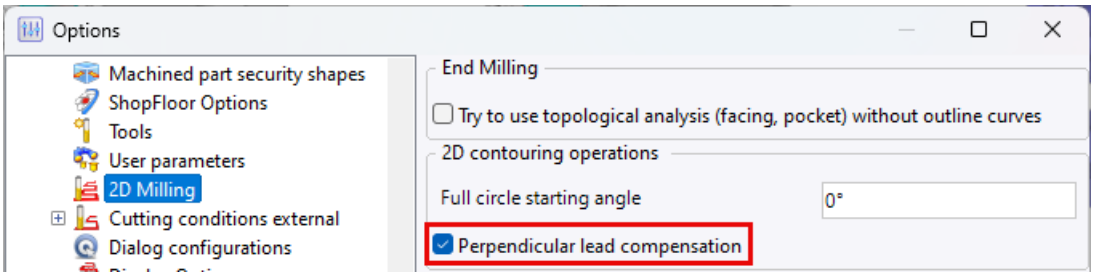
2D milling - Collision detection in case of disabled tool compensation

In corrected mode, when tool compensation is disabled, the collision management has been optimized to align with the behavior of the original mode.

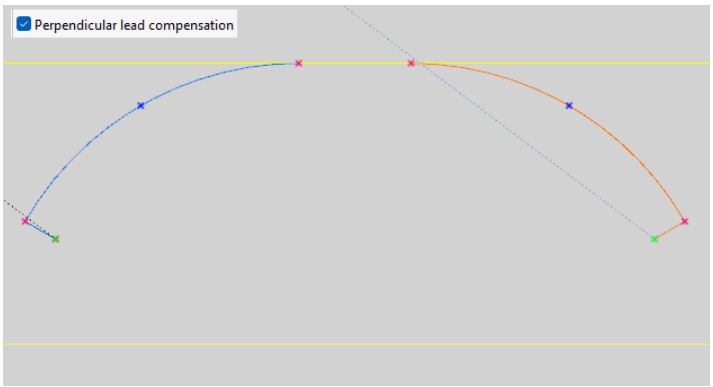
Specifically, if the **Take into account the finish shape** option is checked, the **Compensation method** selected is **Corrected**, the **Use compensation code** option is unchecked and the tool collides with the finish, the operation will not generate a path. Instead, a warning message is displayed in the Events tree.

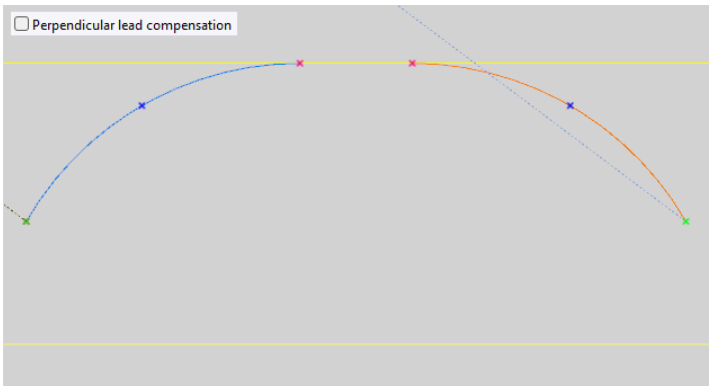
Side milling – Enable/Disable the perpendicular lead compensation

The **Perpendicular lead compensation** option has been added to the general options of **TopSolid** and to the document options.



It can force or not the perpendicular lead compensation on the first movement.





In previous versions, it was forced by default.

This does not apply to external thread milling, hole contouring or integrated contouring.

End and side millings - XY bounding by an enclosing box

In end and side milling operations, the **XY Bounding** section has been added to the **Bounds** tab of the **Geometry** dialog box.

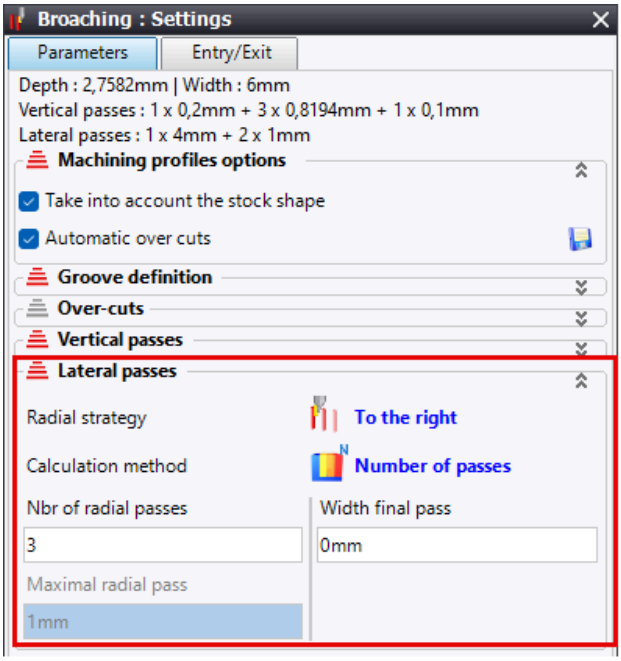
It is now possible to bound the toolpath in XY using a box, as for the roughing operation. To do so, check the **XY Bounding** box. This is a XY bounding in the current WCS. Z bounding is not possible, as it is imposed by the starting and ending altitude of each geometry.

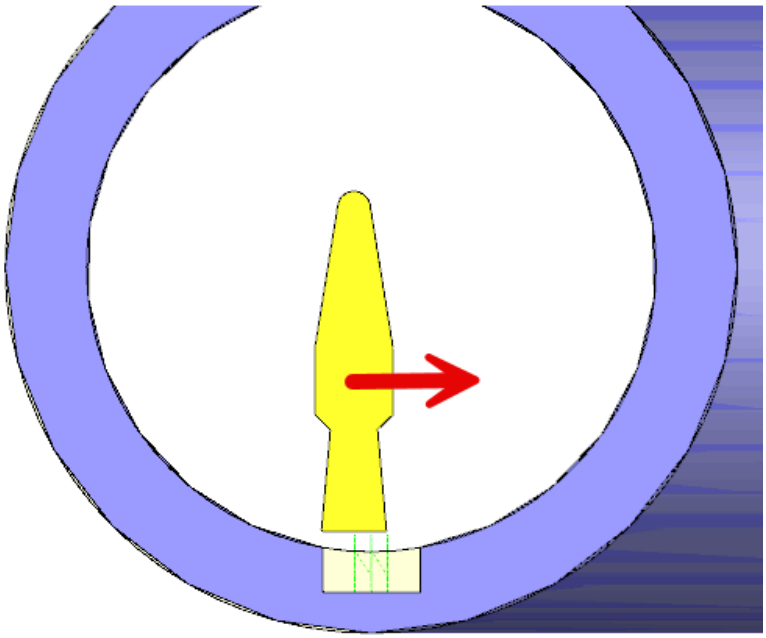
You can also show/hide the XY bounds preview using the new option available in the label.

Broaching - Addition of lateral passes

The broaching operation now allows you to add lateral passes to machine grooves larger than the tool.

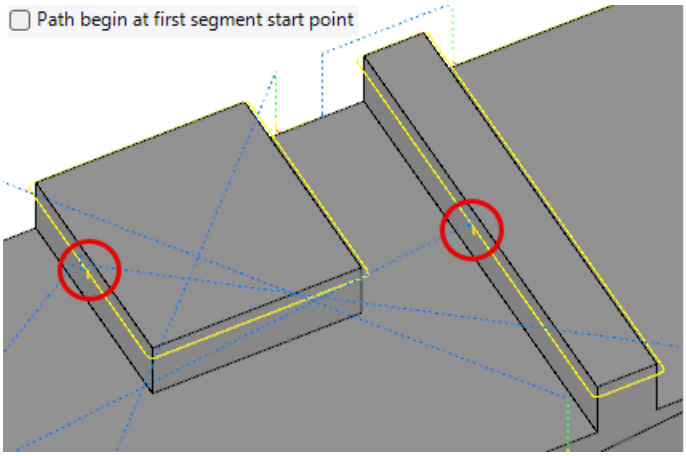
The width is automatically computed in new operations. In cases where the selected geometry is wireframe, automatic computing is not possible, and the tool width is used by default.

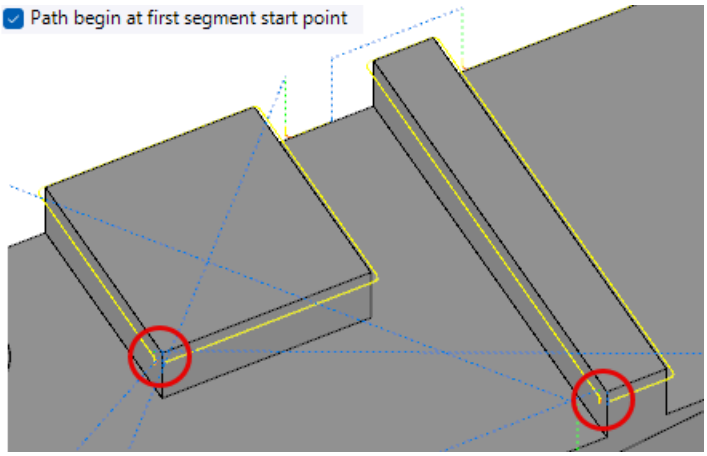




Breaking edges – Starting at the beginning of the first segment

The new **Path begins at first segment start point** option allows the breaking edges operation to start at the beginning of the first segment, when the selected profile is closed, similarly to side milling.

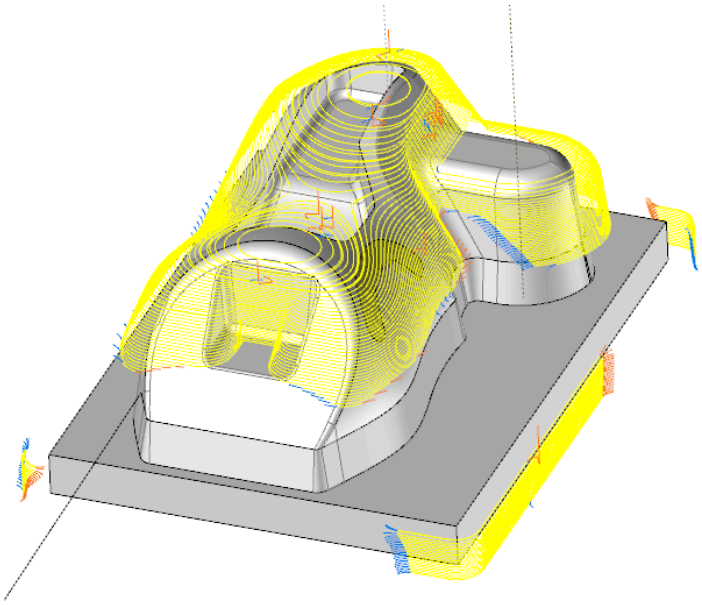




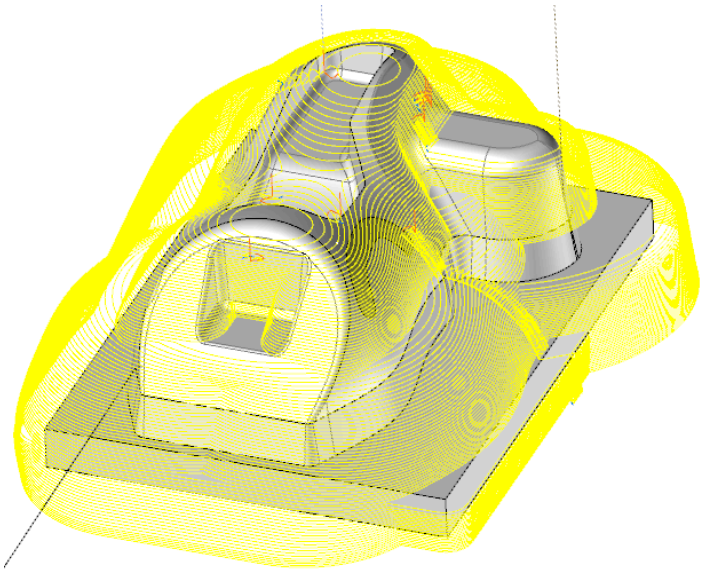
3D milling

3D milling - Collision strategy

To optimize collision detection, the **Collision strategy** parameter has been added to the **Collisions** and **Tool definition** tabs. It allows you to select, where applicable, one of two collision management modes: **Trim** or **Avoid**.



Trim

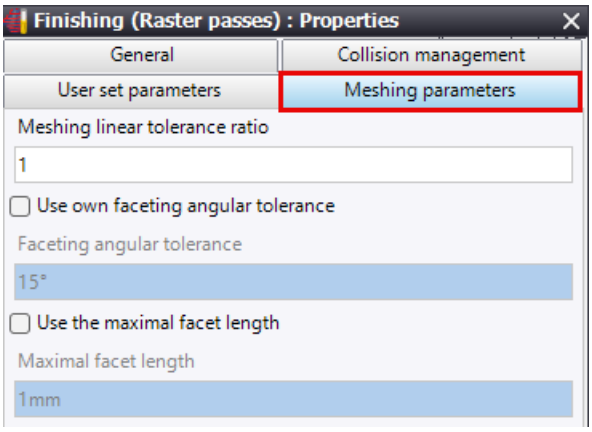


Avoid

3D milling - Faceting ratio and maximal facet length

The new **Meshing linear tolerance ratio**, **Faceting angular tolerance** and **Maximal facet length** parameters are available for 3D operations using faceting.

They can be accessed from the **Meshing parameters** tab of the **Properties** dialog box.



3D contouring - Taking the environment into account

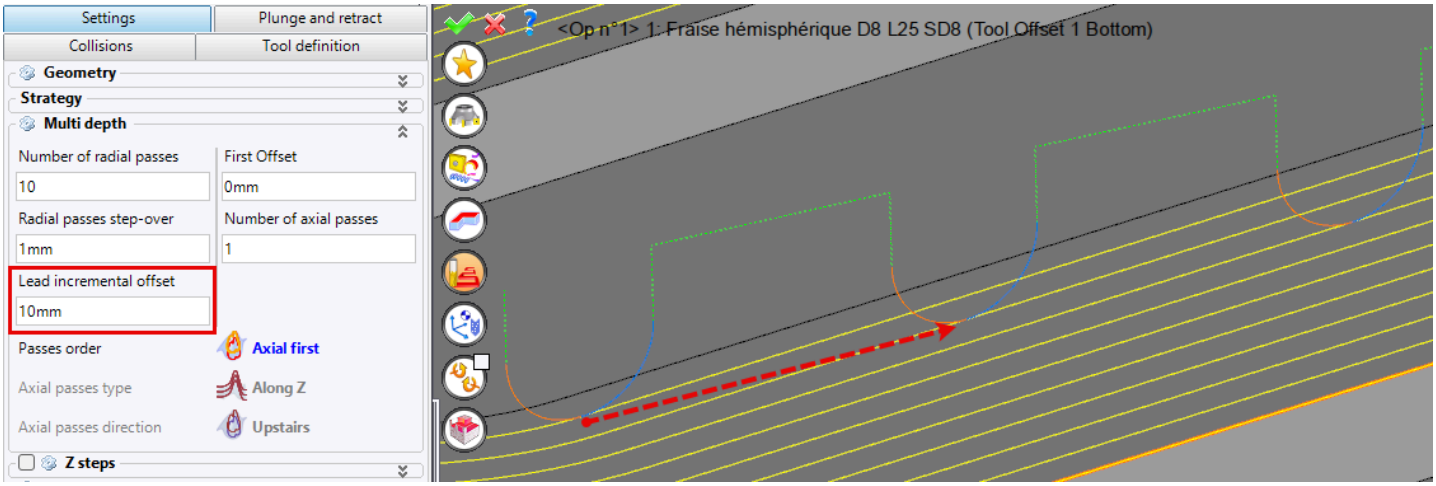
The 3D contouring operation now takes the environment into account.

Local fixture has been added to the **Geometry** dialog box, similarly to 3D operations. A general offset is applied to all fixtures, unlike 2D operations, where a specific offset for each fixture is possible.

3D contouring – Starting point offset

The new **Lead incremental offset** parameter allows you to adjust an offset on the starting point in the 3D contouring operation when there are several radial passes to perform on a closed profile.

The **Axial first** pass order type now allows you to retract at each radial pass.

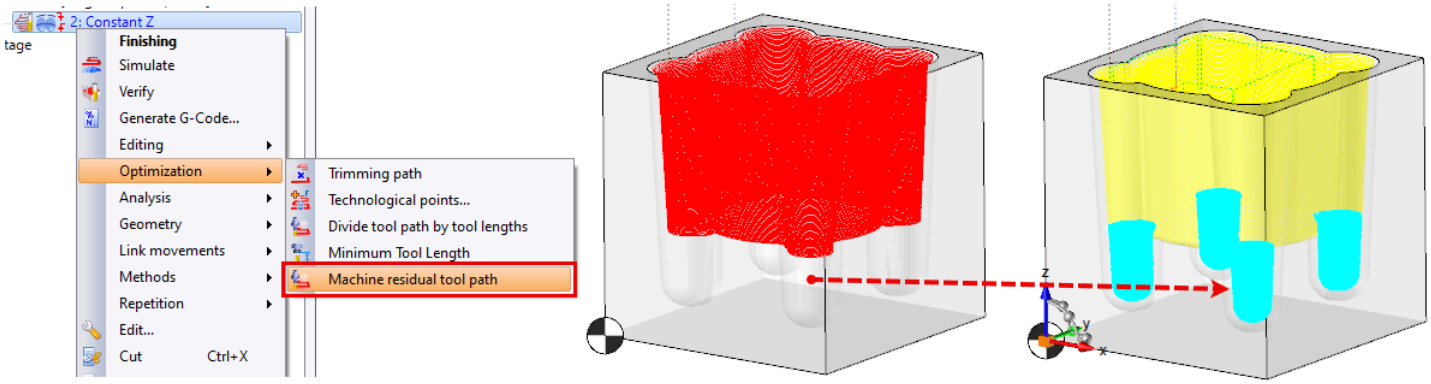


Constant step-over - New location for the “Trim the path at Z min” option

For **Constant step-over**, **Superfinishing** and **Residual machining** finishing operations, the **Trim the path at Z min** parameter has been moved to the **Bounds** tab of the **Geometry** dialog box.

3D finishing - Saving colliding tool paths for residual machining

In a 3D finishing operation, the new **Machine residual tool path** command is accessible in the **Optimization** section, by right-clicking on the machining operation.



Constant Z finishing – Spiral machining and HSM on path

It is now possible to check both **Spiral machining** and **HSM on path** options at the same time.

Finishing - Keeping sharp cutting edges

The **Geometry** tab of the **Constant Z**, **Constant step-over** and **Superfinishing** 3D finishing operations features a new option **Produce sharp cutting edges**.

It allows you to select the sharp edges to keep in order to avoid the toolpath to wrap around them.

High-speed mills - Collision point management

Collision point management is now available for high-speed mills, in both 2D and 3D operations.

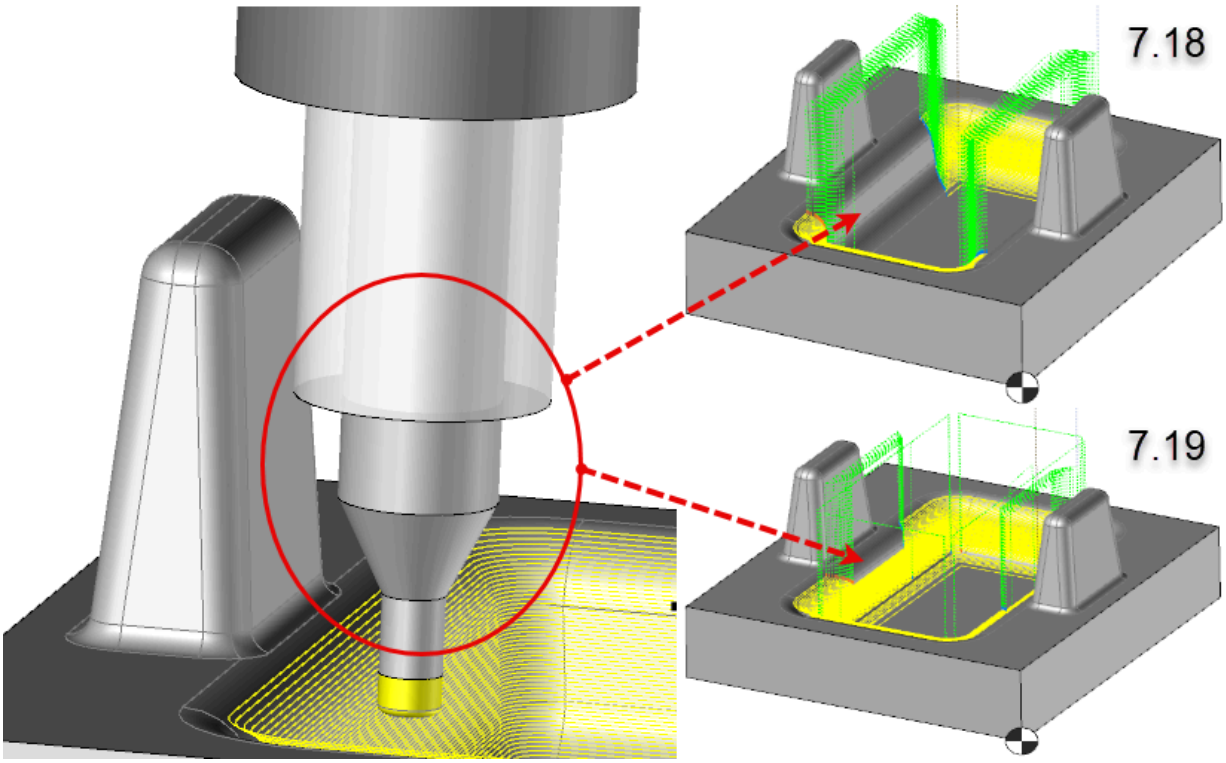
Raster passes finishing - Twin raster passes

The new **Twin raster passes** command is accessible in the **Optimization** section, by right-clicking on a 3D finishing operation. It creates a new operation, offset by half the step value of the previous operation.

This enhancement optimizes the machining process by reducing bumps caused by sudden changes in stocks to leave, which also helps extend tool life.

Dividing path by tool leads out - Improved tool holder management

The collision strategy algorithm has been optimized to take into account the entire shape of the tool holder, rather than just a cylinder. This improvement allows for extended machining with shorter tools.



Residual machining - Precision improvement

The new **Meshing precision** parameter, available in the **Settings** tab, improves the search precision for elements with material left when the result is not convincing.

However, this impacts performance, resulting in a longer computation time.

4D milling

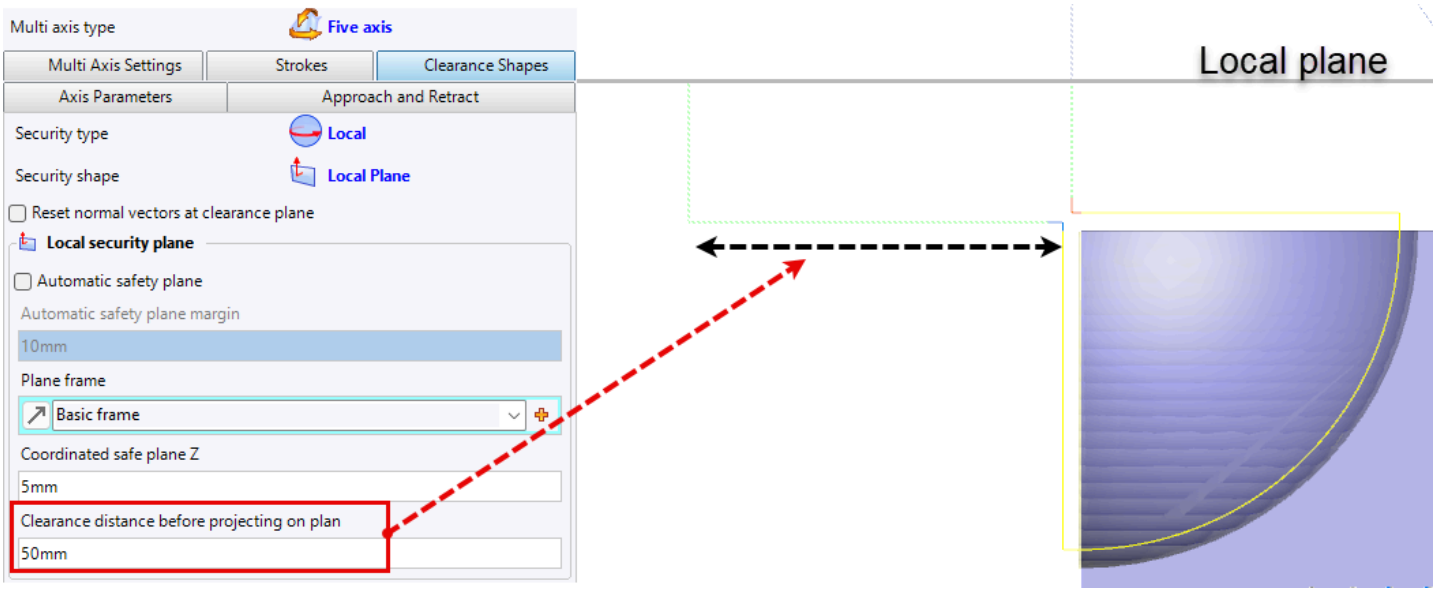
4D milling – Collision point management

Collision point management is now available for 4-axis operations.

5D milling

5D milling – Clearance distance before projecting on plane

When the selected clearance shape type is **Local**, the new **Clearance distance before projecting on plane** parameter enables you to better manage the clearance distance, particularly when the safety plane is parallel to the tool axis, or has a small angle offset.



If the local safety plane is still not reached beyond the specified clearance distance, a perpendicular projection is made to determine the shortest path.

5D milling - WCS creation from a dynamic frame

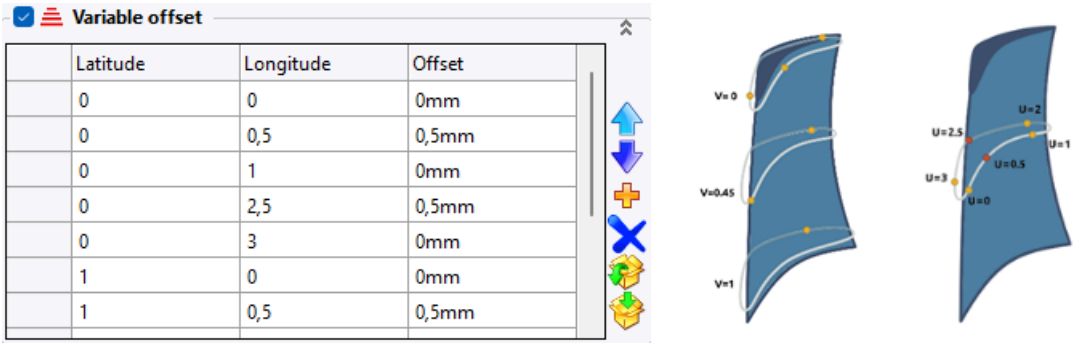
The new **Create on dynamic frame** command is available from the **Operation** tab, in the **WCS Manager** or by right-clicking on a frame.

The dynamic frame can be moved by translation or rotation. The solutions are computed according to its position.

Blade finishing – Variable offsets

It is now possible to define variable offsets in the blade finishing operation in **Blisk** mode.

To do so, enter the table with point coordinates according to latitude (V) and longitude (U), as well as the offset to apply to each point.



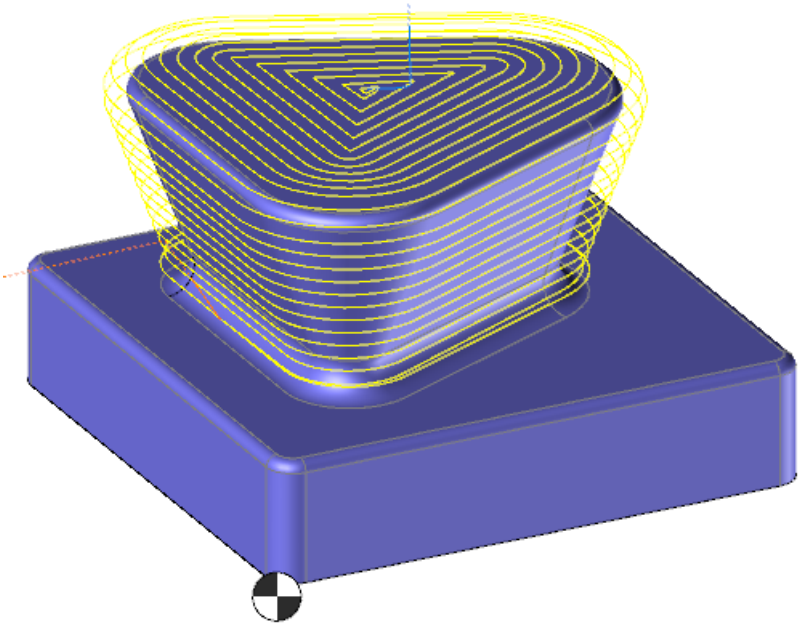
You can export / import these values using a text file.

5D drilling – Normal visualization

Normals can now be displayed on machining paths for 5-axis drilling operations.

5D finishing – Tool center mode

In **Tool center** mode, the **5X Finishing** command using the **ModuleWorks Geodesic** algorithm has been added.



5D automatic machining - Setting step-over according to tool axis

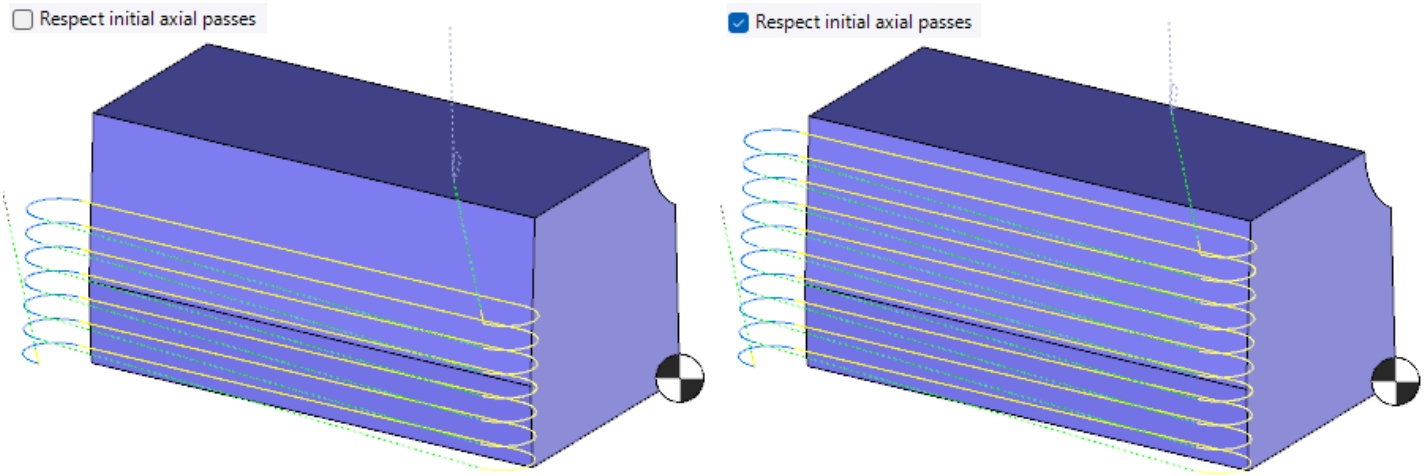
It is now possible to enter the step-over according to the tool axis rather than the scallop height (the two are linked by a formula).

5D automatic machining - Deletion of unnecessary final passes

In the **Geometry** dialog box, the **End passes to remove** option is available under each geometry. Simply enter the number of final passes to delete.

5D automatic machining - First pass depth

The new **Respect initial axial passes** option allows you to add passes to prevent an excessive first axial pass, while respecting the defined axial pass depth as far as possible.



5D automatic machining – U and V restrictions for the faces to machine

U and **V** limits have been added to each selected geometry, similarly to the **Sweeping** operation.

5D automatic machining – Machining with the barrel mill small radius

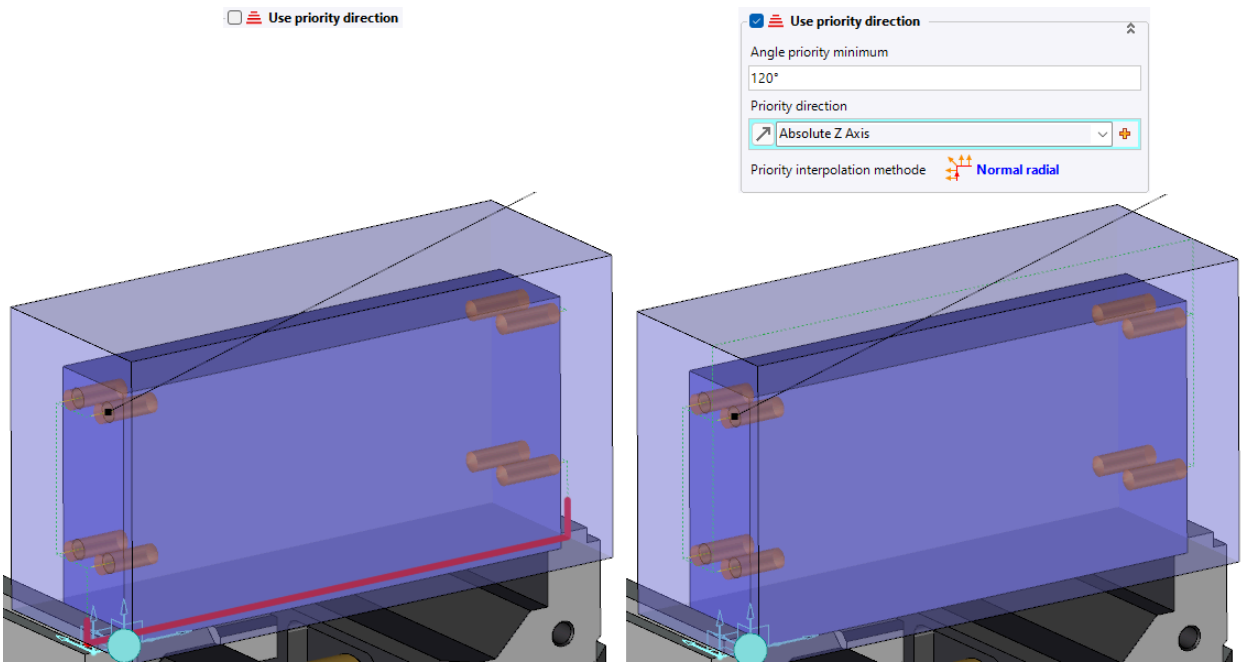
It is now possible to choose which part of the barrel mill to use for path computing.

To do so, in the **Multi Axis Settings** tab of the **Multi Axis** dialog box, two parameters **Targeted point profile section** and **Minimum point profile section** are available.

5D drilling - Priority direction for clearance shape movements

Three new parameters are available in the **Clearance shapes** tab to manage toolpath with movements on clearance shapes (between machined geometries):

- **Angle priority minimum**
- **Priority direction**
- **Priority interpolation method**



Turning

Turning - Reconstruction of spline curves into segments

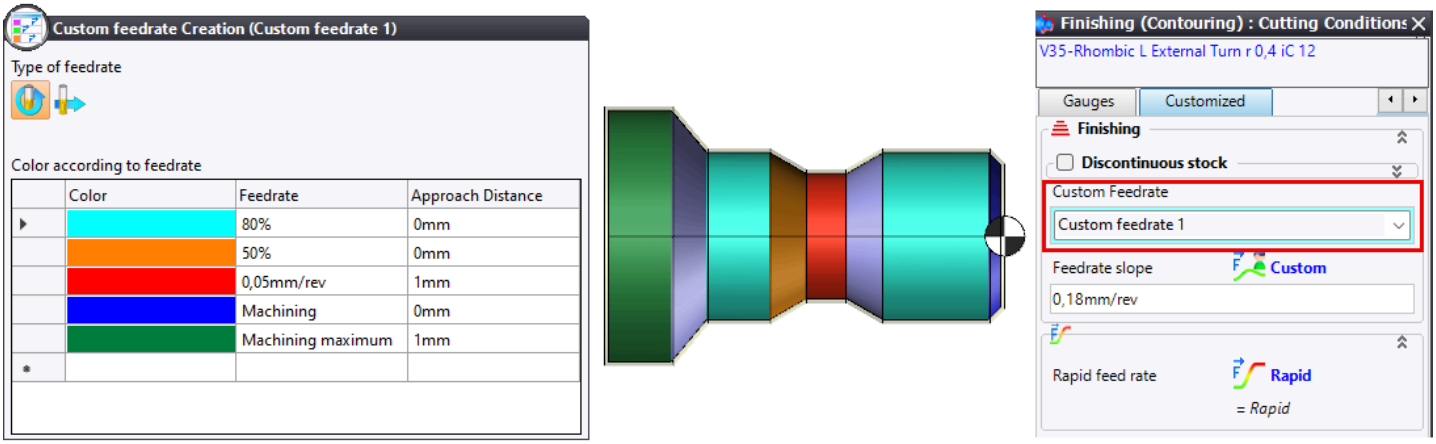
It is now possible to select the spline separation in the spun outline.

The **Spun outline creation** command includes a new **Rebuild method** field with two available modes:

- **Lines**
- **Arcs and lines**

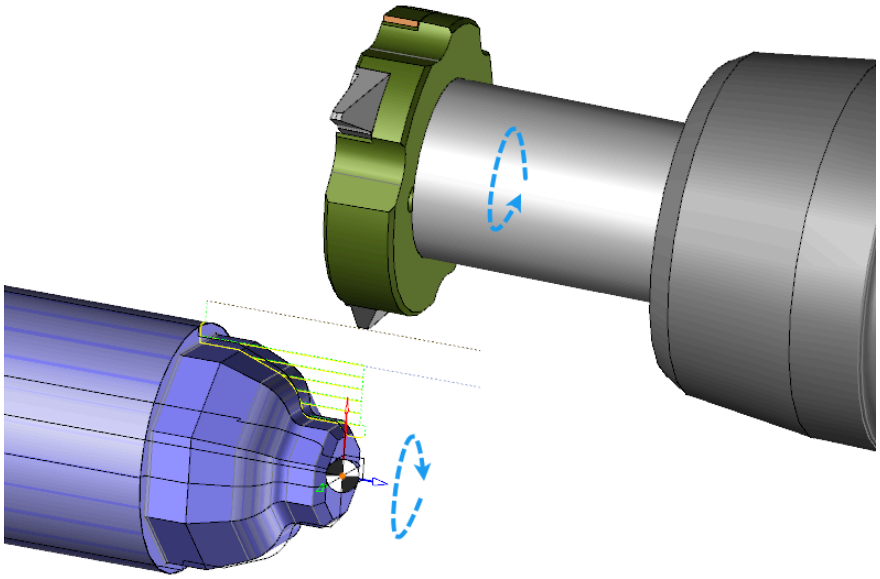
Turning - Linking a feed rate to a face color

It is now possible to adjust the feed rate on different parts of the part to machine according to the color of the faces.



New polygonal turning operation

The new **Polygonal Turning** operation allows the tool and the part to rotate in perfect synchronization to machine flat surfaces.



A new **Polygon Turn** tool type is also available. It has two categories:

- **External polygon turn longitudinal**
- **External polygon turn plunge**

Associated generic tools are also available.

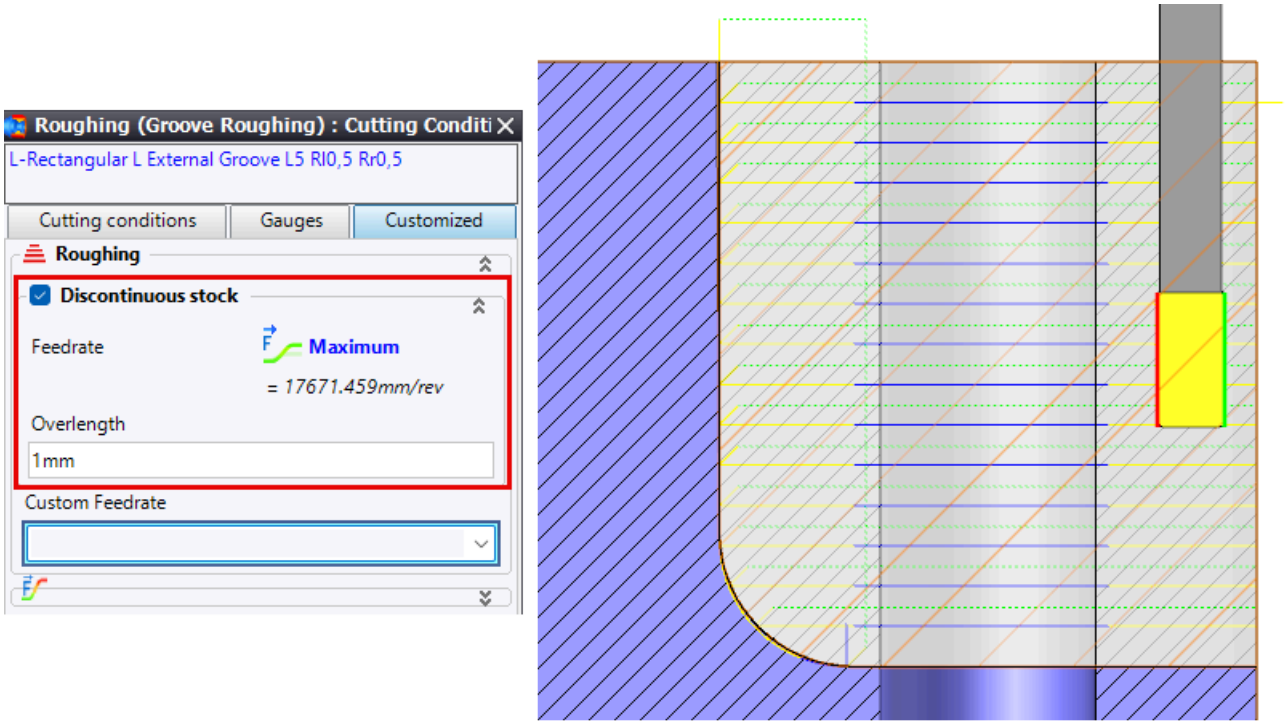
Roughing - Z or X bound inversion

Bounds defined in Z or X in turn roughing operations can be inverted.

To do so, double-click on the yellow **Invert limit** arrow in the graphics area, or check the **Invert limit** option in the **Stock to leave / limits** tab of the roughing operation settings.

Roughing and finishing - Modification of feed rate in discontinuous stock areas

The feed rate can be modified when the tool encounters an area with discontinuous stock that has already been locally machined.



Finishing - Limits in X, Z or using a window

Limits in X, Z or using a window have been added to the finishing operation.

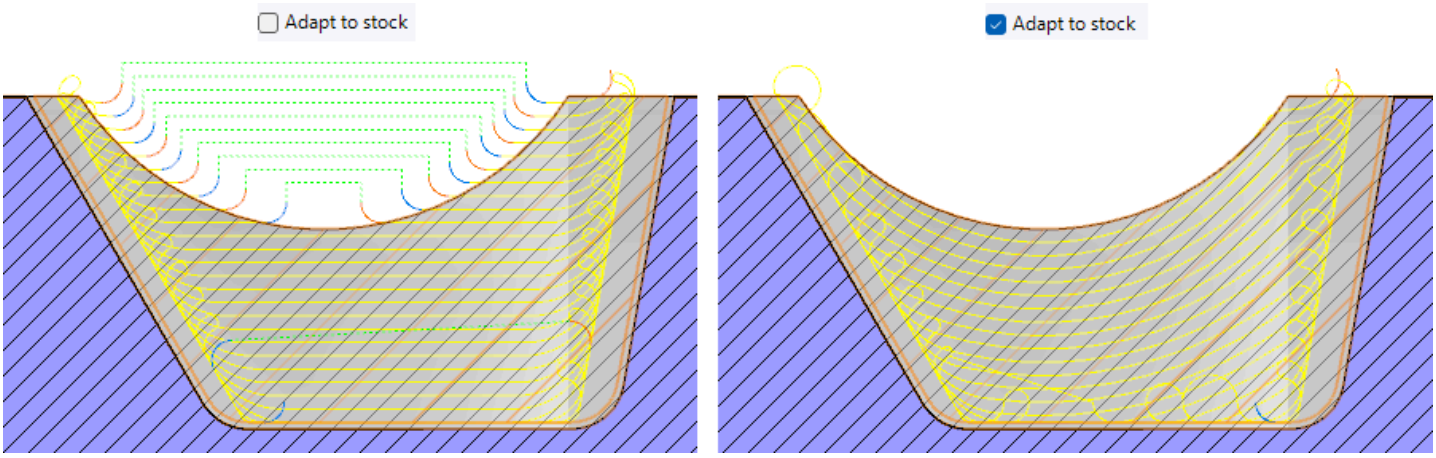
Boost - Stock adaptation

In the operation settings, the new **Adapt to stock** option ensures a smoother toolpath, while reducing link movements in certain cases.

Without stock adaptation, the toolpath follows a straight line movement parallel to the machining direction. When the tool exits the stock, the transition is performed using a rapid link.

With stock adaptation, the path is computed according to the stock profile limited to the machining area. Consecutive parallel passes are created to generate the toolpath.

In this case, the machining direction relative to the stock is taken into account, but not its exact value.



Parting off – Rapid retract

Rapid retract is now possible for all stages.

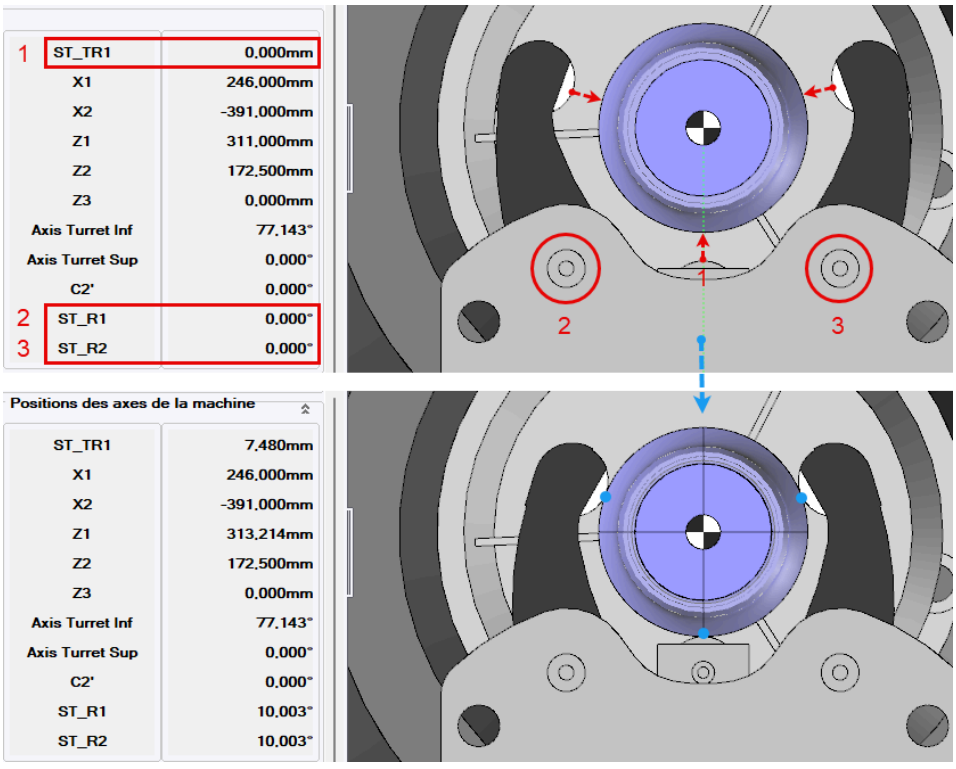
Parting off - Simplified access to parting off

The parting off primitive operation is now available from the **Turning** > **Others** > **Parting off** command.

A dedicated icon has also been added to the turning icon bar.

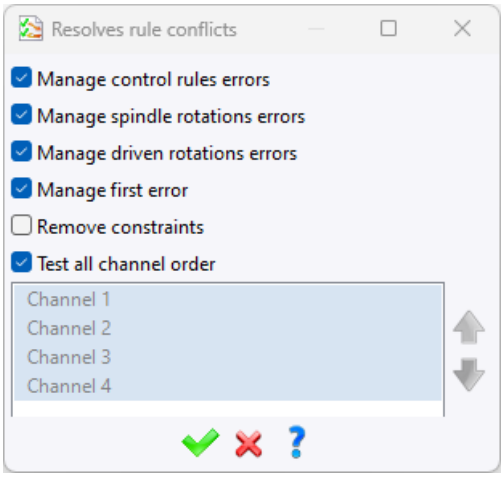
Steady - Steady opening and closing simulation

A new steady definition allows you to simulate opening and closing.



Scenario - Improvement of automatic synchronization

The **Resolves rule conflicts** dialog box, available from right-clicking in the scenario, has been enhanced. It now offers options to resolve control rules conflicts.



Scenario – Displaying coolant type

The machining operation's information thumbnail now displays the operation's defined coolant state, including the coolant type and pressure.

Scenario - Displaying tool change

The TCP text is now displayed each time the tool changes.

Virtual jog and link movements

Link movements - Improved retraction

The automatic tool retraction movement at the end of an operation has been enhanced, particularly for **Internal turning** and **Parting off** operations, to effectively prevent collisions.

Link movements - New criterion in the rules document

The new **Workstation number** criterion makes it possible to test each station number within the rules document.

Link movements and virtual jog – Machine axis motion with predefined positions

In the **Machine axis motion** action, it is now possible to enter predefined axis positions, similarly to movements on coordinates.

The positions include:

- End
- Start
- Max. axis value
- Min. axis value
- WCS
- Change tool
- None

WCS - Linking a frame to several angular solutions

The process for linking a frame to multiple or all angular solutions has been simplified. Using the **Ctrl** or **Shift** key, you can select multiple angular solutions, then apply the desired frame to the selected angular solutions.

Tools and cutting conditions

Cutting conditions - Addition of surface finish quality

Surface finish quality has been added to cutting conditions, with three available values:

- **Roughing**
- **Pre-finishing**
- **Finishing**

This quality can be defined in the abacuses or groups of a cutting conditions document.

In addition, you can select the desired surface finish quality for an operation in the **Cutting conditions** tab, allowing the cutting conditions to adapt accordingly.

Cutting conditions - Secondary operation parameters

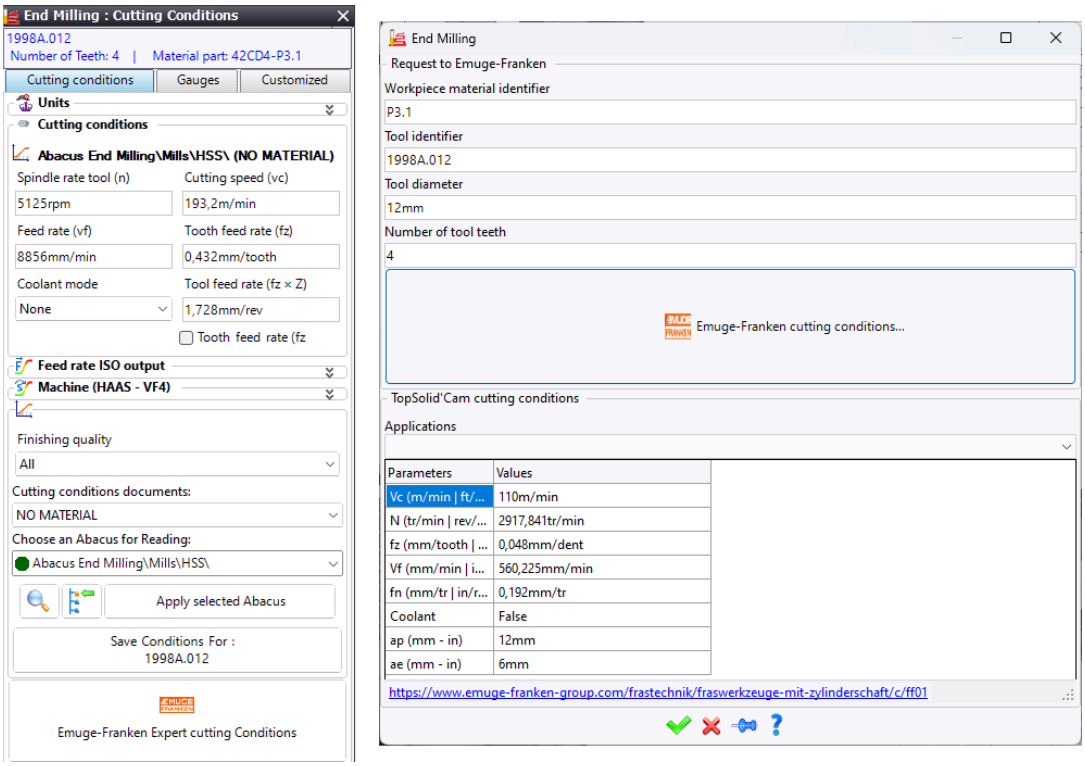
The cutting conditions for secondary operations automatically generated based on a parameter of the primary operation, such as **Roughing on two**, can now be edited.

Cutting conditions - Import/export of specific cutting conditions from Excel

Cutting conditions can be exported and imported in table format. Exports are performed in .csv format, while imports support both .csv and .xls files.

Cutting conditions - Integration of EMUGE FRANKEN cutting conditions

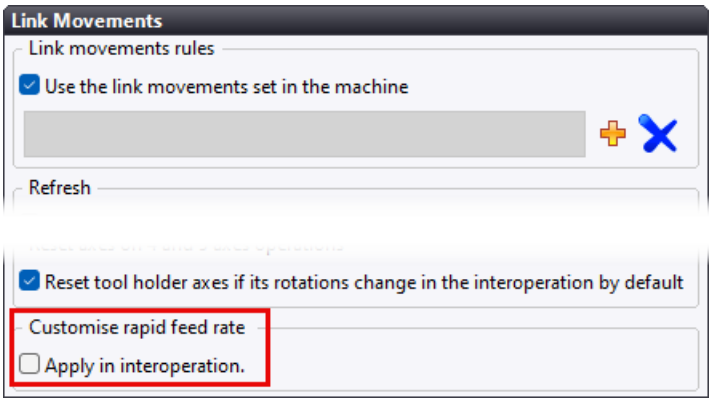
The EMUGE FRANKEN tool manufacturer cutting conditions can now be retrieved in **TopSolid**.



Cutting conditions - Modification of rapid feed rates

The rapid feed rate for each machining operation can now be edited from the **Customized** tab of the **Cutting conditions** dialog box.

Movement is always considered rapid in simulation, verification and collision detection. By default, approach, retract and interoperation movements are not customizable. However, an option is available to modify them.

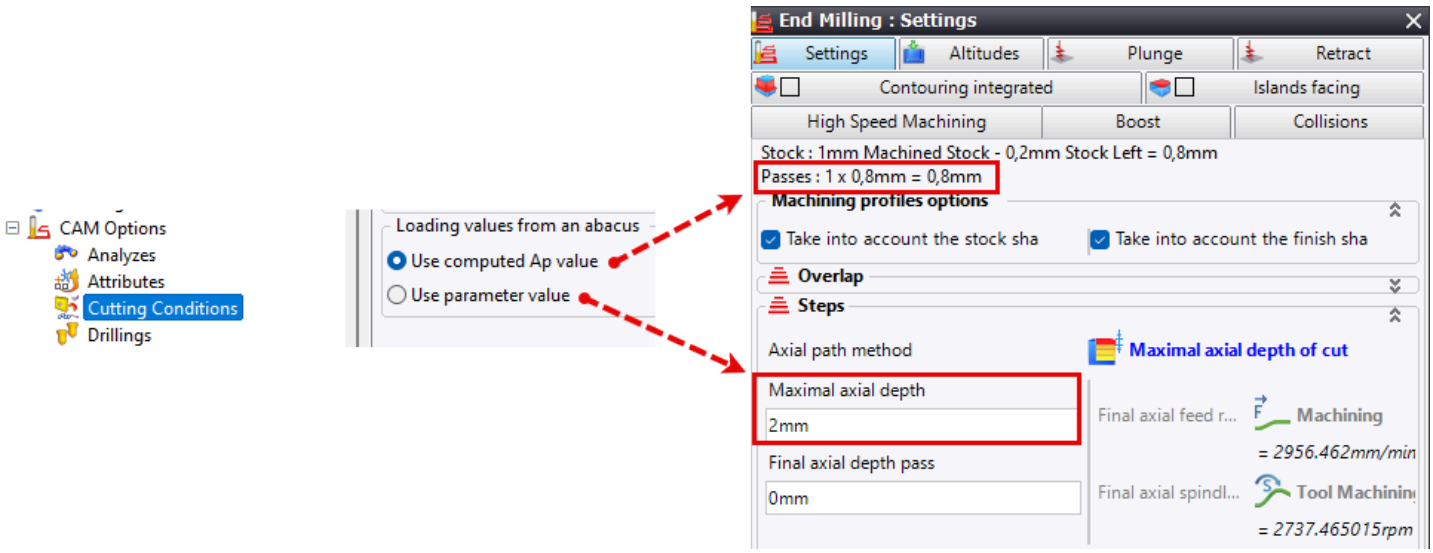


Cutting conditions - Loading abacus values: entered or computed pass depth

The new **Loading values from an abacus** option is available under **Tools** > **Options** > **CAM Options** > **Cutting Conditions**.

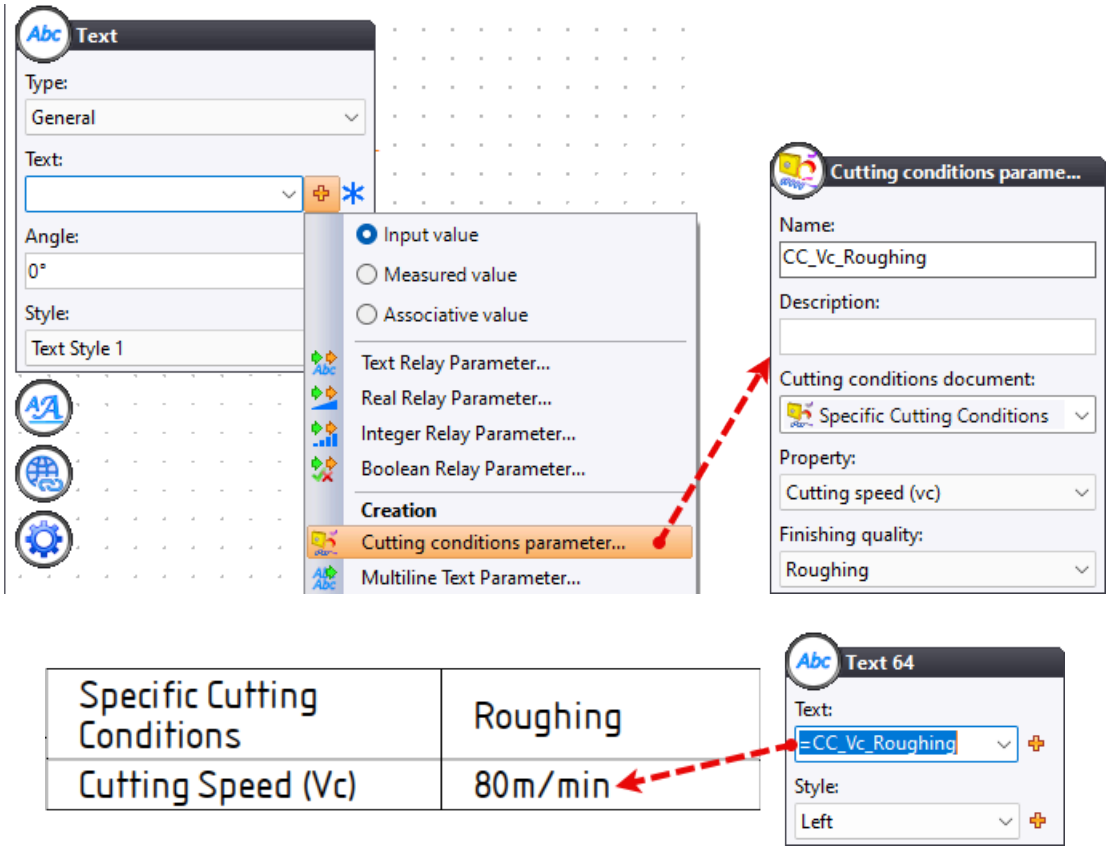
It allows you to specify whether loading of abacus cutting conditions should consider the pass depth defined in the operation, or the value computed by **TopSolid**.

This feature is also useful for machining documents without stock, where the computed depth is always 0.



Cutting conditions - Relay parameter in workshop documents

The new **Cutting conditions parameter** text parameter allows you to retrieve a parameter specific to the tool's cutting conditions within a workshop document.

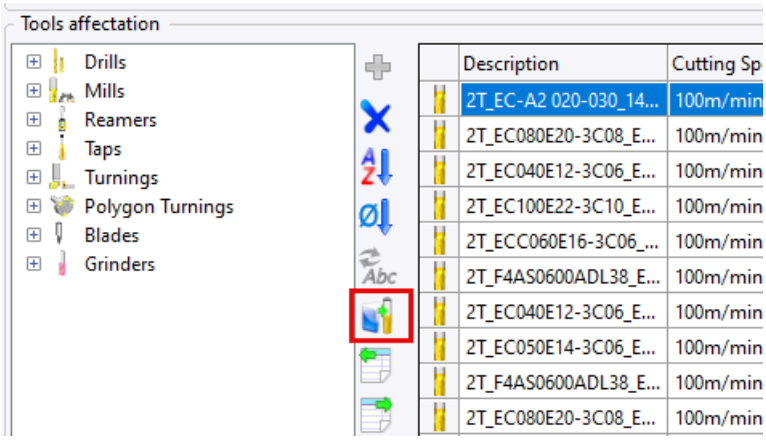


Cutting conditions - Material equivalence document referencing

A material equivalence document can now be referenced in a cutting conditions document. This feature suggests cutting conditions adapted to the machining operation, even if the material differs from that defined in the cutting conditions document, using material equivalences.

Cutting conditions document - PDM tool search

It is now possible to search for one or more tools using the PDM search, in addition to the left-hand explorer in the cutting conditions document.



Cutting conditions document - Modification of the abacus order

The order of abacuses in the cutting conditions document can now be modified. This order is memorized in a property saved in the document, allowing prioritization of one abacus over another, even if they share compatibility, when searching for cutting conditions.

Cutting conditions document - Copying multiple abacuses

You can copy several abacuses at the same time, within the same document or across different cutting conditions documents.

User machining cutting conditions library - Purge unused documents

The new **Purge not used cutting conditions documents** command is available in the context menu of the **Machining User Cutting Conditions** > **Cutting conditions management** library.

It operates in the same way as the **Purge not Used Tools** command from the generic tools library **TopSolid Machining User Tools**.

Tools - Tool selection when creating new operations

When creating a new operation, the new **Do not mount tool for new operations except hole machining** tool selection mode is now available alongside existing **Mount tool for new operations** and **Do not mount tool for new operations** modes.

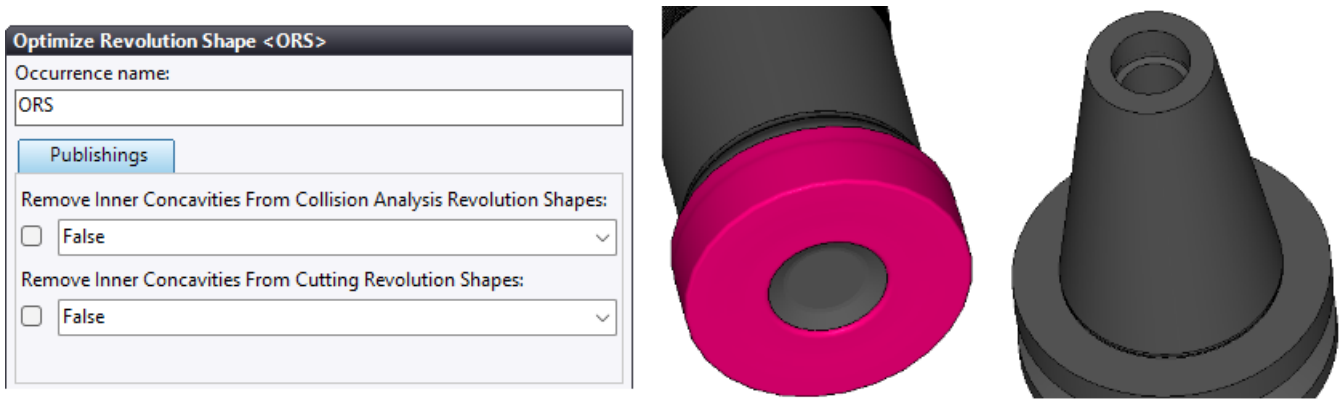
Tools - Addition of profiles and tool compensations from the tool creation wizard

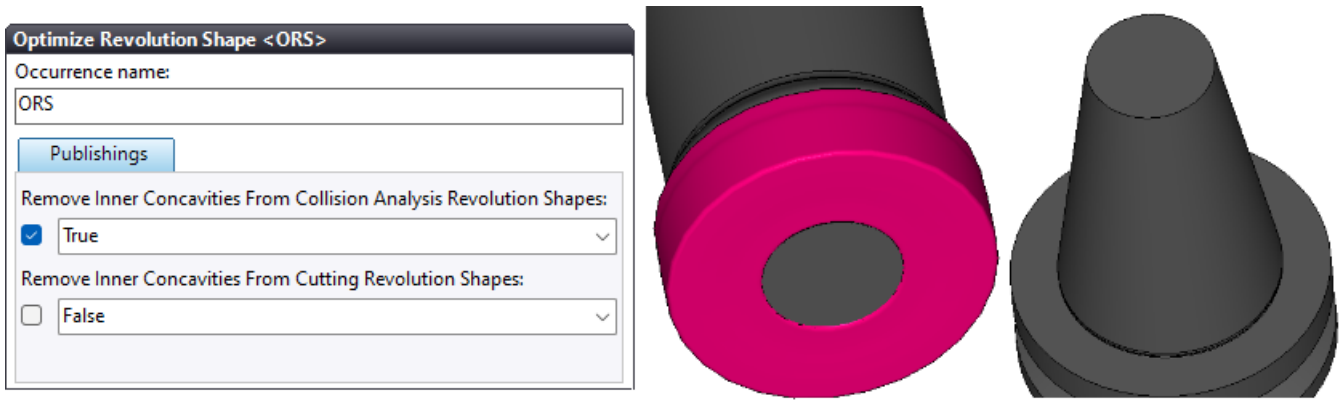
The following functions can be added or deleted directly from the tool creation wizard:

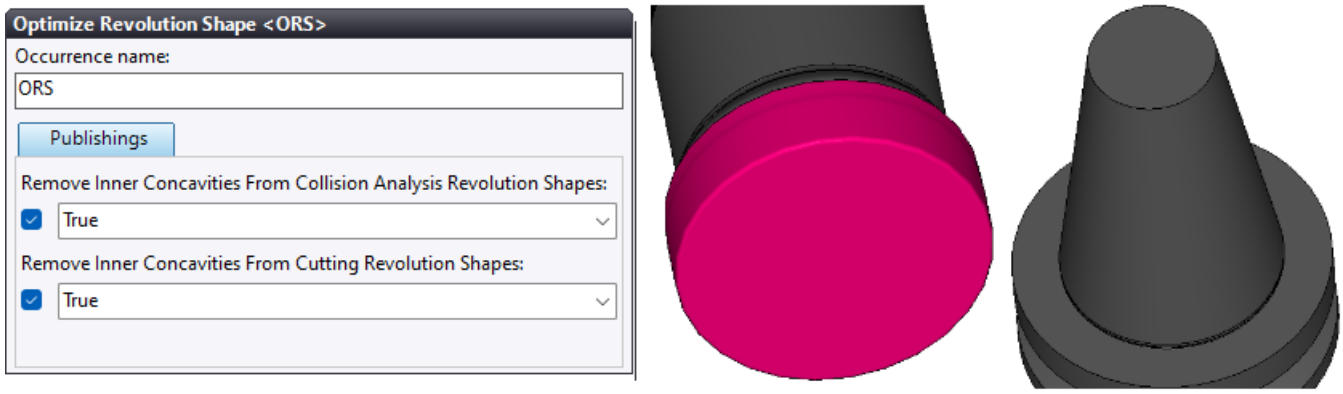
- **Tool Offset**
- **Cutting Shape**
- **Collision Analysis Shape**
- **Cutting Revolution Shape**
- **Collision Analysis Revolution Shape**

Tools - Optimized revolution shape

The new **Optimize Revolution Shape** function allows simplification of the cutting or collision revolution shape during verification.







Verification and simulation

Simulation - Coolant display

The type of coolant used in an operation can now be displayed during simulation.

Simulation - Show/hide non-programmable axes

The new **Show non programmable axes** option has been added to the simulation. It can be accessed from the **Show/Hide** command in the context menu.

Verification - Machine representation selection

It is now possible to choose the machine representation used during verification.

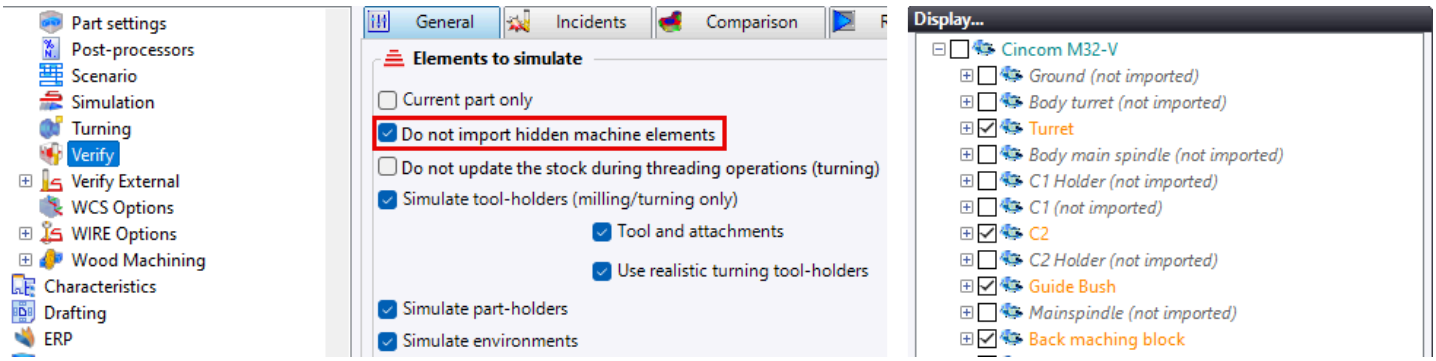
Verification - Show/hide the machine

Similar to simulation, machine elements can now be displayed or hidden by clicking on the machine icon during machine mode verification.

Verification - Selection of machine elements to import

By default, the behavior remains the same as in 7.18 version. However, hidden machine elements from the machining document are imported into verification for collision management.

You can decide whether to import them.



Machine - Rotation axis without limits

Ina machine rotation axis settings, the **Minimum angle** and **Maximum angle** fields can now be empty, enabling definition of a rotation axis with no angular limits.

Machine - Axis rotation direction

The **Direction of rotation** parameter has been added to the machine rotation axis settings. The available options include:

- **Automatic**
- **Clockwise**
- **Counter clockwise**

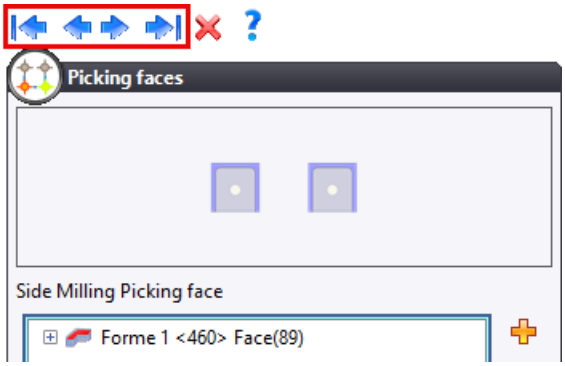
Methods

Methods - Pocket MF bottom area

The new **Area** variable gives the surface area of the pocket bottom after analyzing a pocket MF.

Methods – Simplified navigation in execution dialog boxes

Navigation in method execution dialog boxes has been improved, allowing you to go back or jump to the last page. This functionality also applies to copied operations, whether from the same document or another.



Methods – Customized feed rate and stock to leave

The following element types are now available when selecting or searching for elements:

- **Customized feed rate**
- **Customized stock to leave**

The selection or search result can then be used in the operation parameters.

Methods – New “Edge” geometry type

A new **Edge** geometry type is available in the **Picking element** function, allowing you to select an edge or edge profile when executing a method.

Methods – Finding cutting conditions document

A new **Cutting conditions abacus** document type has been added to the **Find document** function of a method document.

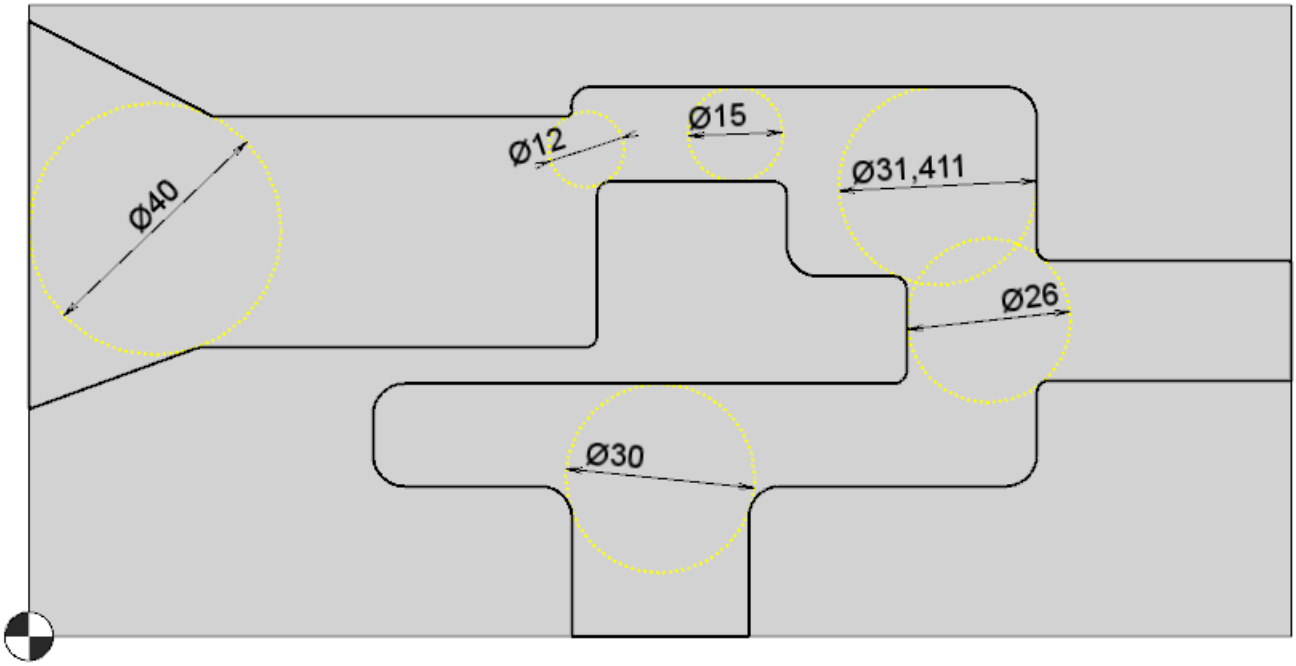
The resulting document variable is only accessible using a formula. When executing the method, **TopSolid** searches for the best-rated abacus in the cutting conditions document found, and applies it to the operation.

Methods – Minimum/maximum compatible tool diameter

The end milling analysis features new variables:

- **Diameter of smallest compatible tool**
- **Diameter of largest compatible tool**

They are used to specify the smallest diameter that can access all areas of the part (selected faces), or the largest diameter that can machine at least part of the machining area.



Methods – User variable update

The new **Update user variable** action, when integrated into a loop, retrieves the smallest or largest value of a parameter from a set of elements.

For example, it can find the smallest vertical radius among a batch of pockets.

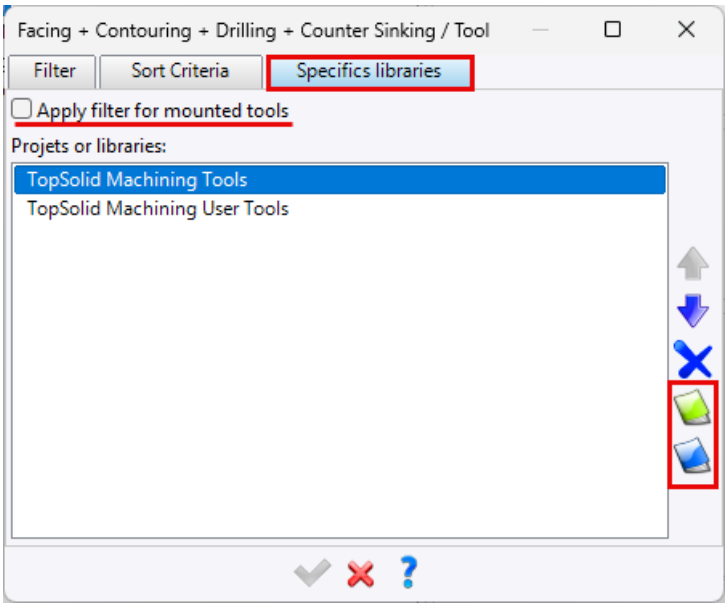
Methods – Tool selection during method execution

If no compatible tool is found during method execution, the tool selection dialog box is displayed, allowing you to search for a tool in the catalog or to directly create a generic tool.

Methods – Tool search in specific libraries or projects

Tool searches can now be directed to specific libraries or projects through the **Specific libraries** tab.

A new **Tool document library name** variable has also been added. This applies to the tool definition document and is available as a property in machining operations.



Methods - Tool already in use

With the new Boolean **Tool already in use** variable, you can choose between several tools already in use.

Methods - Search for similar hole MF elements

When executing a method, the **Find similar cylinders** section has been added when selecting hole MF type elements.

Methods - Reference to a TopSolid parameter

You can now use the **Reference a TopSolid parameter** variable in an operation to keep a link to the parameter. Previously, only the parameter value at the time of method application was used.

Methods - Automatic WCS for two-part machinings

A second automatic WCS is available for two-part machinings (drilling/2 or rough turning/2).

Methods - Submethod execution optimization

If a submethod execution in the primary method is conditioned by a VB formula and the test result is false, the action is simplified, reducing the method execution time.

Miscellaneous updates

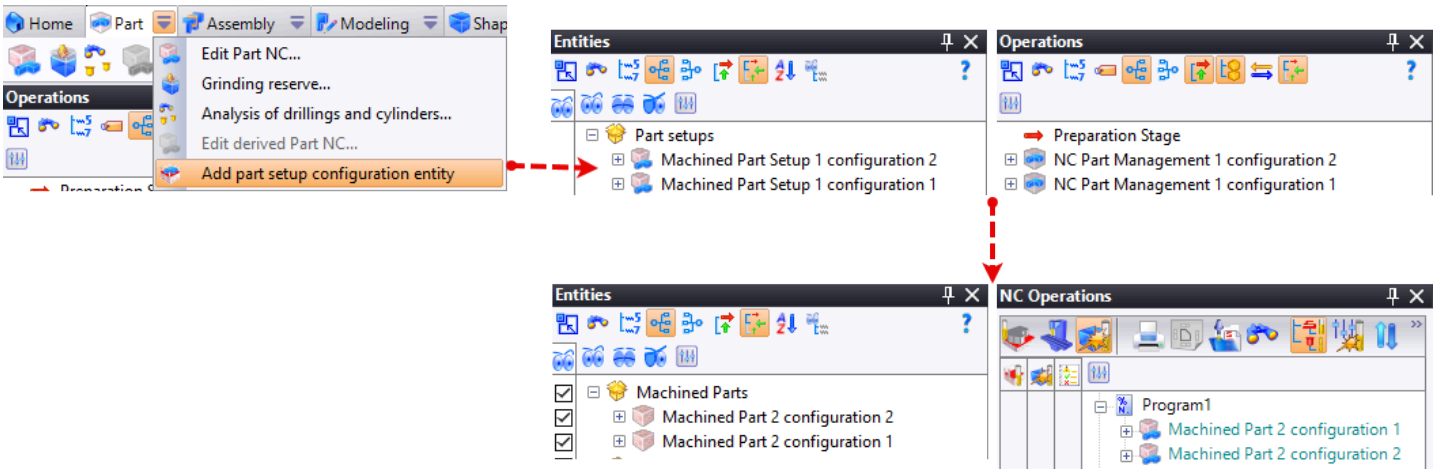
Machined part setup – Multi-configurations for NC parts

The **Add part setup configuration entity** command allows you to create multiple configurations for NC parts, each with specific environments.

When creating the machining document, one NC part per configuration in the machined part setup document is created in separate machining stages. Their stock is linked, similarly to part repositioning.

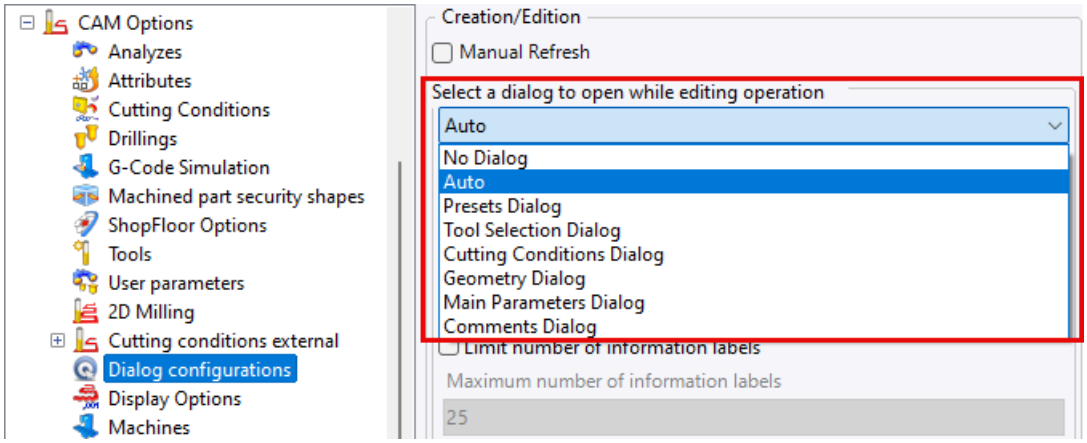
The stock is defined in the first configuration and is also applied to subsequent configurations.

These configurations follow the chronological evolution of the part machining process, and their order is immutable.



Operation editing - Default display of the last tab used

When editing a machining operation, you can automatically display the subdialog box and the last tab used.

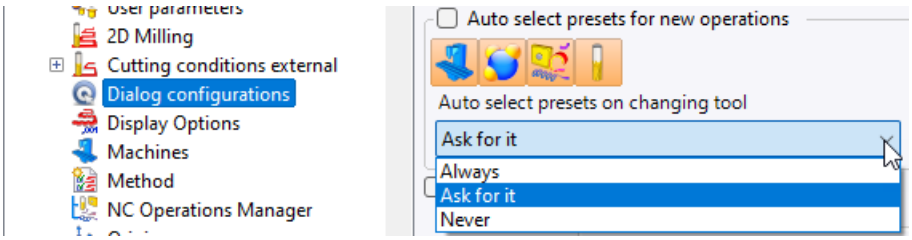


Geometry analysis – Default origin

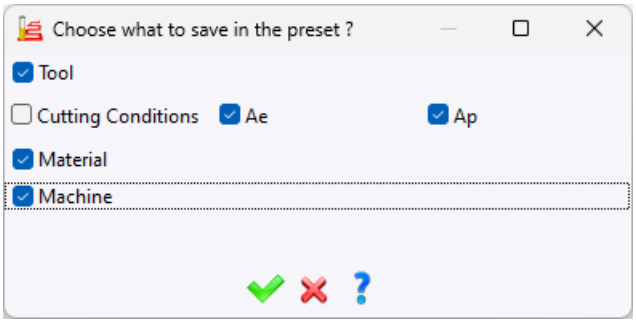
If only one part origin exists in the current stage, it is now selected by default in the geometry analysis.

Presets - Automatic selection of presets

You can now automatically select a compatible preset for the created operations. If several presets are compatible or none is found, the presets dialog box will be displayed.



Additional save options for presets are available.



Environment - Group definition using graphical selection

You can now group environments using graphical selection.

Programs - Renaming programs

A new **Rename programs** command is available in the program manager for single-channel machining documents. It can also be accessed from the **Change program** command's dialog box.

Programs are renamed in sequence order.

Programs - Adding programs from the change program dialog box

A new **Add new program** command has been added to the **Change program** command's dialog box for single-channel machining documents.

Analysis view - Display axis min/max values

The new **Show max/min values** option is available in the analysis window.

Normals - Differentiate normals with one or two solutions

In the normals display options, a new color has been added for normals with a single angular solution.

Security elements - Addition of the sphere

The sphere element has been added to the **Security elements** folder in the Entities tree. As a reminder, these elements are created automatically in the machining document.

Default parameters - Import without deleting existing parameters

When importing a default parameter set, if **TopSolid** detects existing value sets with the same name, it will assign a new name to the imported set by adding a number to the original name.

Default parameters - Saving a parameter to the current parameter set

A new **Show the parameters save buttons** option is available in the **CAM Options** > **Dialog configurations** section of **TopSolid**'s general options.

When this option is enabled, you can save a parameter locally in the current default value set, if it is modifiable. A disk-shaped button appears next to the field when selected, provided the value differs from that of the selected set.

Hold down the **Ctrl** key and click on the button to select another value set to modify.

Technological points - Improved selection of element's start and end points

When a start or end point is selected, the selected path portion is now saved, ensuring correct recalculation of the point on the correct element.

Repetition - Echoing operations in repetitions from a WCS

When creating a repetition from a WCS, the operations within this WCS are now visible in the graphics area.

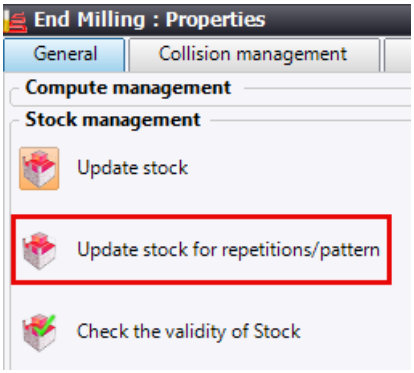
Repetition - Resetting modified parameters in repeated operations

When one or more modified repeated operations are selected in the NC Operations tree, the new **Repetition** > **Reset to source operation** command is available by right-clicking on them.

The modified values then return to the initial value of the parent operation.

Repetition - Update stock for repetitions

The **Update stock for repetitions/patterns** option has been added to the operations properties. It defines the initial state of the **Update stock** option for the repetitions instances.



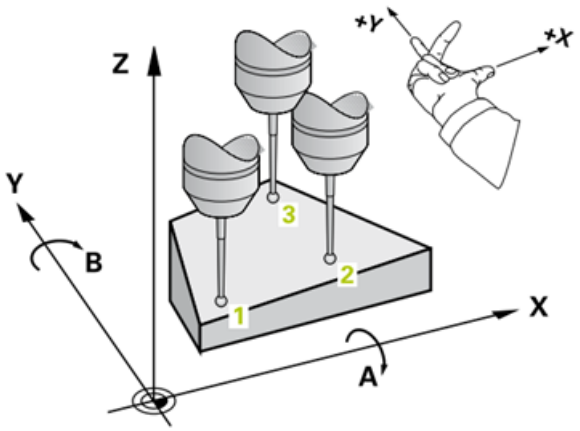
Control points - Alignment

The new **Alignment** control points mode allows a machine axis to be aligned by controlling two elements (two points, holes or tenons).

Control points - Plane measurement

The new **Plane measurement** control points mode allows you to measure the slope of a plane, using three points.

Points 1 and 2 determine the X+ orientation, while point 3 defines the Y+ direction.



Workshop document - Extracted stock from last operation in tool sequence

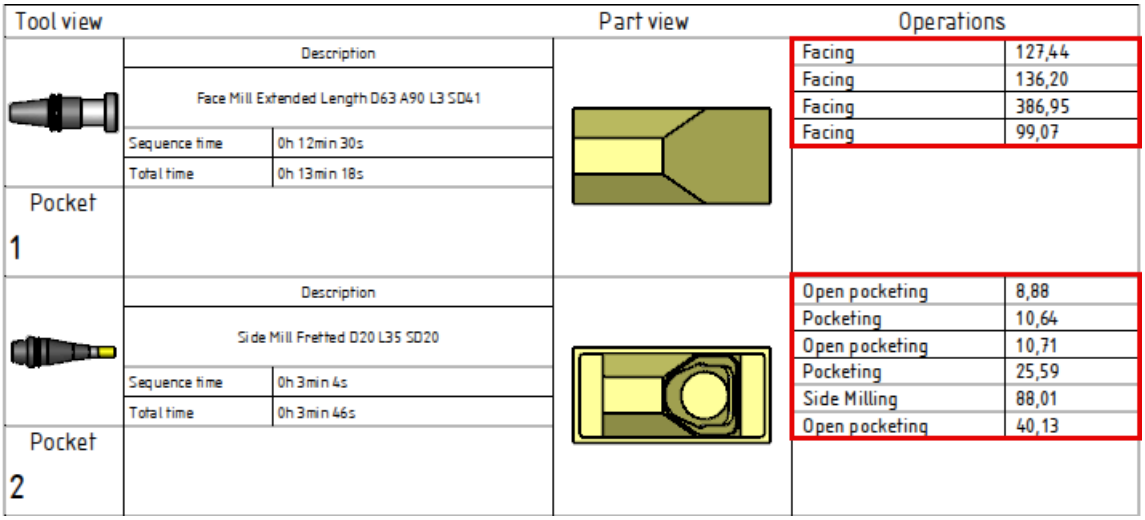
In a workshop document containing the **Tool change machining list** chapter, the extracted stock for the tool sequence now reflects the state after the last operation in the sequence (previously, it was the stock of the first operation in the sequence).

Workshop document - Driven point parameter with simplified text

A new **DrivenPointType** parameter displays only the description of the **Tool offset** function, not the full text of the offset.

Workshop document - Displaying consecutive operations by tool

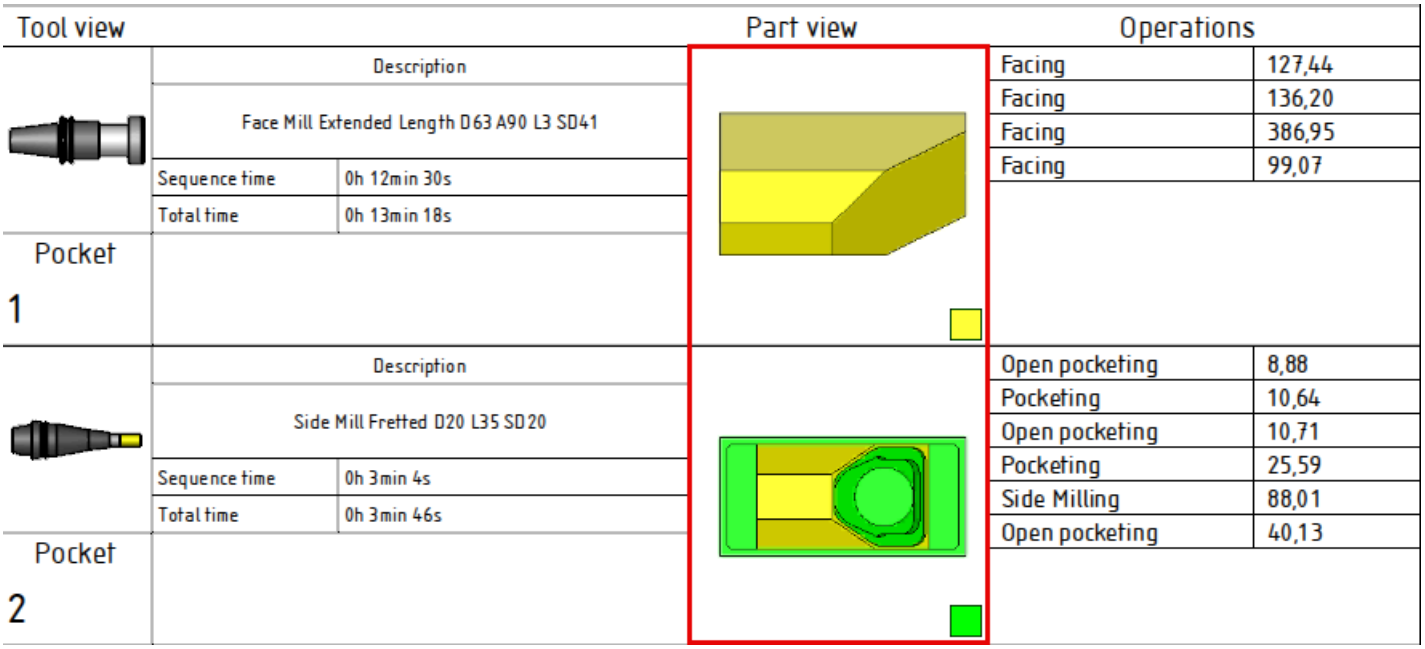
A new **Consecutive machining operations by tool** list has been added to the property table. It generates the list of machinings performed by a single tool.



Tool view		Part view	Operations	
	Description		Facing	127,44
	Face Mill Extended Length D63 A90 L3 SD41		Facing	136,20
	Sequence time 0h 12min 30s		Facing	386,95
	Total time 0h 13min 18s		Facing	99,07
Pocket 1				
	Description		Open pocketing	8,88
	Side Mill Fretted D20 L35 SD20		Pocketing	10,64
	Sequence time 0h 3min 4s		Open pocketing	10,71
	Total time 0h 3min 46s		Pocketing	25,59
Pocket 2			Side Milling	88,01
			Open pocketing	40,13

Workshop document - Stock update color by tool

Model views displaying the stock can now display the machined faces by the operation in the same color as the tool used, as in verification.



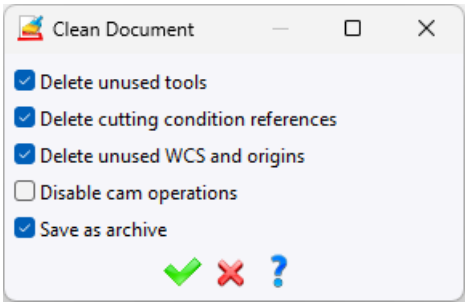
Tool view		Part view	Operations	
	Description		Facing	127,44
	Face Mill Extended Length D63 A90 L3 SD41		Facing	136,20
	Sequence time 0h 12min 30s		Facing	386,95
	Total time 0h 13min 18s		Facing	99,07
Pocket 1				
	Description		Open pocketing	8,88
	Side Mill Fretted D20 L35 SD20		Pocketing	10,64
	Sequence time 0h 3min 4s		Open pocketing	10,71
	Total time 0h 3min 46s		Pocketing	25,59
Pocket 2			Side Milling	88,01
			Open pocketing	40,13

Workshop document - Machine component data

Machine component data can now be retrieved in workshop documents.

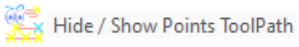
UI - Cleaning the document

The new **Clean Document** command, available in the **Operations** tab, offers the following options.



UI - Display/hide toolpath points

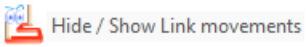
Anew icon in the graphics area allows you to show or hide machining operations' toolpath points. These points are the same color as the toolpath feed rate and are positioned at the extremities of the entities to go over.



In addition, blue points are displayed in the center of circular arcs.

UI - Display/hide toolpath link movements

A new icon in the graphics area allows you to display or hide link movements' toolpath points.



UI - Canceling all calculations

You can cancel the execution of the entire operation range by pressing the **Esc** key. Non-executed operations will need to be re-executed.

UI - End-of-program display

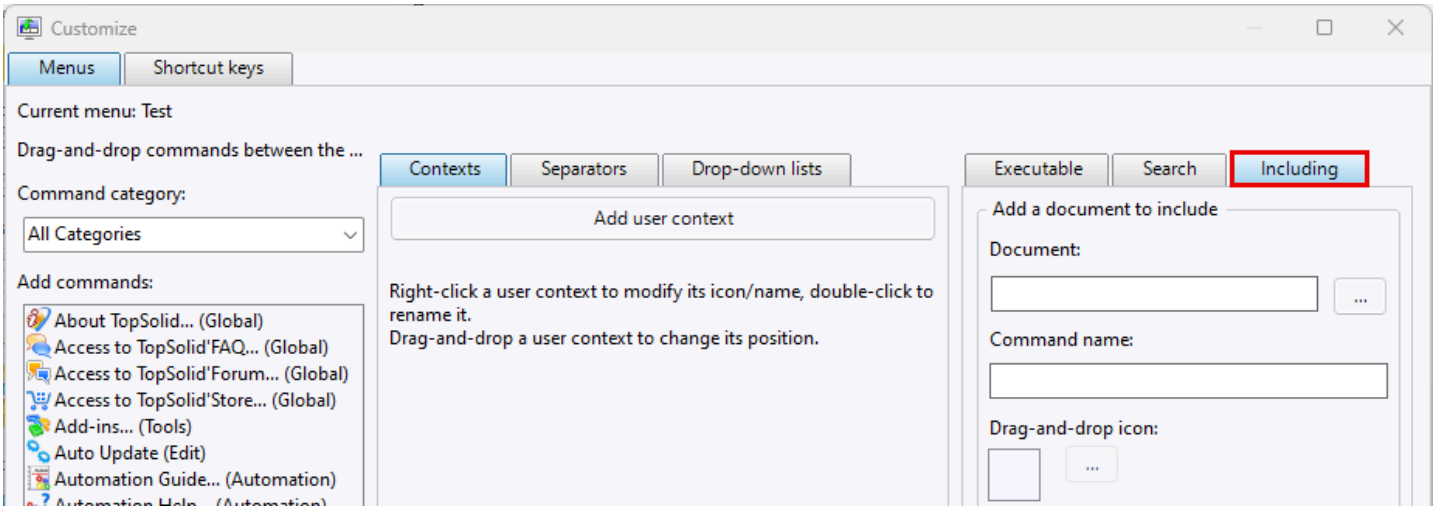
A line marking the end of a program appears in the NC Operations tree if a description is entered.

UI - Display operation number during editing

The operation number is now displayed before the tool name in the graphics area, when editing it.

UI - Menu: shortcut to include a document

A new **Including** tab in the **Customize** command's menu allows you to create shortcuts to a document to be included, contained in a library or project.



UI - Display/hide universal machine

It is now possible to display or hide the universal machine (planes/axes, etc.) by clicking on the machine icon, as for other machines.

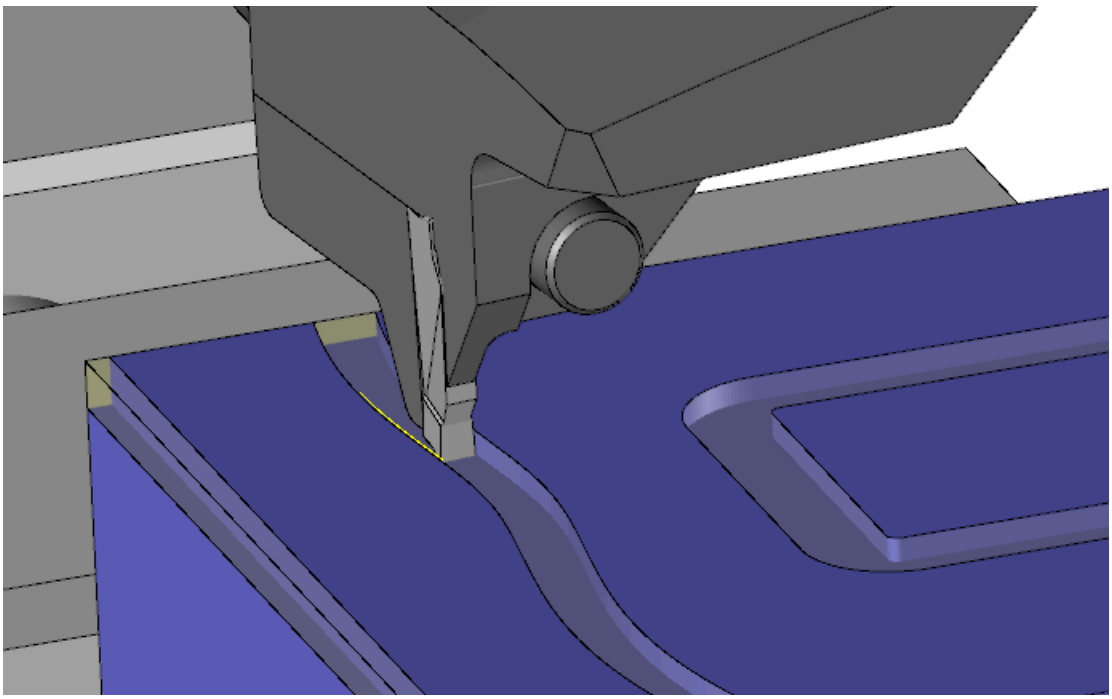
Hale Machining

Hale machining is a machining process in which neither the tool nor the part rotates around a power axis. The tool orientation is continuous, perpendicular to the machined profile.

The **Hale Machining** type has been added to the following operations:

- **Side Milling**
- **Finishing** (**Raster passes** / **Morphing** / **Constant step-over** modes)
- **3D Contouring**

It can be accessed from the **Multi Axis** dialog box by selecting **3X + Hale Machining** in the **Multi axis type** field.



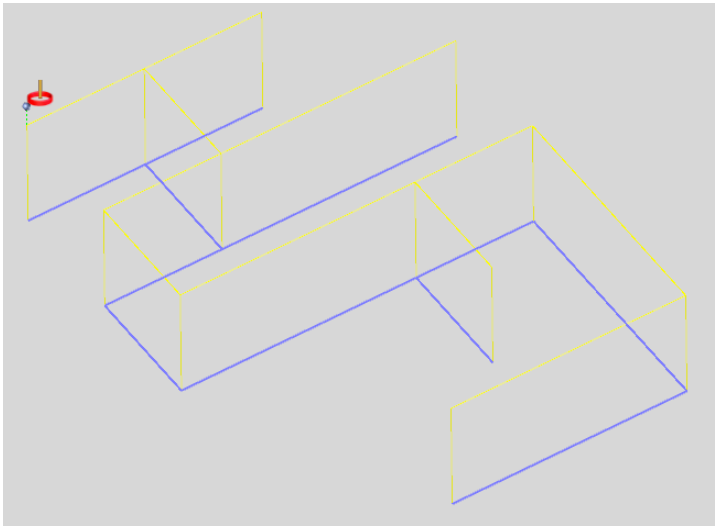
Wire

Wire - Verification in machine mode

Machine mode verification is now available in wire machining documents.

Wire - Cutting self-intersecting profiles

Wire cuts can be created using self-intersecting geometries, even when based on a self-intersecting profile.



Wire - Workshop document: threading points list

The list of threading points now only includes the thread points used in the cuts.

Wire - Selectable element filtering

When attempting to select an unsupported element, the **Set wire geometry** and **Cutting** commands are no longer available in the context menu.

Wire technology - Cutting type addition

A new **NC Type** field has been added to the **Wire EDM Machine** tab of the machine properties. It can be accessed from the **Machine** > **Machine Properties** command. It also appears in wire technology documents, with the default value **None**.

Additionally, a cutting types list is available for each NC type. The cutting type has been added to wire document filters and methods, as well as in a new column in the wire technology selection.

When importing a wire technology, the cutting type is assigned in the technology document. It is also included in the technology document's name.

Wire method - Exclusion of lead in and out technologies from operation count

In the **Select technologies** function, the **Wire cutting number** field now excludes the lead in / lead out technologies present in the wire technology document.

Cam Operator

Required check-in for modified documents

A new **Force check-in document when save** option is available in the **Cam Operator** section of **TopSolid**'s general options.

This ensures that modified documents must be checked into the vault before they can be saved.

User properties in document previews

User-defined properties are now visible in the preview of machining documents.

Simulation and verification on multiple operations

You can now perform simulation or verification on multiple operations simultaneously.

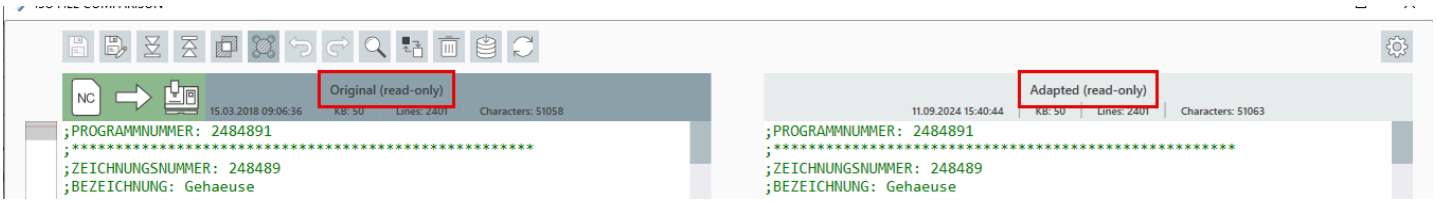
What's New in TopSolid'ShopFloor 7.19

Discover the latest features of **TopSolid'ShopFloor**, **TopSolid 7**'s manufacturing data management software!

Integrated NC program editor

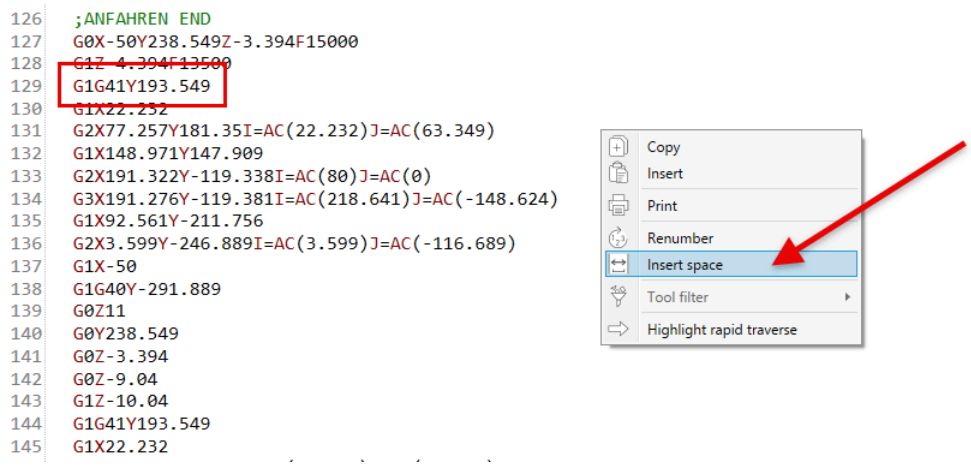
Read-only files

The ISO file comparison method has been optimized. Now, ISO files that cannot be modified are automatically marked as read-only.



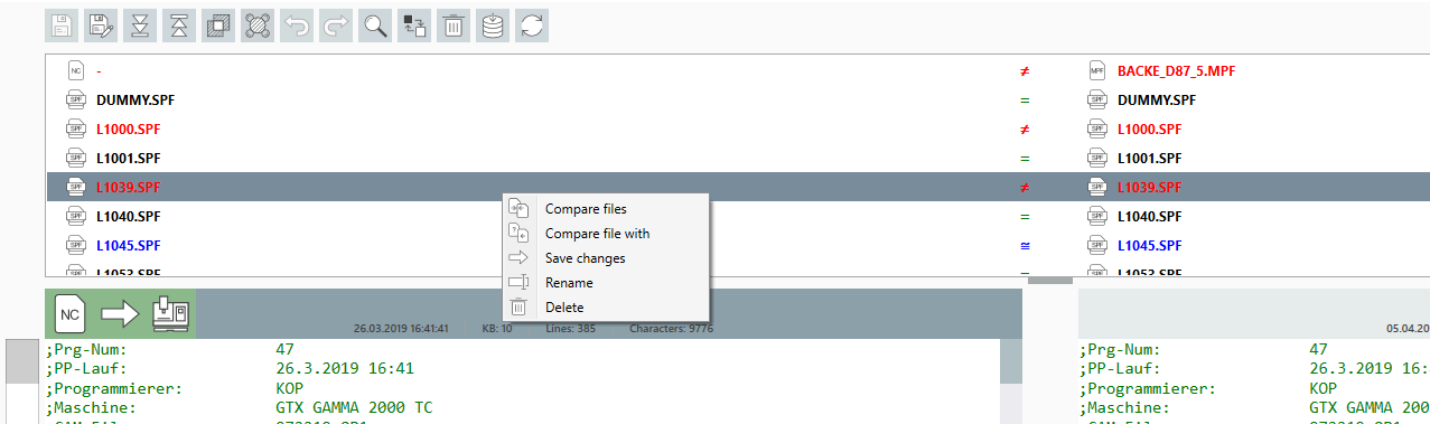
Automatic space insertion

A new feature has been added to the NC program editor to automatically insert spaces between coordinates, significantly improving code readability.



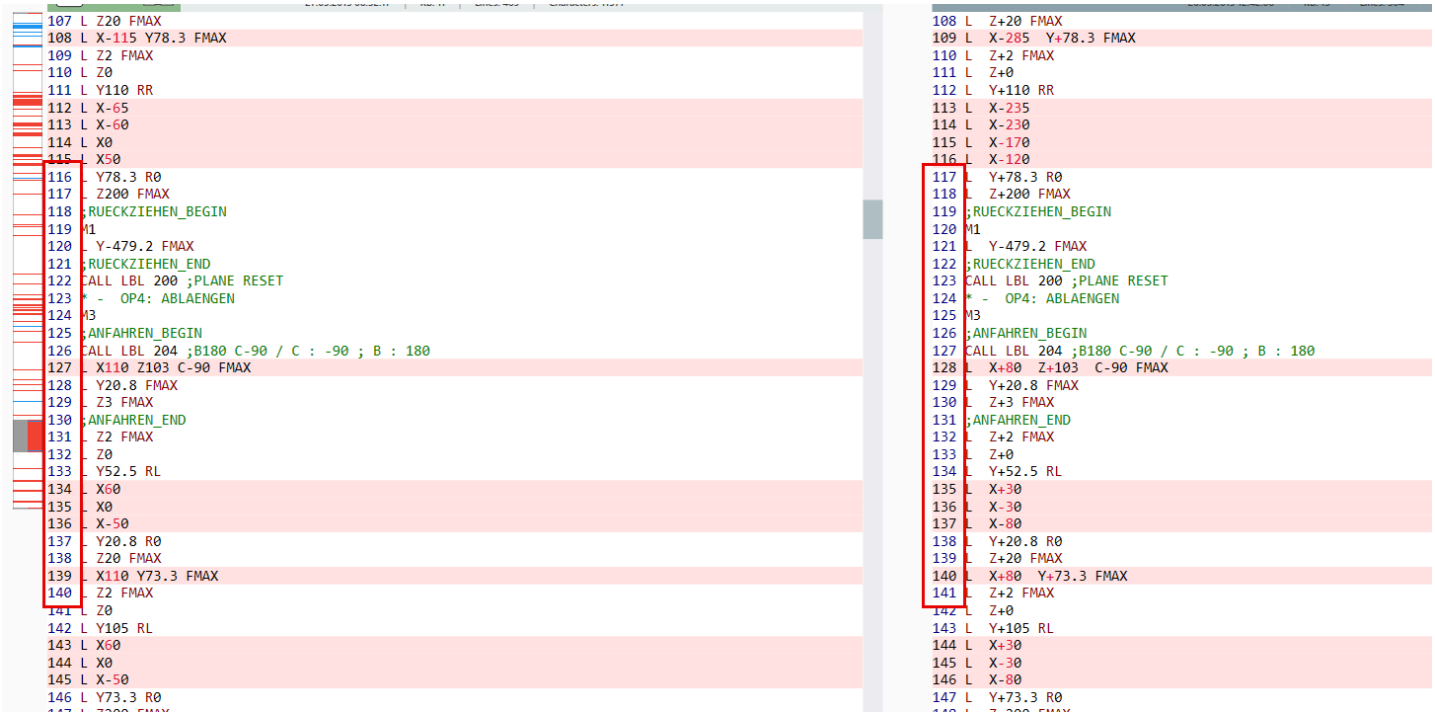
Improved pre-comparison for WPD folders

Pre-comparison for WPD folders has been optimized to distinguish between relevant and irrelevant differences, making comparisons more precise. In addition, the context menu has been enhanced with new commands —**Save changes**, **Rename** and **Delete**— improving operation efficiency.



Block number differences

Differences between block numbers are no longer highlighted.

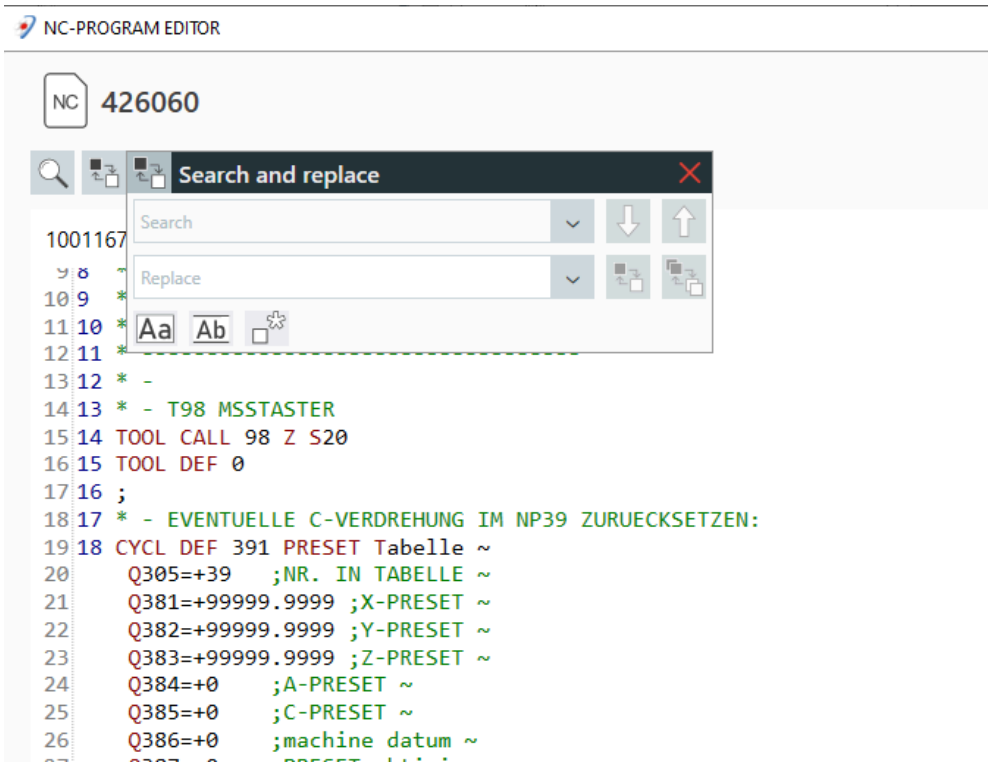


```
107 L Z20 FMAX
108 L X-115 Y78.3 FMAX
109 L Z2 FMAX
110 L Z0
111 L Y110 RR
112 L X-65
113 L X-60
114 L X0
115 L X50
116 L Y78.3 R0
117 L Z200 FMAX
118 ;RUECKZIEHEN_BEGIN
119 M1
120 L Y-479.2 FMAX
121 ;RUECKZIEHEN_END
122 CALL LBL 200 ;PLANE RESET
123 * - OP4: ABLAENGEN
124 M3
125 ;ANFAHREN_BEGIN
126 CALL LBL 204 ;B180 C-90 / C : -90 ; B : 180
127 L X110 Z103 C-90 FMAX
128 L Y20.8 FMAX
129 L Z3 FMAX
130 ;ANFAHREN_END
131 L Z2 FMAX
132 L Z0
133 L Y52.5 RL
134 L X60
135 L X0
136 L X-50
137 L Y20.8 R0
138 L Z20 FMAX
139 L X110 Y73.3 FMAX
140 L Z2 FMAX
141 L Z0
142 L Y105 RL
143 L X60
144 L X0
145 L X-50
146 L Y73.3 R0
```

```
108 L  Z+20 FMAX
109 L  X-285  Y+78.3 FMAX
110 L  Z+2 FMAX
111 L  Z+0
112 L  Y+110 RR
113 L  X-235
114 L  X-230
115 L  X-170
116 L  X-120
117 L  Y+78.3 R0
118 L  Z+200 FMAX
119 ;RUECKZIEHEN_BEGIN
120 M1
121 L  Y-479.2 FMAX
122 ;RUECKZIEHEN_END
123 CALL LBL 200 ;PLANE RESET
124 * -  OP4: ABLAENGEN
125 M3
126 ;ANFAHREN_BEGIN
127 CALL LBL 204 ;B180 C-90 / C : -90 ; B : 180
128 L  X+80  Z+103  C-90 FMAX
129 L  Y+20.8 FMAX
130 L  Z+3 FMAX
131 ;ANFAHREN_END
132 L  Z+2 FMAX
133 L  Z+0
134 L  Y+52.5 RL
135 L  X+30
136 L  X-30
137 L  X-80
138 L  Y+20.8 R0
139 L  Z+20 FMAX
140 L  X+80  Y+73.3 FMAX
141 L  Z+2 FMAX
142 L  Z+0
143 L  Y+105 RL
144 L  X+30
145 L  X-30
146 L  X-80
147 L  Y+73.3 R0
```

Search and replace

The NC program editor now offers enhanced search and replace functionality. Press the **F3** or **Shift+F3** to initiate searches, with results highlighted and numbered. Advanced options such as **case sensitivity**, **whole-word search** and **regular** expressions are now available. You can also switch between **Search** and **Search and replace**. Finally, the search mask is now movable and integrated directly into the editor window for a smoother experience.

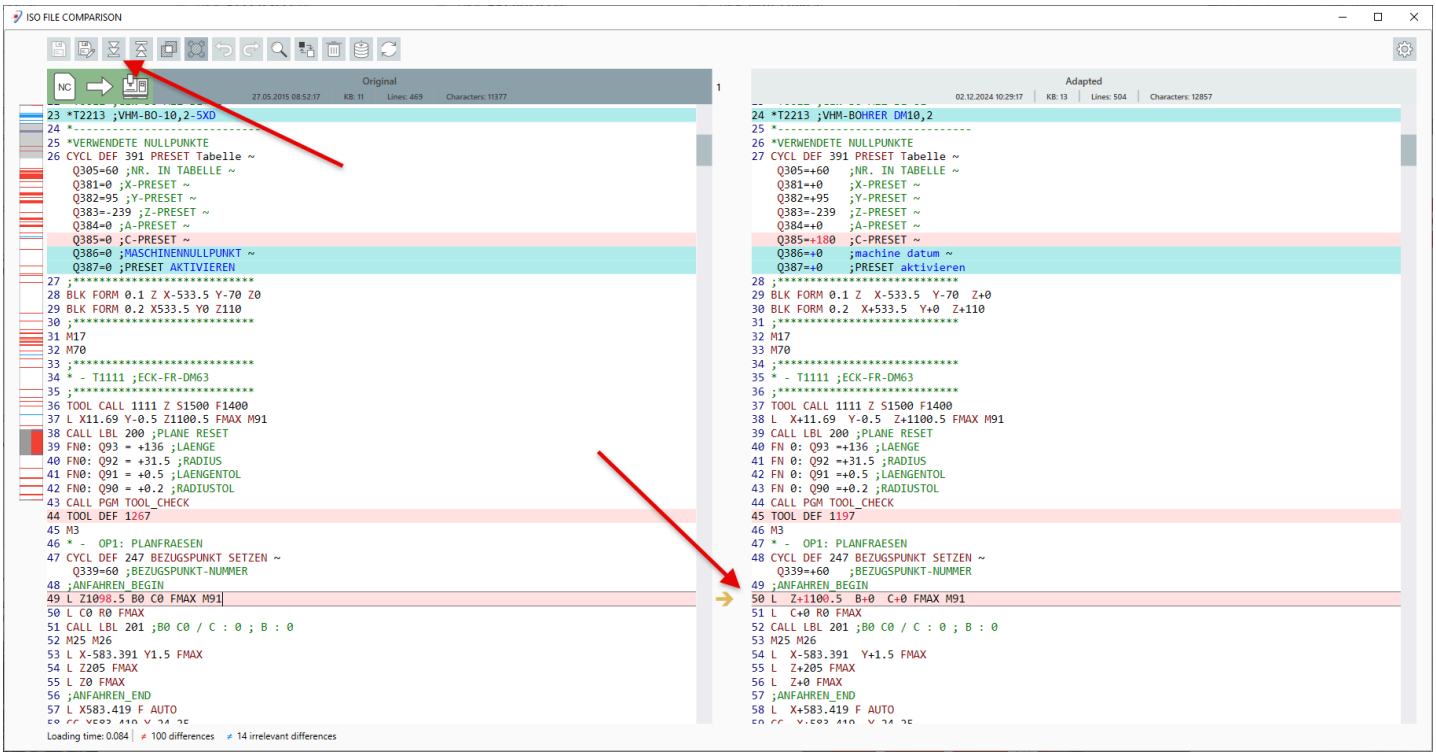


Keyboard shortcuts

New shortcuts improve navigation in the NC editor:

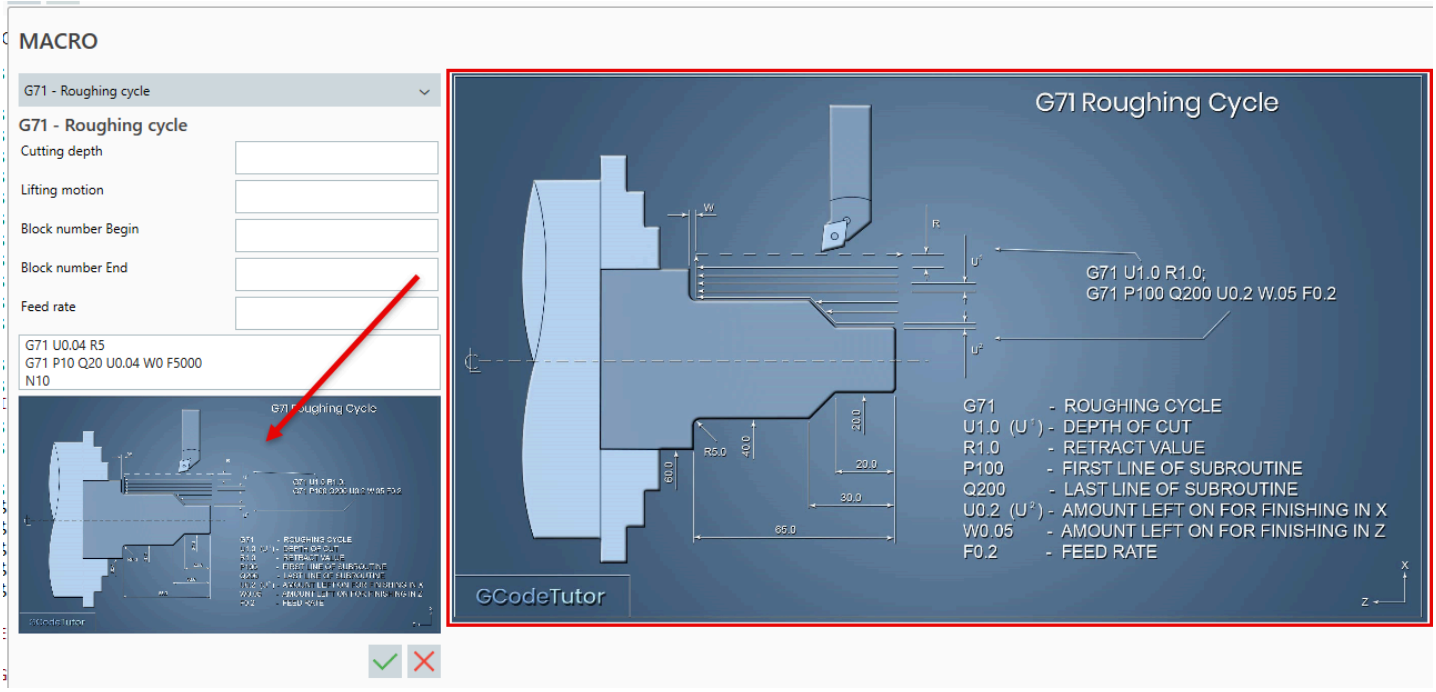
- **Ctrl+N**: Go to the next difference.
- **Ctrl+B**: Go to the previous difference.
- **Ctrl+O**: Apply the difference to both sides.

The existing shortcuts **Ctrl+L** and **Ctrl+R** remain for left and right actions, respectively.



Macro

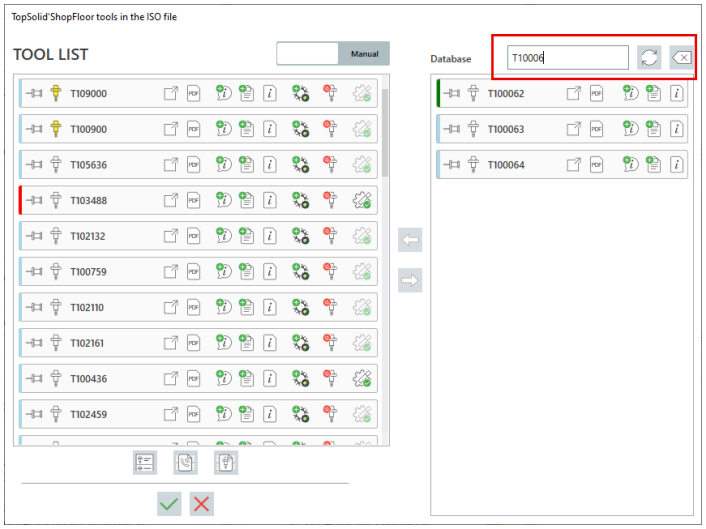
When you click on the preview image of a macro in the NC editor, it is enlarged and displayed next to it. A second click reduces the preview.

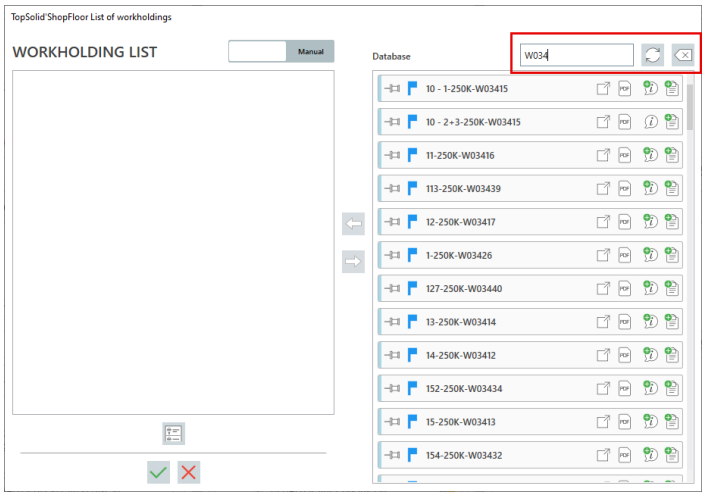


Program Manager

Tool/Workholding list

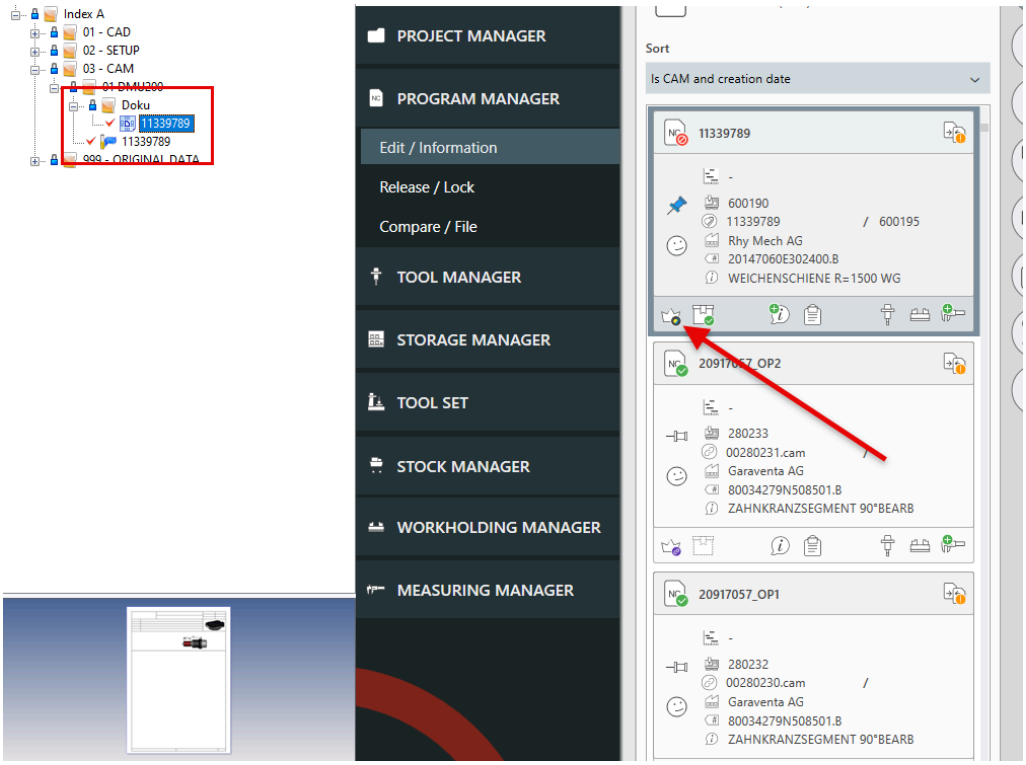
It is now possible to search directly in the tool list and workholding list database in **Manual** mode, simplifying specific entry searches and optimizing workflows.





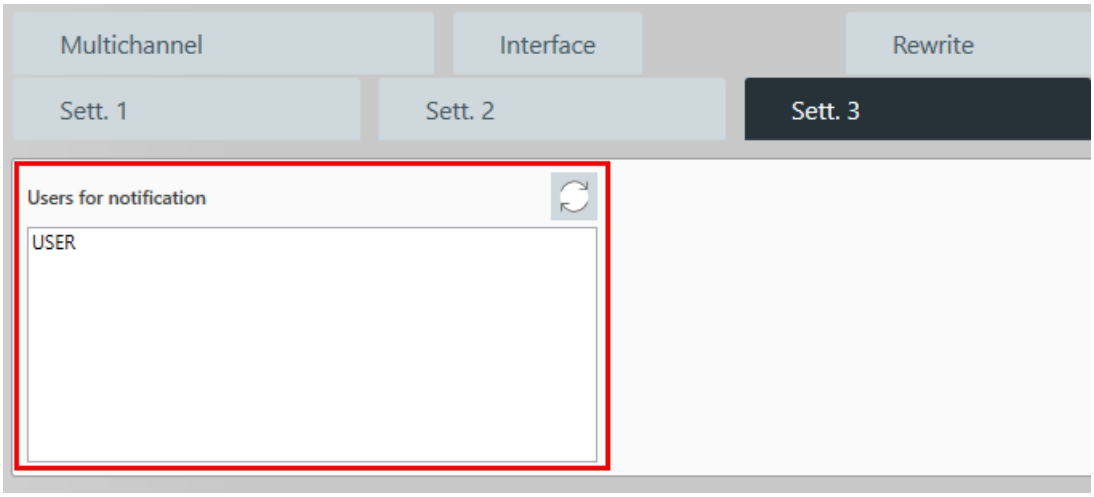
Viewer package

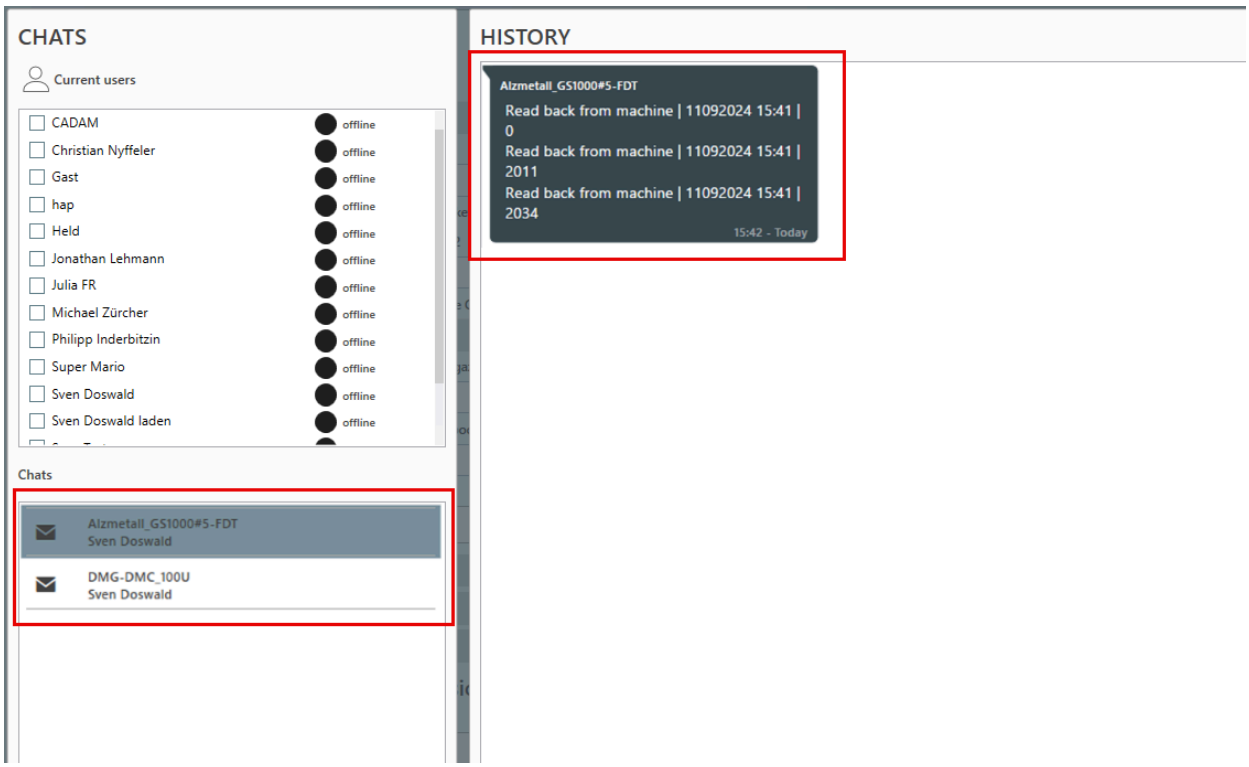
A check-in process is now performed before a visualization package is created, ensuring that all changes are correctly applied.



ISO file read-back notifications

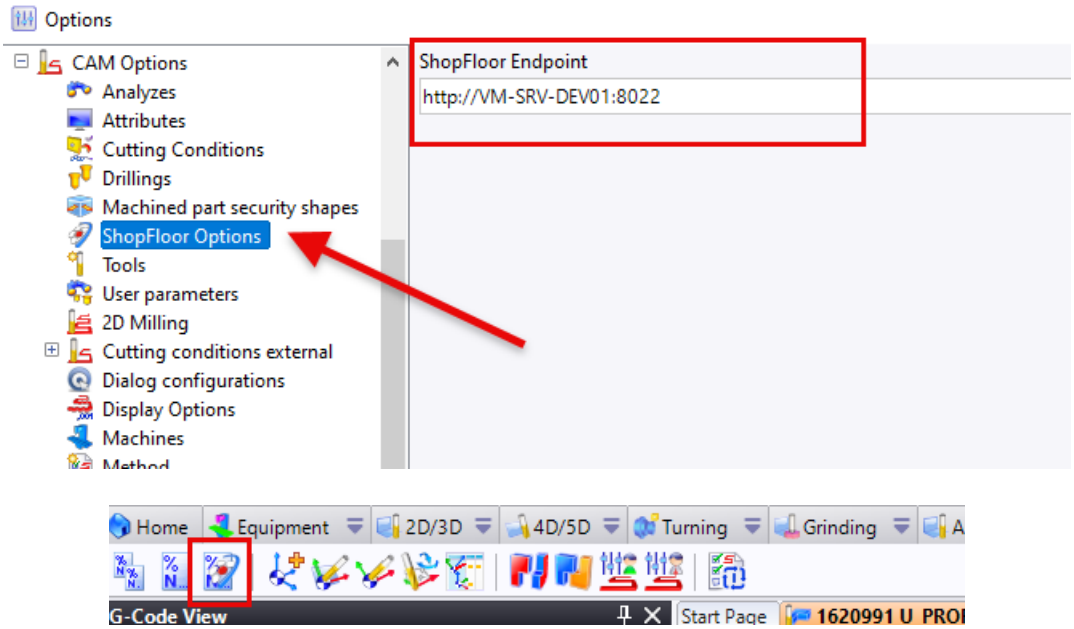
You can now define, for each machine, a list of users to be notified when ISO files have been read back. Notifications are generated just before the daily cache reread.





TopSolid'Cam <> TopSolid'ShopFloor Program Manager interface

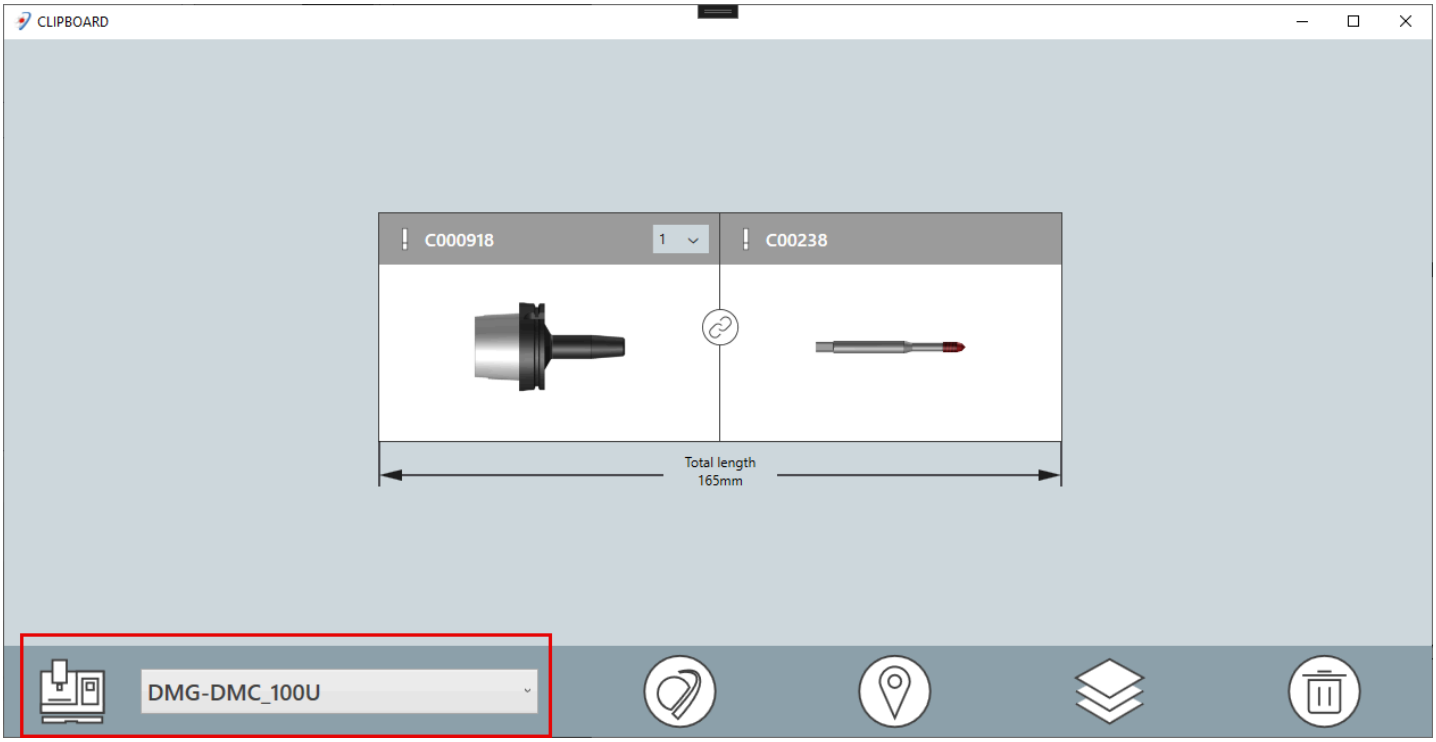
A new interface has been established between **TopSolid'Cam** and the **TopSolid'ShopFloor Program Manager**. When enabled, an icon generates ISO code directly for **ShopFloor**. During generation, the next available program number is automatically assigned, and in **ShopFloor**, the ISO file is linked to the corresponding **CAM** file.



Tool Manager

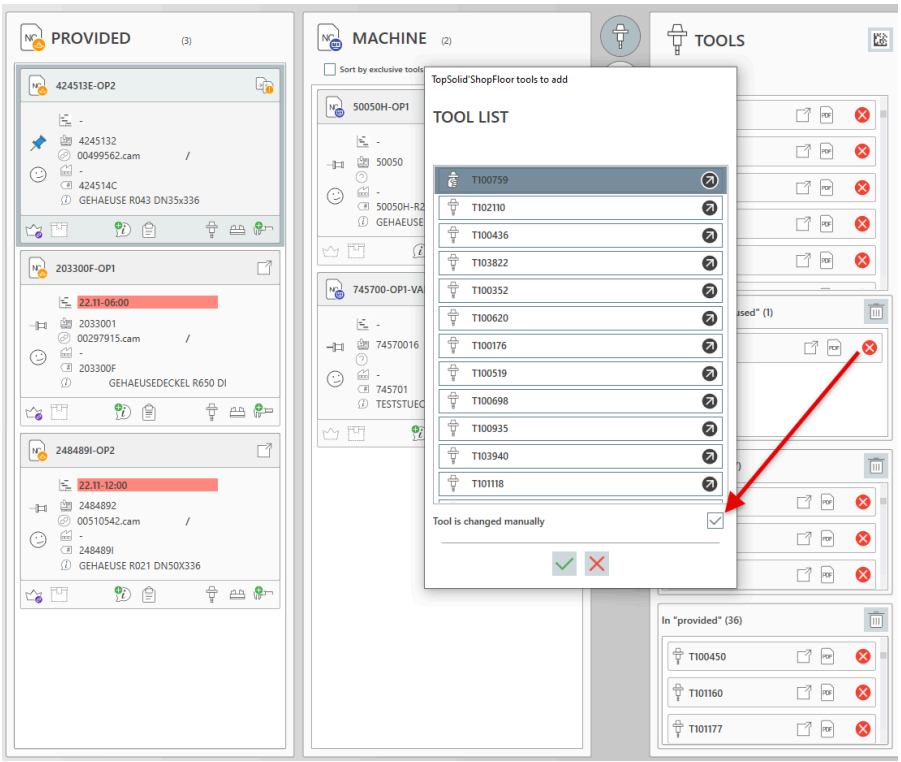
Clipboard

A new option lets you display and modify the active machine in the clipboard. This feature makes it easy to quickly switch from one machine to another, ensuring that the correct machine is selected.



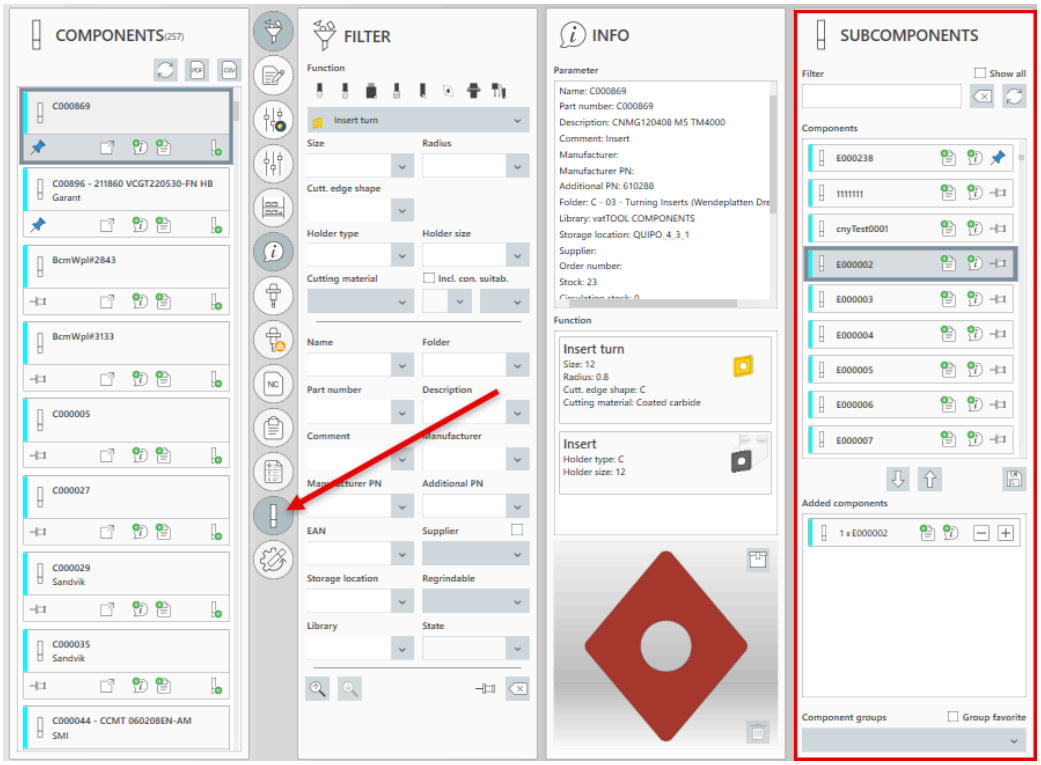
Temporary manual changes

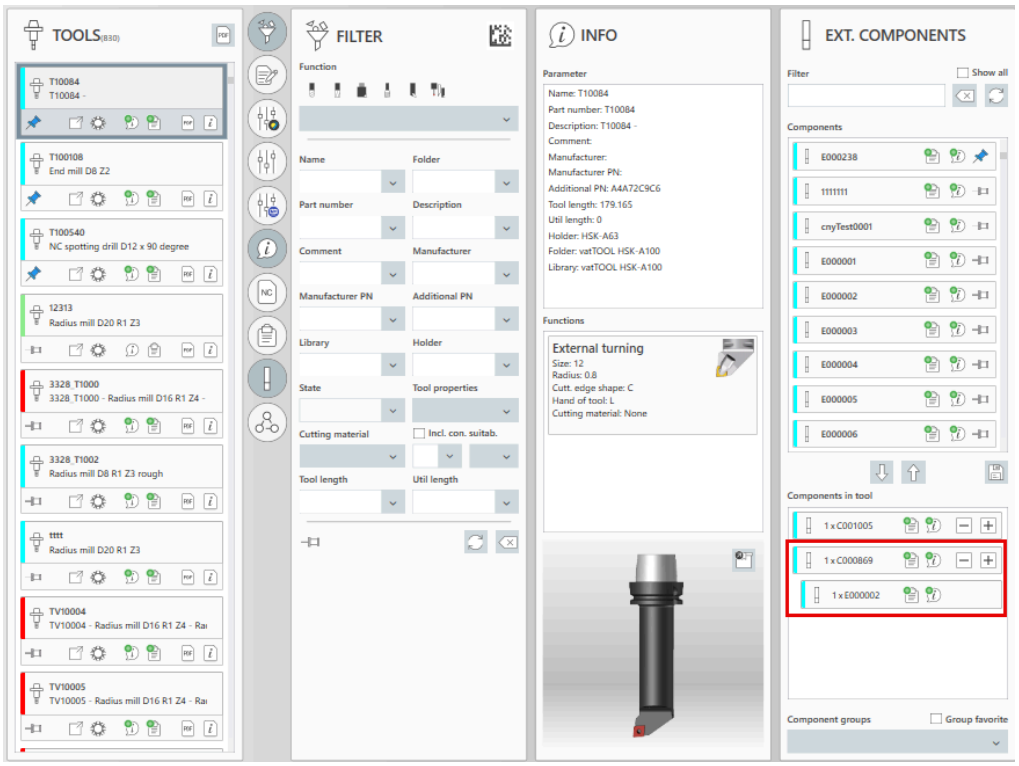
If a tool must be manually replaced temporarily (e.g., due to stock shortages), an option can be enabled when loading the ISO file onto the machine. This option deactivates automatically once the tool has been removed from the magazine, ensuring smooth replacement and optimized management.



Subcomponents

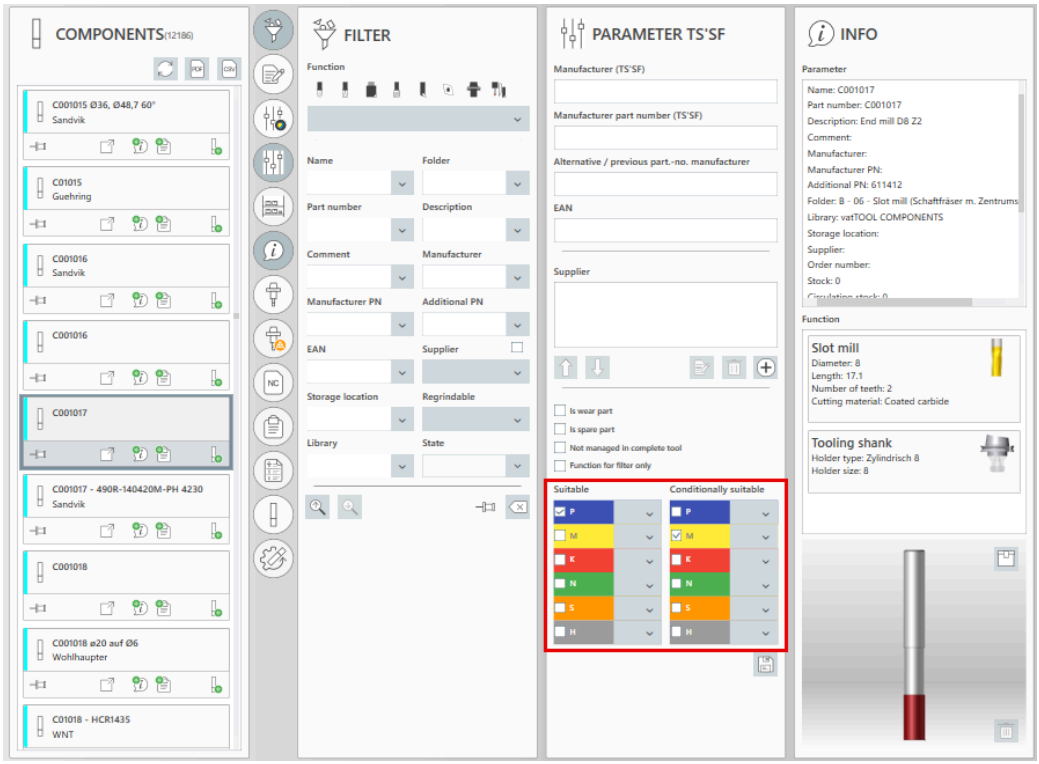
Subcomponents can now be defined for tool components, such as a screw for an insert. When the main component is used in a tool, the corresponding subcomponents are automatically added.

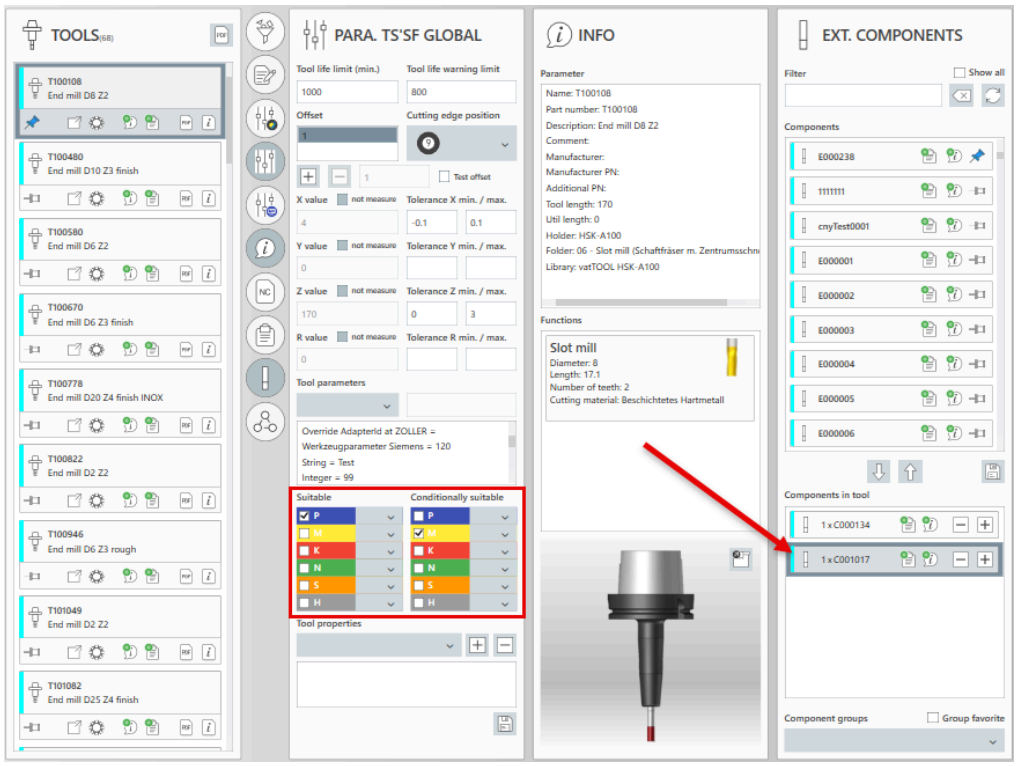




Material usage inheritance

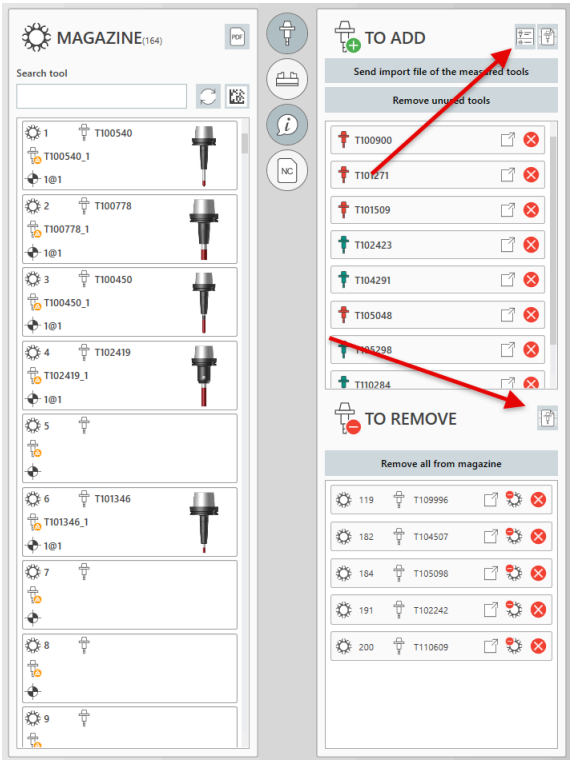
Automatic inheritance of material usage from components to tools is now possible. A client restart is required for the new definition to take effect and for the changes to be visible in the tool.





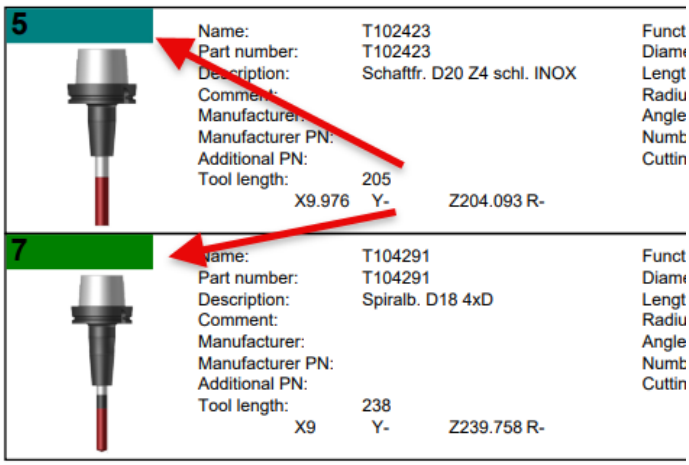
Tools to add/remove - PDF

Tools to be removed are now listed in the tool list. You can also generate a PDF list containing these tools.



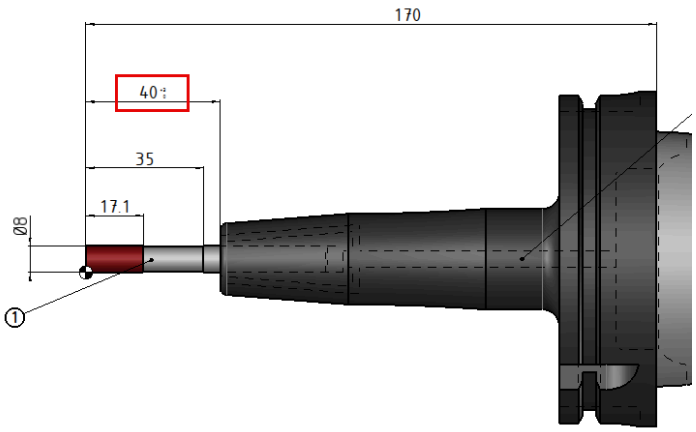
Tool state in setup/dismantle list

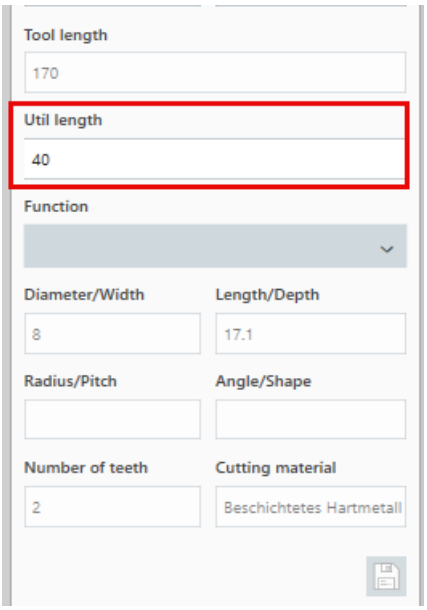
The state of tools to be set up is now transferred to the setup/dismantle list.



Util length

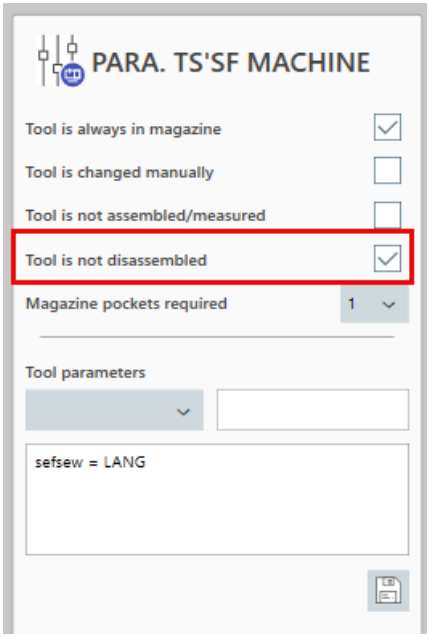
The new **Util length** parameter is automatically retrieved from **TopSolid**, but can be adjusted manually if required.

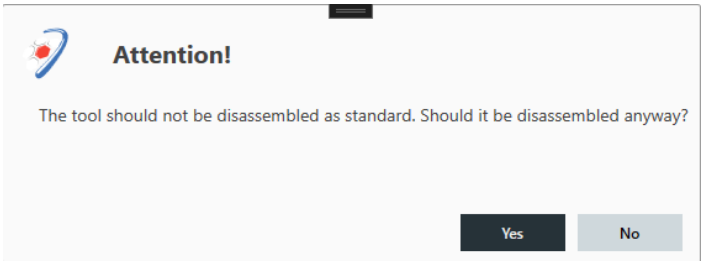




Non-dismantling tool

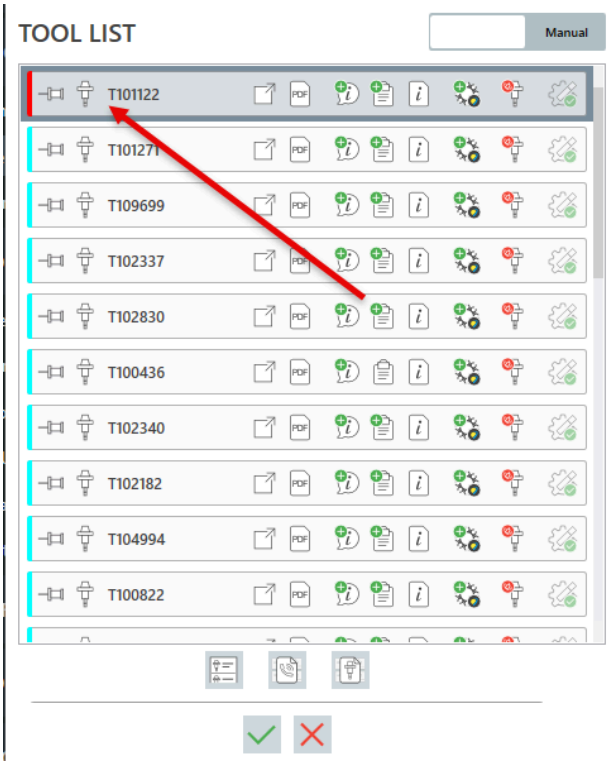
Tools can now be marked as "Tool is not disassembled". If dismantling is attempted, a warning message appears, but the tool can still be disassembled.

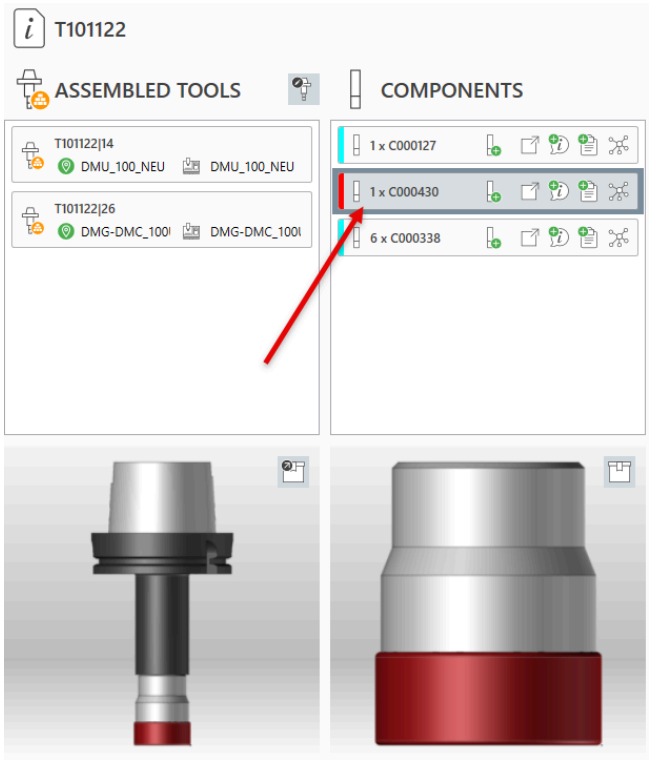




Component state inheritance

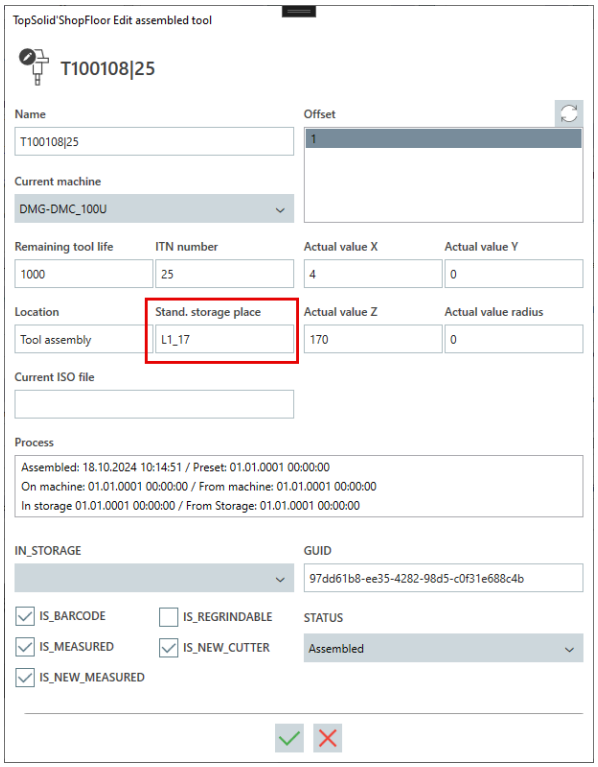
Component state is now transferred to the complete tool. For example, if a complete tool contains a locked component, it is visually indicated as locked even if its actual state remains unchanged. When a program is released or provided, it is automatically checked for locked components. If detected, a warning message is displayed.





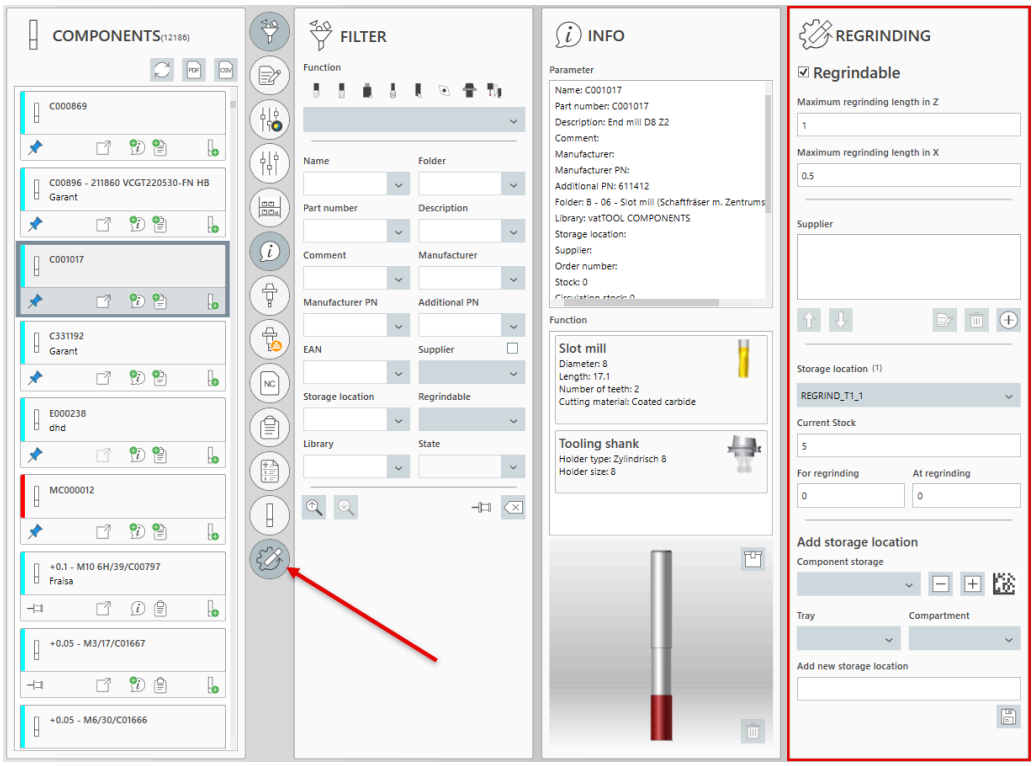
Standard storage place

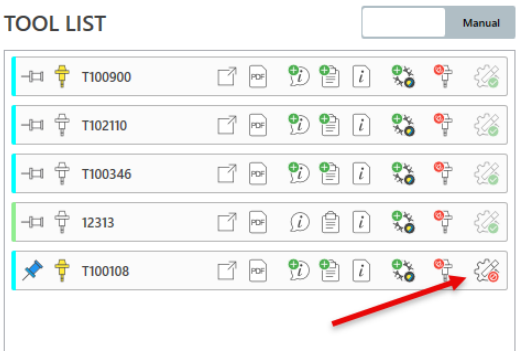
When an assembled tool is linked to a standard storage place, a notification now appears in the following cases: when it is loaded from the magazine, if the standard place corresponds to the machine's standard storage, and when it is disassembled.

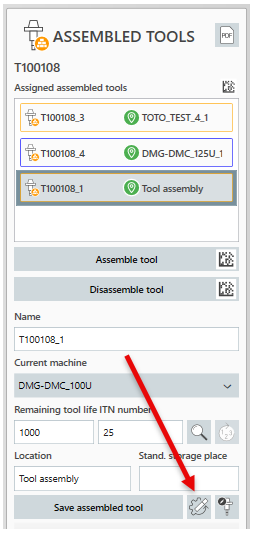


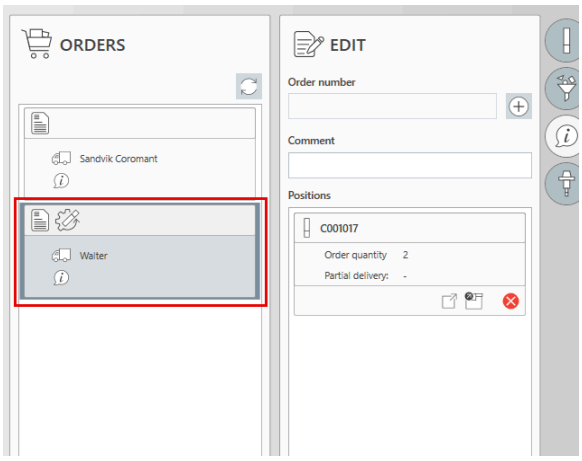
Regrinding tool management

Components requiring regrinding are now managed.



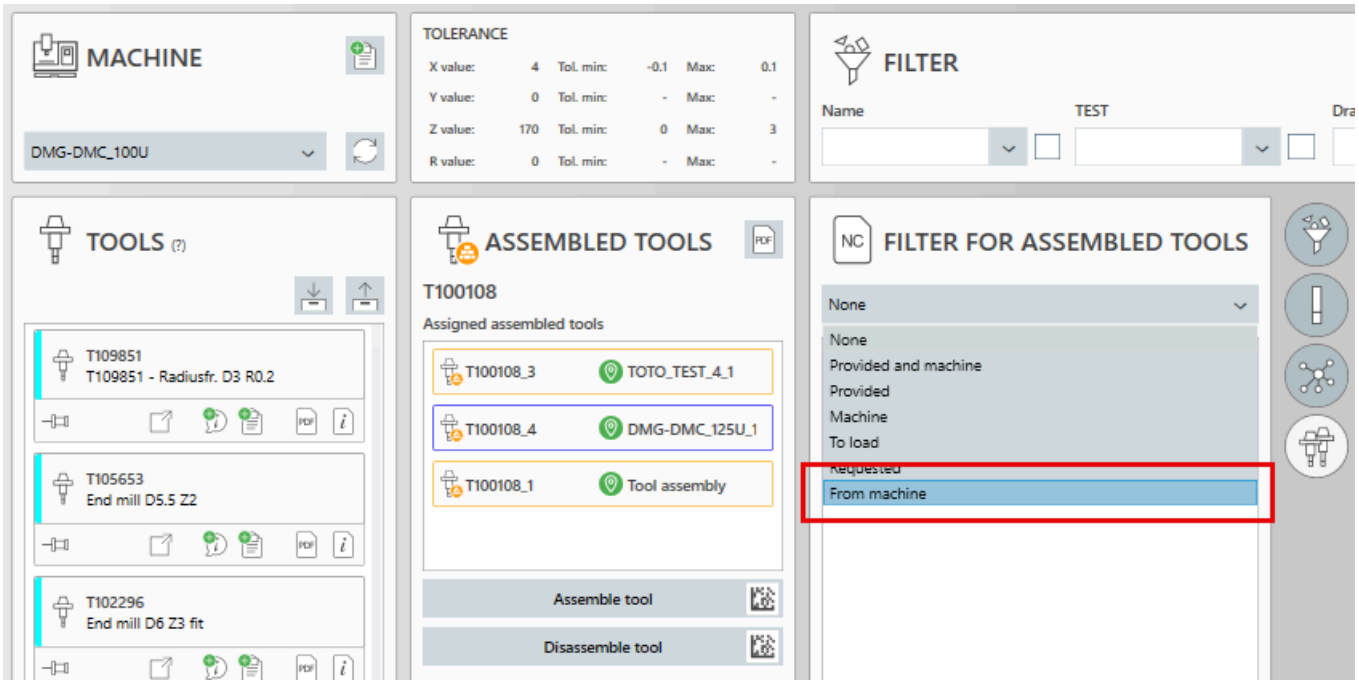






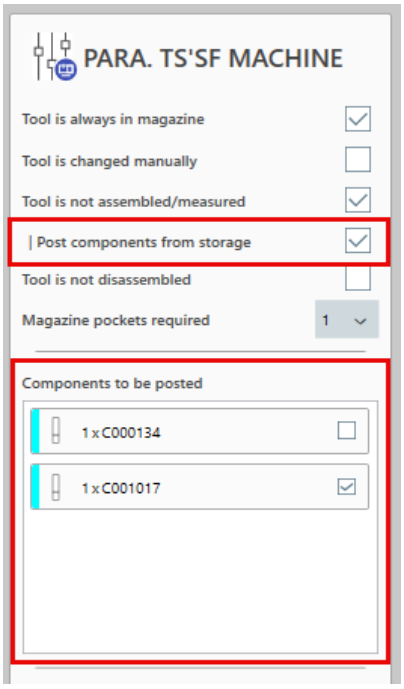
New filter for assembled tools

A new **From machine** filter is available for assembled tools, allowing specific selection of tools returning from the machine.



Extension of the removal list during assembly process

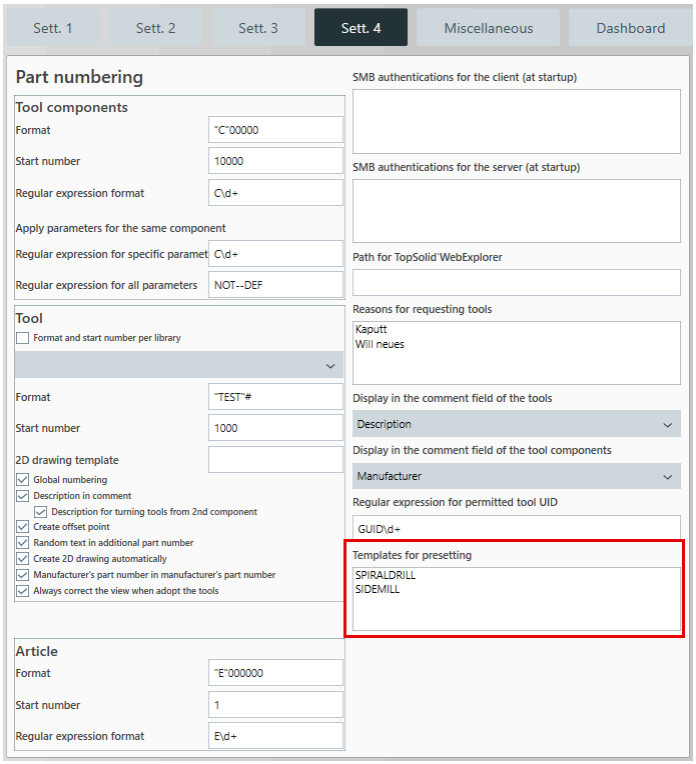
You can now define certain complete tool components that, while not requiring assembly or measurement, still need to be removed from storage. These components are automatically added to the removal list during the assembly process.

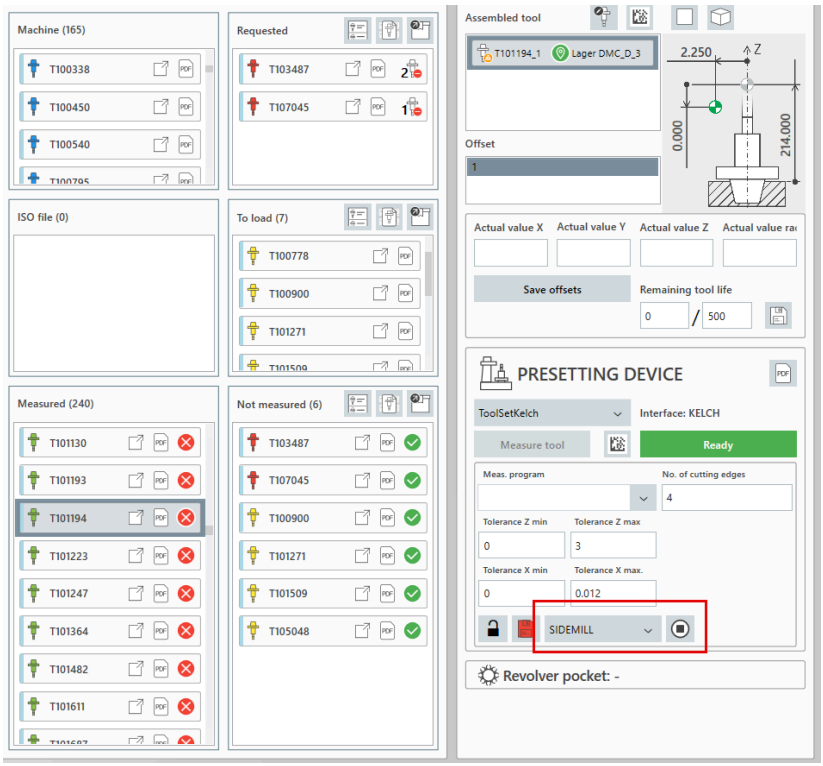


Tool Set

Preset templates

Specific templates for tool presetting can now be defined and selected directly in the **Tool Set**, enhancing efficiency and user experience.

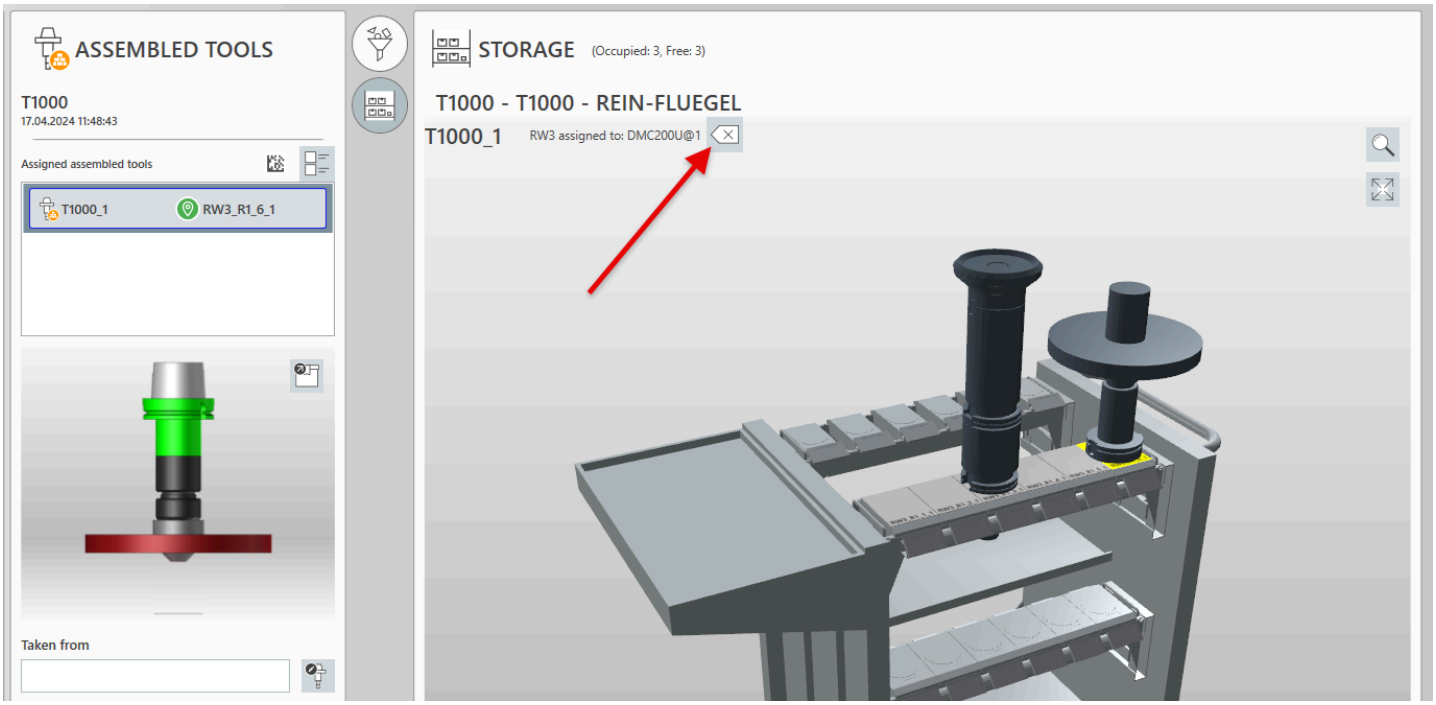




Storage Manager

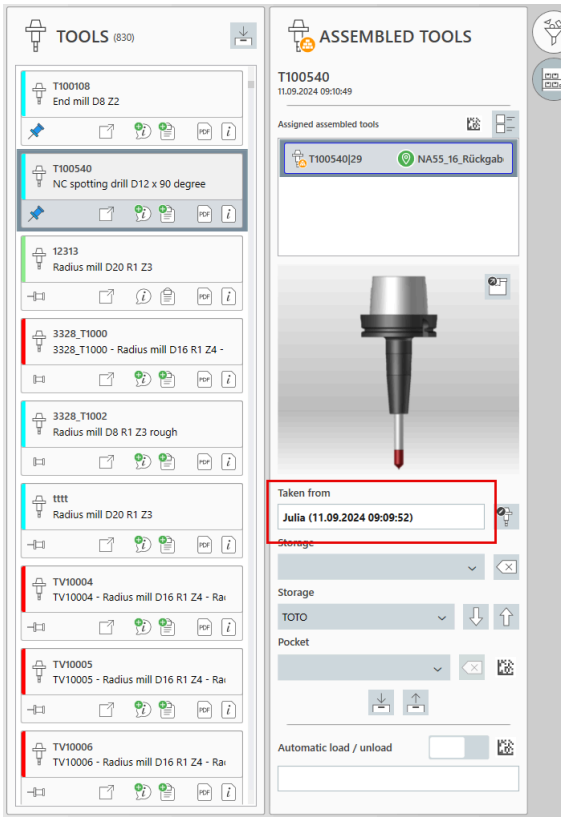
Assignment removal

It is now possible to remove the assignment of an intermediate reserved storage space, offering greater flexibility in the management of storage resources.



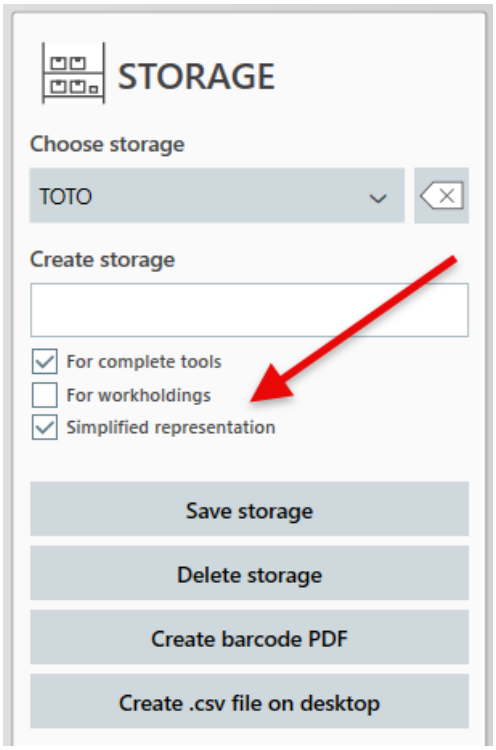
Display of tool removal date and time

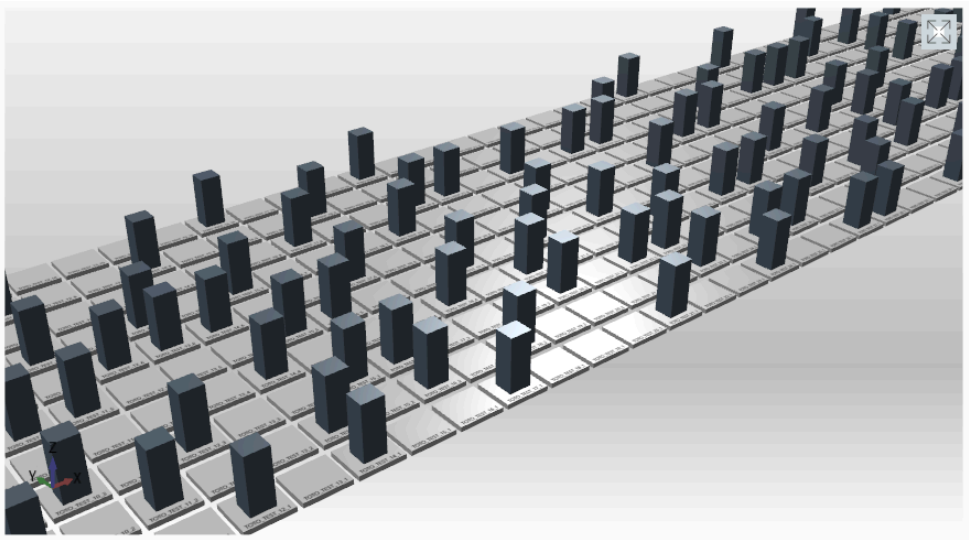
The **Storage Manager** now displays the date and time the tool was removed from storage, improving traceability.



Simplified representation

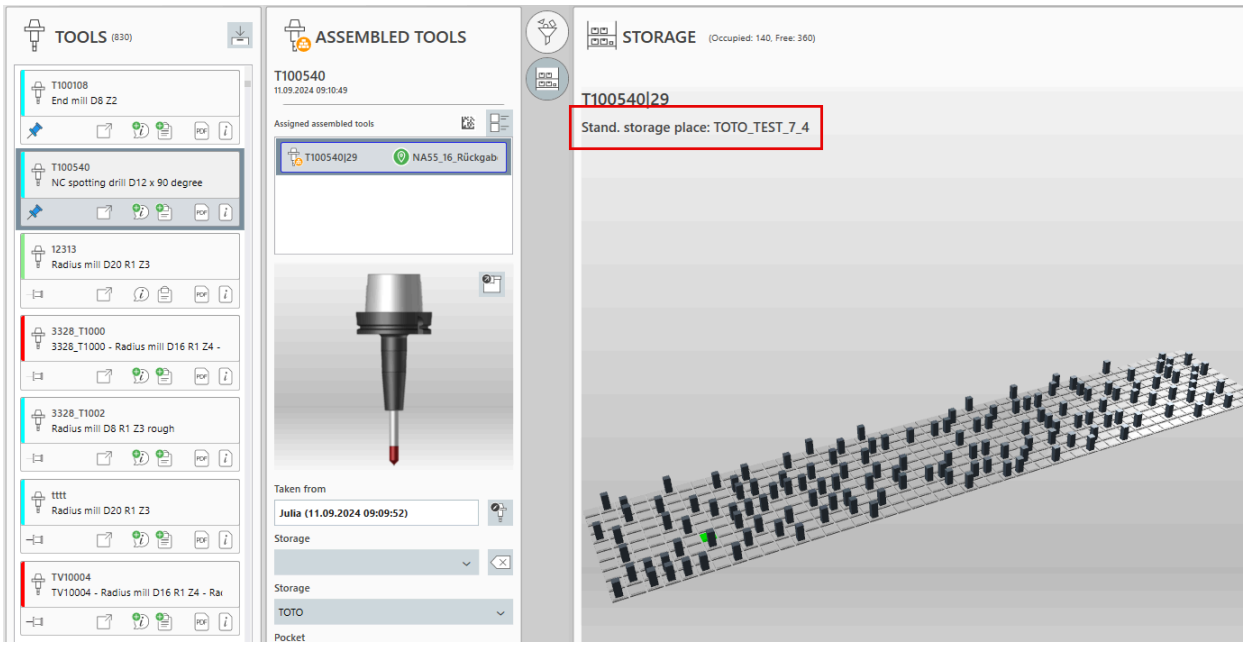
A simplified representation is now available for large storages with numerous pockets, optimizing loading times and improving efficiency.





Standard storage place

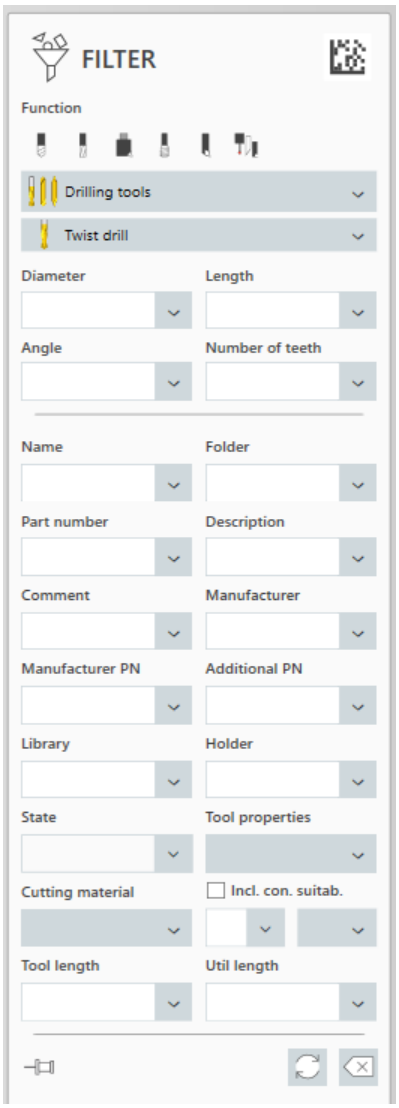
When an assembled tool with a stored standard storage place is selected, the standard storage place is displayed as text. If you attempt to load this tool into a pocket stored as a standard storage place for another tool, a warning message appears and the action is canceled.

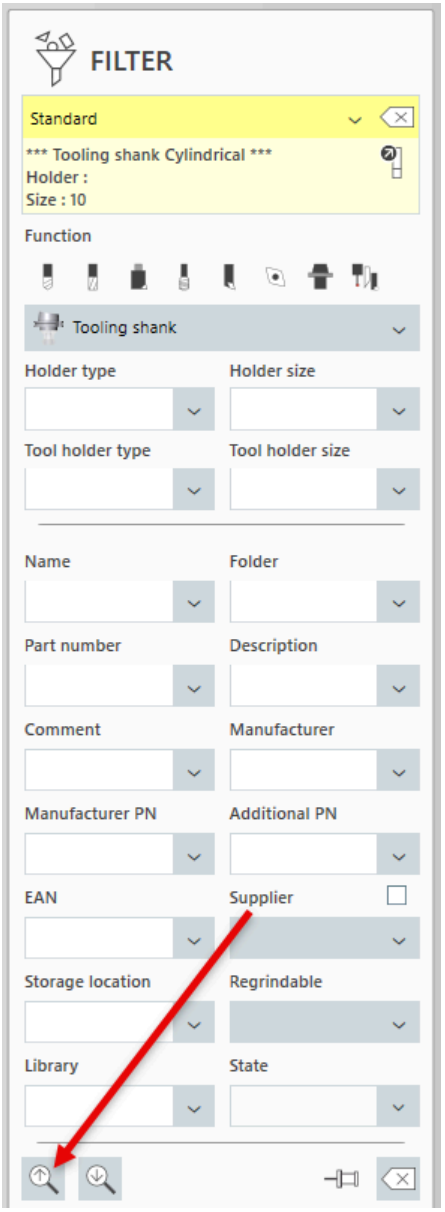


Filter

Improvements

The filter for complete tools and tool components has been completely redesigned for smoother, more dynamic user experience. Performance has been significantly improved, boosting process efficiency. Mapping has also been integrated, enabling the compatible components to be filtered (e.g., a drill with a cylindrical tool shank can be paired with a cylindrical, Weldon or shrink-fit tool holder). Finally, you can now filter by functional groups or select tools and components with no assigned function.

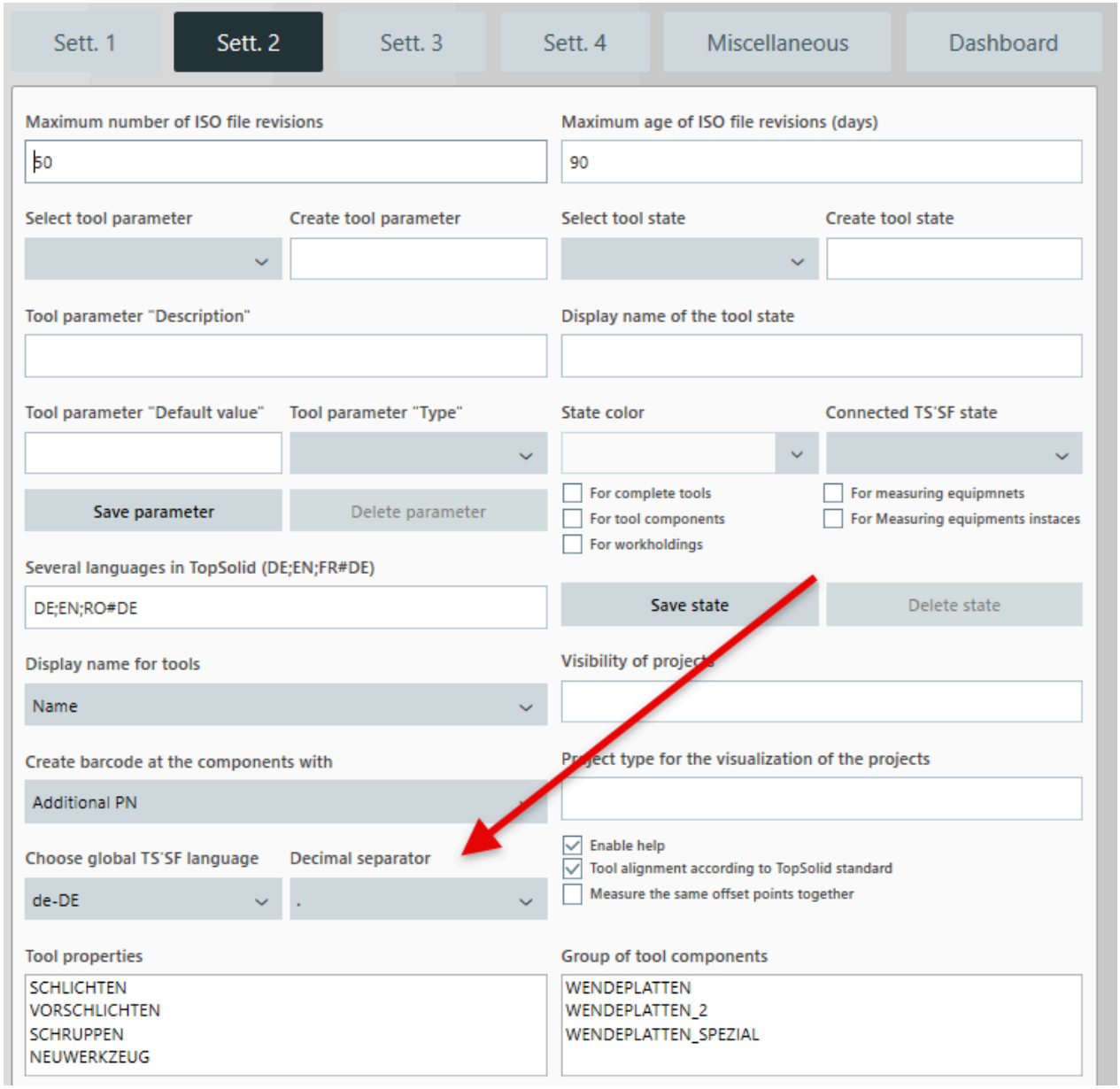




Settings

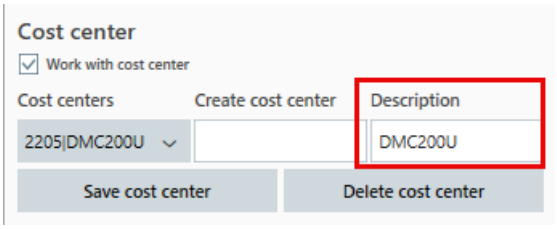
Decimal separator

The decimal separator is now configured independently of Windows settings in **ShopFloor**, simplifying operations.

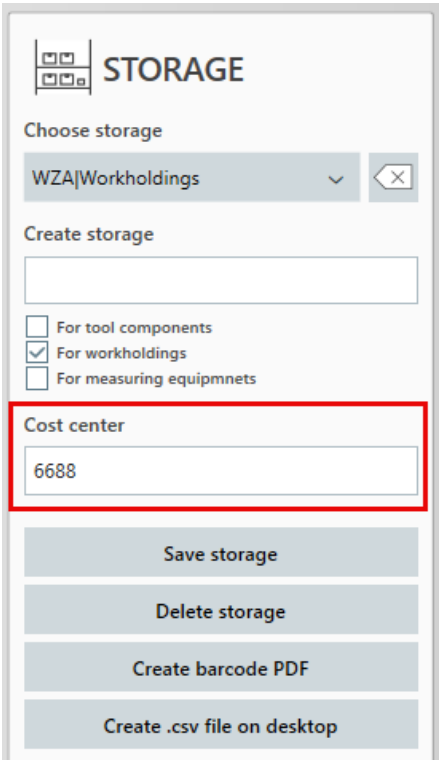


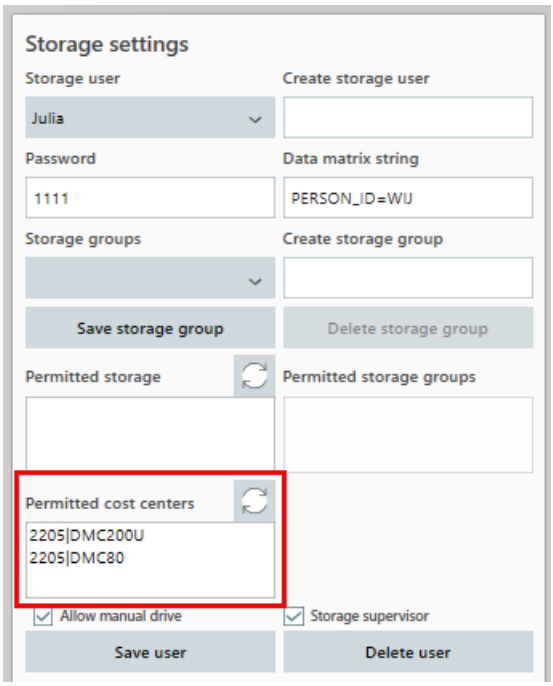
Cost centers

Cost centers can now include a description.



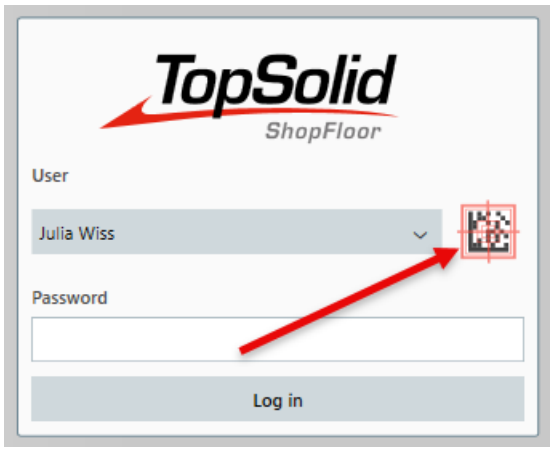
It is also possible to define specific cost centers for component storages, and assign them to individual storage users for authorization.





Login

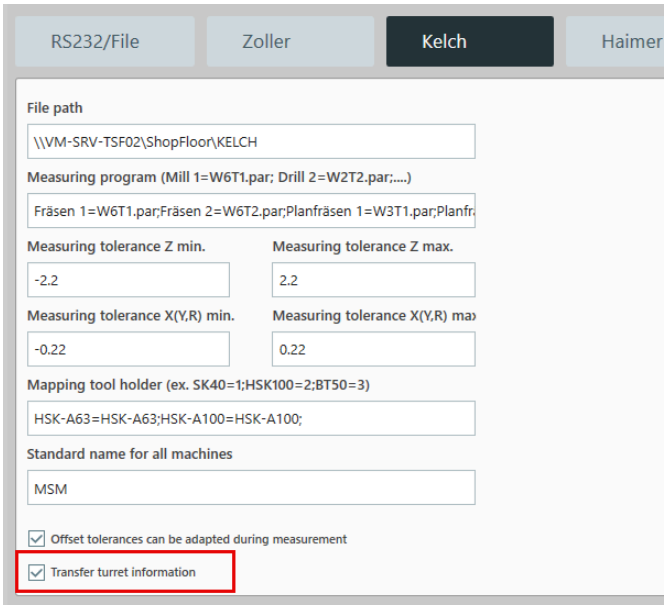
ShopFloor users can now log in using a Data Matrix code, provided such a code is registered for the selected user. This feature is automatically enabled whenever a Data Matrix is available.

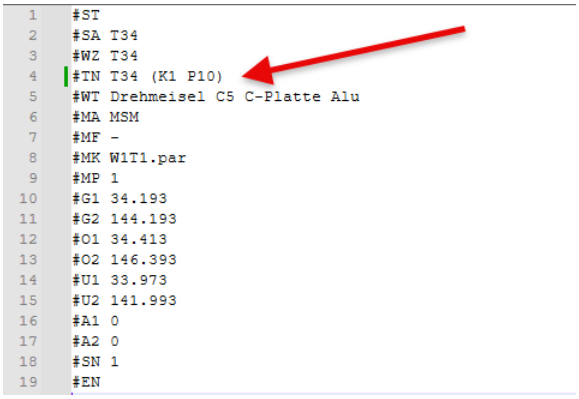


Interfaces

Kelch: Turret information transfer

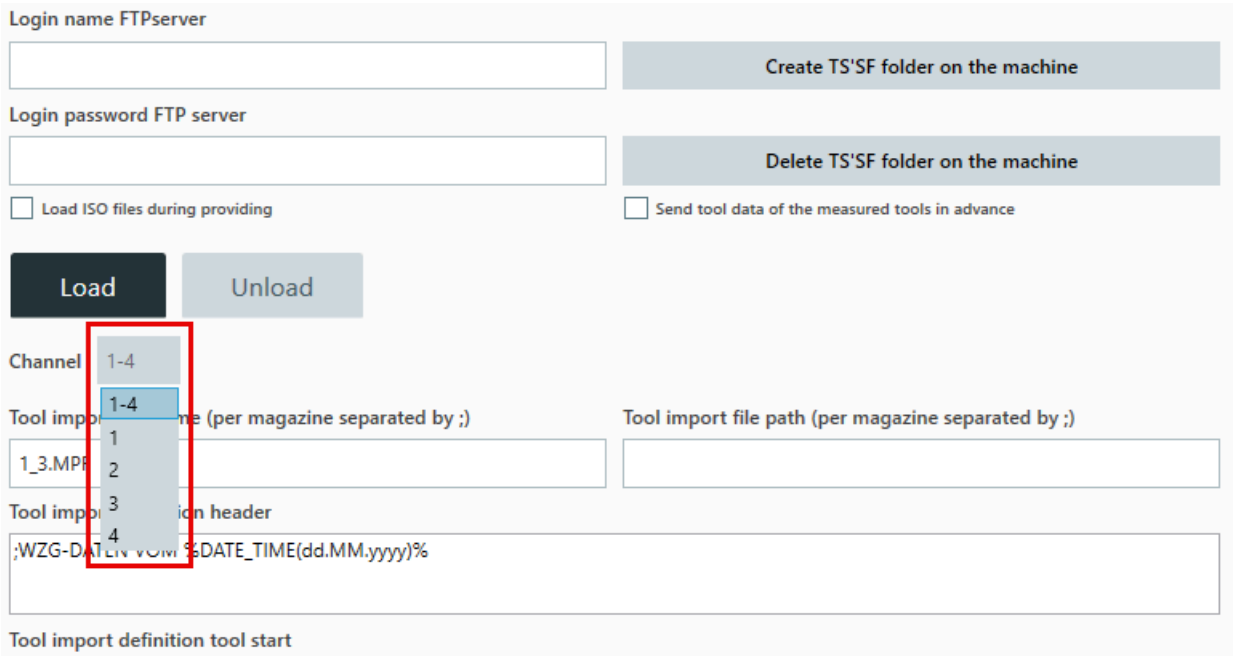
In the **Kelch** interface, it is now possible to transfer turret-related information. This option can be selected in **Settings** > **Configure tool presetter** > **Kelch**.





Tool data transfer

You can now generate one file per channel in the tool data transfer interface.

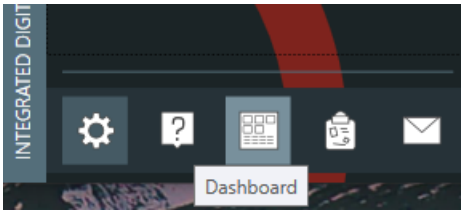


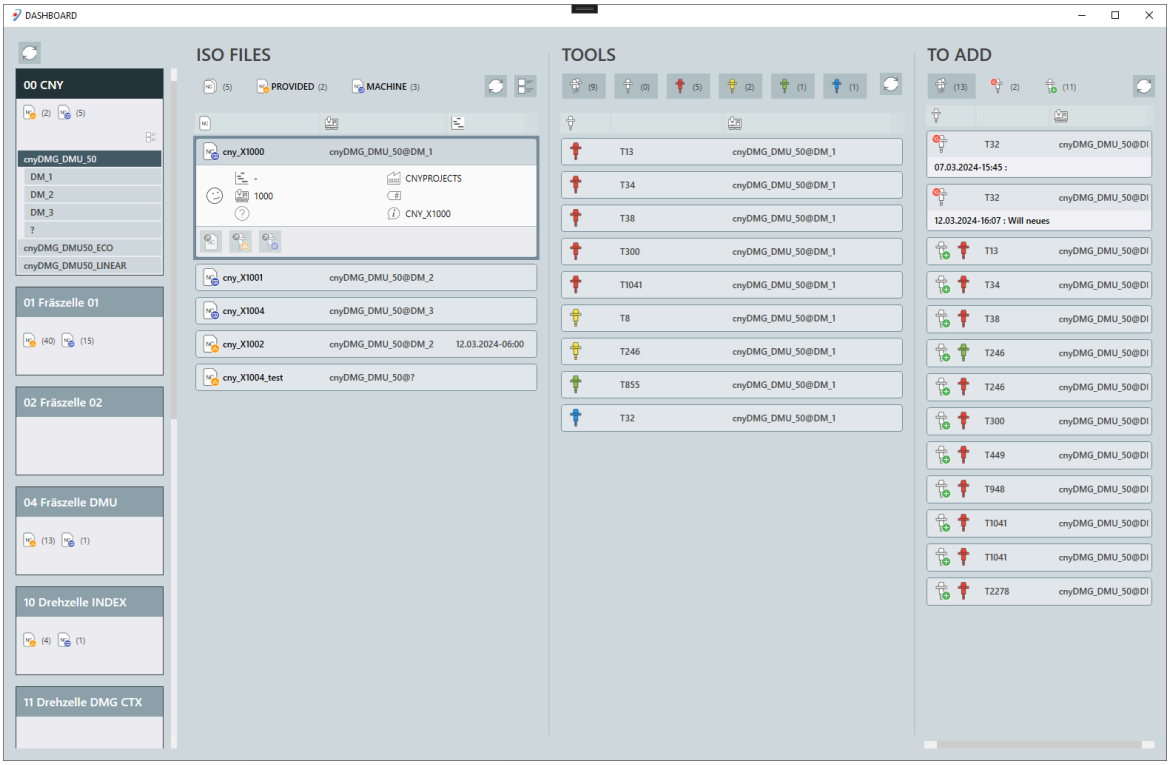
Dashboard

The new dashboard offers an optimized overview of ISO files (available on the CNC and on the machine), as well as tools for machine groups, individual machines and their magazines.

It displays all required tools, either for all ISO files or only for the selected file, specifying their status (on the machine, in the magazine or for assembly). If an ISO file is not assigned to a magazine, you can consult the status of tools in the various magazines to facilitate decision-making.

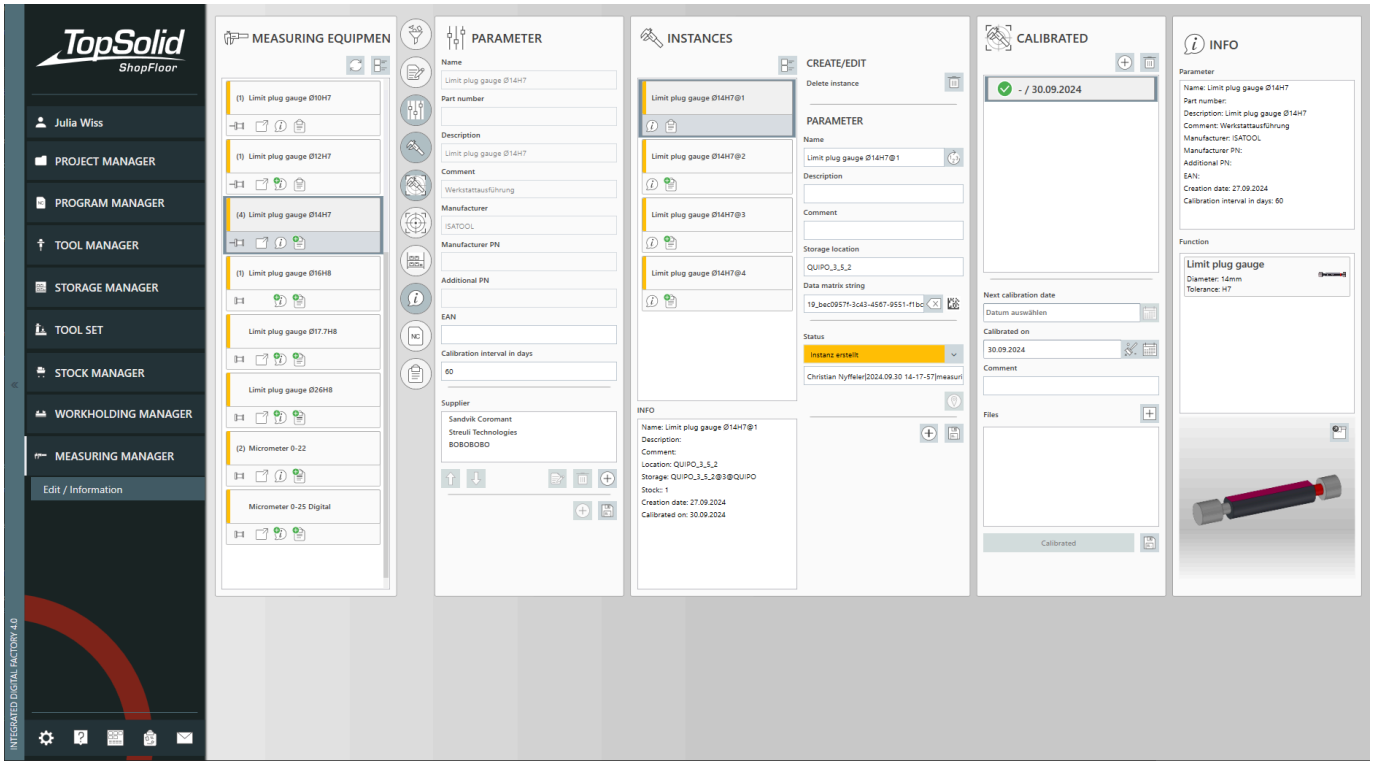
Additionally, the dashboard provides an overview of requested tools to be added. Tools can also be transferred directly from the dashboard to interim storage or assembly.

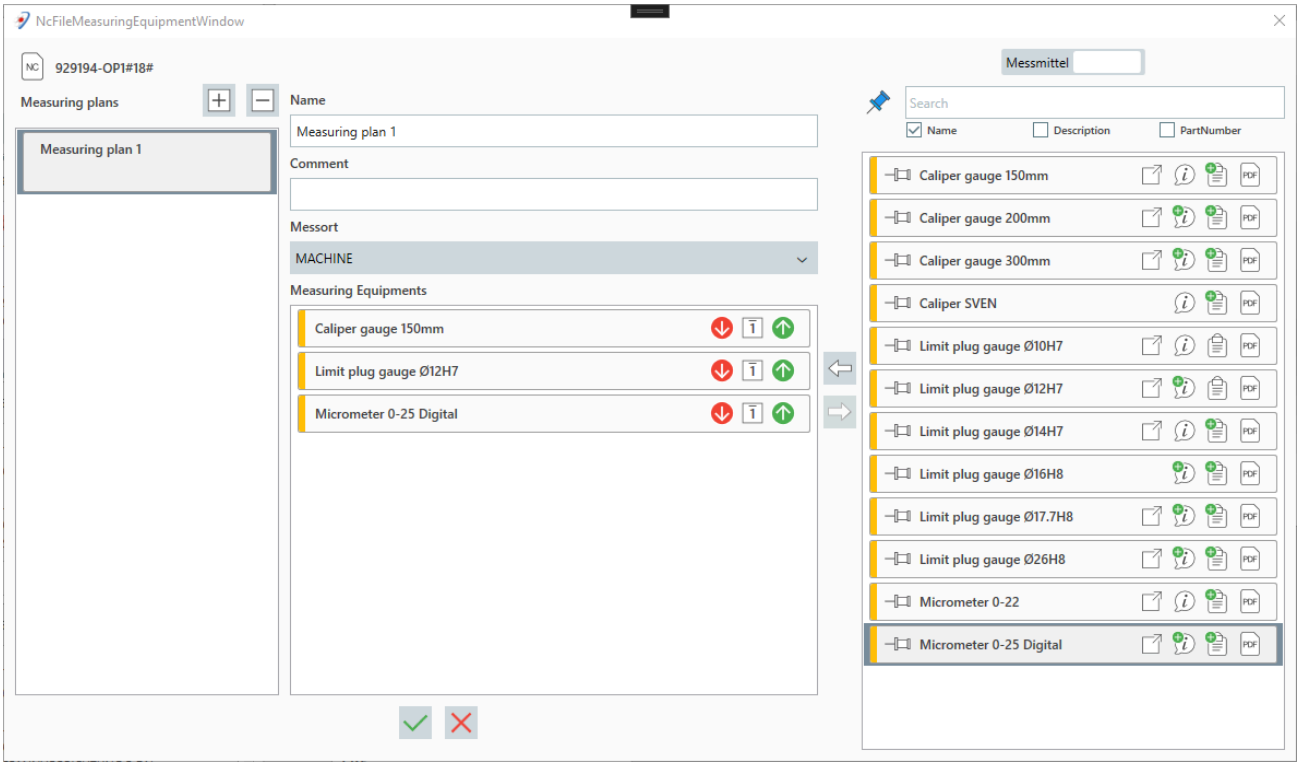




Measuring Manager

The new **Measuring Manager** module provides a central platform for managing measuring equipment. You can store detailed information such as descriptions, storage locations and statuses, as well as calibration data, including due dates for the next required calibration. ISO files can now be linked to the corresponding measurement plans and required equipment.





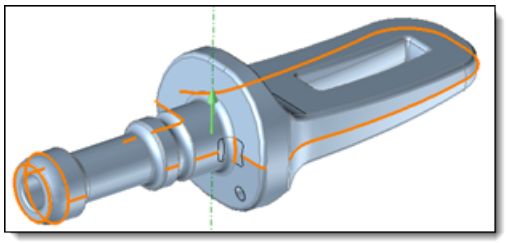
What's New in TopSolid'Tooling 7.19

Explore the latest enhancements in the **Split** (Block split), **Mold**, **Electrode** and **Progress** (Progressive die) applications of **TopSolid 7**!

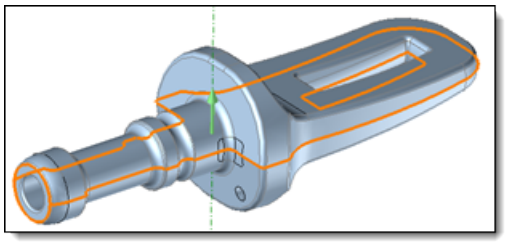
TopSolid'Split (Block split)

Candidate edges

The new **Isoclines** option in the **Candidate Edges** command calculates and imprints silhouette curves only on the visible faces of the part, following the selected molding axis.

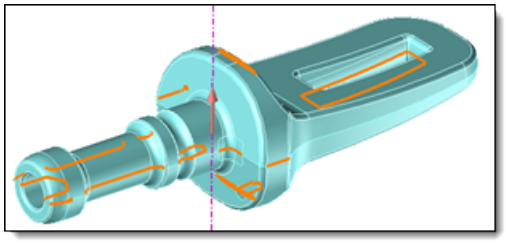


*Example of searching for candidate edges in **Precision** mode. A closed path cannot be created due to the selection of unnecessary edges.*



*The **Isoclines** option automatically imprints the missing silhouette curves along the outer contours, ensuring the creation of a closed path.*

This feature eliminates the need to perform an imprint operation on the part before defining the parting edges. Indeed, isocline imprinting generates numerous curves, many of which are not essential for parting line design. This previously required the manual selection of relevant part faces to optimize the process.

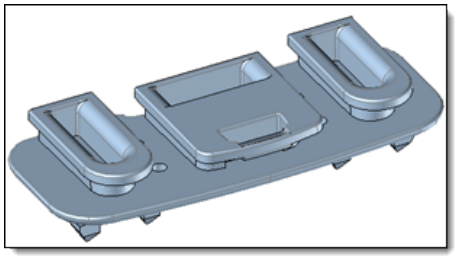


Preview of the silhouette curves suggested by the imprint process for the same part.

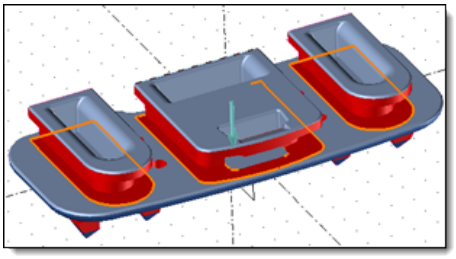
Shadow curves

The new command **3D Sketch** > **Operations** > **Shadow Curves** creates curves from a graphical analysis that visualizes shadow areas on a part in a given direction.

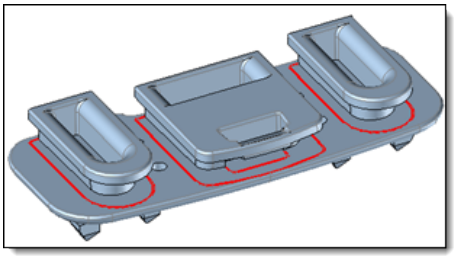
Areas visible in the given direction, those in the opposite direction and undercuts are differentiated by specific colors. The curves delimiting these areas, which do not already correspond to edges on the part, are then generated.



The part to be analyzed has undercut areas in a given direction.



The graphical analysis identifies the "invisible" areas in this direction.

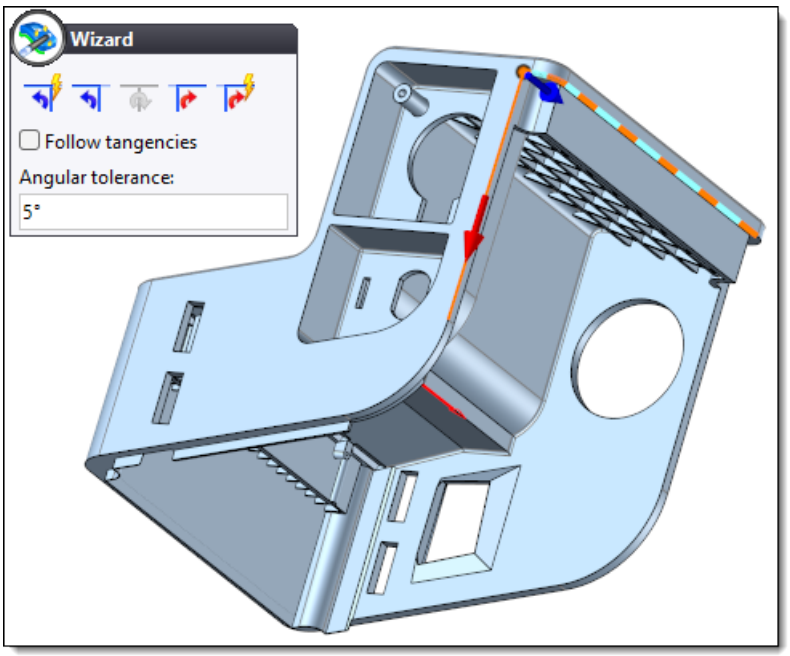


The curves delimiting these areas are then created in a 3D sketch.

Once the analysis is complete, the necessary curves can be imprinted on the part. This feature facilitates, for example, the definition of parting edges for a moving insert in the split.

Parting edge creation wizard

The new **Wizard** command, available during parting edge creation via the special inputs icon , provides an intuitive way to define edge paths. You can select the next edge either graphically using an arrow or via keyboard shortcuts, with the path being progressively built.

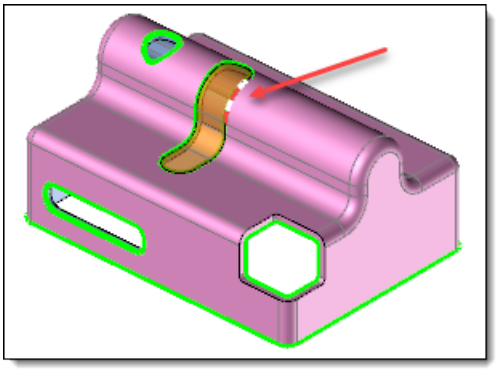


You can also customize keyboard shortcuts for the command's main arrows in the **Parting Edges** section in the document options.

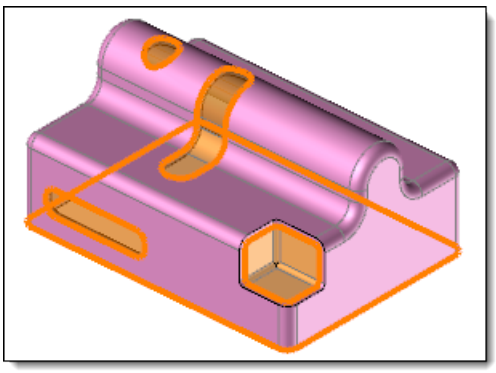
Shut-off surfaces

Enhancements to the **Shut Off Surfaces** command include reorganized dialog box and updated options:

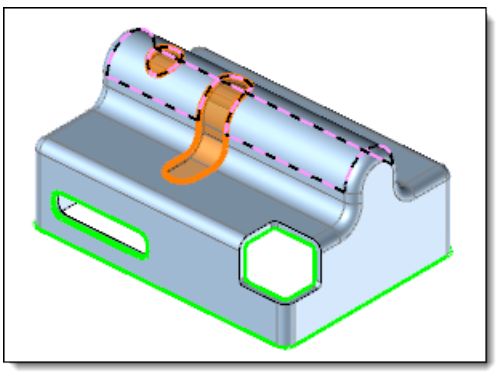
- **Edges** mode: It is no longer necessary to select a reference face. Simply select an edge of a closed loop to create the hole's shut-off surface.



- **Shape** mode: Improved edge loop search excludes the "outer" loop as a hole to be filled.



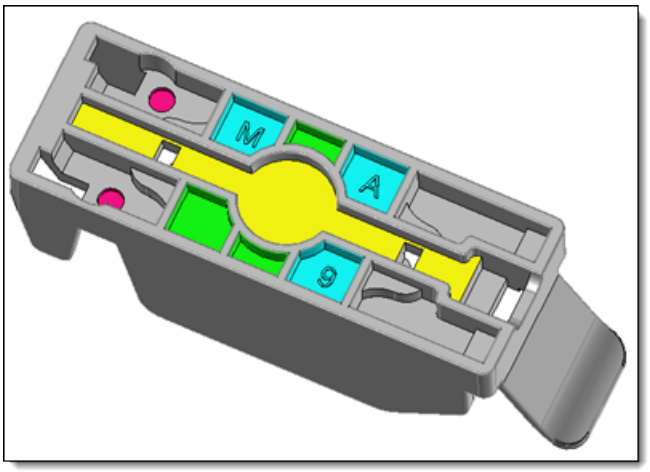
- **Face** mode: Shut-off surfaces connected to a supporting face are now created. You can choose to ignore specific loops or select only the edge loops to be considered.



Split document options

In the **Split Blocks** document options, new settings are available under the **Attributes** > **Molded Set** section to define the color of the shape with shrink generated by the block separation process. You now have the option to choose a global color for the part and/or display or remove the colors of the part faces.

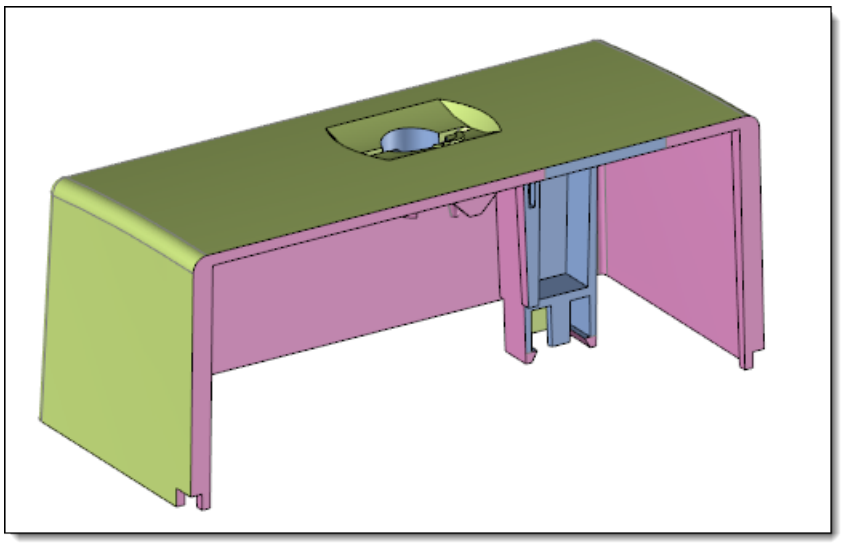
It is recommended to configure these options in a template document to automatically apply them in all future studies.



Export with conversion

You can now export the shape to split from the **Split Blocks** document, while retaining the defined colors of the molded areas. This feature facilitates client validations with clear identification of the insert zones.

To do so, you need to create a representation that includes this shape, and then export it in the **parting** stage.



TopSolid'Mold

Non-synchronized design

The creation of a complex mold study often involves modeling a large number of parts, which can significantly increase modeling and management time when these parts are synchronized in the mold document.

You can now work on a mold in a non-synchronized way with three new part inclusion modes, accessible in the mold document options:

- **Mold Base**: The plates of the standard model are derived and included in the mold without synchronization.
- **Core Cavity Blocks Inclusions** The assembly parts from the block split document are derived and included in the mold without synchronization.
- **Mold Set Inclusions** The parts from the injection or ejection assembly are derived and included in the mold without synchronization.

To avoid synchronizing these parts in the mold, operations must be performed on parts edited in place or via the assembly in non-synchronized mode.

Restoration of graphic selection

Previously, some parts included in the mold could lose their graphic selection, making it impossible to select them when launching a command.

You can now restore the graphic selection of a part by right-clicking on it in the **Parts** folder of the Entities tree and selecting the **Restore Picking** command.

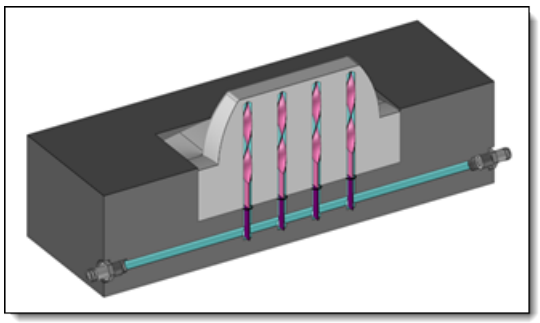
Cooling circuit

Existing sketch

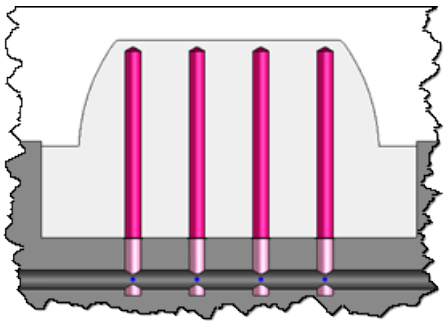
You can now choose to use an existing 2D or 3D sketch when creating a cooling circuit.

Double baffles

Double baffles can now be created in a cooling sketch. This feature enables the definition of two baffle lines and automatically inserts two baffle components when the baffle line passes through two plates.



Two self-locking baffles are automatically added when creating baffle lines.

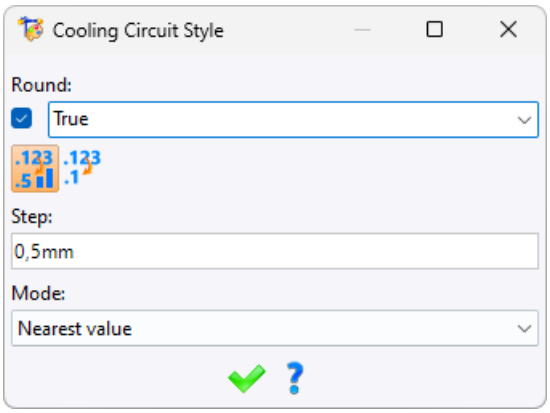


The lengths of the two holes are defined by the cooling style.

Cooling circuit style

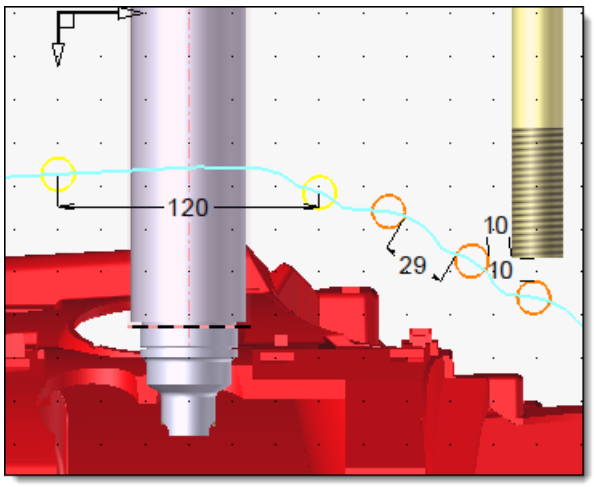
The cooling circuit style now supports defining the two double baffle holes and their associated components.

The **Round** option allows you to adjust the depths of the baffle lines according to a defined step.

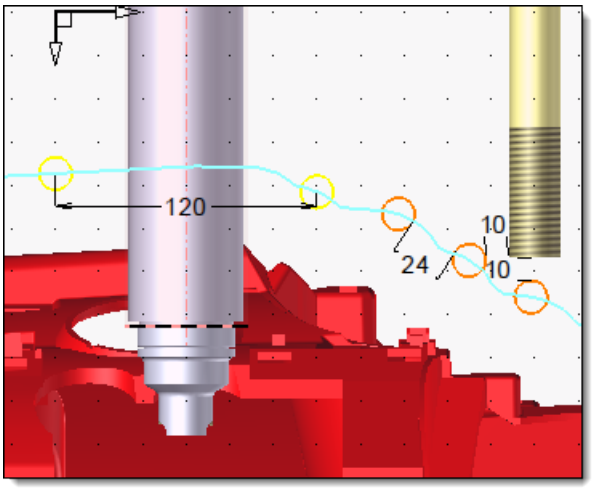


Drilling distribution wizard

The drilling distribution algorithm has been optimized to ensure a more uniform distribution across each interval.



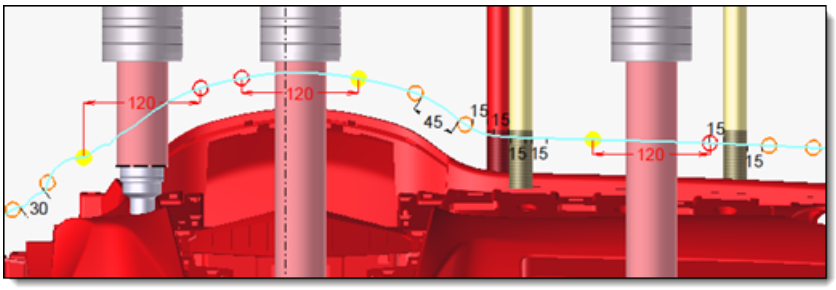
Version 7.18: The interval between the yellow hole and the first orange hole was significantly smaller than the interval between the following 29mm holes.



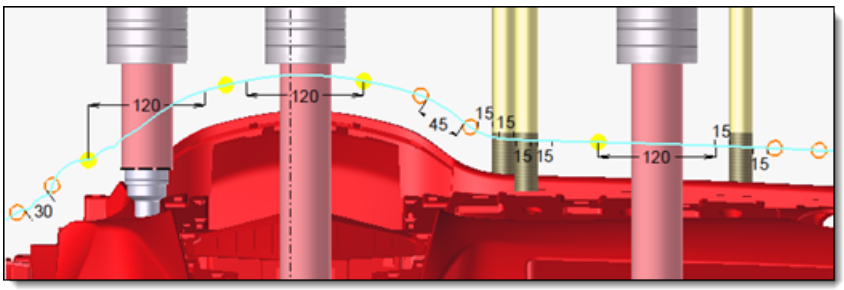
Version 7.19: The holes are now evenly distributed within the interval between the yellow hole and the element to be avoided.

For cases where hole distribution cannot meet all input data, several management modes for cooling parts are now available:

- **Strict**: This mode displays errors and prevents validation of the dialog box until they are corrected.



- **Allow invalid drillings**: This mode corresponds to the non-strict mode of version 7.18. Errors are not displayed, validation of the dialog box is allowed, and the invalid holes are still created.
- **Delete invalid drillings**: Errors are not displayed, validation is allowed, but invalid holes are not created.
- **Solve**: Errors are automatically corrected.



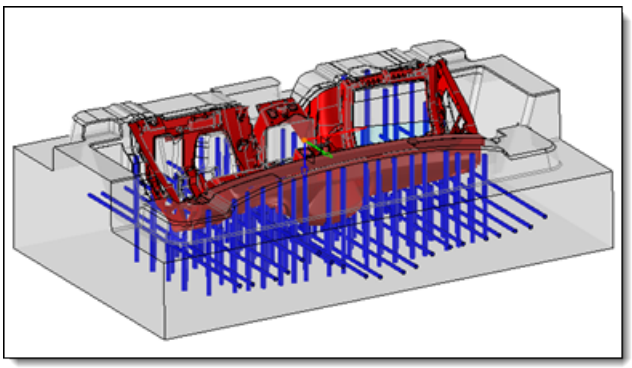
In the example above, a hole is placed in the middle of the two invalid holes on the left (120mm), and no hole is placed on the right (200mm).

Baffle line distribution wizard

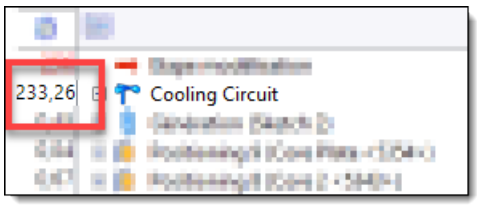
The baffle line distribution algorithm benefits from the same improvements as the drilling distribution.

Enhanced performance

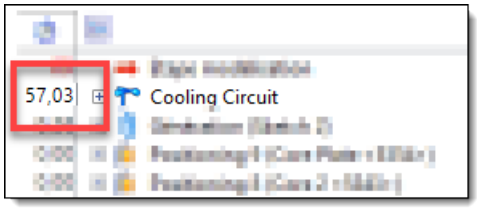
The previously time-consuming calculation of cooling baffle depths over very large core-cavity block has been considerably optimized.



Example of a 75% reduction in calculation time.



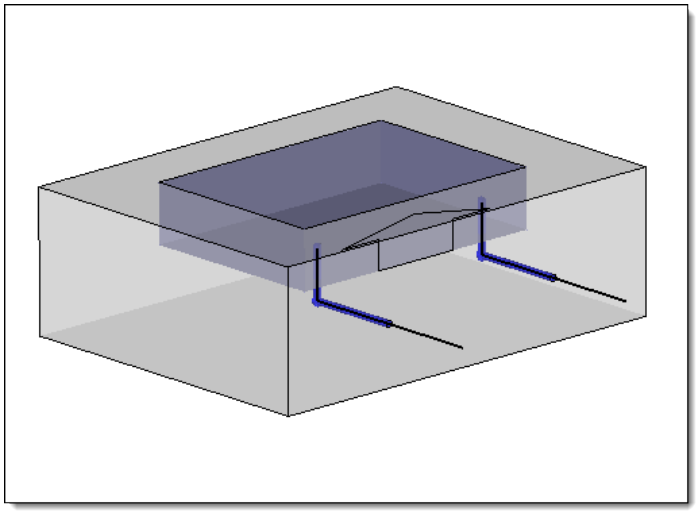
Calculation time before enhancement.



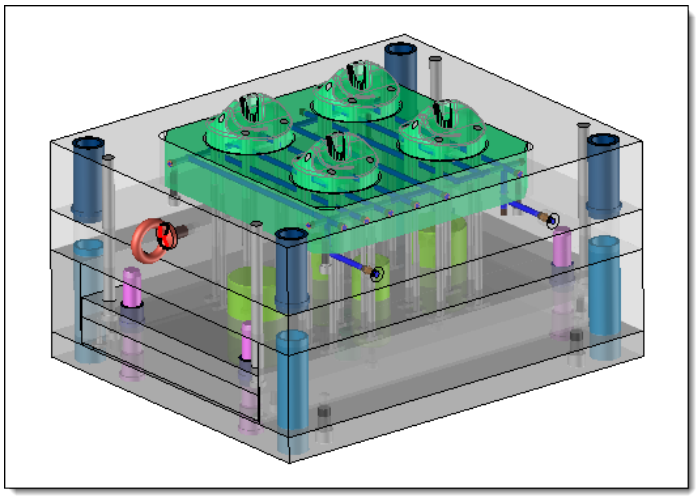
Calculation time after enhancement.

Method

You can now define a method to automatically generate a cooling circuit. This feature allows you to quickly create standard circuit parts, which are commonly shared across different molds, and then easily modify them to adapt the final circuit to your specific needs.



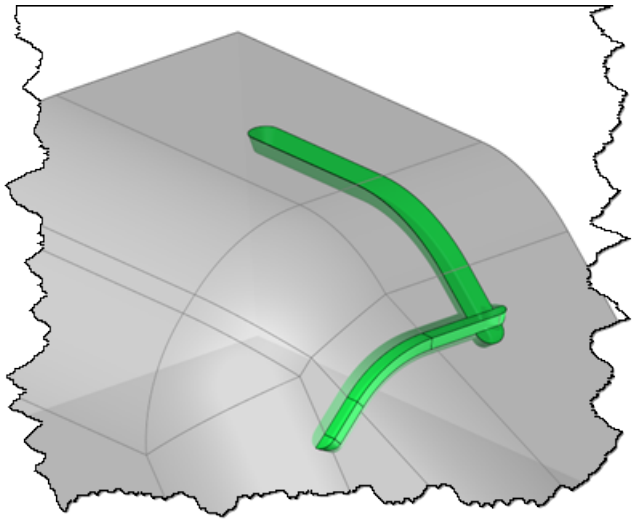
The method automatically generates the cooling circuit between two parts.



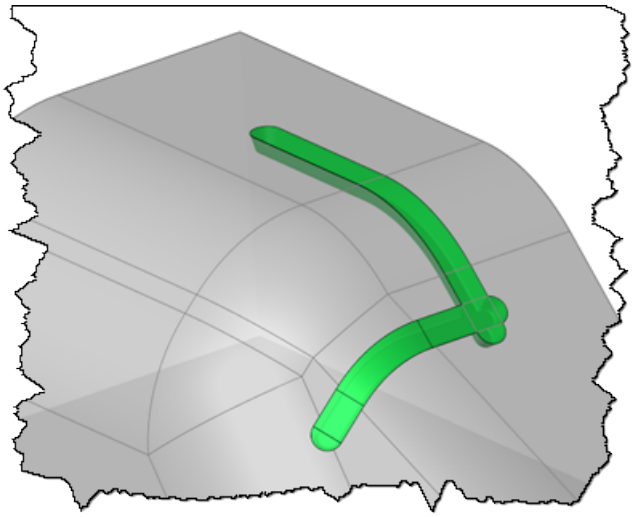
Once the circuit is created, it can be adjusted to add any missing drillings.

Runner circuit

The **Runner Circuit** command features a new projection mode, **Normal to shape**, which allows the runner section to be oriented perpendicular to the shape.



Previously, the sections were oriented by the mold opening direction, sometimes limiting the precise positioning of the circuit.

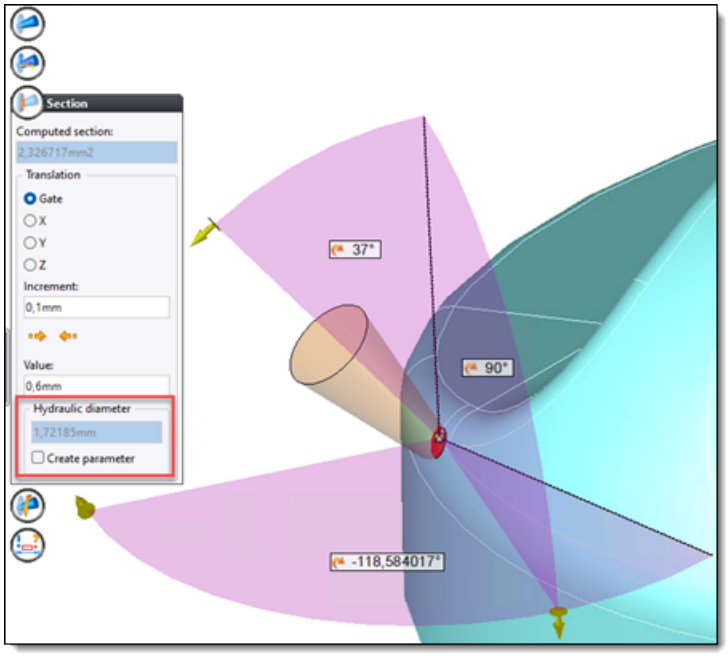


*With the **Normal to shape** orientation mode, the circuit can now follow all faces of a shape, regardless of their normals.*

Gate

Hydraulic diameter

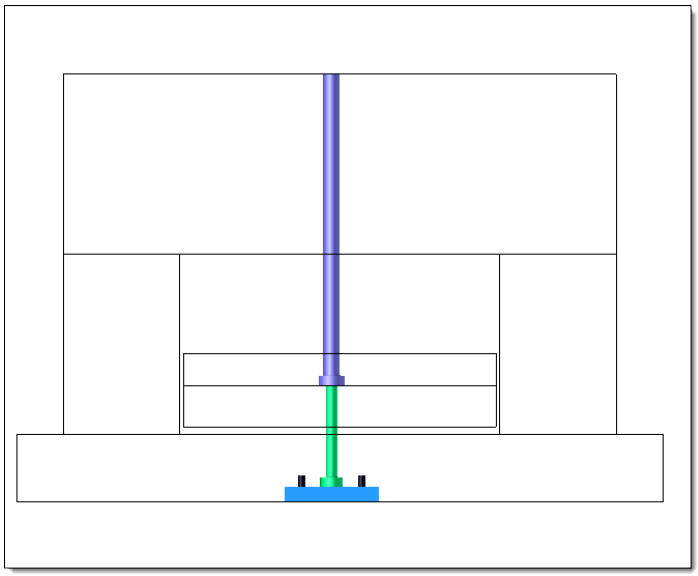
The hydraulic diameter is now automatically calculated at the injection point when including a gate. This diameter value is displayed in the command dialog box, and it can be used as a parameter for later use.



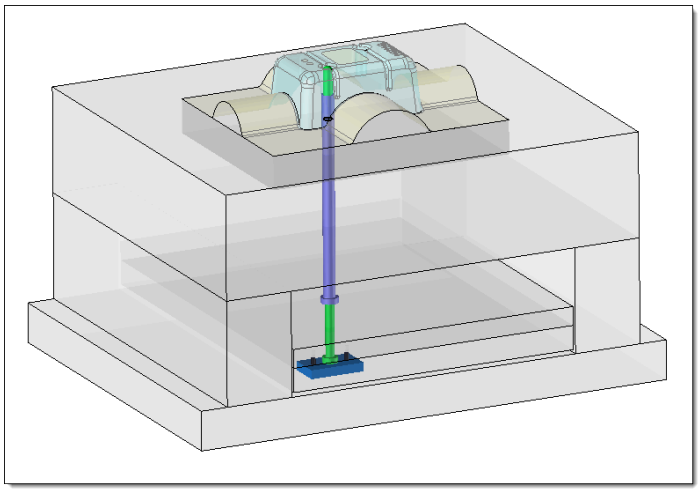
Pin

Method

You can now create a method for including "equipped" pins in a single operation. This feature allows you to define common pin combinations based on your needs, and standardize them for future use.



Example of a method including a cylindrical pin and a tubular pin, with a precision plate.

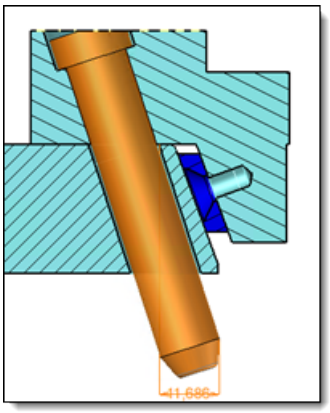


All components are automatically included and adjusted in the mold in a single operation.

Angle pin

Component

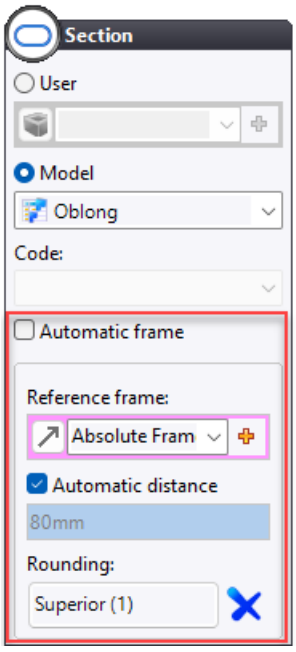
The **Angle Pin** command includes a new mode, **Angle pin retainer**, which simplifies the positioning and dimensioning of angle pins. The angle pin is automatically positioned and dimensioned based on the retainer diameter, and its head is trimmed by the retainer. Only the length of the angle pin can be adjusted to control the desired stroke value.



Angle pin pocket

Angle pin pocket positioning

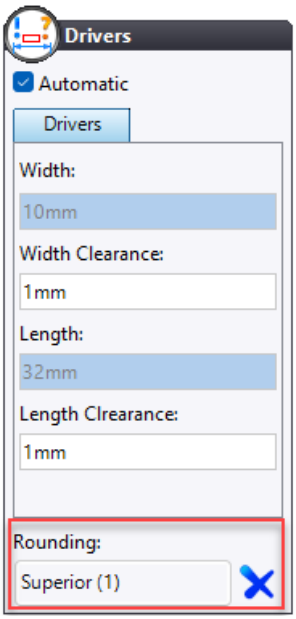
Previously, the angle pin pocket section was automatically centered around the selected angle pin. Now, you can define the position of the sketch center from a frame, such as the mold origin, and round off the distance between this frame and the sketch center.



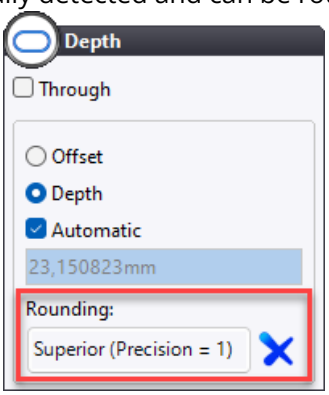
Angle pin pocket dimensions

You can now create angle pin pockets using rounded dimensions, thanks to an option available in both of the sections below. To activate this feature, check the **Automatic** box.

- **Drivers**: The minimum driver values are automatically associated with the sketch drivers, with the option to round them off.

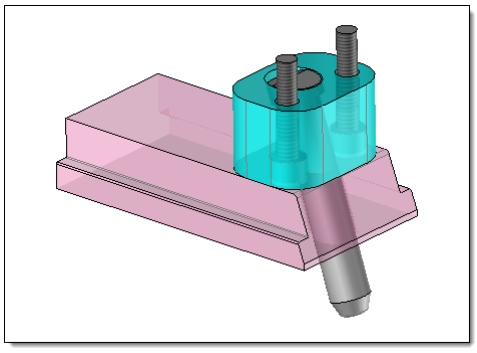


- **Depth**: The minimum depth is automatically detected and can be rounded off if required.



Method

You can now create a method to include angle pins and their pockets. This feature enables the design of standardized movement assemblies, including various components such as slides or angle pin retainers. The associated processes and the angle pin pocket are also automatically applied when using the method, optimizing the design process.



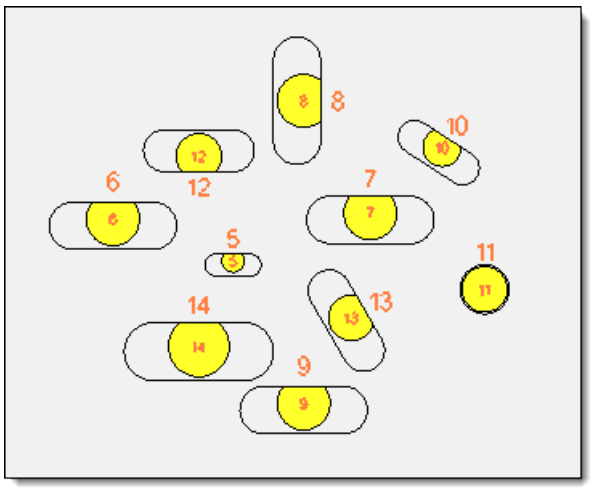
Marking

The **Pins Marking** and **Drillings Marking** commands have been optimized to improve the legibility of markings on 3D parts.

A new advanced option, **Color**, now lets you assign a color to the edges generated by the marking operation or to the sketch of the lightened marking, making them easier to identify.

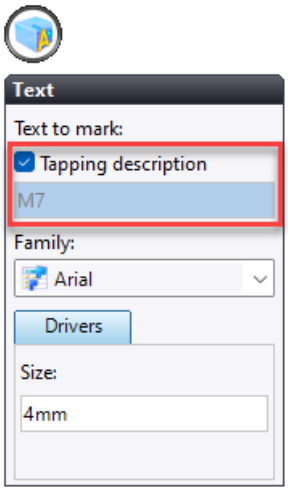
Pin marking

A new advanced option, **Automatic positioning**, automatically fills in the radius and angle values required to position the text on the plate. These values are automatically calculated based on the selected pin diameter and its keying orientation.



Drilling marking

A new option allows you to mark the tapping description (e.g., M6, G1/4, etc.) when the selected hole is tapped. This marking is associative with the tapping performed.



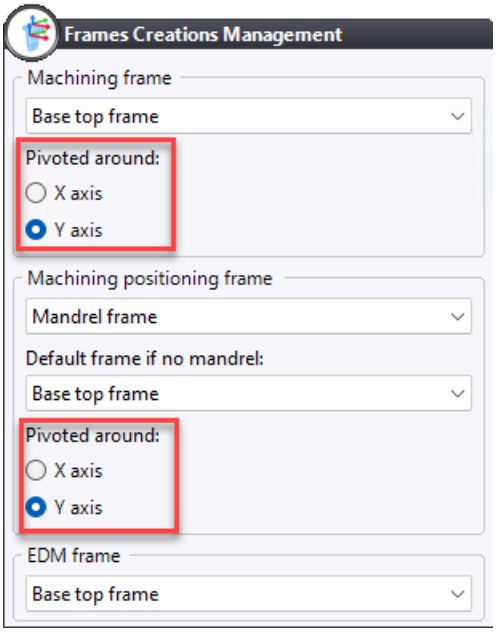
In addition, the advanced **Automatic positioning** option automatically fills in the radius value for positioning the text on the plate based on the selected hole diameter.

TopSolid'Electrode

Electrode frames

Previously, the machining and machining positioning frames, automatically defined during electrode creation, were rotated 180° around the X axis of the electrode base frame.

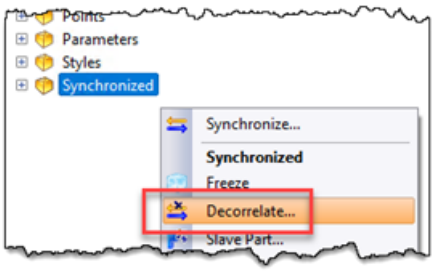
You can now choose to rotate the base frame around its X or Y axis.



Decorrelation

The **Decorrelate** command, which was previously only available in Assembly or Mold documents, can now be used in an **Electrodes** document.

This allows you to decorrelate synchronized parts directly from the **Synchronized** folder in the Entities tree. In addition, operations created in the **Electrodes** document are now rebuilt in part documents, without any link to the **Electrodes** document.



TopSolid'Progress (Progressive die)

Restoration of graphic selection

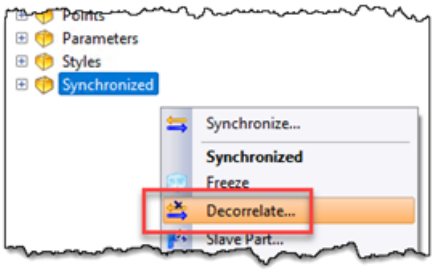
Previously, some parts included in the progressive die could lose their graphic selection, making it impossible to select them when launching a command.

You can now restore the graphic selection of a part by right-clicking on it in the **Parts** folder of the Entities tree and selecting the **Restore Picking** command.

Decorrelation

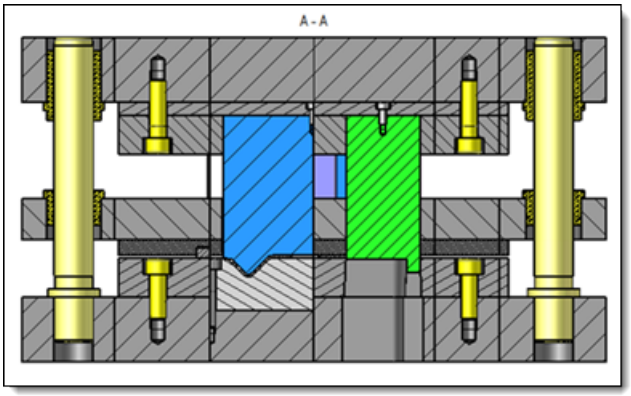
The **Decorrelate** command, which was previously only available in Assembly or Mold documents, can now be used in a **Progressive Die** document.

This allows you to decorrelate synchronized parts directly from the **Synchronized** folder in the Entities tree. In addition, operations created in the **Progressive Die** document are now rebuilt in part documents, without any link to the **Progressive Die** document.

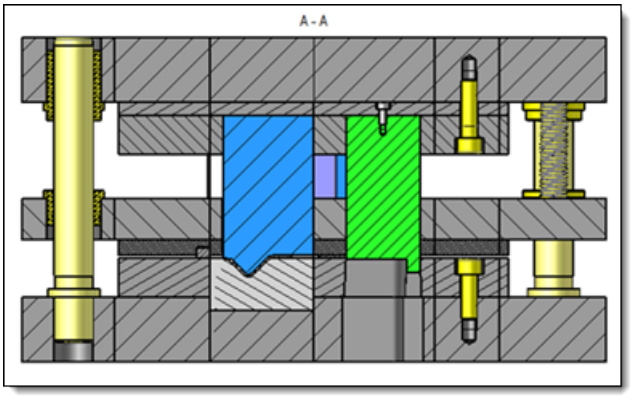


Hiding components

A new **Hide Components** command, available in a drafting document, allows you to hide a component and its associated process on a projected view. Additionally, missing hatching is automatically rebuilt to ensure correct representation.



In the example above, several components are symmetrically placed.



The section view can be lightened by hiding these components and their process.

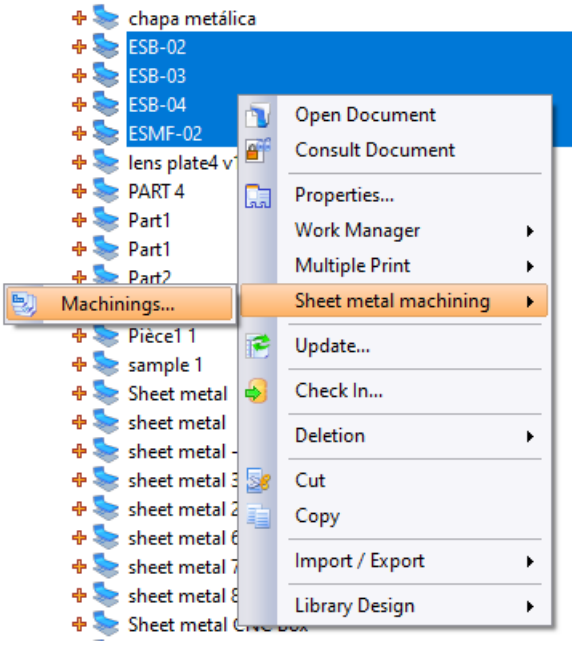
What's New in TopSolid'Cut 7.19

Explore the latest innovations in **TopSolid'Cut**, **TopSolid 7**'s cutting software for the sheet metal industry!

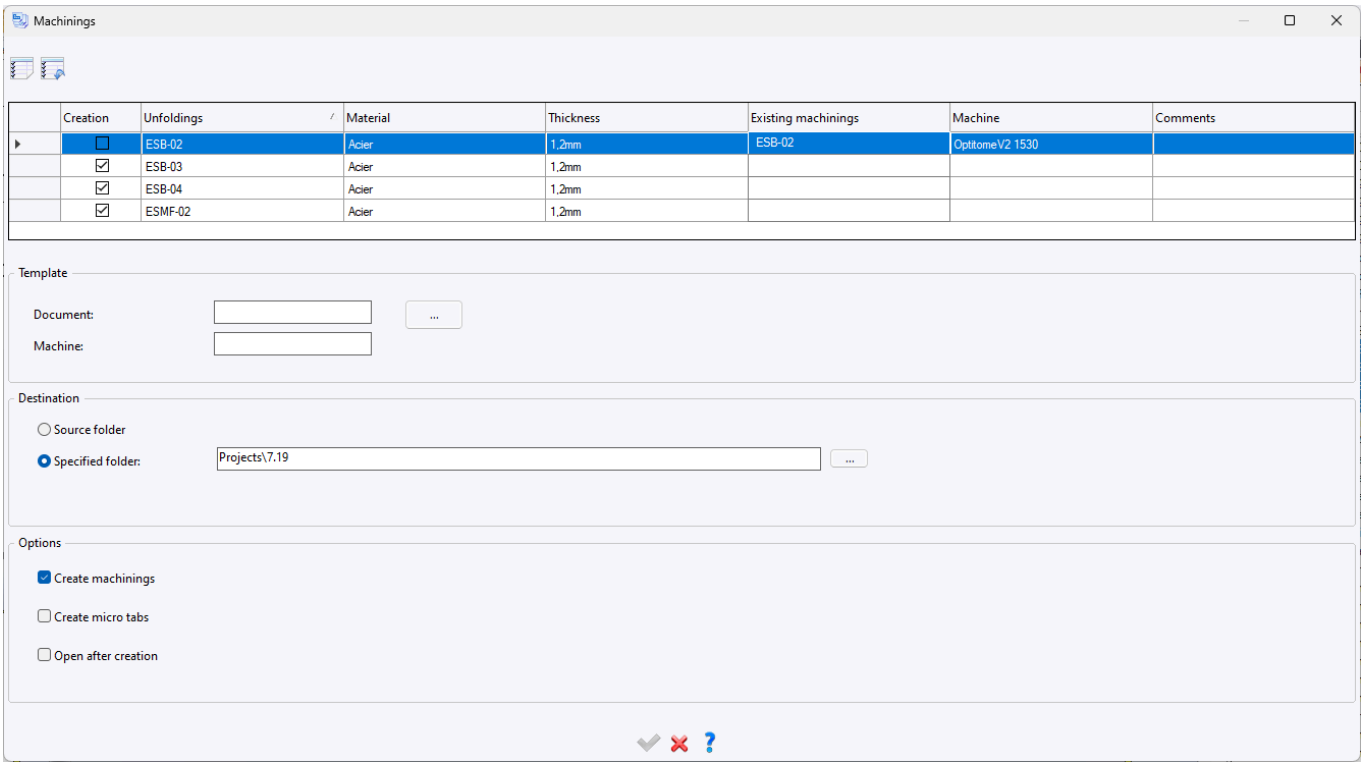
Cutting

Multiple sheet metal machining

A new **Machinings** contextual command is available in the project tree, enabling quick creation of multiple sheet metal machining operations.



This command allows you to select only unfolding documents, and works in a similar way to sheet metal machining processing. Simply configure the dialog box by selecting the desired template and confirm.

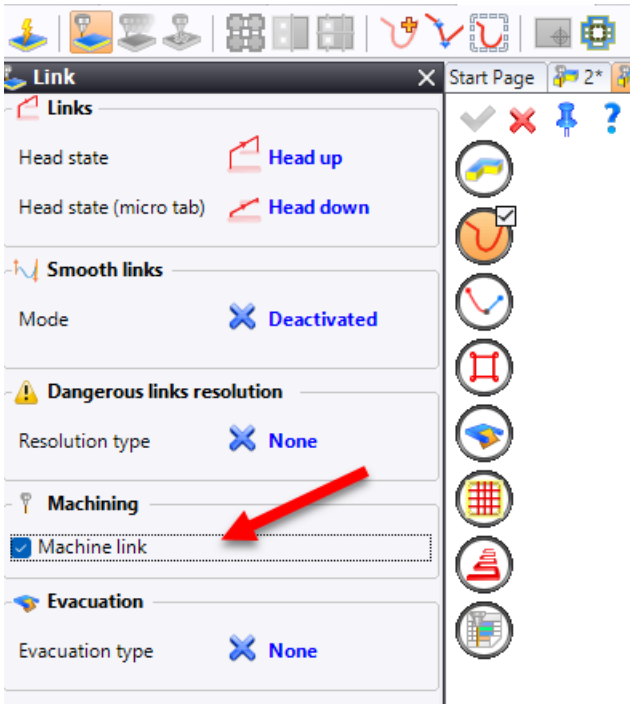


In the example above, three sheet metal machining operations will be created as machining already exists for the first unfolded part.

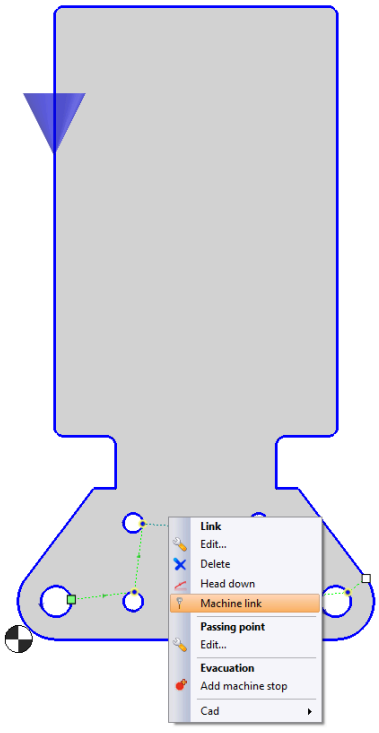
Link machining

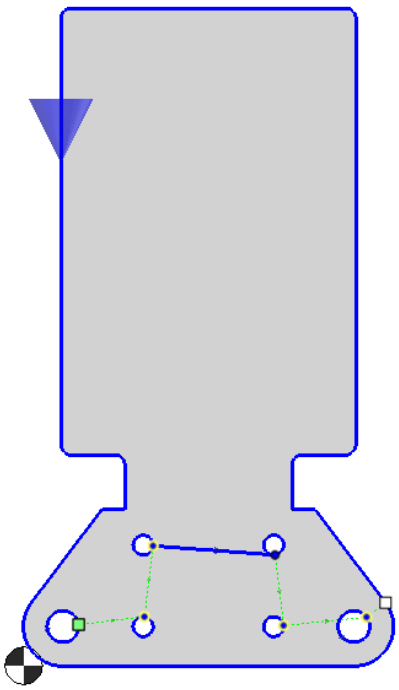
It is now possible to machine a link. This option is accessible from the **Link** tab in a cutting operation and is compatible with the following operations:

- **Cutting**
- **Center punching**
- **Waste cutting**
- **Skeleton cutting**
- **Common cutting**



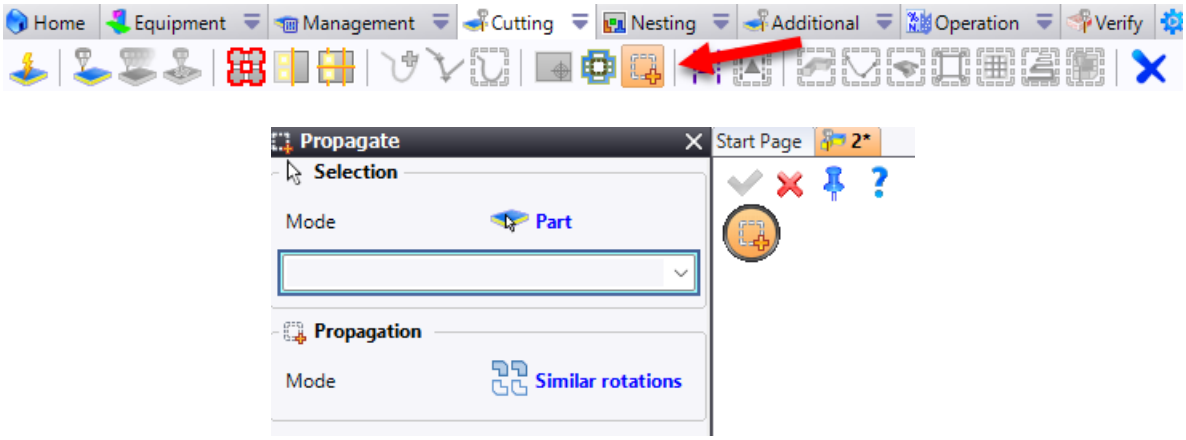
This feature can also be selected by right-clicking on a compatible link.





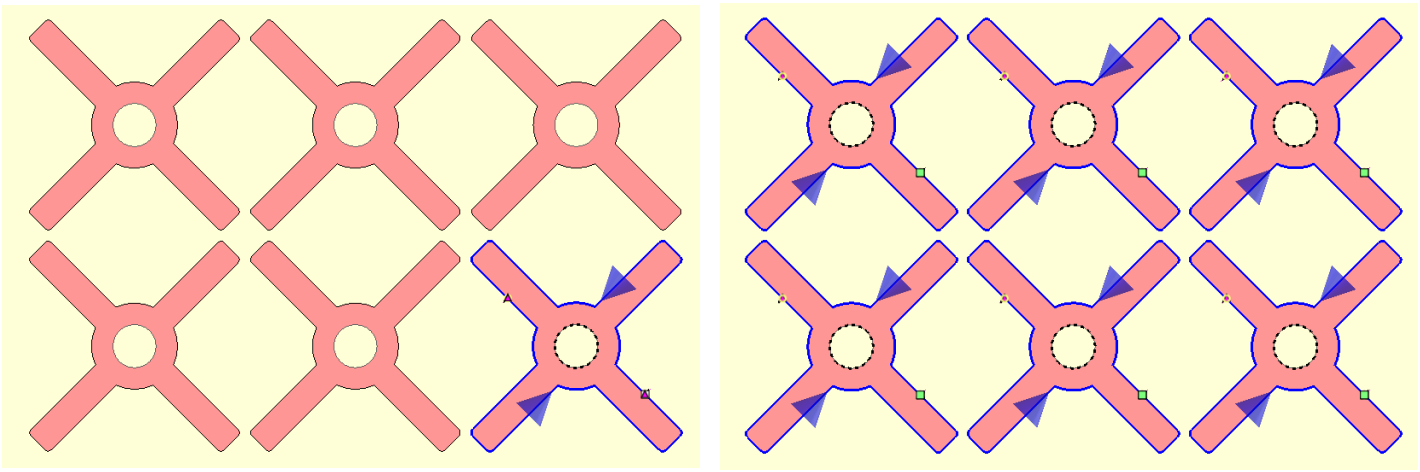
Machining propagation

The **Cutting** tab features a new **Propagate** command, which lets you select a part, contour or machining operation, then propagate it to all similar parts.



Supported machining types include:

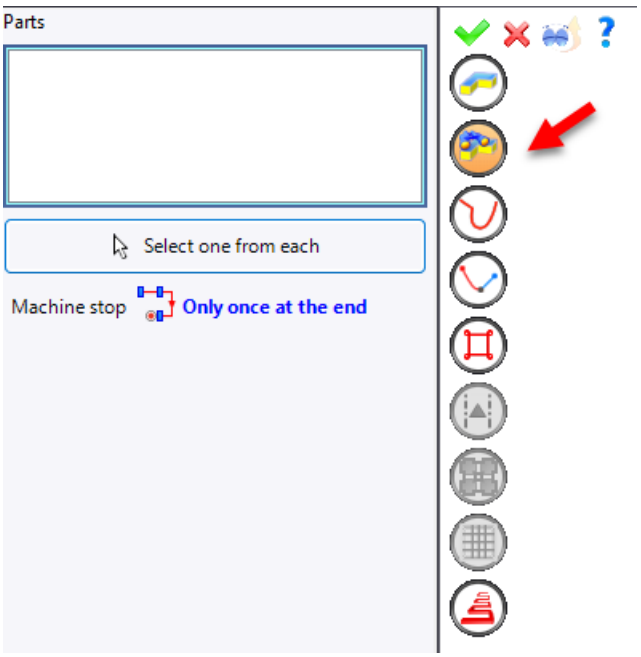
- **Cutting**
- **Center punching**
- **On the fly cutting**
- **Micro tabs**
- **Geometry to skip**



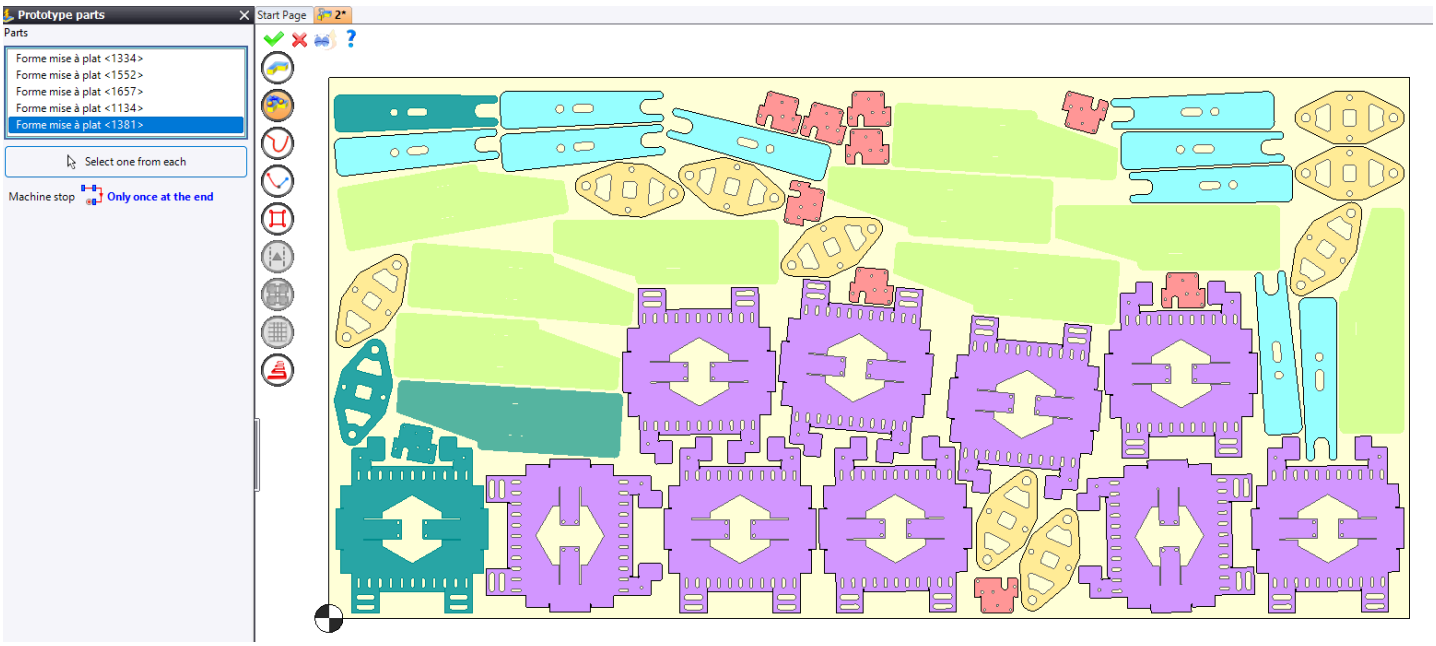
In this example, the machining operations have been propagated from the bottom right-hand part to all identical parts.

Prototype parts

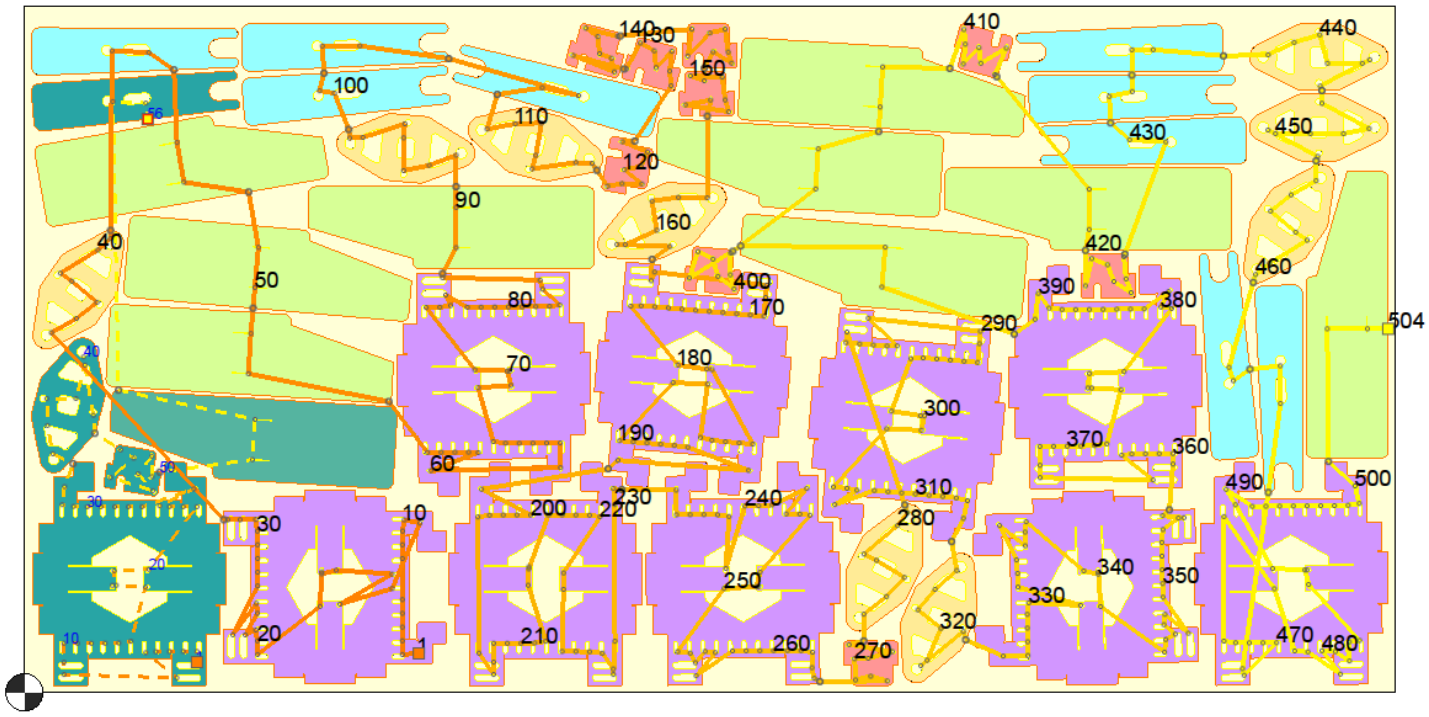
A new **Prototype parts** tab is available in the **Automatic Machinings** command, allowing you to select parts to be excluded from the overall machining path and machined first.



Parts can be selected individually or via the **Select one from each** button, which automatically picks one part of each type. A stop is applied either after machining each prototype part or upon completion of all prototype parts.

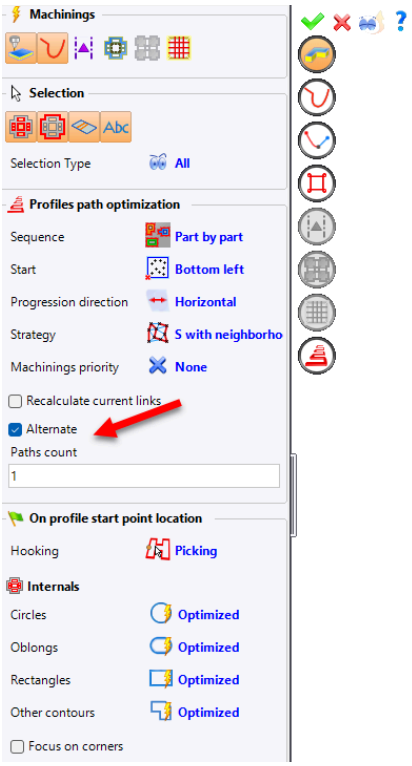


Additionally, prototype part tool paths are previewed separately from the main tool path.

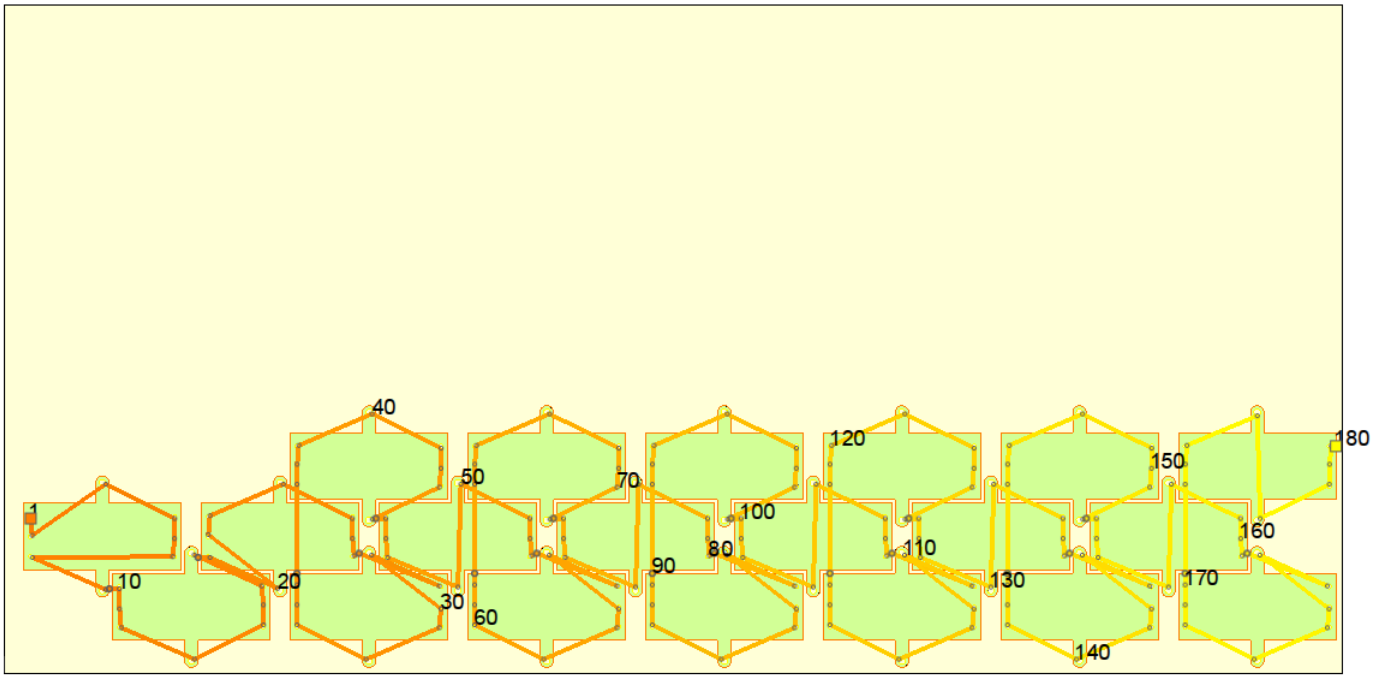


Heat management

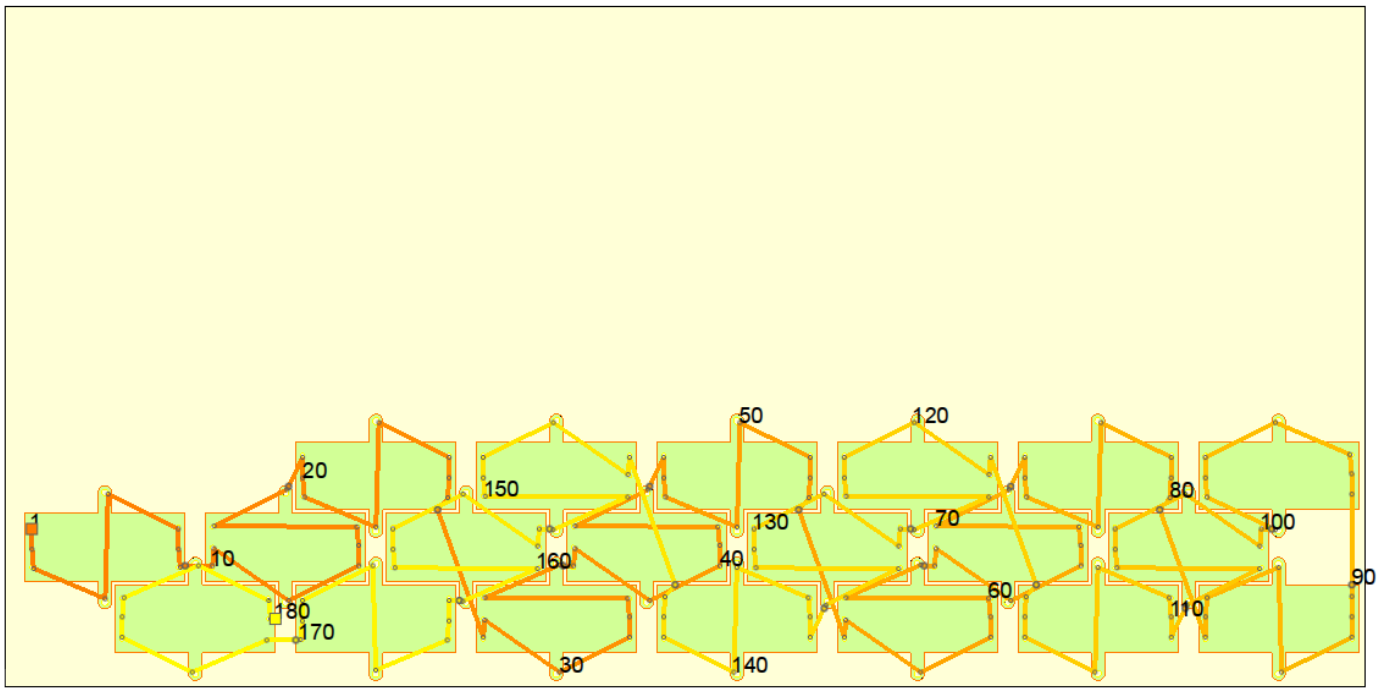
The **Automatic Machinings** command features a new **Alternate** option, allowing you to define the number of passes to reduce heat concentration on a specific sheet metal area.



The tool path alternates between machining passes and skipping an equal number of operations, as specified, to prevent overheating the same zone.



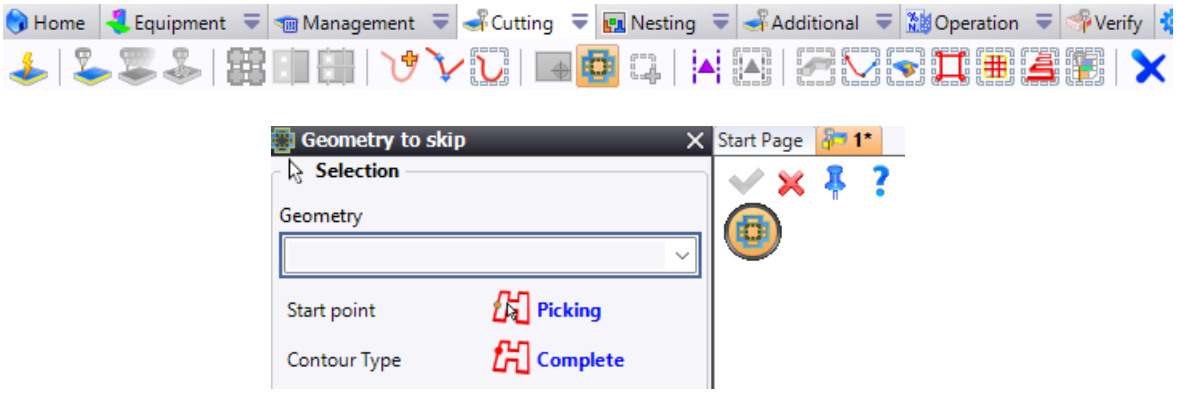
Example with 1 pass.



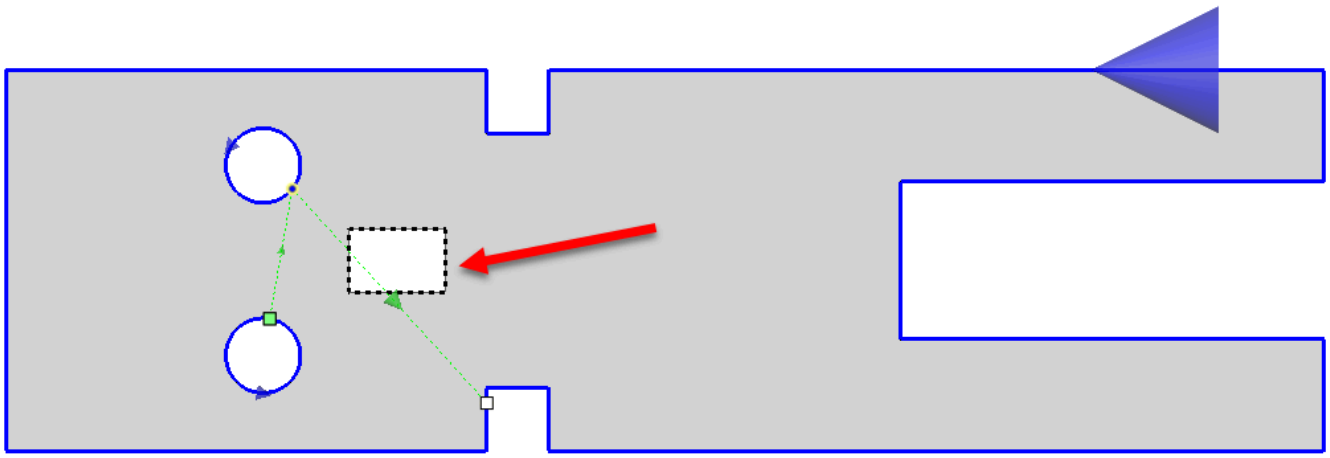
Example with 2 passes.

Ignored geometry

The new **Geometry to Skip** command in the **Cutting** tab lets you exclude a contour or a portion of a contour from machining.

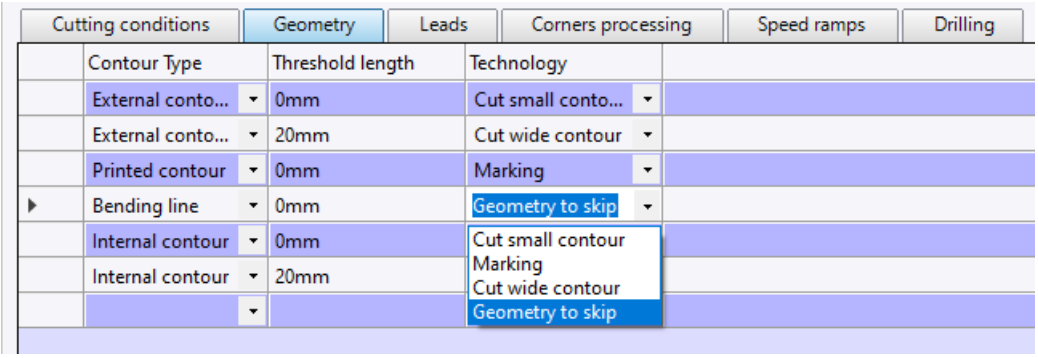


By default, ignored geometry is represented by black dotted lines, but the color can be customized via **Tools** > **Options** > **CAM Options** > **Sheet Metal Cam Options** > **Machining Attributes**.



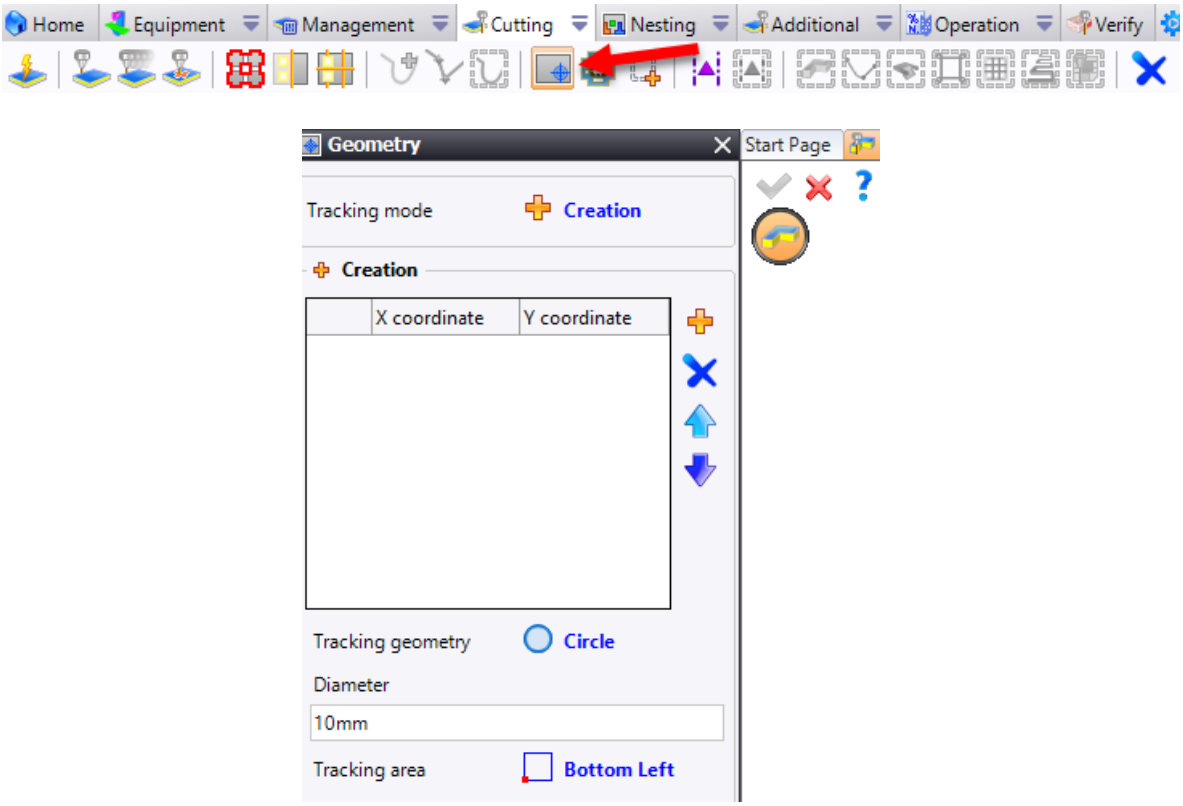
This option is also available in automatic machining, although it is not possible to simultaneously generate machining operations and geometries to be ignored.

You can set the geometries to be ignored in the **Cut Technology** document, under the **Geometry** tab, by filling in the **Technology** column.

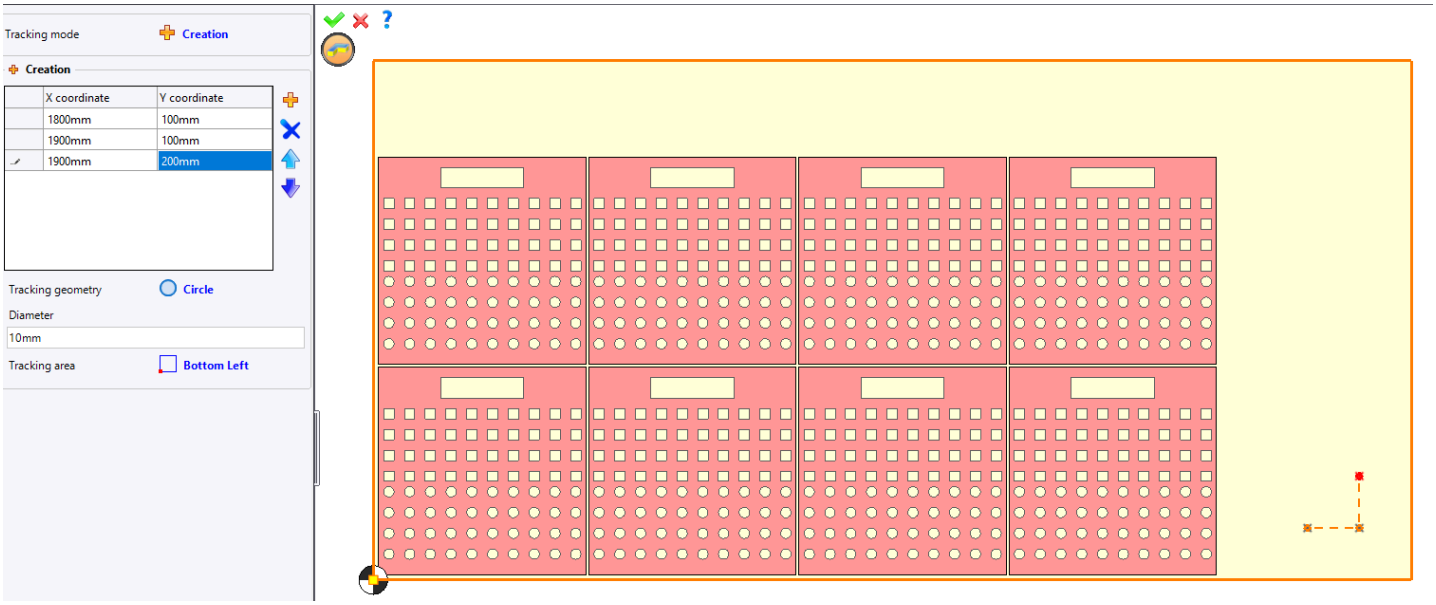


Optical tracking

The new **Optical Tracking** command in the **Cutting** tab enables compatible machines to verify sheet squareness by analyzing specific geometries before machining.



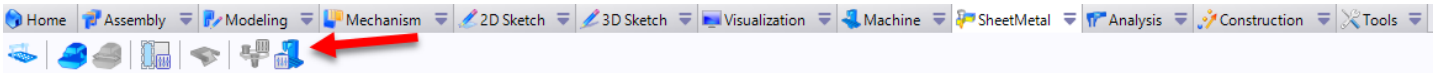
You can use existing geometries or create new ones as needed.

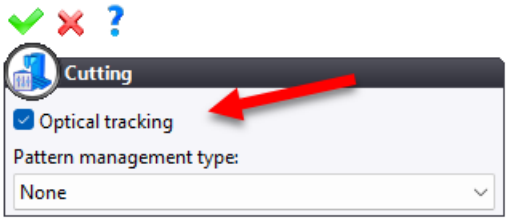


To use this feature, the following conditions must be met:

- The document must be in **For ISO code generation** mode.
- The selected machine must support optical tracking.
- A sheet metal format must be defined.

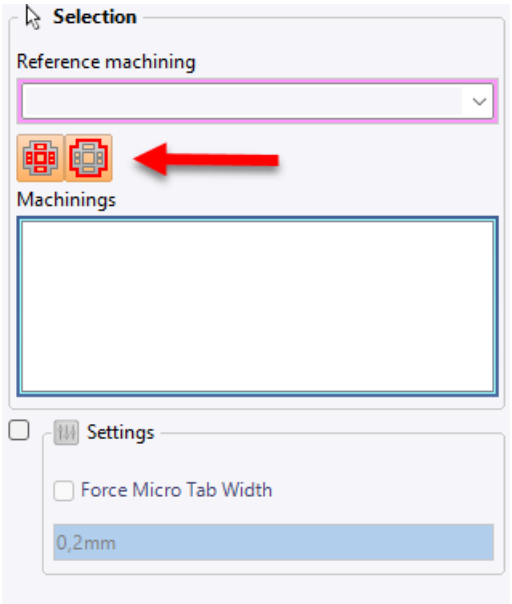
The **Optical tracking** option can be activated in the machine document via the **Machine Setting** command in the **Sheet Metal** tab.





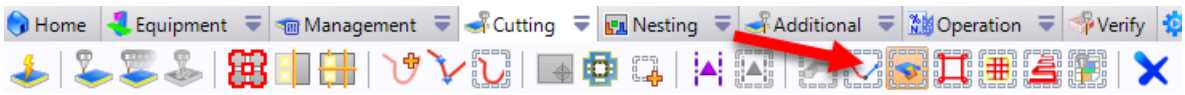
Multiple edit filter

A new filter is now available in multiple edit commands, allowing only internal or external machining operations to be selected.

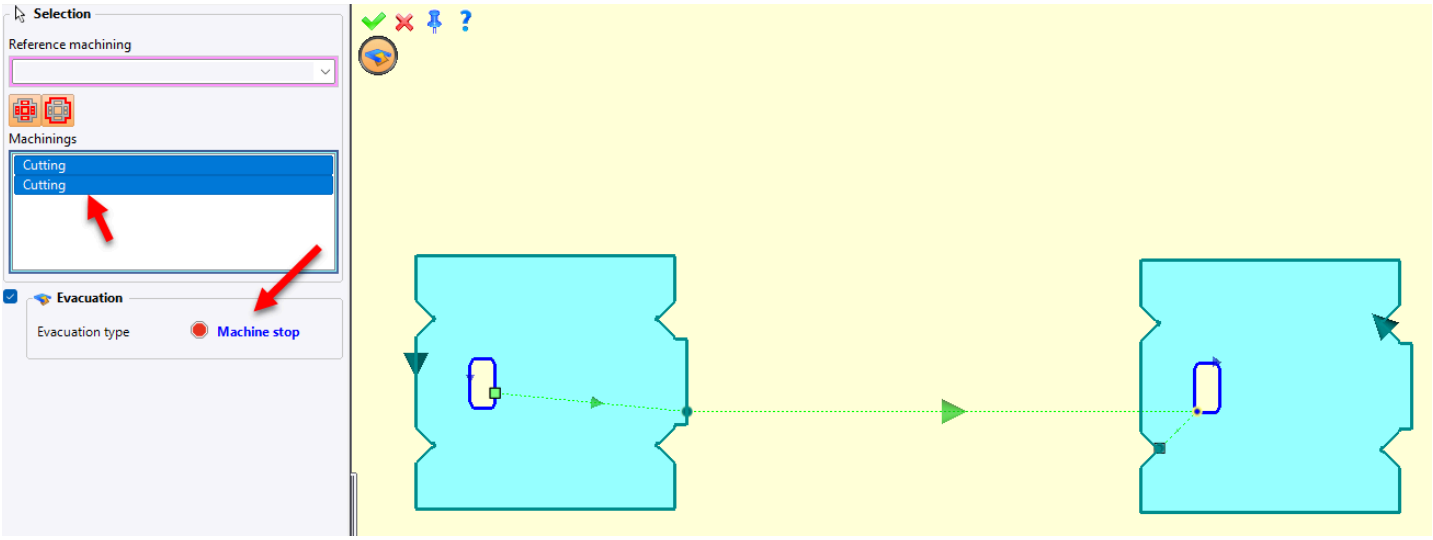


Evacuation editing

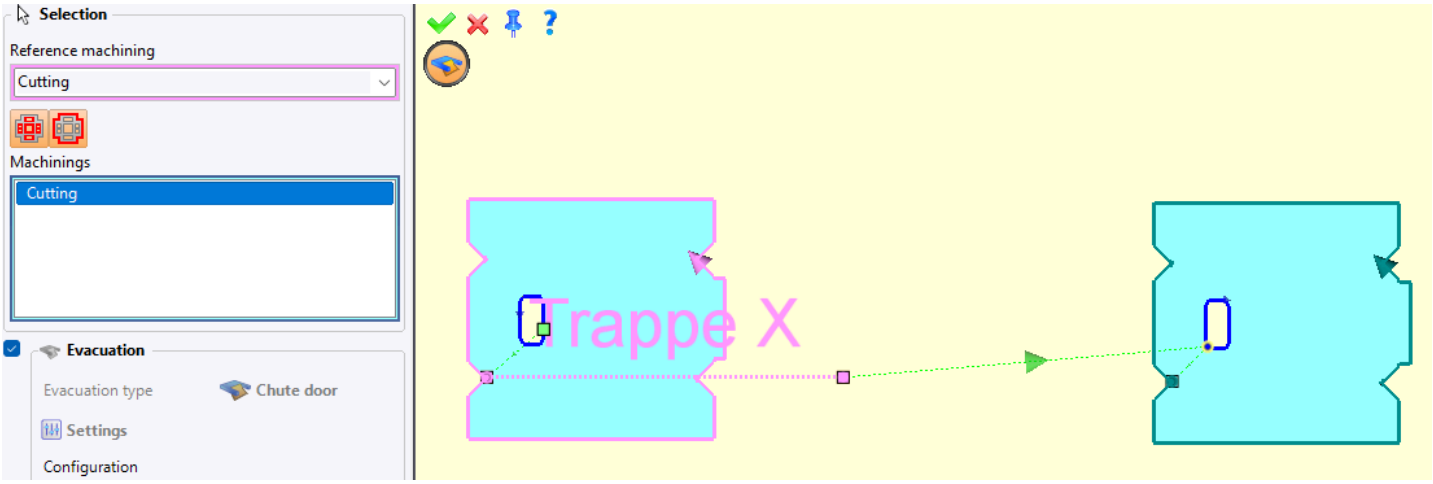
The new **Edit Evacuations** command in the **Cutting** tab provides several practical options:

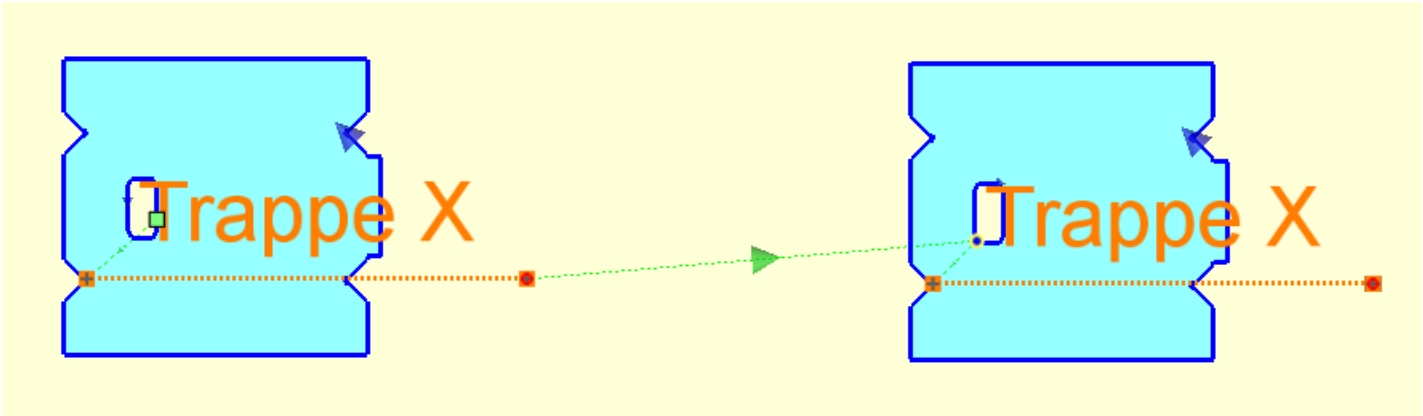


- Add a machine stop to all selected machined parts.



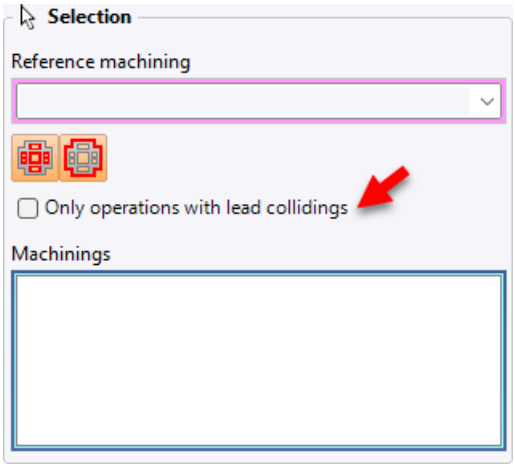
- Apply an existing chute door from one cut to another.





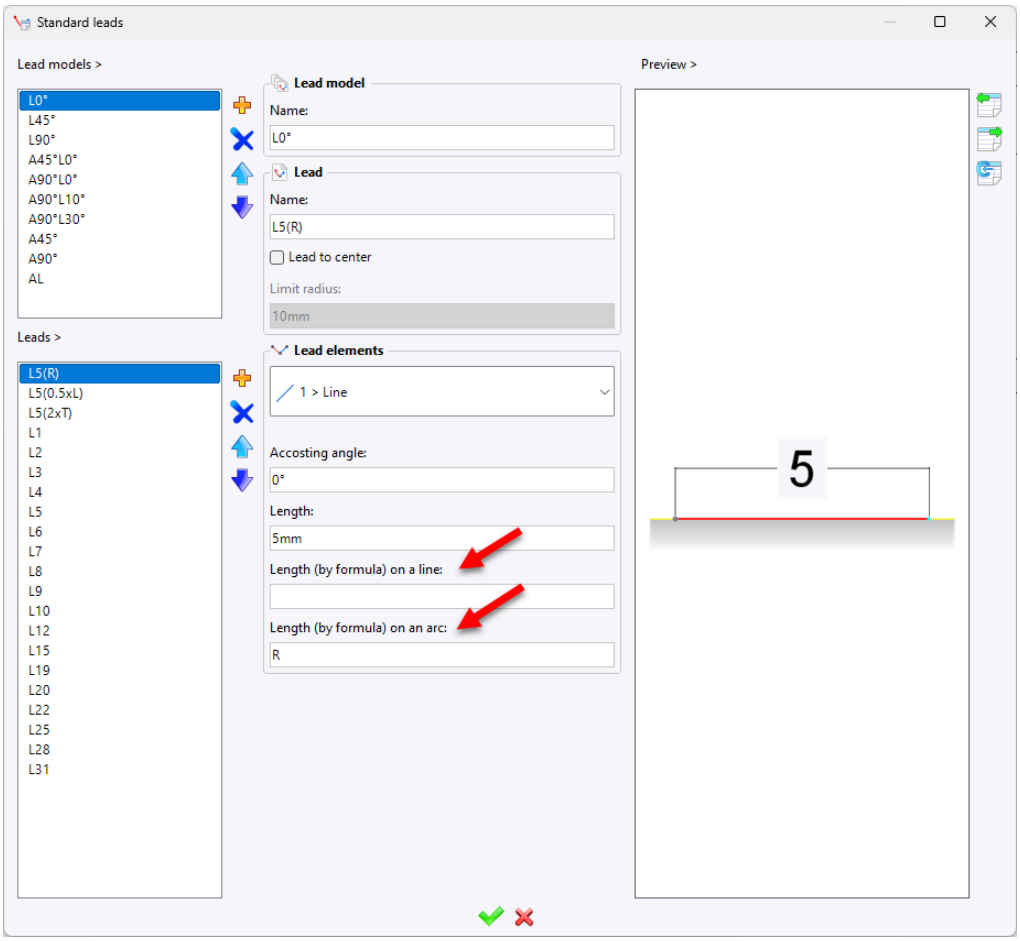
Lead editing

The **Edit Leads** command now includes an option to select only operations with collisions.



Lead formulas

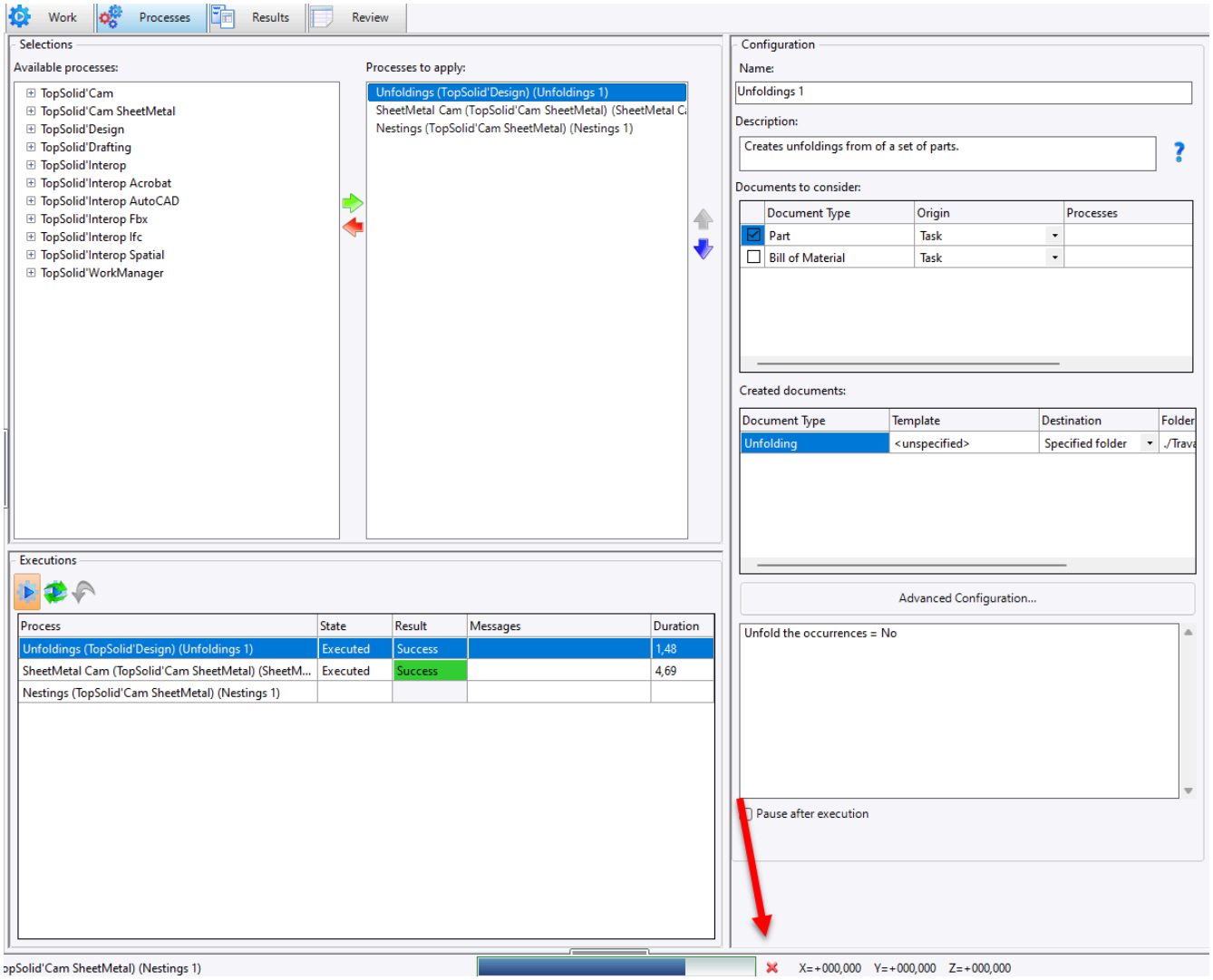
You can now assign multiple formulas to the same lead, allowing different formulas to be applied depending on whether the lead is used on a line or an arc.



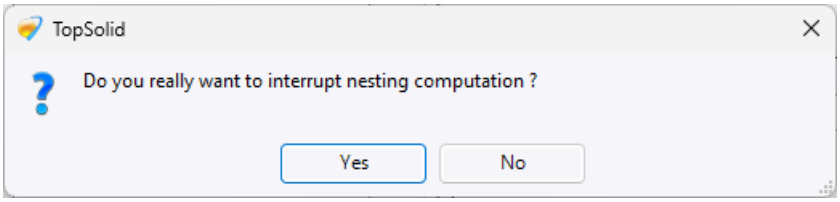
Work

Process cancellation

You can now cancel a process in progress in the work document by simply pressing the **Esc** key or clicking on the red cross at the bottom of the screen.



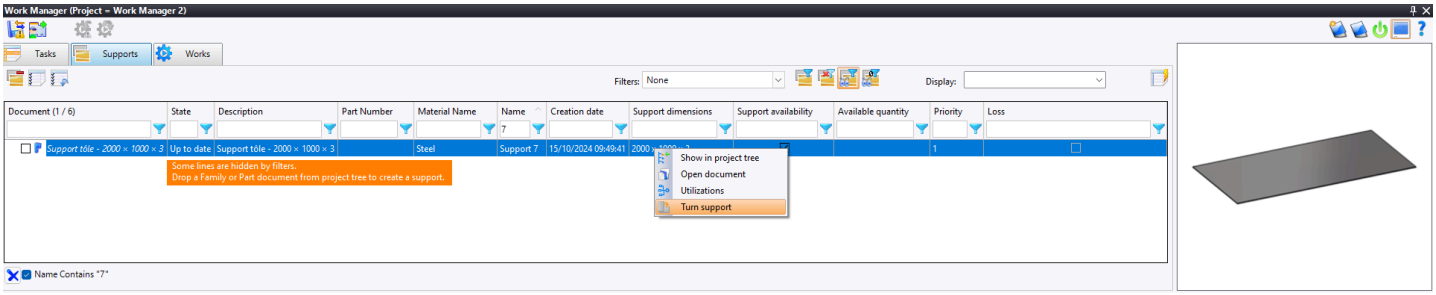
A confirmation message will appear after any of these actions.



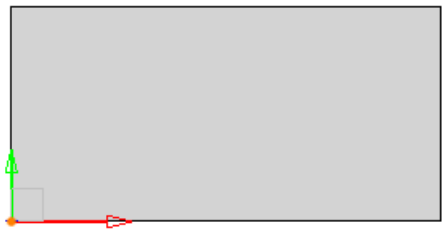
Work Manager

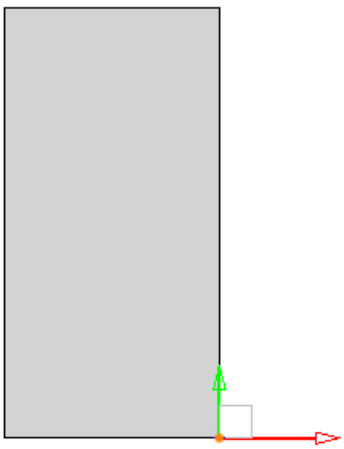
The **Turn Support** contextual command is now available in the **Supports** tab of the Work Manager, allowing you to apply a rotation to the selected support.

This feature is only available for supports that are not derived from a family.



Support before rotation	Support after 90° rotation





What's New in TopSolid'Inspection 7.19

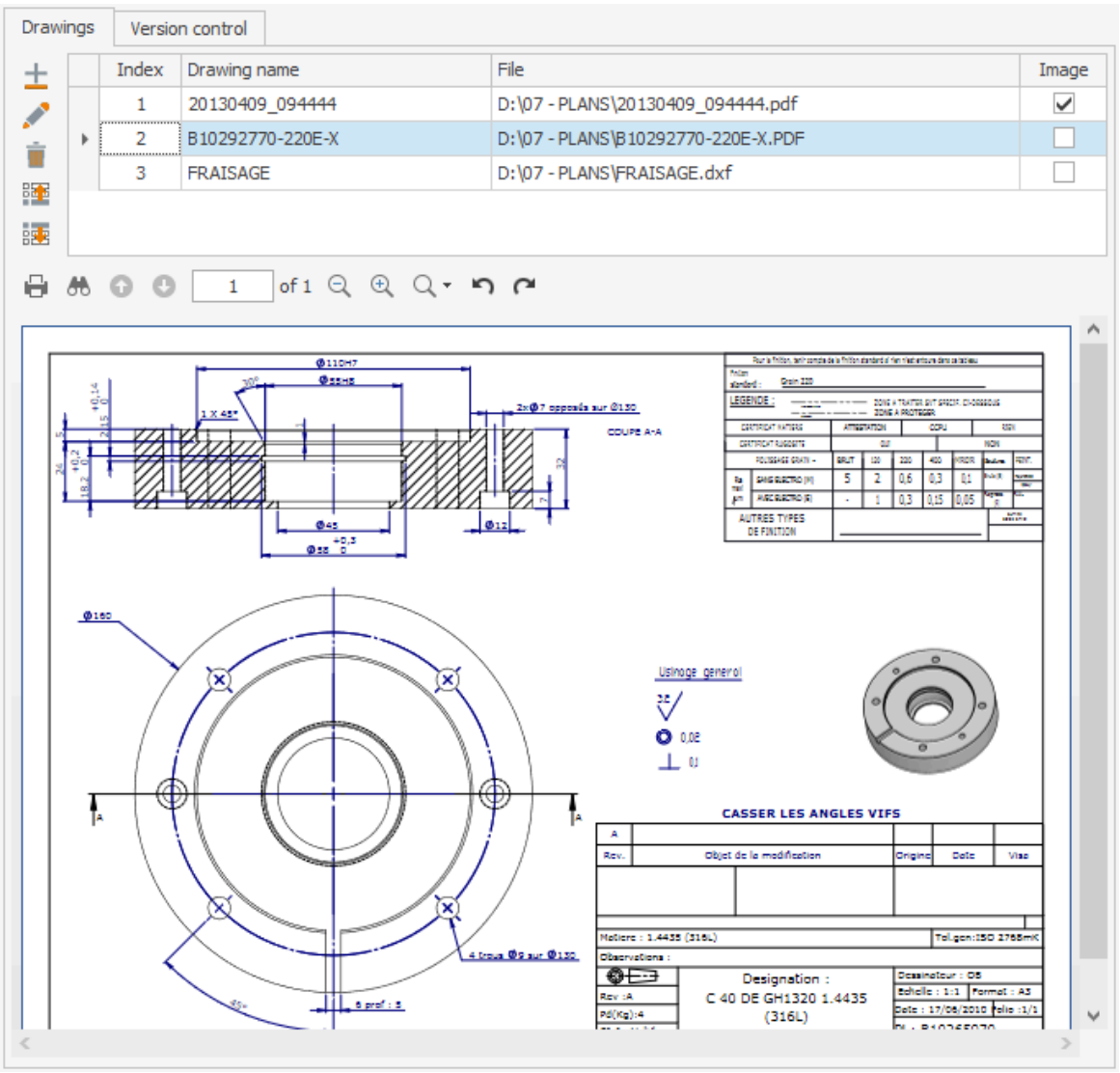
Discover the latest features in **TopSolid'Inspection**, **TopSolid 7**'s quality control software!

Process

Multi-drawing management

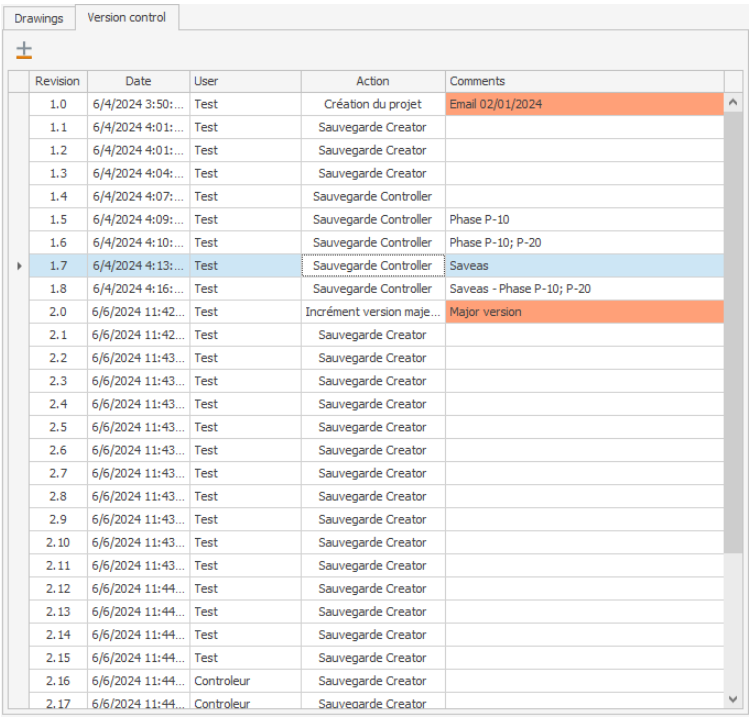
You can now capture dimensions from multiple part drawing files.

This feature allows the integration of files in various file formats and structures, with the use of multiple tabs to efficiently manage your entire inspection project.



Version management

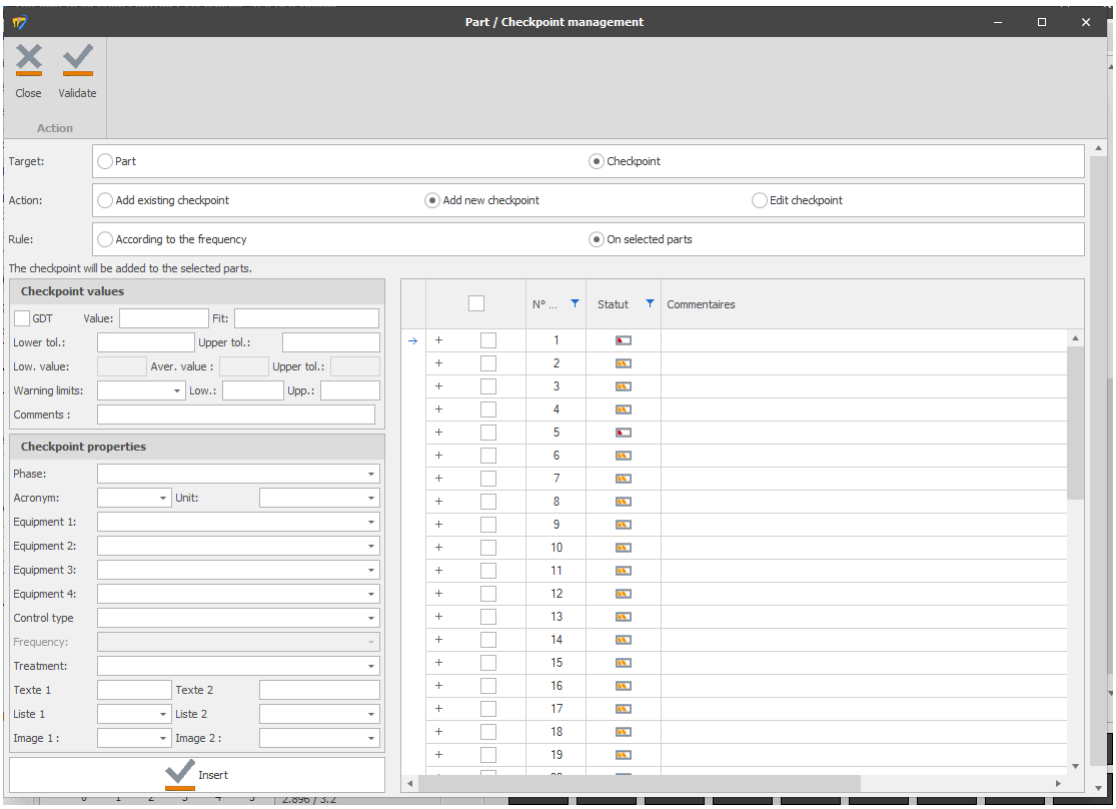
Track your project's lifecycle with ease. In line with the new ISO standard, any updates to an Inspection project are now automatically versioned. Each save in **TopSolid'Inspection** or any action within **TopSolid'Erp** automatically increments the version number, whether for a major or minor revision.



Revision	Date	User	Action	Comments
1.0	6/4/2024 3:50:...	Test	Création du projet	Email 02/01/2024
1.1	6/4/2024 4:01:...	Test	Sauvegarde Creator	
1.2	6/4/2024 4:01:...	Test	Sauvegarde Creator	
1.3	6/4/2024 4:04:...	Test	Sauvegarde Creator	
1.4	6/4/2024 4:07:...	Test	Sauvegarde Controller	
1.5	6/4/2024 4:09:...	Test	Sauvegarde Controller	Phase P-10
1.6	6/4/2024 4:10:...	Test	Sauvegarde Controller	Phase P-10; P-20
1.7	6/4/2024 4:13:...	Test	Sauvegarde Controller	Saveas
1.8	6/4/2024 4:16:...	Test	Sauvegarde Controller	Saveas - Phase P-10; P-20
2.0	6/6/2024 11:42...	Test	Incrément version maje...	Major version
2.1	6/6/2024 11:42...	Test	Sauvegarde Creator	
2.2	6/6/2024 11:43...	Test	Sauvegarde Creator	
2.3	6/6/2024 11:43...	Test	Sauvegarde Creator	
2.4	6/6/2024 11:43...	Test	Sauvegarde Creator	
2.5	6/6/2024 11:43...	Test	Sauvegarde Creator	
2.6	6/6/2024 11:43...	Test	Sauvegarde Creator	
2.7	6/6/2024 11:43...	Test	Sauvegarde Creator	
2.8	6/6/2024 11:43...	Test	Sauvegarde Creator	
2.9	6/6/2024 11:43...	Test	Sauvegarde Creator	
2.10	6/6/2024 11:43...	Test	Sauvegarde Creator	
2.11	6/6/2024 11:43...	Test	Sauvegarde Creator	
2.12	6/6/2024 11:44...	Test	Sauvegarde Creator	
2.13	6/6/2024 11:44...	Test	Sauvegarde Creator	
2.14	6/6/2024 11:44...	Test	Sauvegarde Creator	
2.15	6/6/2024 11:44...	Test	Sauvegarde Creator	
2.16	6/6/2024 11:44...	Controleur	Sauvegarde Creator	
2.17	6/6/2024 11:44...	Controleur	Sauvegarde Creator	

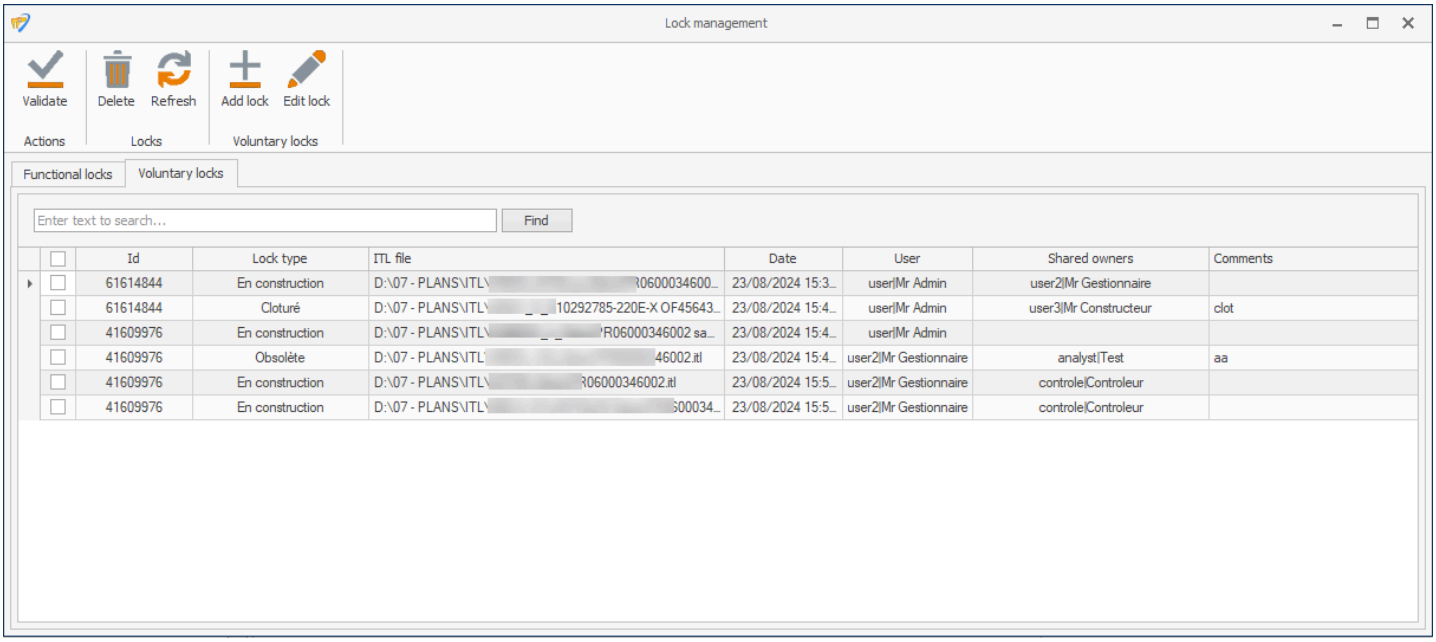
Adding checkpoints in Controller

This new feature improves workshop flexibility. During measurement sessions in **Controller**, you can now directly interact with parts or checkpoints. You can add characteristics to be inspected on all or a specific selection of parts, include additional parts for inspection, or adjust existing characteristics.



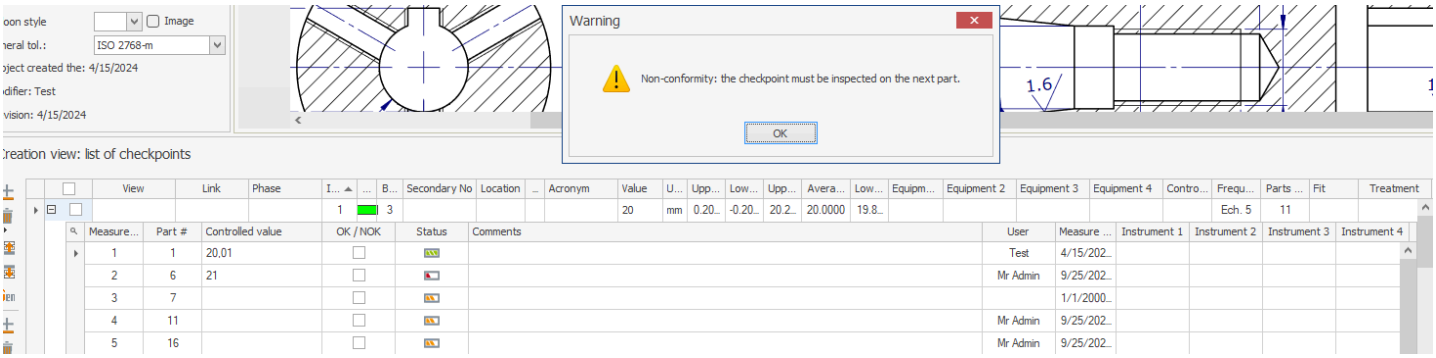
Voluntary project locking

You can lock a project to close it or restrict access to authorized persons only, enhancing project security and control.



Non-conformance management

Take scrap parts into account with new non-conformance management features. If a part has a non-conforming dimension, it can now be rejected. Additionally, for sampling frequencies, non-conformity of one dimension can trigger measurement on the next part.



Distributed percentage frequency

A new checkpoint frequency type allows you to distribute a defined percentage of parts for inspection across the entire series, ensuring optimal adherence to sampling frequencies.

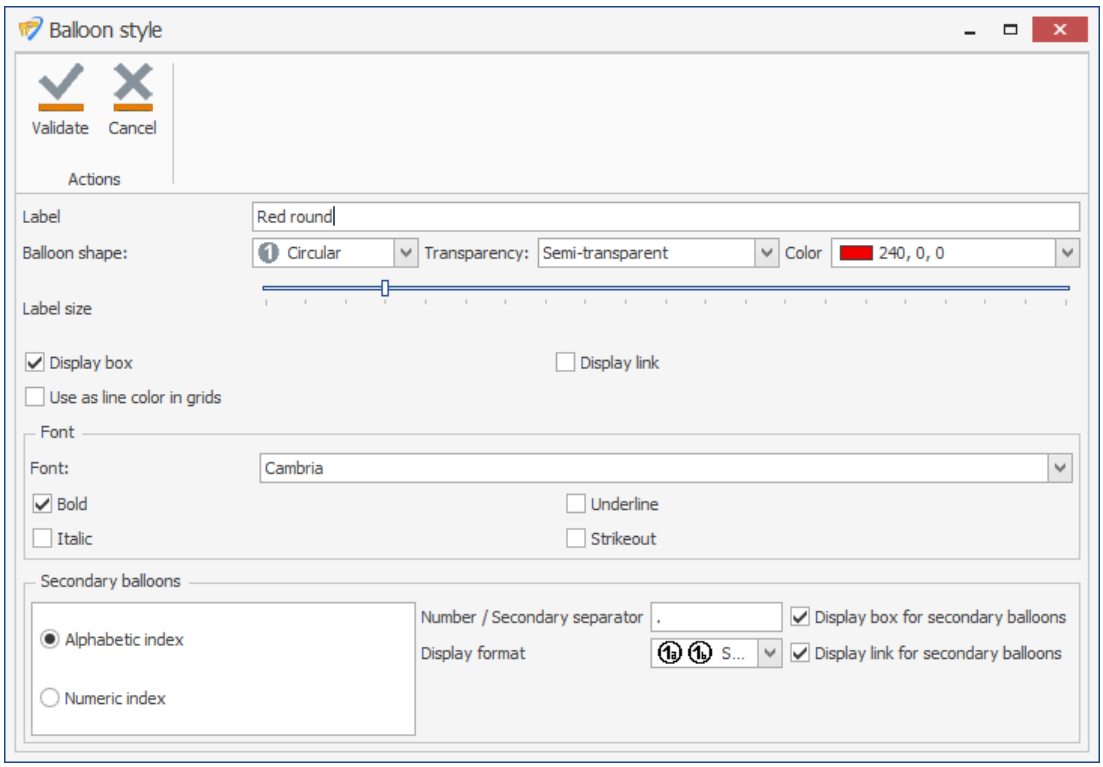
Deleting checkpoints without renumbering

This new feature streamlines project reuse by allowing you to delete a checkpoint without automatically renumbering the balloons.

Usability

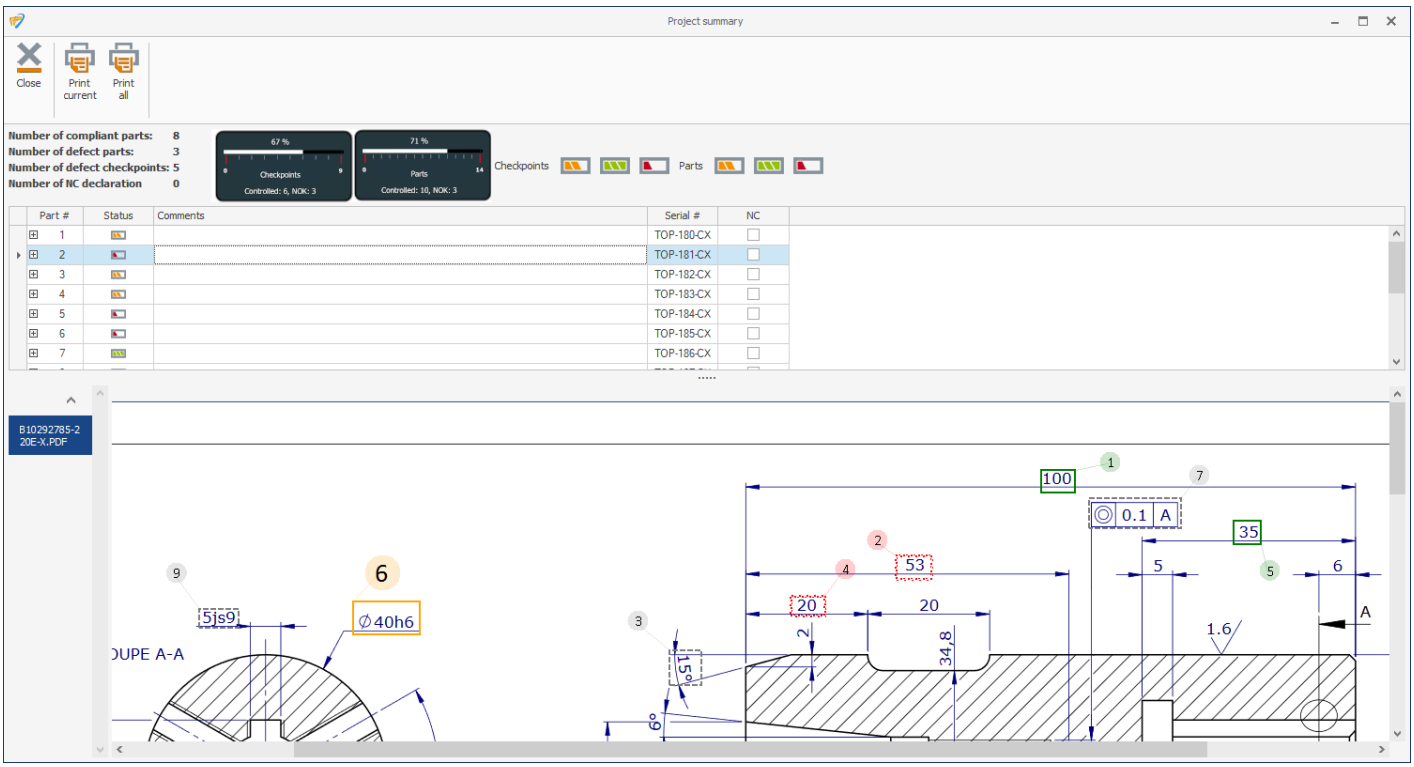
Balloon style

To simplify balloon formatting, display options such as color, shape and transparency are now grouped into customizable styles, accessible with a single click from the project or label formatting window.



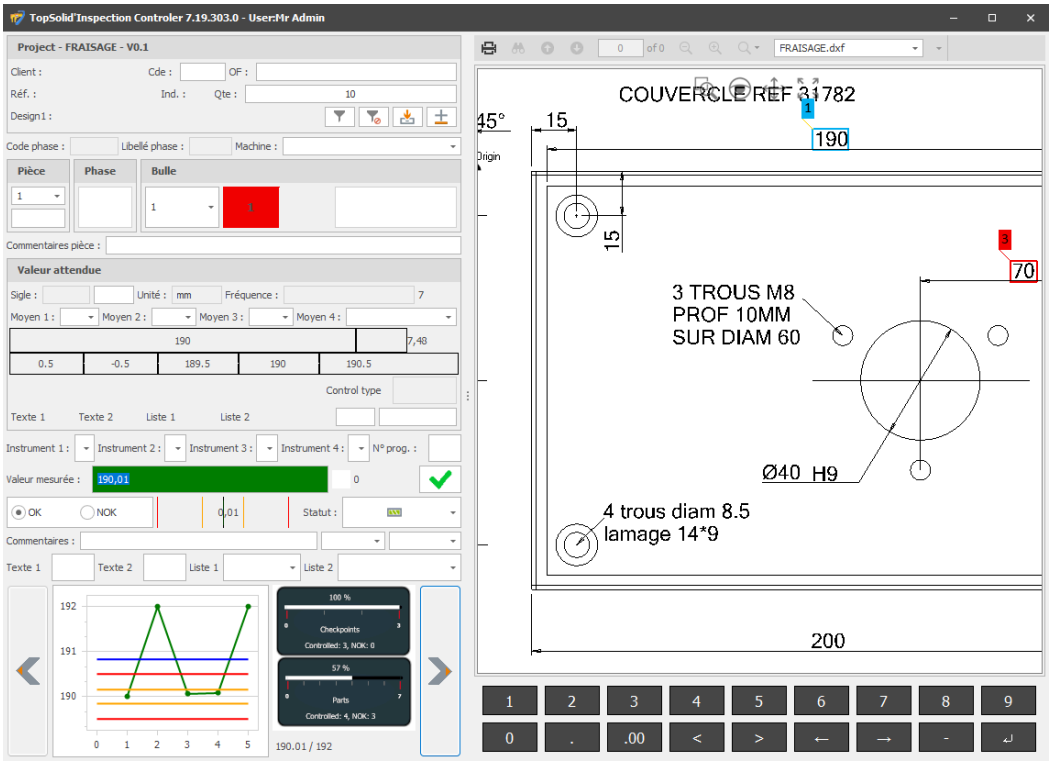
Non-conformance summary

This feature prints out a color-coded drawing for each part, highlighting the conformity of dimensions and providing a clear and quick overview of non-conformities.



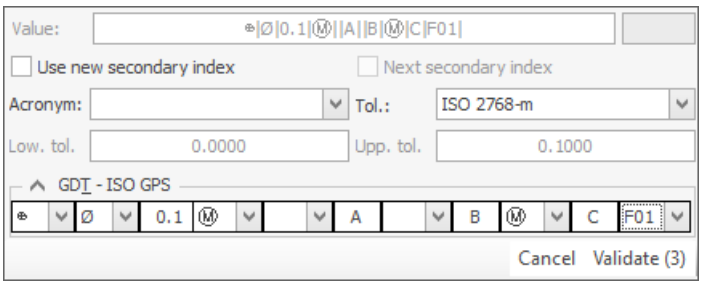
DXF drawing display in Controller

The **Controller** application now makes it easy to display digital drawings in DXF or DWG format.



Complex geometric tolerances

The functionality for entering ISO GPS geometric tolerances (GD&T) has been enhanced, enabling more precise management of complex tolerances in your projects.



Single balloon for secondary indexing

Customize the layout of your ballooned drawings. When duplicating a balloon with indexing, only one balloon will be displayed, simplifying the organization of information on the drawing.

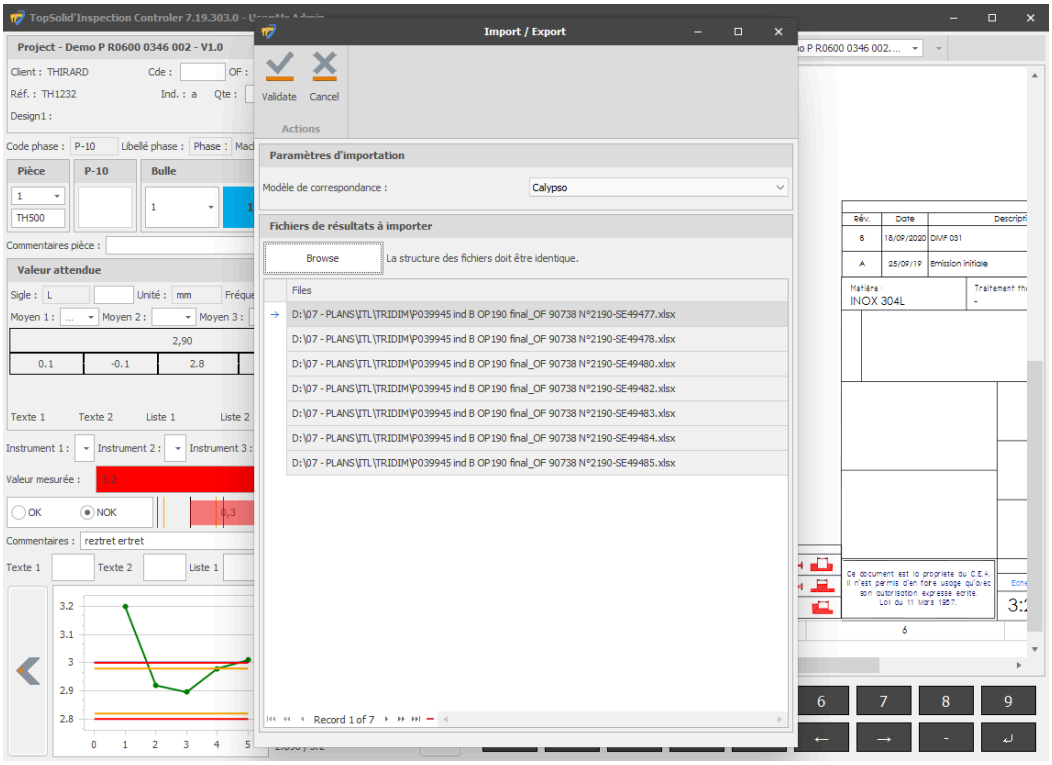
Disabling graphic animation

You can now disable graphic animations in **Creator** and **Controller**, optimizing display performance.

Import

CMM import from Controller

It is now possible to import results directly from the workshop station. **Controller** supports importing external reports, allowing for additional properties to be filled in during the import process.



Location on CAD import

When creating a project from a **TopSolid'Design** drawing, you can easily locate the balloons and automatically focus on the dimension to be inspected.

CMM import by balloon number

A new feature allows you to import CMM reports using the part's balloon number and serial number, instead of balloon index and part index. This ensures greater compatibility with various import formats.

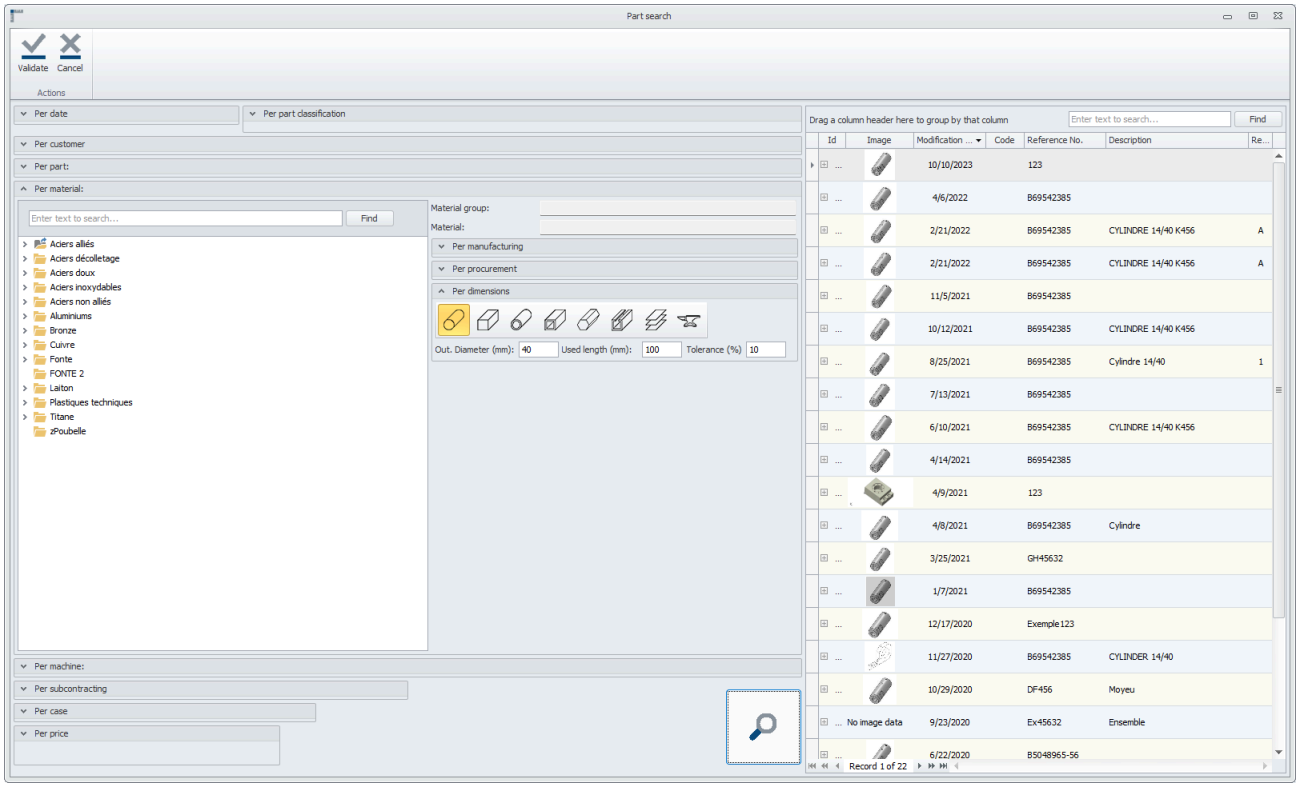
What's New in TopSolid'PartCosting 7.19

Discover the latest features of **TopSolid'PartCosting**, **TopSolid 7**'s part costing software for manufacturing!

Performance

Part search by material dimensions

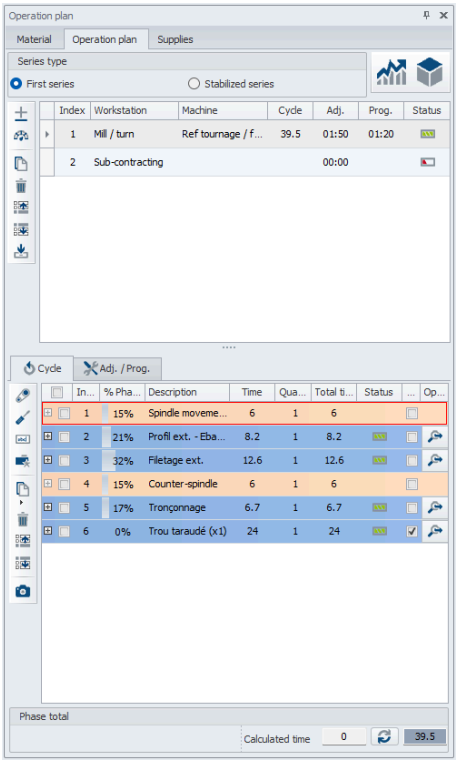
Easily identify similar parts based on their material characteristics to optimize analogy-based cost estimation. The multi-criteria search window now lets you filter by material dimensions.



Usability

Configurable time unit and rounding

Customize the calculation time unit to align with your business needs. Cycle times can be displayed in hours, minutes or seconds, with rounding adjustments according to your specifications.



Process

Batch quantity

Optimize manufacturing by defining the quantity per batch. The quantity per batch is added to the total number of parts to be supplied. Implementation times are now calculated for each batch, rather than for the whole series.