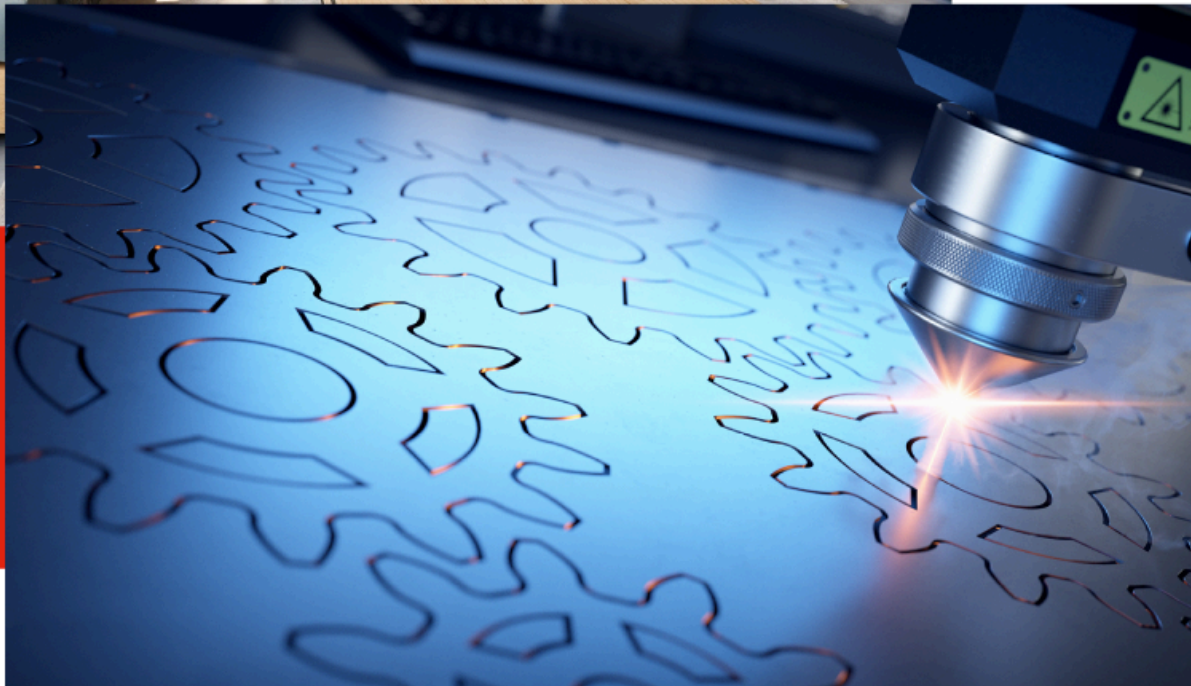


**TopSolid**  
2024



| What's New  
TopSolid 6.25



2024, **TOPSOLID SAS**.  
7, Rue du Bois Sauvage  
F-91055 Évry, France  
Web: [www.topsolid.com](http://www.topsolid.com)

All rights reserved.

**TopSolid®** is a registered trademark of **TOPSOLID SAS**.

**TopSolid®** is a product name of **TOPSOLID SAS**.

The information and the software contained within this document are subject to change without prior warning and should not be construed as a commitment by **TOPSOLID SAS**.

The software covered by this document is supplied under license and may only be used and duplicated in compliance with the terms of this license.

Hard copy or digital materials provided during training or accessible on-line as part of the training represent a protected original work that is the property of the training organization. They cannot be reproduced in part or in full without the express consent of the training organization.

All the texts, comments, works, illustrations and images reproduced on these materials are protected by copyright worldwide. Any use other than that designed for the purposes of training is subject to the prior authorization of the training organization, subject to prosecution. The Customer shall refrain from using, reproducing, representing, lending, exchanging, transmitting or transferring and more generally exploiting all or part of the documents without the prior written consent of **TOPSOLID SAS**. The Customer shall also refrain from extracting all or part of the data and/or transferring it to another material, and from modifying, adapting, arranging or transforming it without the prior written consent of the training organization. The Customer is only granted a right of use, to the exclusion of any transfer of ownership in any form whatsoever. Therefore, only the reproduction and representation of the content authorized by the French Intellectual Property code on a screen, and a single hard copy for archiving purposes, are authorized, for strictly personal purposes, and for professional use.

The Customer also undertakes to not take part in competing, directly or indirectly, with the training organization by transferring or communicating these documents to anyone.

Version 6.25

Rev. 01

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

# Contents

- Welcome to TopSolid 2024..... 1**
- What's New in TopSolid'Wood 6.25.....2**
  - User interface.....2
  - Design..... 11
  - Drafting..... 28
  - WoodWOP interface.....32
- What's New in TopSolid'WoodCam 6.25..... 33**
  - Cutting conditions..... 33
  - Label..... 33
  - Tools..... 35
- What's New in TopSolid'Sheetmetal 6.25.....36**
  - V7 import..... 36
  - Multi-document import..... 37
  - Cutting..... 38
  - Punching..... 47
  - Cutting and punching..... 55
  - Tasks manager.....57
  - Tubes..... 61



# Welcome to TopSolid 2024

This document presents the main improvements and evolutions introduced in version **6.25** of **TopSolid v6** software. The innovations described here are only part of the new innovative features incorporated in the software.

To find out what else is new, kindly reach out to your nearest agency.

# What's New in TopSolid'Wood 6.25

This chapter describes the new features in the version 6.25 of TopSolid'Wood.

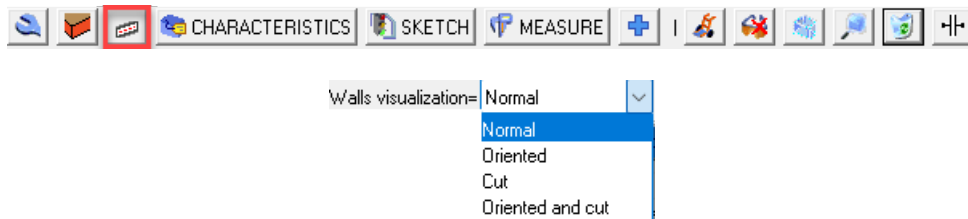
## User interface

### Wall command at neutral mode

You can now use the **Neutral mode** command on the graphics area to create walls.

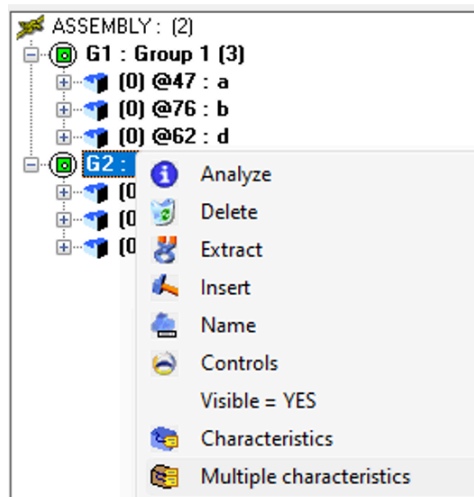


You can now use the **Neutral mode** command on a wall to manage its visualization.



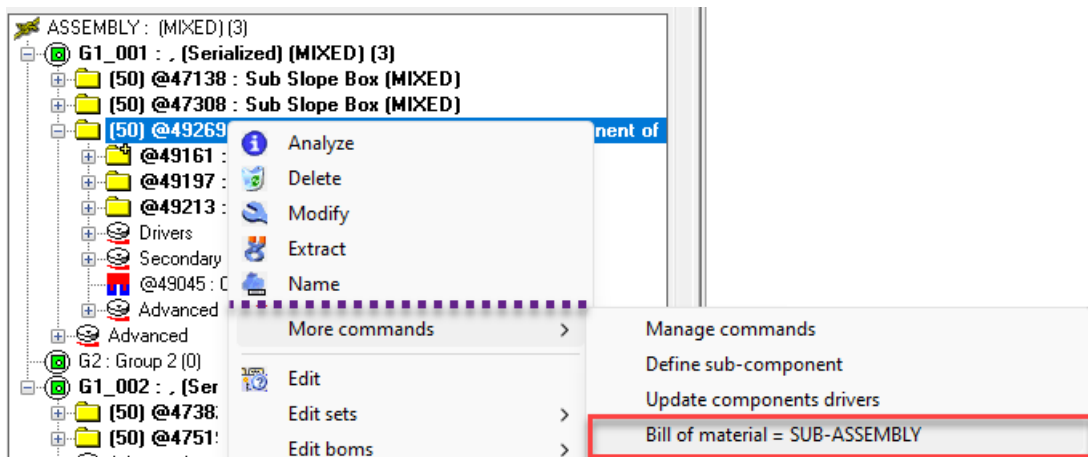
### Definition of multiple parts from the construction tree

The multiple part definition command is now available in the context menu from the construction tree. The main assembly and the first node of the tree provide the **Multiple characteristics** command which displays the selected line(s) in the multiple part definition window.



## BOM mode modification

You can now define the **Single unit** or **Sub-assembly** bill of material mode of a sub-component directly from the final assembly. In previous versions, this feature was only possible on the first node of the tree.

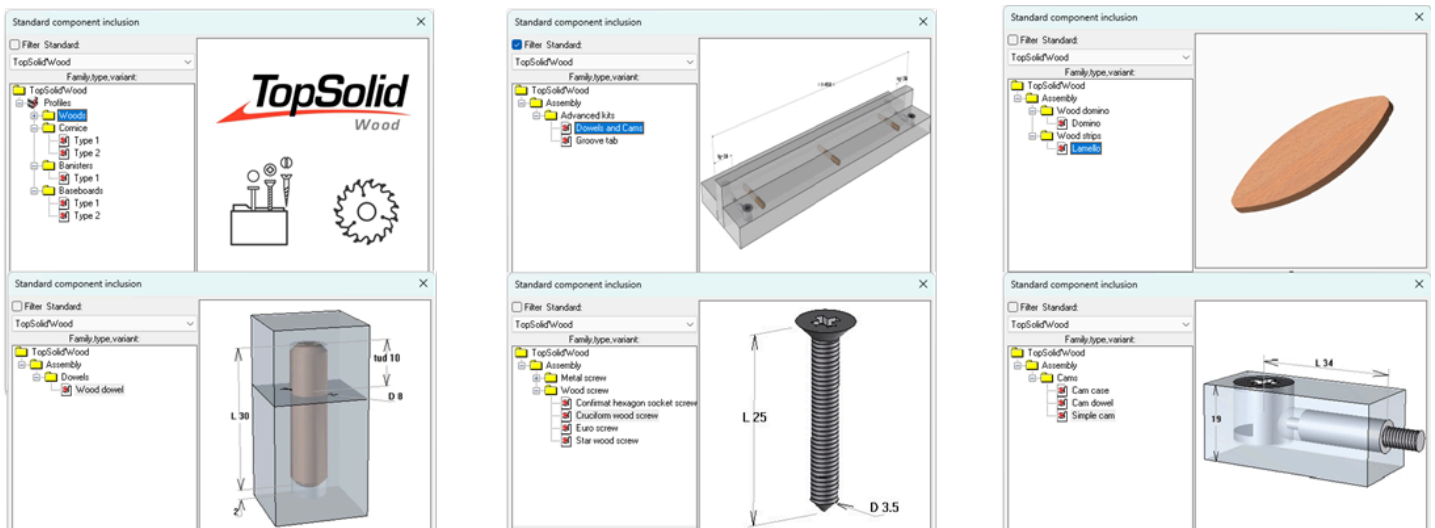


## Hardware filtering

The **Dowel assembly**, **Cams and dowels**, **Screw**, **Lens**, **Extruded component** and **Assembly kit** assembly commands now only display compatible hardware.



You can use this standard component filtering mechanism to create your own libraries.



Add the corresponding command line to the **lib.cfg** file at the root of your standard library:

### Dowel

```
D_COMPO_ZWOO_PIN "Family" "Type" "Variant"
```

### Assembly kit:

```
D_COMPO_ZWOO_KIT "Family" "Type" "Variant"
```

### Lens

```
D_COMPO_ZWOO_LENS "Family" "Type" "Variant"
```

## Cam

```
D_COMPO_ZWOO_ECCENTRIC "Family" "Type" "Variant "
```

## Screw:

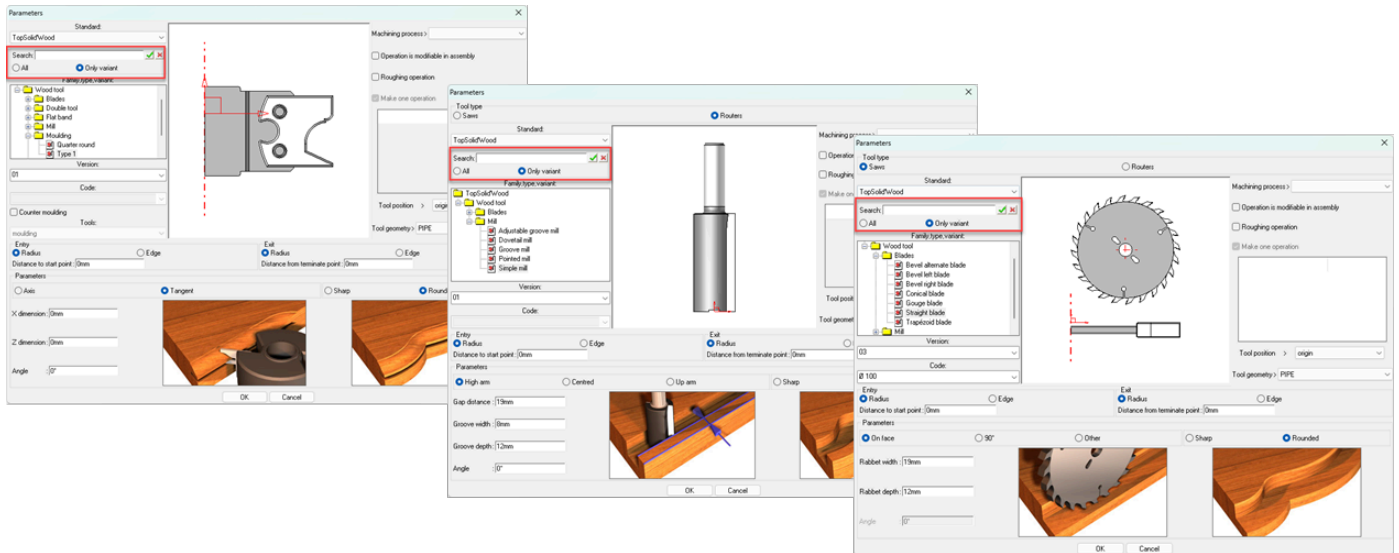
```
D_COMPO_ZWOO_SCREW "Family" "Type" "Variant "
```

## Extruded component

```
D_COMPO_ZWOO_EXTRUDEDCOMPONENT "Family" "Type" "Variant "
```

## Machining tool filtering

The **Groove**, **Rabbet** and **Moulding** machining commands only display compatible tools. You can use this standard component filtering mechanism for your own libraries.



Add the corresponding command line to the **lib.cfg** file at the root of your standard library:

```
D_SH_OP_ZWOO_SHAPING_FILTER_SLOT "Family" "Type" "Variant "
```

```
D_SH_OP_ZWOO_SHAPING_FILTER_FOLD "Family" "Type" "Variant "
```

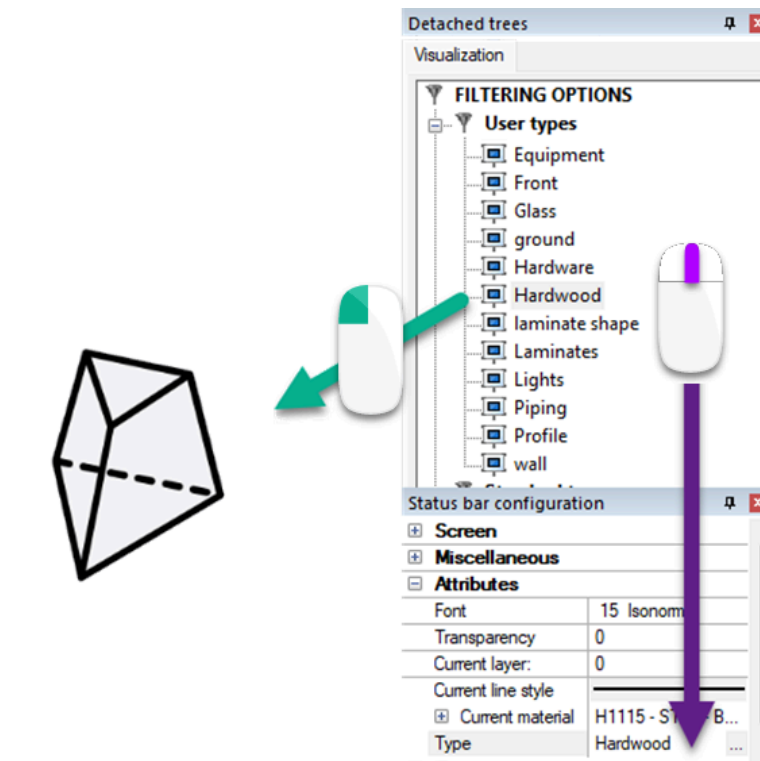
```
D_SH_OP_ZWOO_SHAPING_FILTER_MOULDING "Family" "Type" "Variant "
```



## Part typing

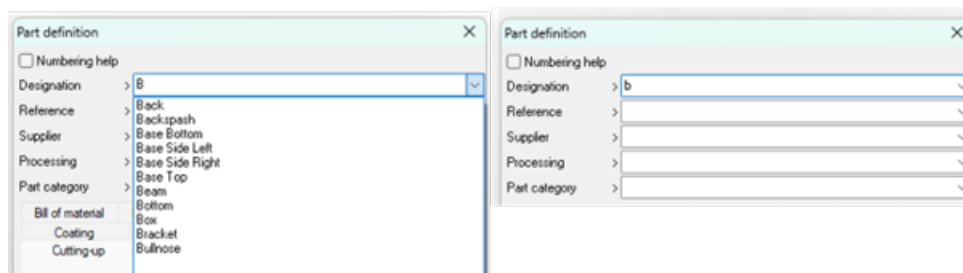
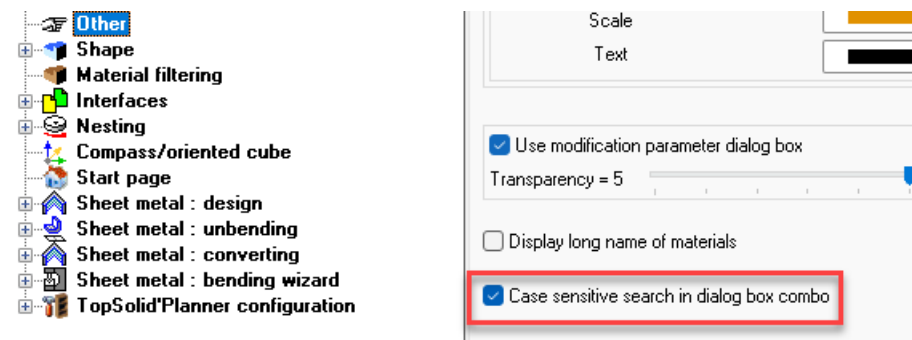
The Visualization tab of the construction tree features two new shortcuts for typing parts. You can:

- either use the wheel click to set the default current value;
- or make a drag & drop into the graphics area for manual selection.



## Search by completion

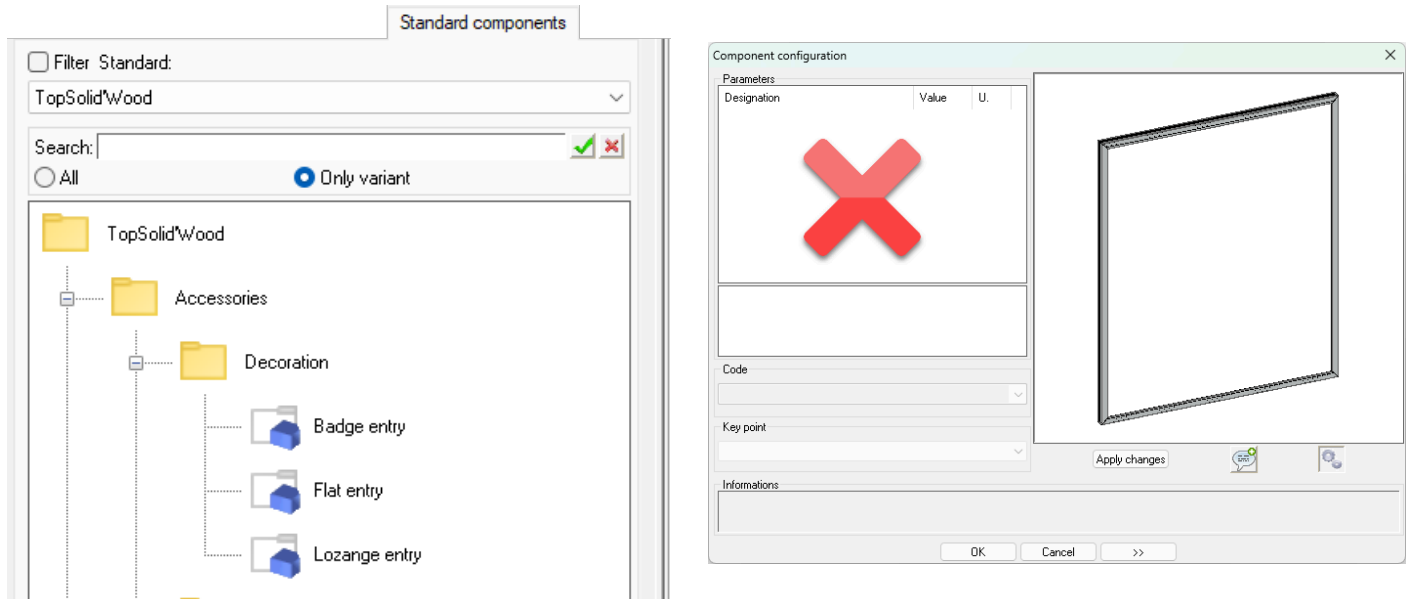
The new Case sensitive search in dialog box combo option, available in Tools > Options > Other, ensures exact syntax when searching by completion.



## Including a standard component from the tree

### Parameter management

When including a standard component from the tree, the component configuration dialog box is no longer displayed when no parameter is available.

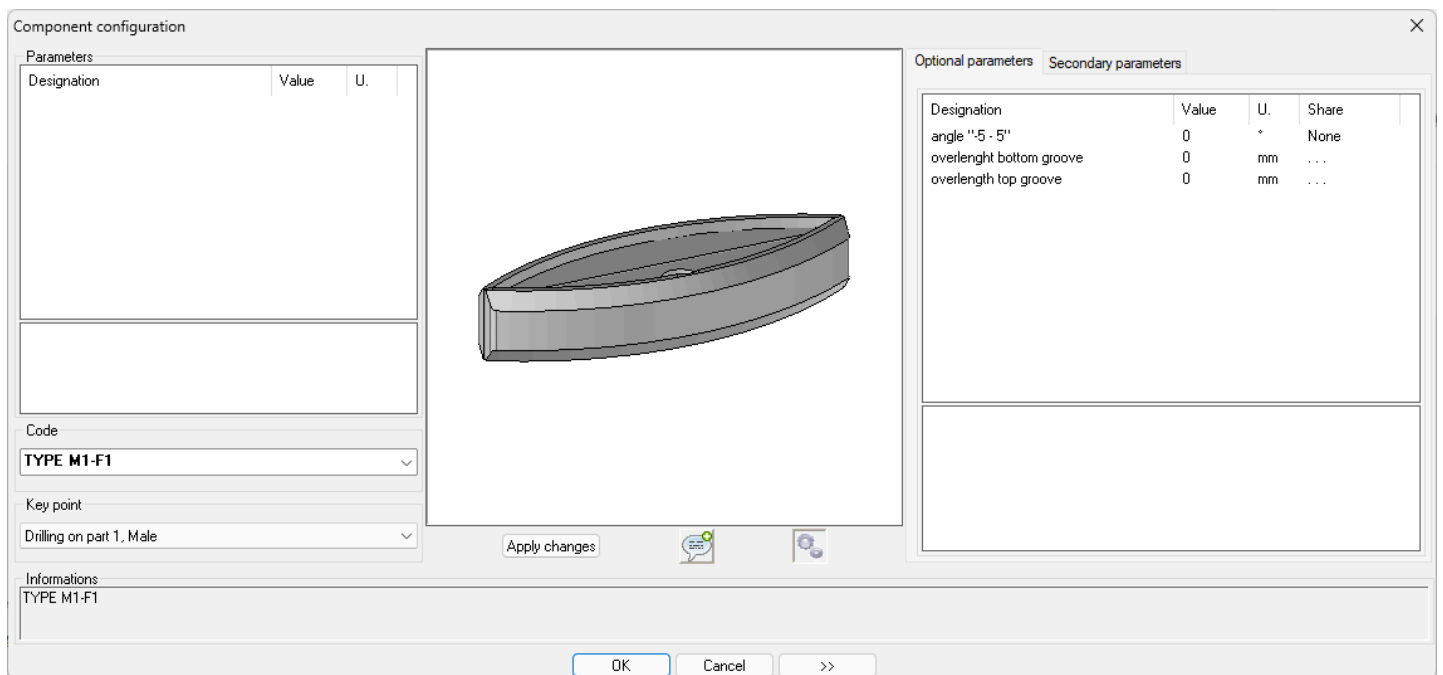


### Advanced mode

The **>>** mode extends the side panel in the component configuration window by default when the component has optional or secondary parameters.

In cases where the component does not have one, the side panel collapses.

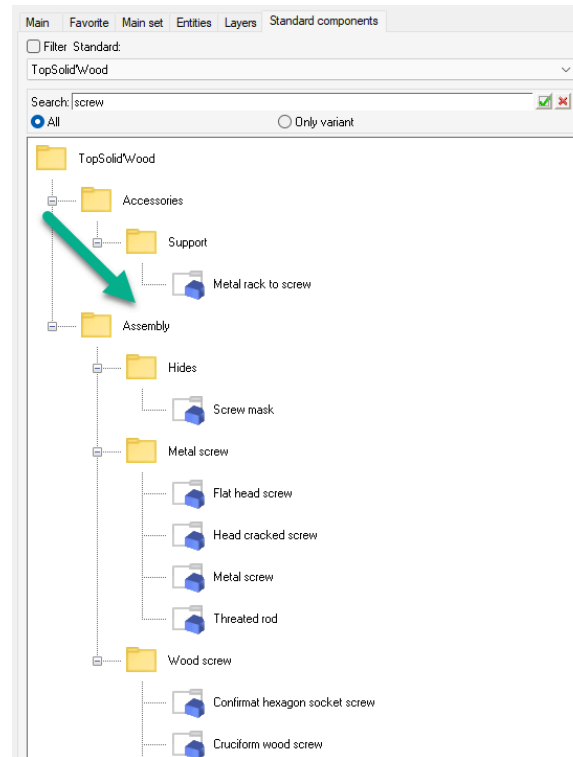
The **<<** mode displays the configuration window in minimalist mode and advanced parameters are not shown.



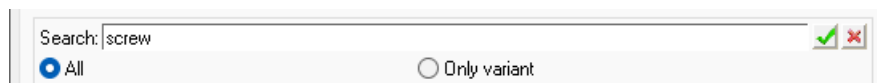
**Note:** In the **<<** mode, when a component does not have a main parameter but has advanced parameters, the side panel is still expanded.

## Component search

When searching for a component, the search results tree is automatically expanded.



The search for input text is carried out either on the standard component structure (**Family/Type**), or on the variant only, as in previous versions.

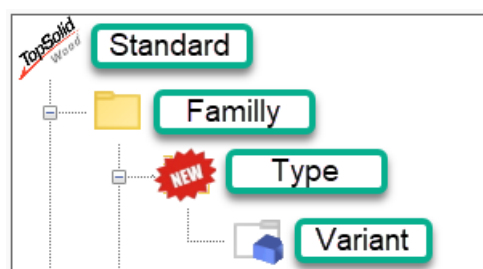


## Standard library: customizing the image of the component type

You can customize the preview image for the standard component type.

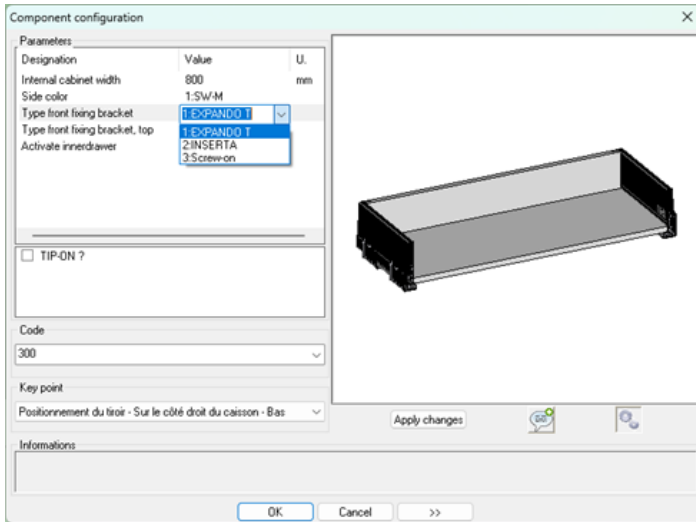
It must meet the following criteria:

- Its name must be "family type name.png".
- Its dimensions should be 40x40 pixels.
- Its location should be at the root of the standard component.



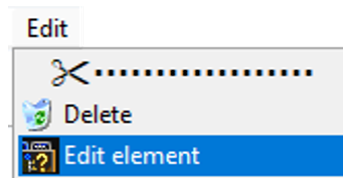
## Designation of predefined parameters

The standard component configuration dialog box now displays predefined parameter designations.

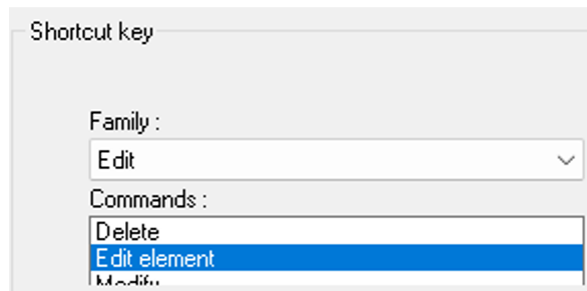


## Element editing

The new **Edit > Edit element** command allows you to search for an element and to display it in the tree.



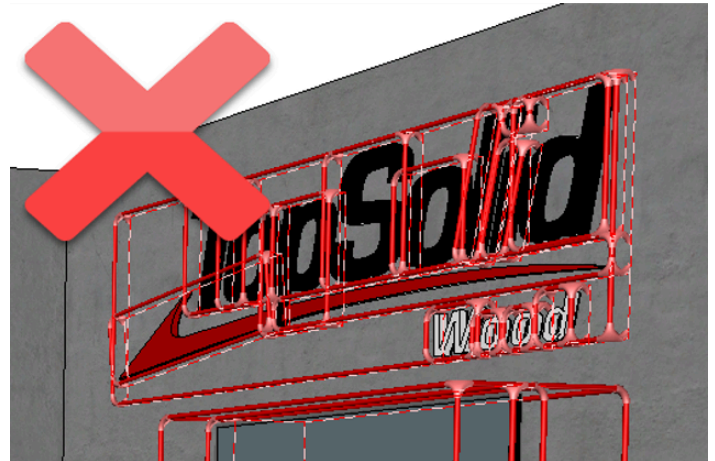
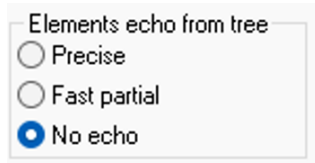
You can configure a keyboard shortcut for the command in **Tools > Options**.



## Selection echo in the configurators

You can now manage the echo of parts in the wood configurators (**Material, Properties, Export, Nesting, Machining and Drafting**).

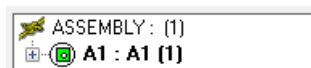
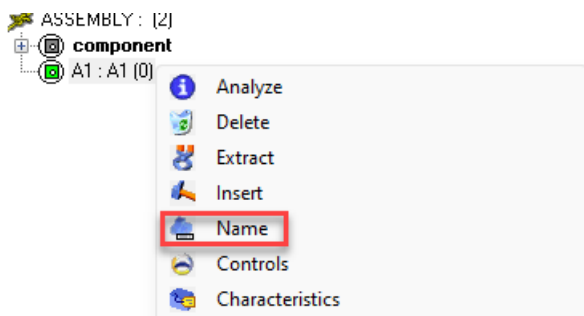
The echo setting for elements in the tree defined in **Tools > Options > Other** is applied to these wood commands.



## In-place subassembly

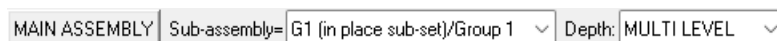
### Naming

The context menu on an **in-place subassembly** now includes a naming command.



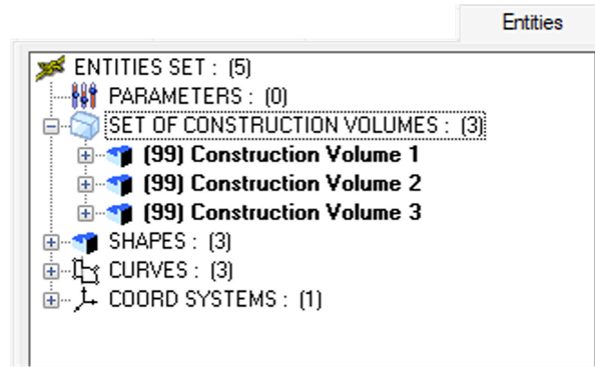
### Designations of in-place subassemblies in configurators

The **TopSolid'Wood** configuration features now display in-place subassembly designations in the subassembly selection drop-down menus.



## Construction volumes

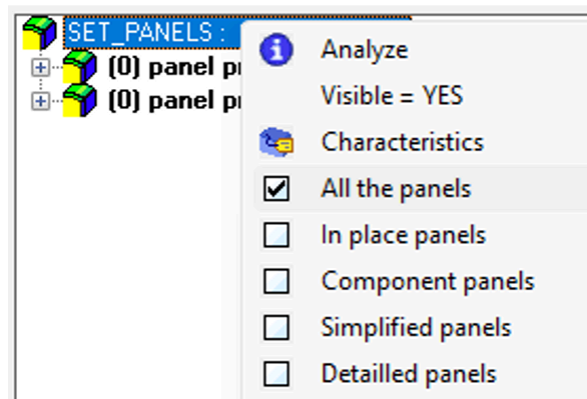
The **Entities** tab features a new section for organizing construction volumes.



## Panel set

A new set named **Panels set** displays all document panels. Several filters are available for a finer display:

- **In place panels:** panels designed in the document.
- **Component panels:** panels from components.
- **All panels:** the two previous panel types.

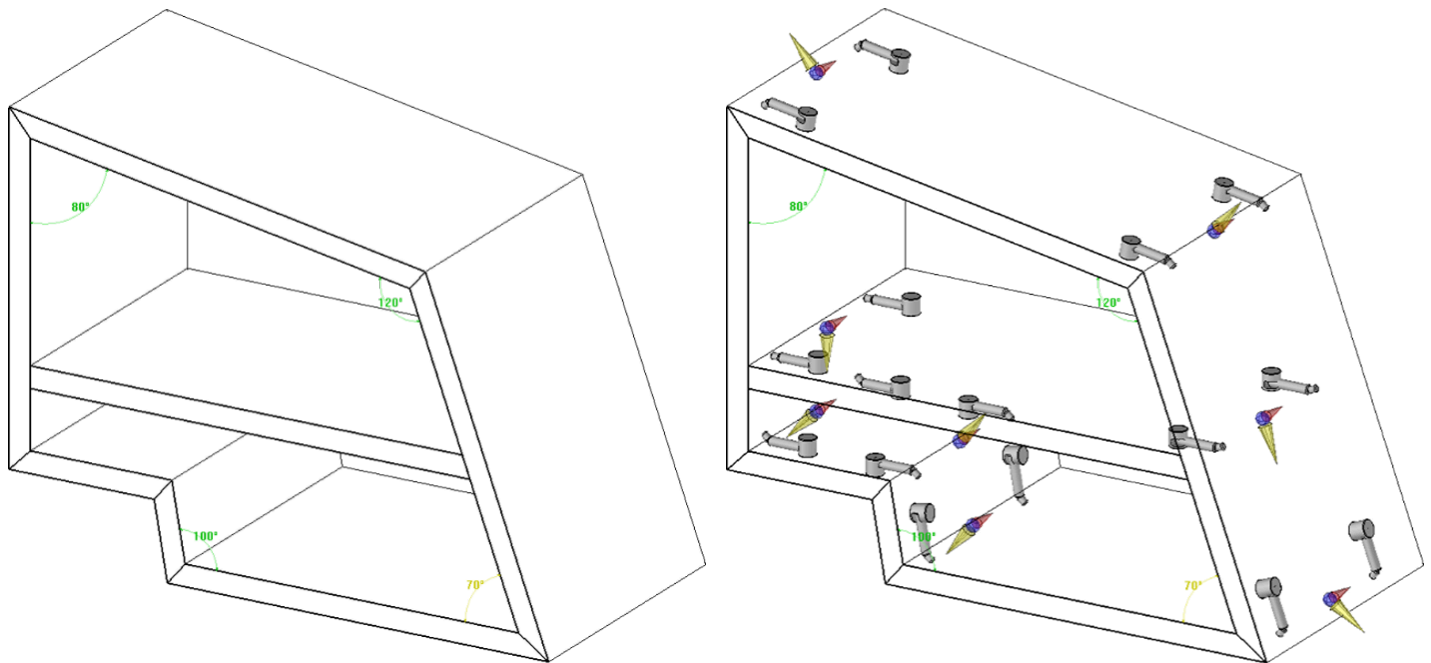


The **Simplified panels** and **Detailed panels** filters verify the panel representations.

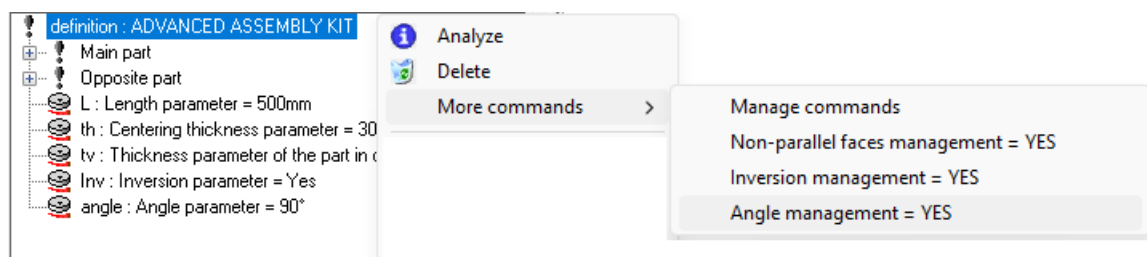
## Design

### Assembly kit

#### Kit on non-perpendicular parts



When activated, the new **Angle management** command creates an angular parameter under the assembly kit section. It is only available when the **Non-parallel faces management** and **Inversion management** modes are activated.



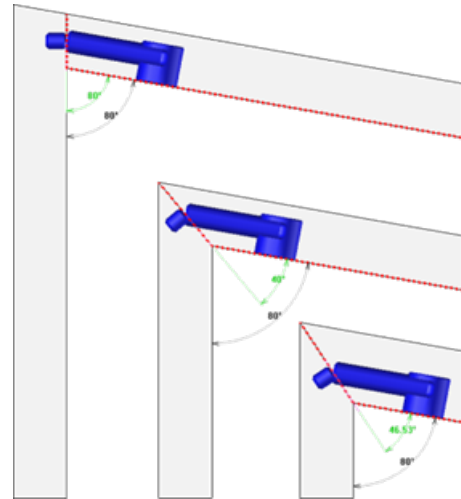
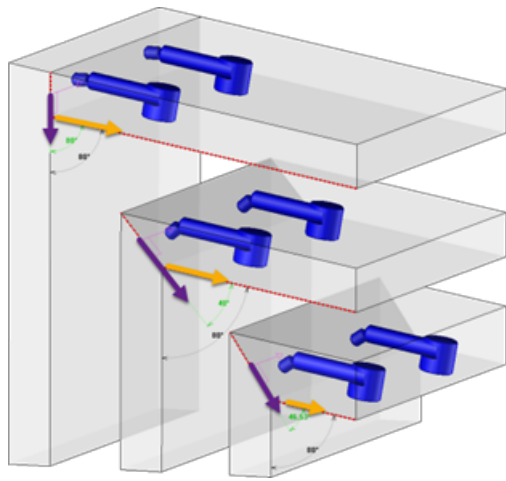
Assembly kits from earlier versions can be upgraded in the same way.

When the kit is included, each assembly proceeds to its own angular measure and provides the result to the kit, thereby unlocking new possibilities for managing angular links.

The measured angle is the result of two planes/directions.

The kit's positioning coordinate system gives the first plane of the measure, and the Y+ direction of the positioning coordinate system (see the purple arrow in the image below) gives the first direction.

On the second part to be assembled in relation to this Y+ direction, the second plane/direction is identified. The angle corresponds to these two planes in the directions of the purple and yellow arrows.



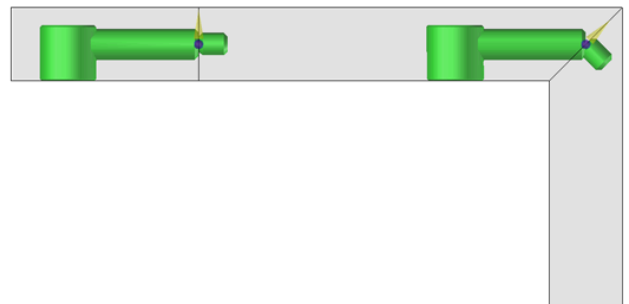
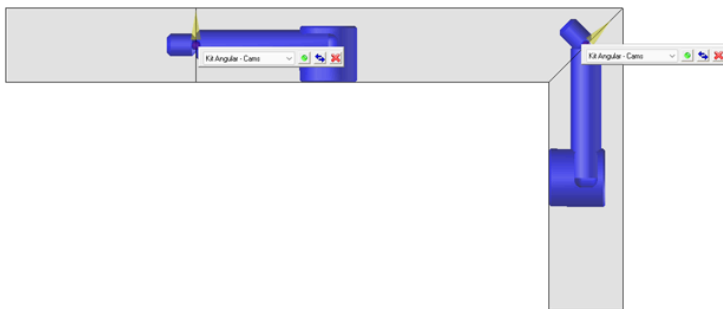
### Opposite definition in automatic assembly

Automatic assembly can now choose between the main definition or the opposite definition.

### Definition reversal

When using the **Automatic assembly** command, a new blue sphere gives access to kit modification.

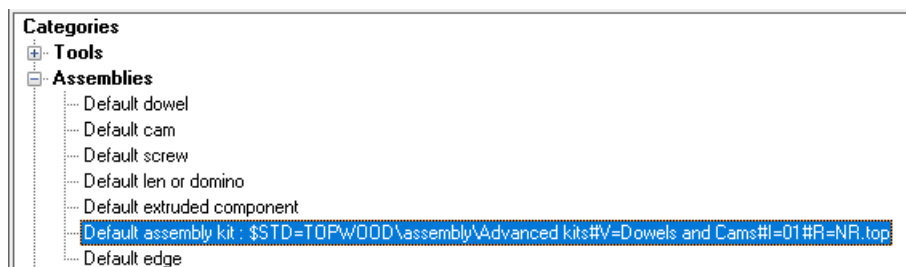
A new "blue arrows" icon allows you to select the opposite definition of the kit.



*The kit is positioned in the opposite direction.*

### Memorization of the last kit used

In **Tools > Options > TopSolid'Wood configuration > Files > Assemblies**, you can now define the assembly kit that will be used by default in each new document.

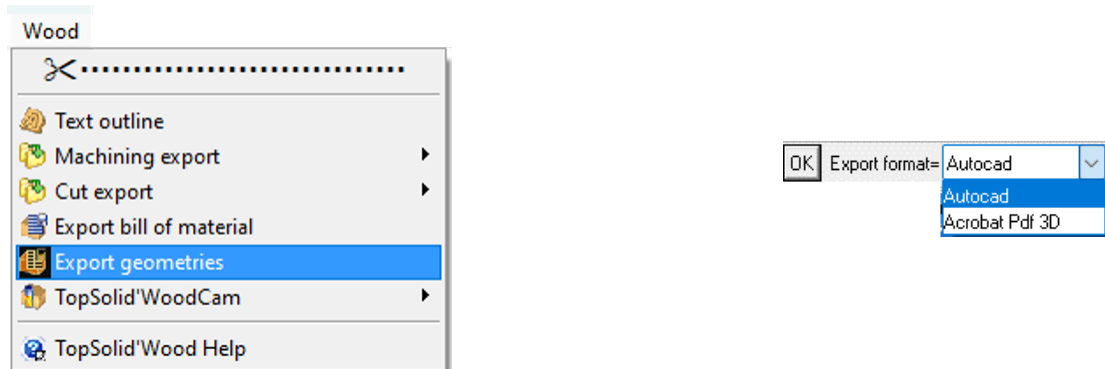


In addition, **TopSolid** memorizes the last assembly kit used, allowing you to use more assembly commands successively without having to select a kit each time. The default value is reset at the next **TopSolid** start.



## Multi-export of geometries

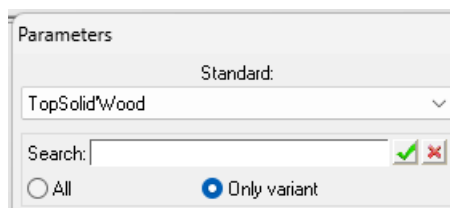
Based on the same principle as multi-export of machining operations, the new **Wood > Export geometries** command allows part geometries multi-export in **DXF** or **PDF** format.



**Note:** In **Tools > Options > TopSolid'Wood configuration > Files > Parts selection sorting**, you can define a BOM template to control the name of generated files.

## Search in wood machining operations

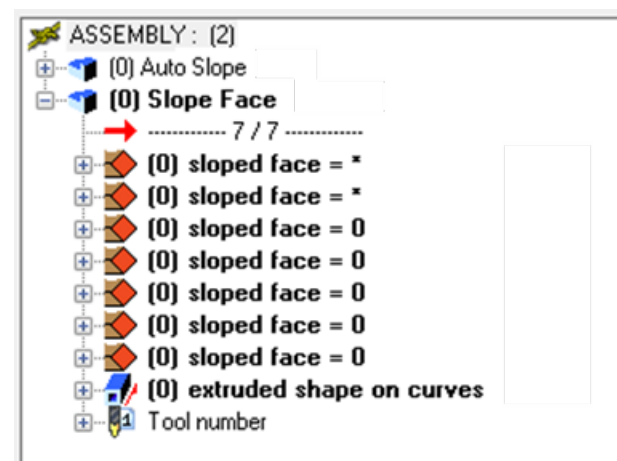
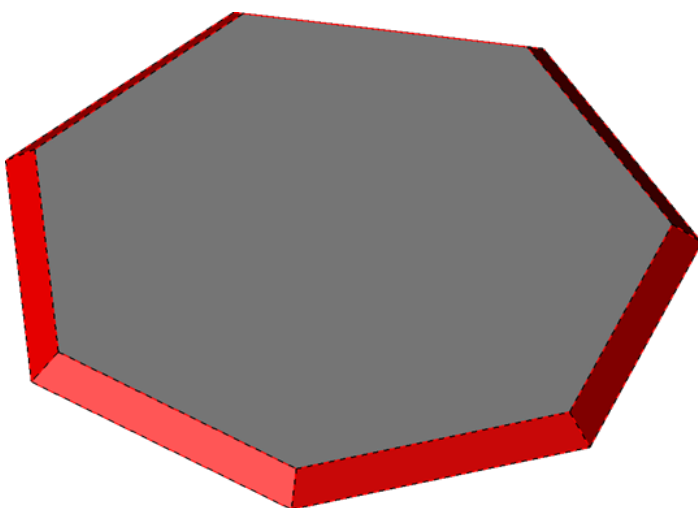
The search is now applicable to wood machining operations (groove, rabbet and moulding).



## Sloped face user machining

When creating a sloped face user machining, the new **Automatic** option automatically detects the lateral faces adjacent to the reference face selected.

An operation is created for each selected face.



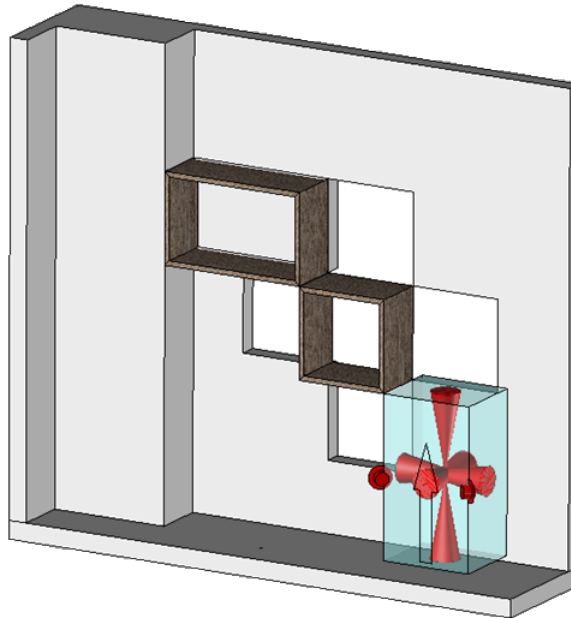
## Driver block

### Manual housing

The inclusion of a driver block component in **Housing** mode offers a new option for manually controlling the six planes that define the component's position/dimensions.



The first two planes define the front and back of the driver block. The following planes are free and define in pairs the width and height of the driver block.



### Driver block and catalog code

The notions of driver block and catalog code can now be combined.

When a catalog code fills in the parameters for a driver block (for example, block.x, block.y and block.z), the inclusion process no longer requests for these parameters, and the catalog takes precedence.

Code	block.x (mm)	block.y (mm)	block.z (mm)	hp (mm)
500	500	500	700	100
600	600	500	700	125
700	700	500	700	150

Columns management dialog:

- Column
- Main set
- Parameters
  - block.x
  - block.y
  - block.z
  - hp
- Shapes
- Components
- Texts

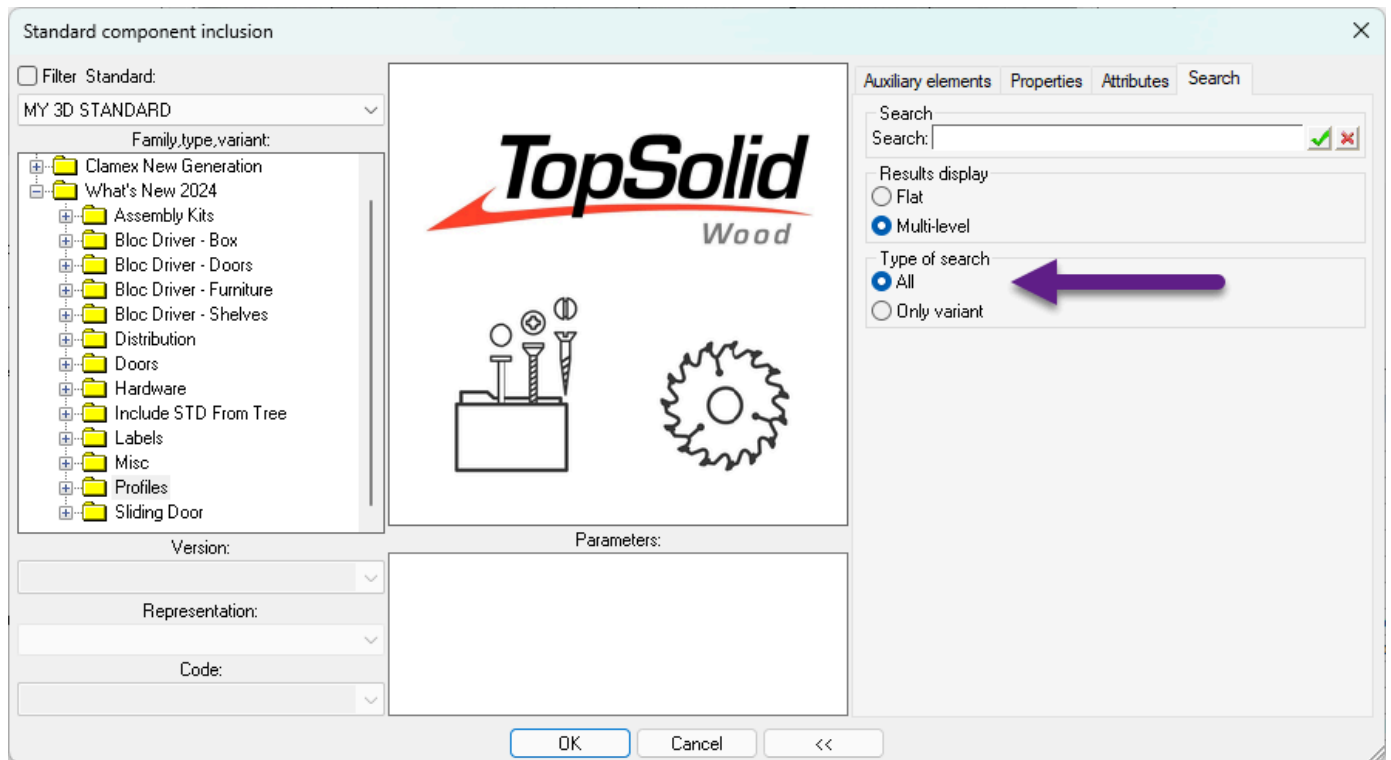
Code selection dropdown: OK Code= 500 (selected), 600, 700

The catalog code modification resizes the driver block values.

In cases where the driver is forced, it takes over the catalog.

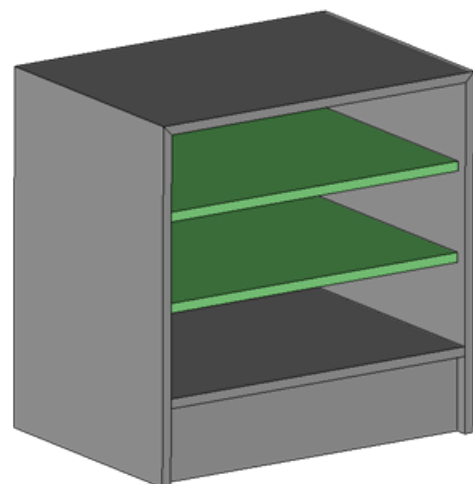
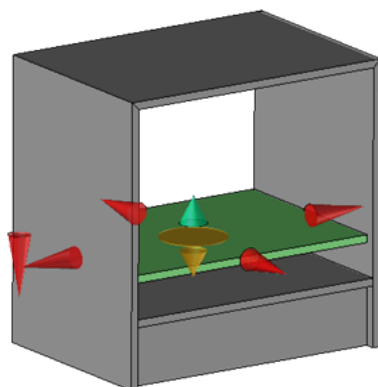
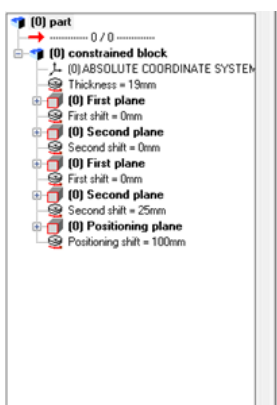
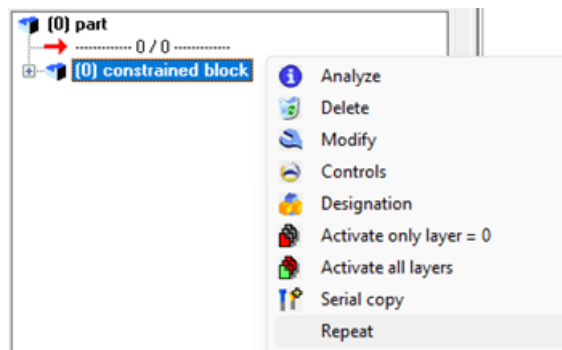
## Standard component search

Advanced search, on the entire structure or only on the variant, is also available when including a component via the **Include standard** command.



## Constrained block distribution

It is possible to repeat a constrained block using the new **Repeat** context command.



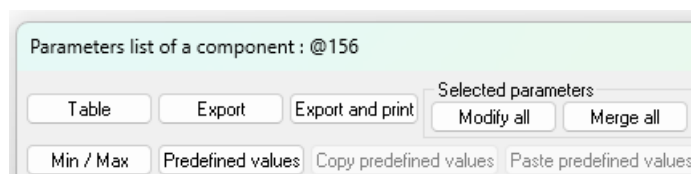
Two modes are available:

- **Distribute:** start face, end face and quantity are specified as for a distribution when creating the constrained block.
- **Constrained linear:** start and end faces are specified, then the distribution definition window opens as when creating the constrained block.

In the situation where the constrained block to be repeated has kits and edges, the repetition also applies to the latter.

## Multiple driver modifications

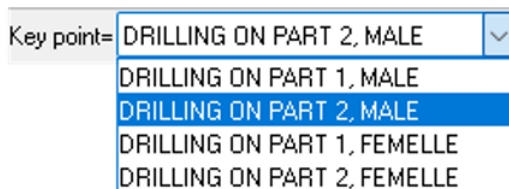
When modifying a component's parameters, the new **Modify all** and **Merge all** buttons can be used to modify or merge the selected parameters.



## Key point memorization when including multiple components

When inserting a component containing **several key points**, the selection of a key point other than the first in the list will be saved if a second component is positioned.

Once the first component has been validated, a second component can be positioned with the last key point used.



## Standard default component configuration

Every **standard library** can now define the default component type using a configuration word. For each library, add the `D_COMPO_FIRST` line indicating the default type:

```
D_COMPO_FIRST "Standard" "Family" "Type"
```

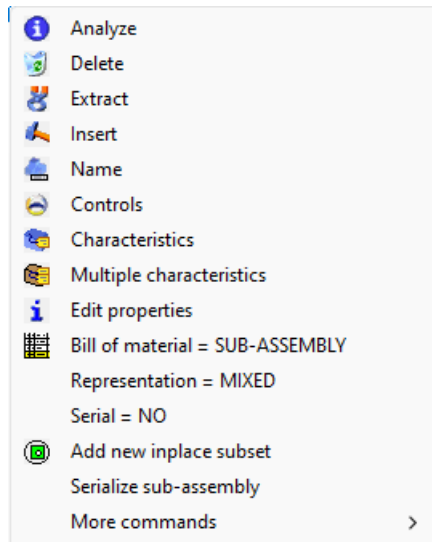
Standard/family/type names should be entered exactly as they appear in the **TopSolid** interface (unlike the file name on the disk). It is strongly recommended to pay attention to uppercases added by **TopSolid** and to use quotation marks when the name contains spaces.

## Declaring a subcomponent

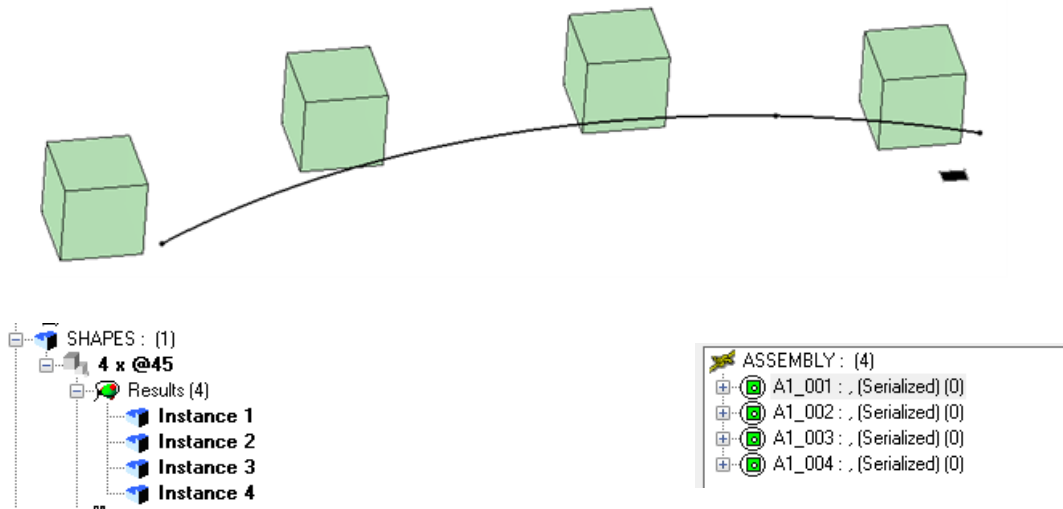
Declaring a subcomponent initializes the code, variant and deletion by default.

## Serial subassembly

The new **Serialize subassembly** command serializes a subassembly via a repetition, distribution or group.

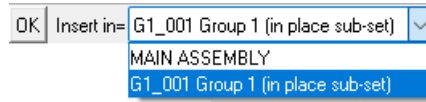


The number of subassemblies created depends on the number of repeated or distributed parts, or of the number of elements in the group.

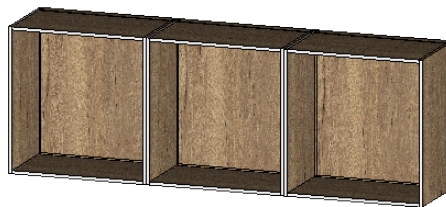


## Serial subassembly and serial component copy

When including a component, depending on the settings defined in **Tools > Options > Components > Components management > Choose insertion set**, it is possible to define the in-place subassembly in which the component must be created.

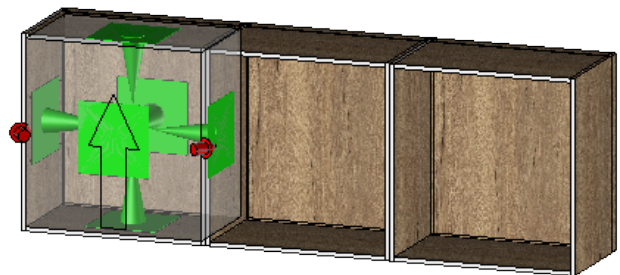
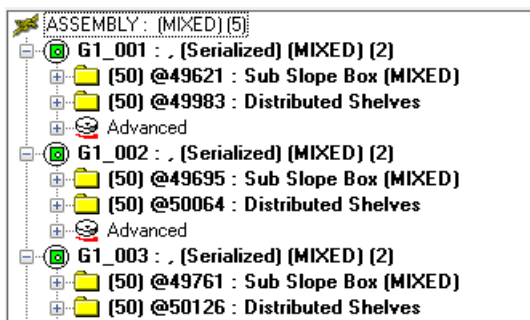


In the situation where the in-place subassembly is serialized, an automatic serial copy of this component occurs within each in-place subassembly.

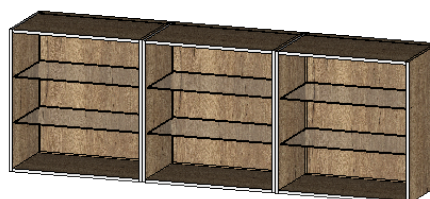


Here, the cabinet component has been copied and distributed to the correct in-place subassemblies.

We will include a shelf component in each cabinet.



The shelf component is positioned and its six publishings connect to the six publishings of the cabinet.

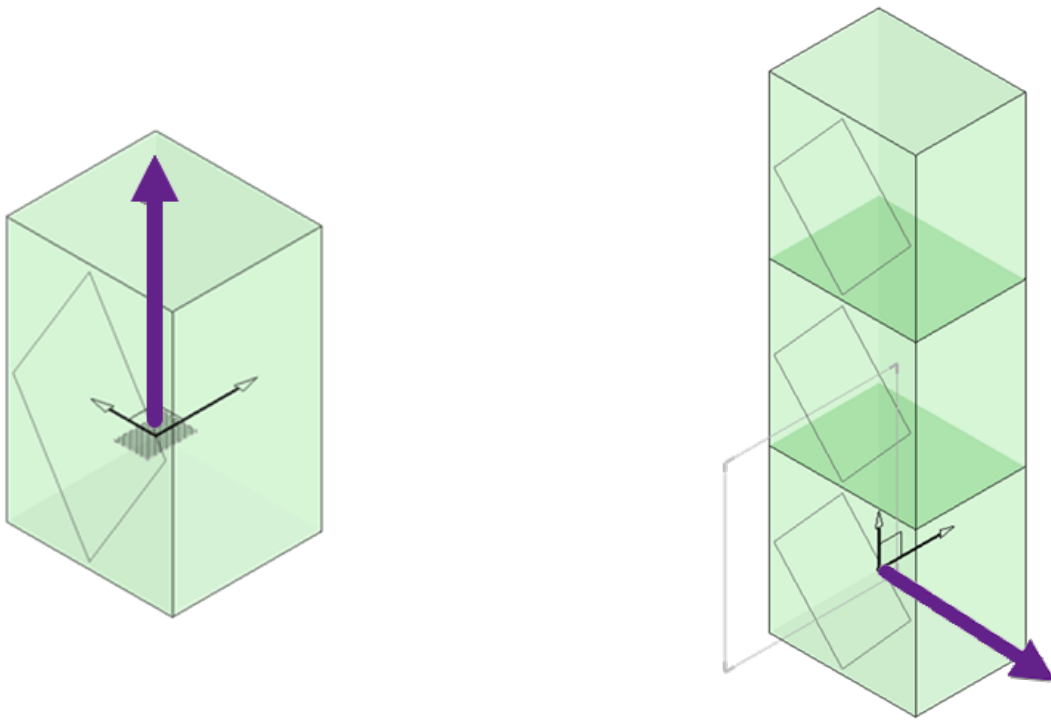


In this precise case, **TopSolid** will be able to serialize the copy of this shelf component and copy/distribute the other shelf components into each in-place subassembly.

## Cartesian serial copy

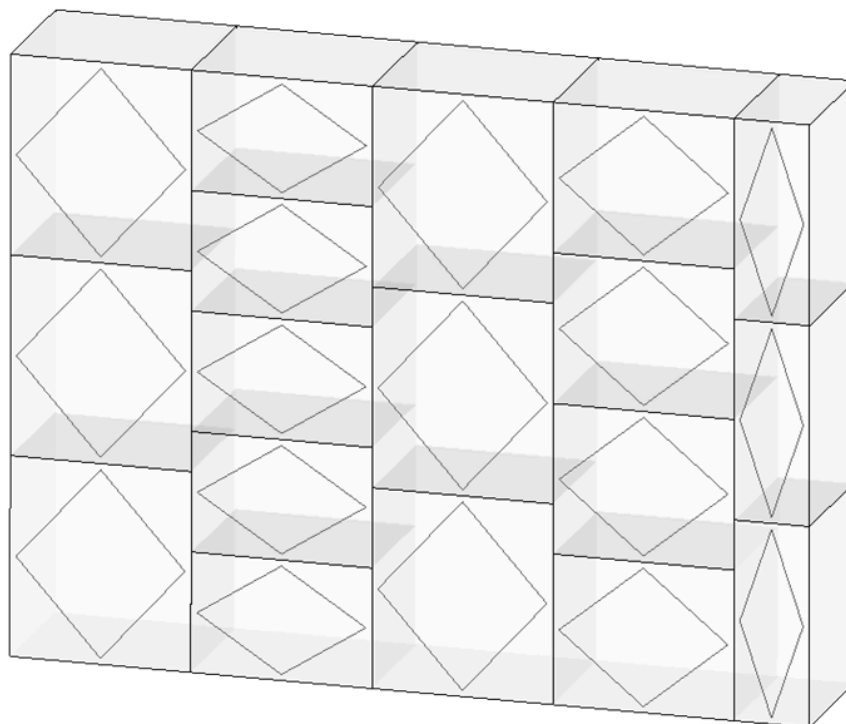
Let's take the example of a vertically distributable component (on the left below). This component is included and distributed vertically in alternate mode (on the right below). The distribution is defined as a driver.

We declare a horizontal distribution of this block of vertical volumes (on the right below).



On a new document, we insert this vertical unit, then distribute it horizontally.

Modifying certain elements generates the following example.

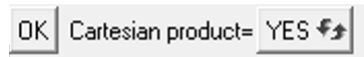


The driver block is included in one of the boxes, then the serial copy is launched.



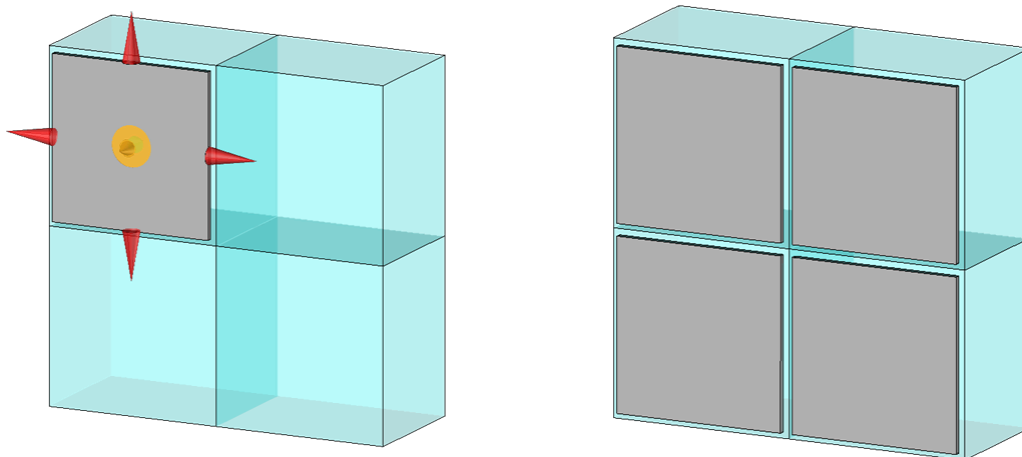
Depending on the selected volume, the components are distributed to the corresponding volumes.

In this scenario, the new **Cartesian product** option proposes the cartesian product of the serial copy.

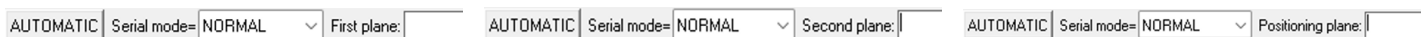


### Serial copy of constrained block

When creating a constrained block automatically or by face extrusion on one of the faces of a distributed or repeated volume, it is now possible to serialize the constrained block operation.



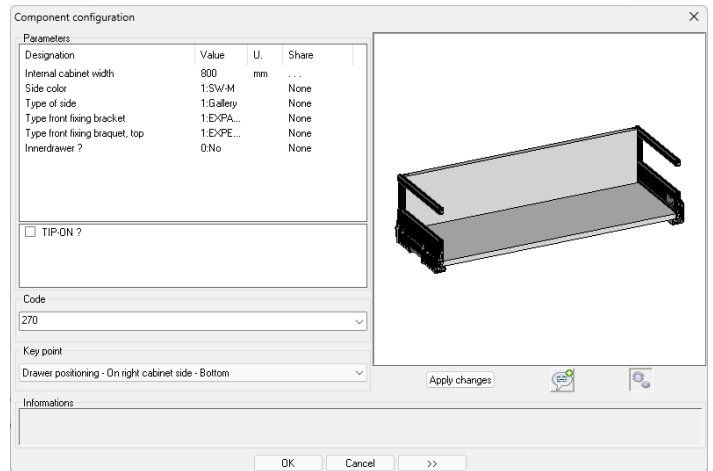
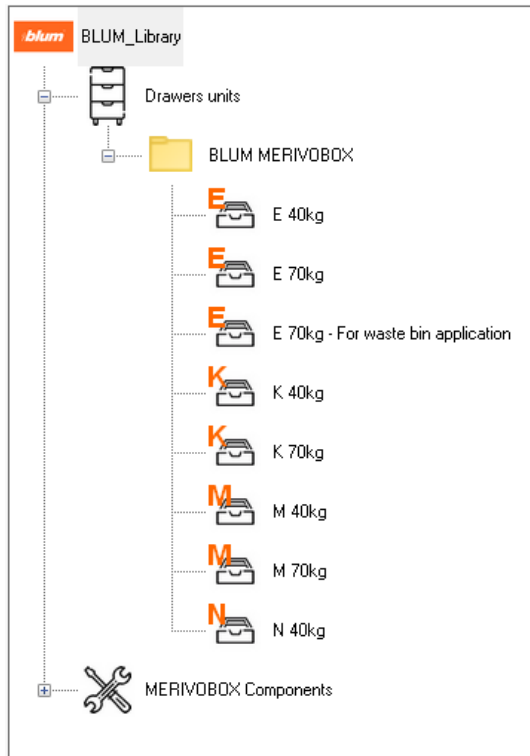
After clicking on the **Serial copy** button, you can define the desired positioning for the copied constrained block.





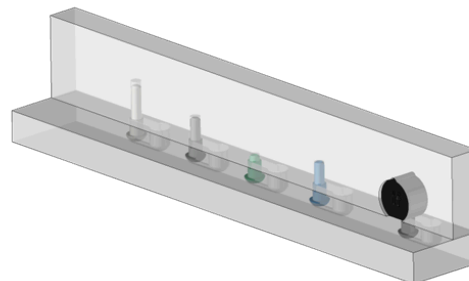
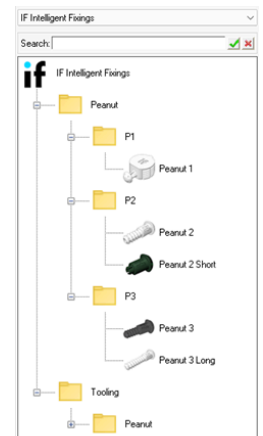
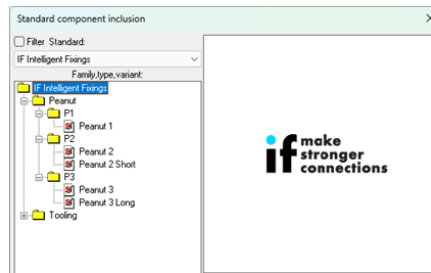
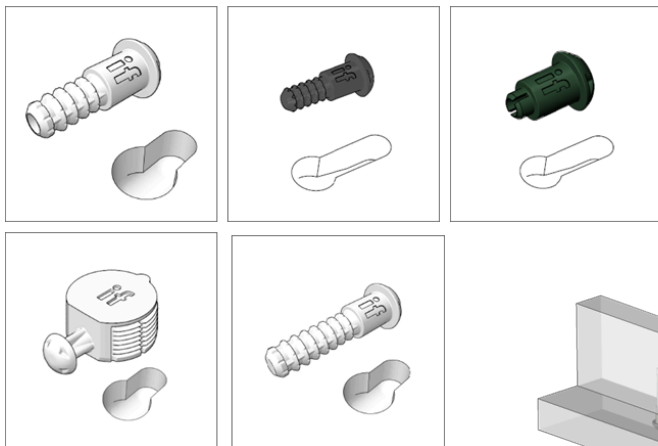
## New Blum Merivobox library in TopSolid'Store

The new Blum Merivobox library is available on [TopSolid'Store](#).



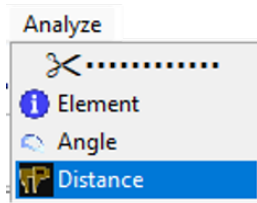
## New IF Intelligent Fixings library in TopSolid'Store

The new IF Intelligent Fixings library is available on [TopSolid'Store](#).



## Distance measures

In the **Analyze > Distance** command, the state of the **Delete previous measures** button is now saved for the session.

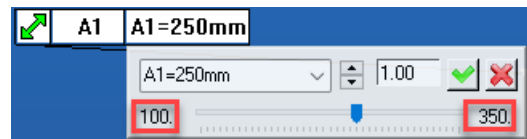
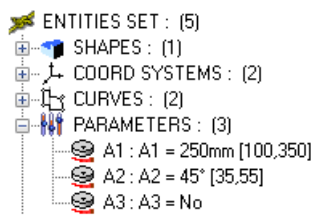


## Value labels

In previous versions, when modifying the numerical value of a parameter, **TopSolid** displayed a label in the graphics area, allowing the value to be edited via a field or cursor. Several improvements have been made.

### Management of min/max values

The min/max values of a parameter are now taken into account.

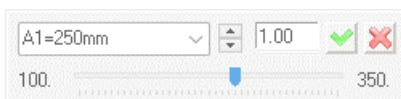


## Transparency management

You can now define the default value for label transparency in the top.cfg file:

```
D_DOC_PARAM_EDIT_INPLACE_TRANSPARENCY 0
```

The number after the configuration word determines the label's transparency (0 = no transparency).



Transparency: 5



Transparency: 10

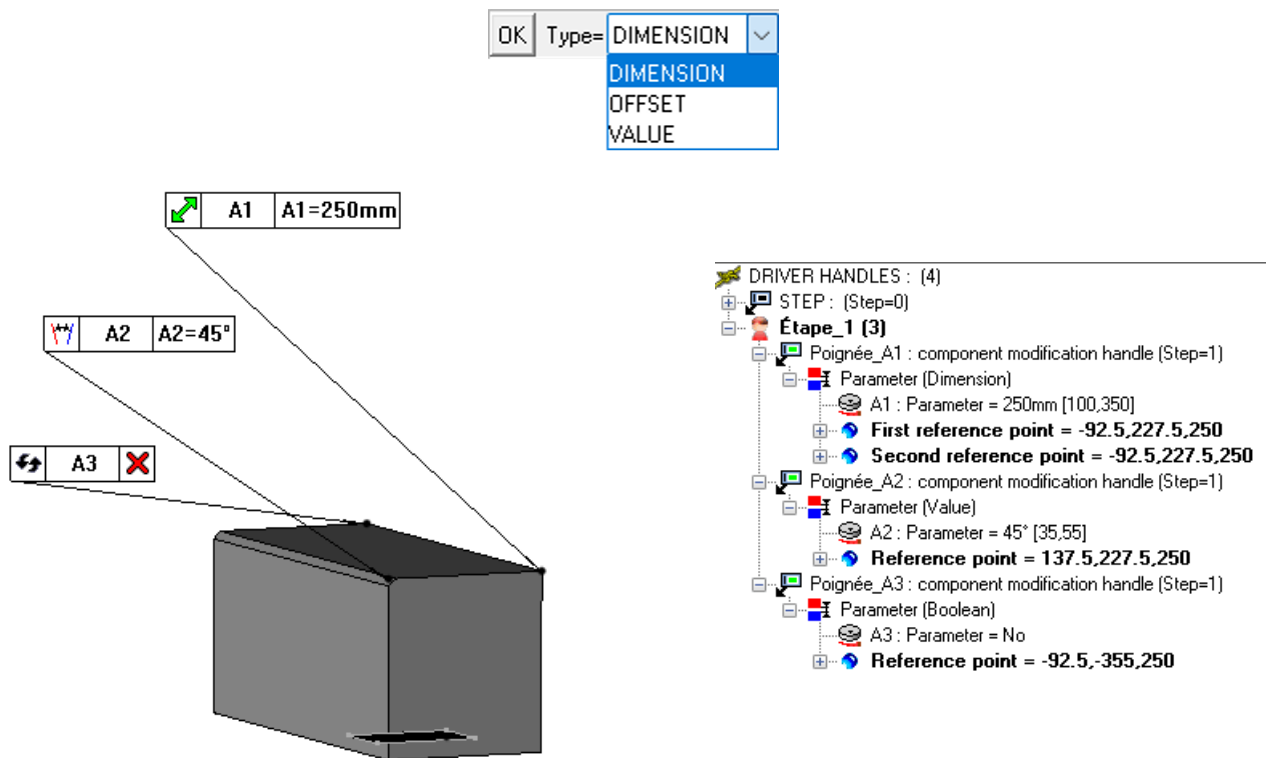
## Handles (parameters, components and drivers)

Improvements have been made to enhance certain concepts, such as parameters, components and drivers by using handles. They make it easier to modify parameters when modifying drivers of the included component.

Creating a handle generates a set of handles for managing the next evolutions.

## Parameters

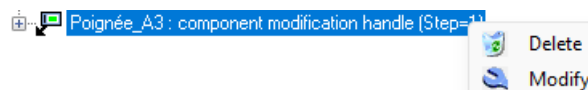
It is now possible to create a handle via the context menu by right-clicking on a parameter and selecting the **Add driving handle** command.



When creating a handle, it is necessary to name it and enter a number and description for the step.

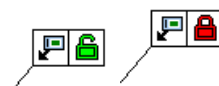


To modify the handle, right-click on the handle line in the handle set and select the **Modify** command.



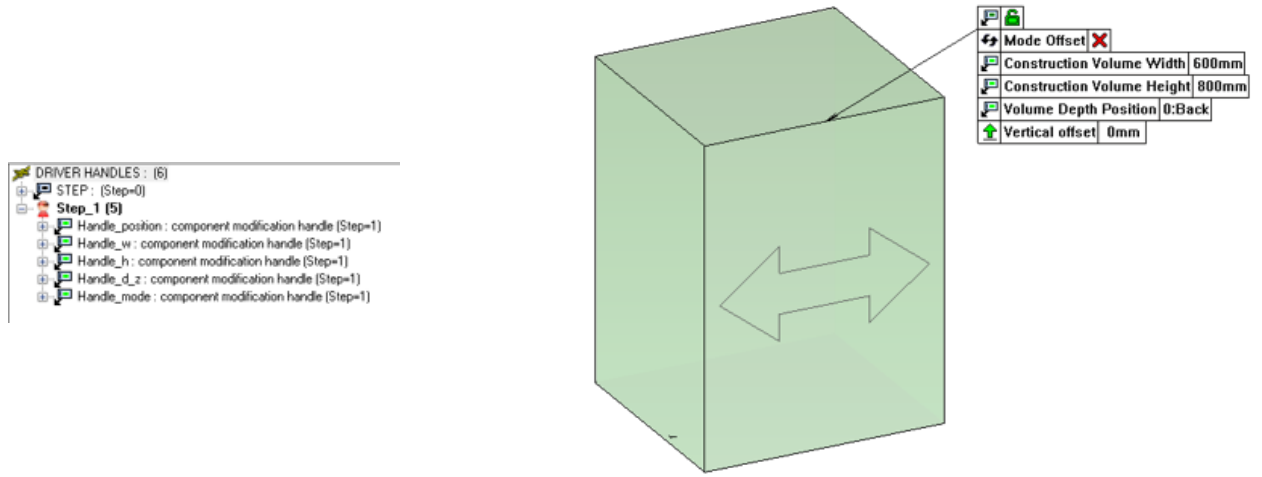
## Update

A new lock icon allows you to delay the calculation of the set of values for a handle.

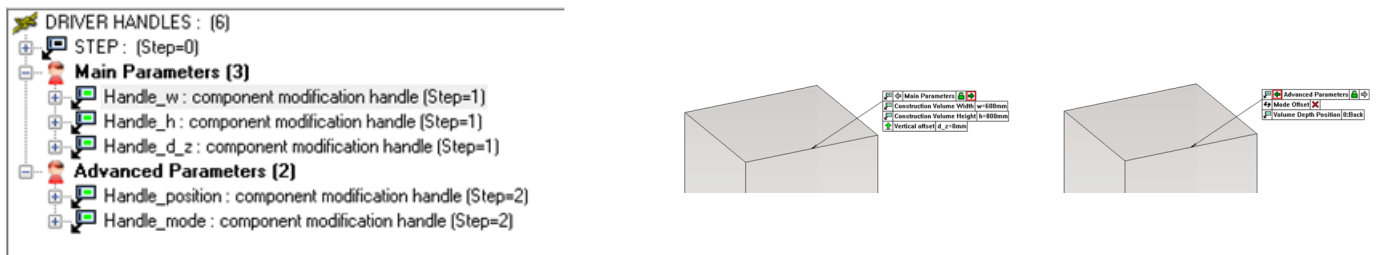


## Step management

If you wish to display a large number of handles, it may be wise to organize them in steps.



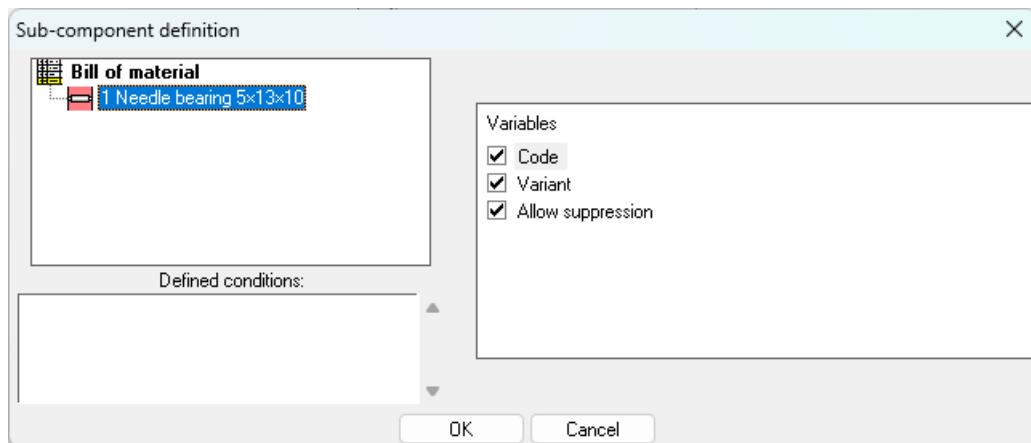
Example of handles grouped in the same step.



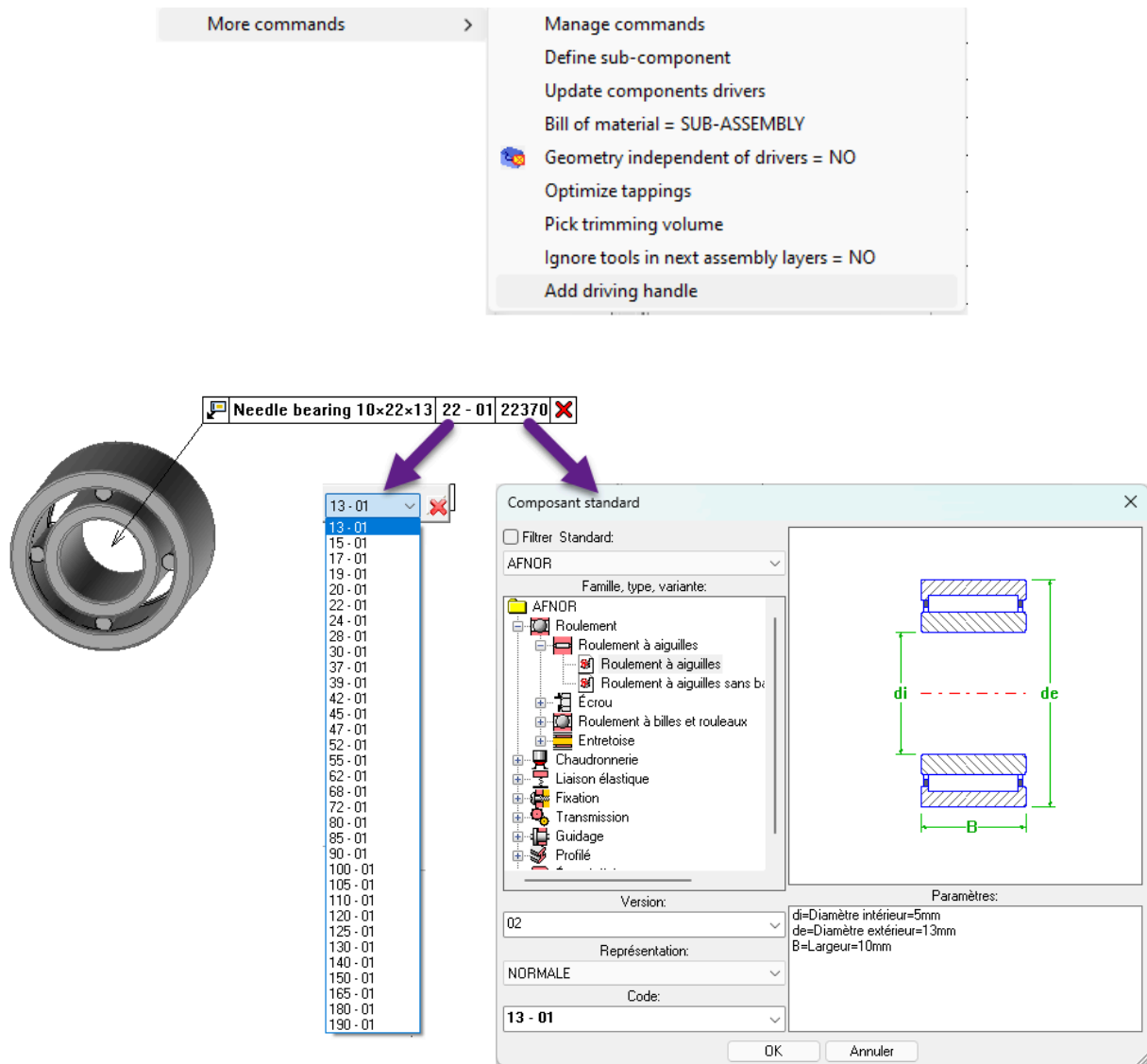
Example of handles organized in two steps.

## Components

A standard component defined as a subcomponent is used to create a component handle.

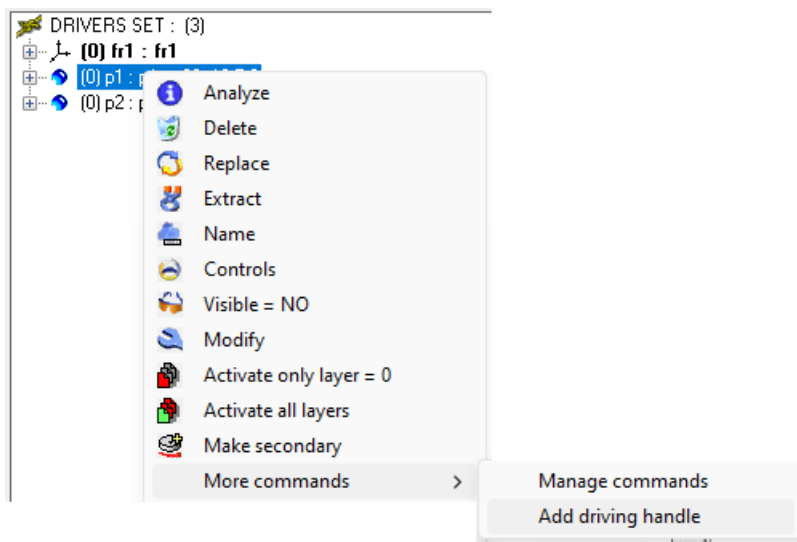


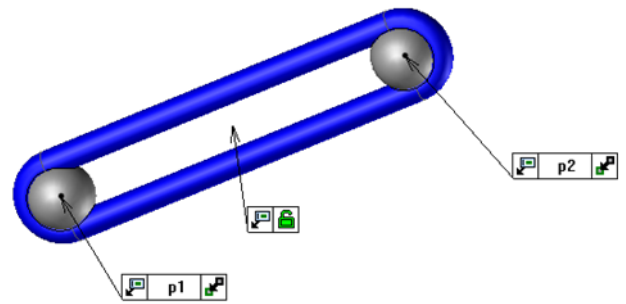
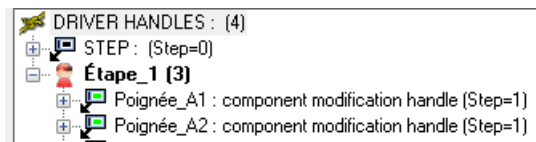
Simply right-click on the subcomponent in the main tree and select the **Add driving handle** context command. Depending on the subcomponent variables, the handle displays more or less options: interchanging the variant, catalog code choice and deletion.



### Geometric point drivers

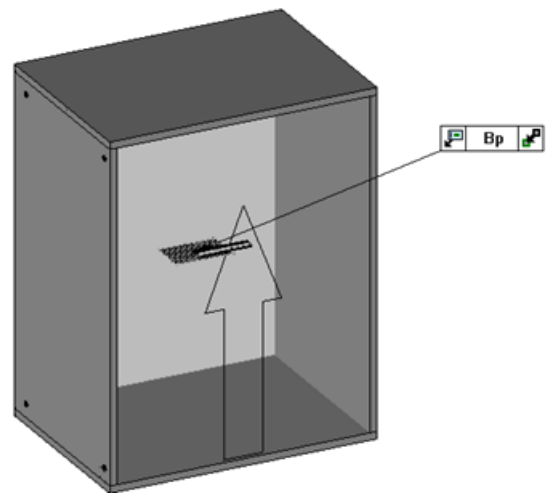
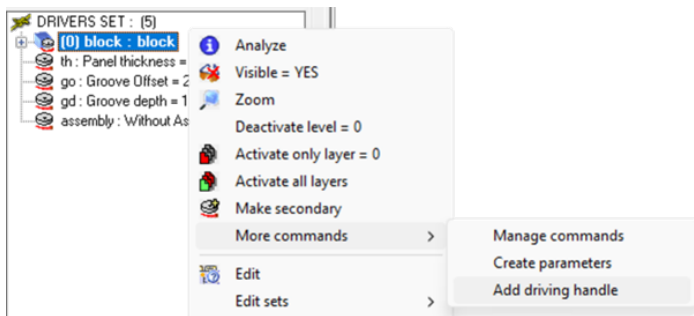
Geometric point drivers are eligible for handle creation, which simplifies the modification of geometric drivers when the component is included.





## Driver block

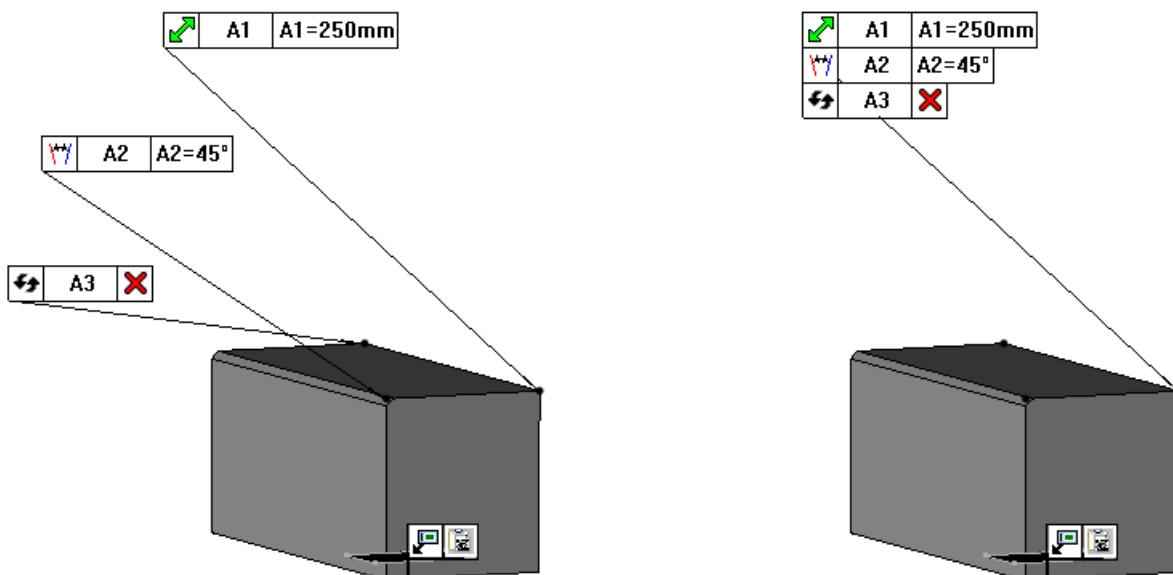
Driver blocks are now eligible for handle creation. The handle of a driver block gives access to the six arrows of the driver block.



## Grouping handles in the graphics area

It is possible to group and order the various handles created within a document.

Drag and drop the desired handle to hook it onto the others. Releasing the handle near another handle will attach the latter to the adjacent handle.

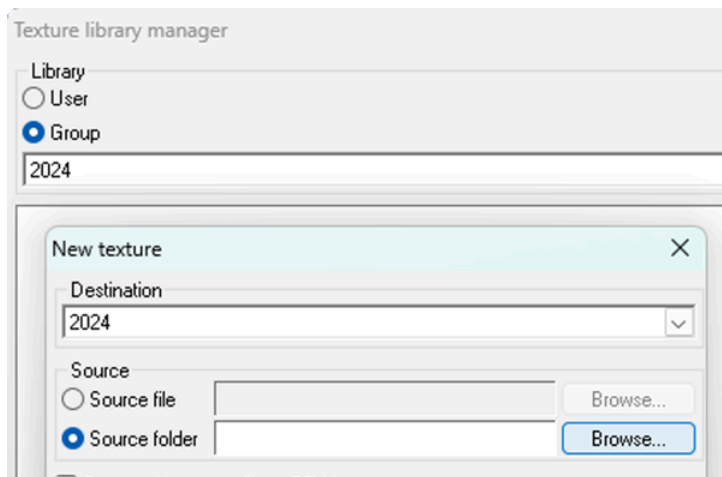


Once the handles have been grouped, the first handle is used to move the batch of handles in a single block.

You can ungroup one or more handles to generate several handle groups.

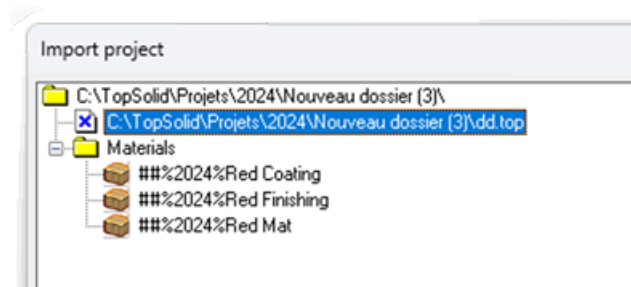
## Simplified texture import

The import path for a texture folder is now memorized, making future texture reimports quick and easy.



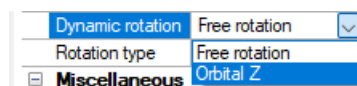
## Import/Export of project finishings and coatings

It is now possible to import or export a project's finishings and coatings.



## Dynamic rotation

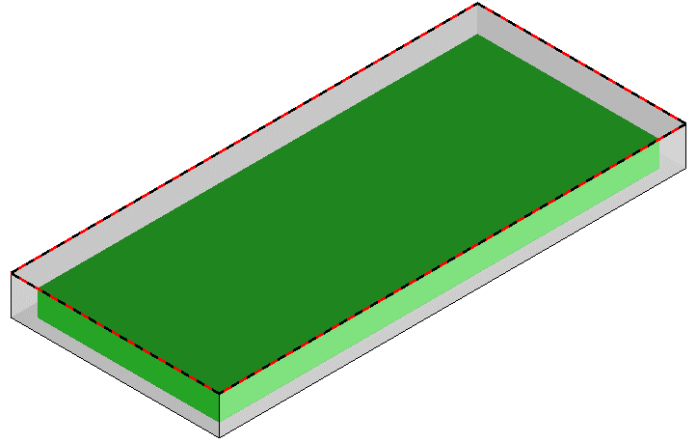
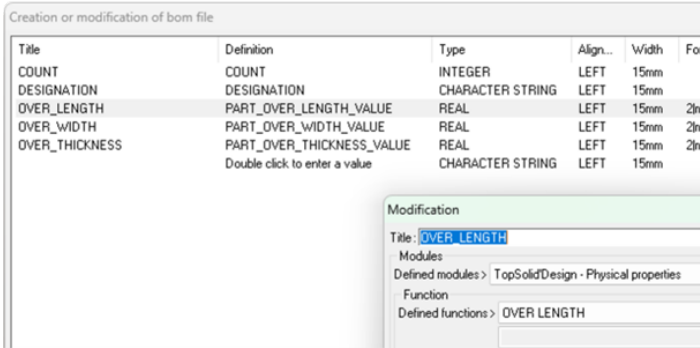
The dynamic rotation mode selected is now saved for the session.



## Drafting

### Part overdimensions

The new **Overlength**, **Overwidth** and **Overthickness** BOM properties can be used to retrieve the value of the overdimension in X, Y, Z. These are available in the BOM file editor of the **TopSolid'Design - Physical properties** module.



1	Part	182.5	112.5	35
COUNT	DESIGNATION	STOCK_LENGTH	STOCK_WIDTH	STOCK_THICKNESS

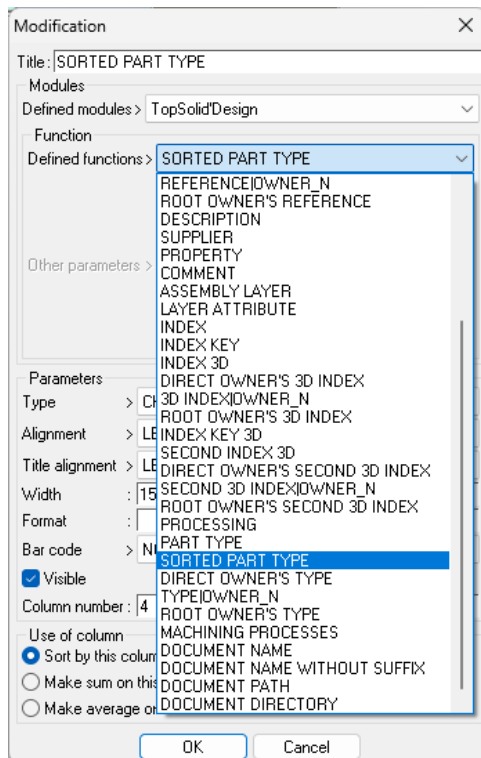
  

1	Part	20	10	5
COUNT	DESIGNATION	OVER_LENGTH	OVER_WIDTH	OVER_THICKNESS



## Alphabetical sorting for part type

The new **Sorted part type** BOM property sorts part types alphabetically.



1	Part2	A / B / C	A / C / B
1	Part1	A / B / D	D / B / A
NOMBRE_TOTAL	DÉSIGNATION	SORTED_PART_TYPE	PART_TYPE

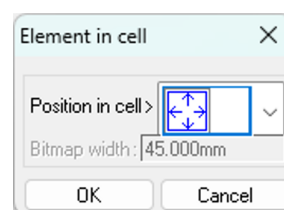
**Important:** If the part type name contains `<space>/<space>`, it will be misinterpreted and split in two.

## QR code in the title block

It is possible to add a **QR code** to the title block of a drafting document. This can be calculated from:

- a manual text or drawing property;
- a BOM property defined in the multi-drawings template.

Go to the title block and select the **Insert** command in the desired cell. Click on the **QR Code** button, select the source to convert in QR code, then validate its position in the cell.



The QR code is placed in the corresponding cell on the title block.

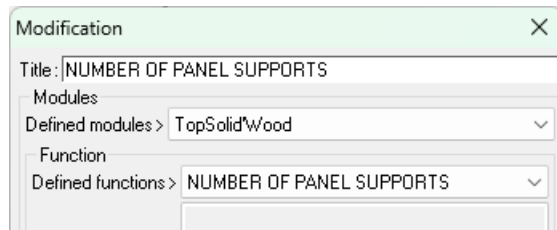
**Note:** In the case of multiple drawings, the QR code matches the property calculated by the bill of materials.



**Important:** Some characters are not usable in QR code conversion.

### Number of panel supports

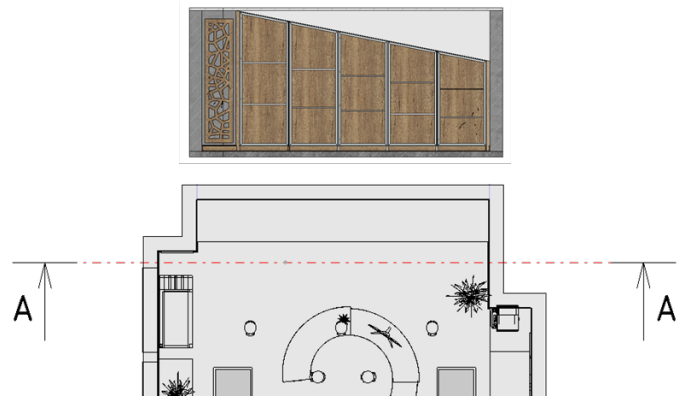
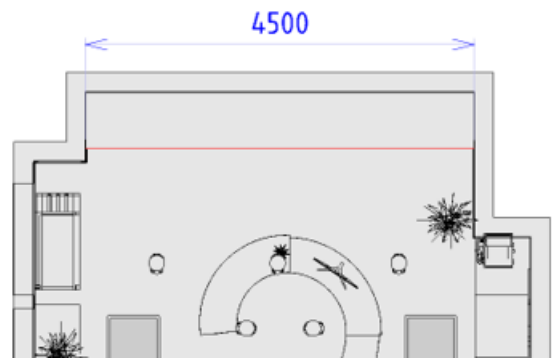
The number of supports for a panel entity is now available in the TopSolid'Wood > Number of panel supports module.



1	Multi Support	3
1	Mono Support	1
COUNT	DESIGNATION	NUMBER_OF_PANEL_SUPPORTS

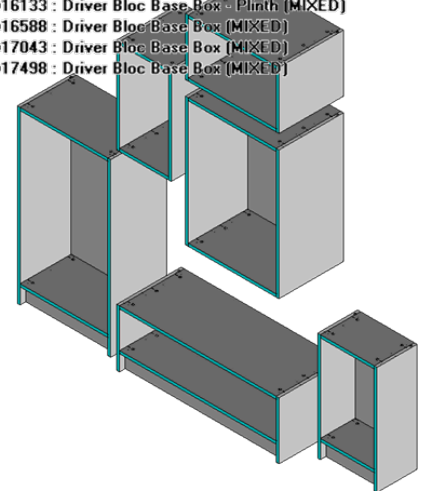
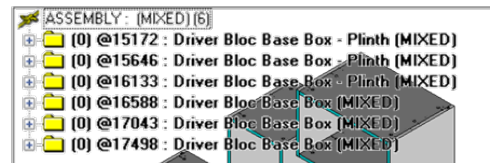
## Fast dimensioning and full section at neutral mode

The Fast dimension and Full section commands are available at the neutral point.



## Script and multiple drawings

When creating multiple drawings via a script, the main set can now be selected.

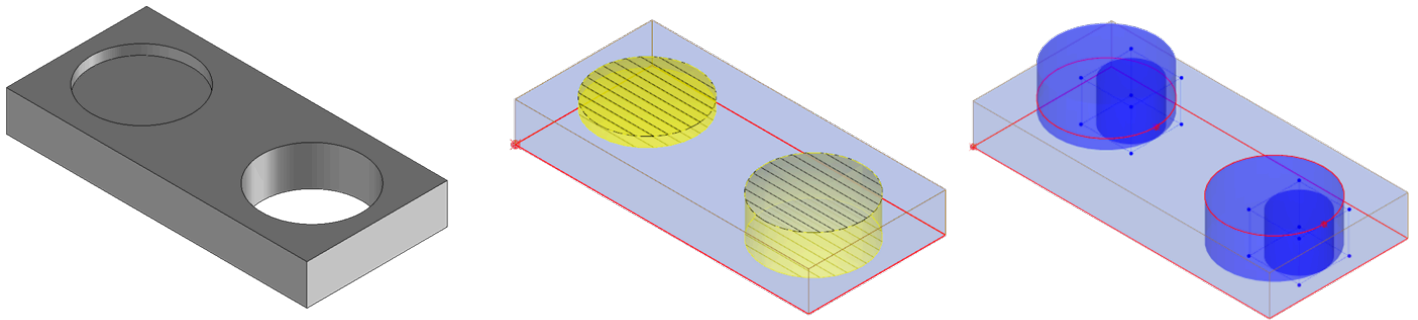
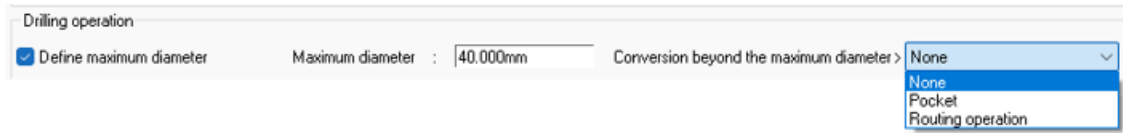


## WoodWOP interface

### Converting a drilling beyond maximum diameter

During a woodWOP export, if a drilling operation exceeds the maximum diameter defined in **Tools > Options**, it is now possible to choose the machining strategy to adopt:

- pocket conversion;
- routing conversion;
- do not export the operation.



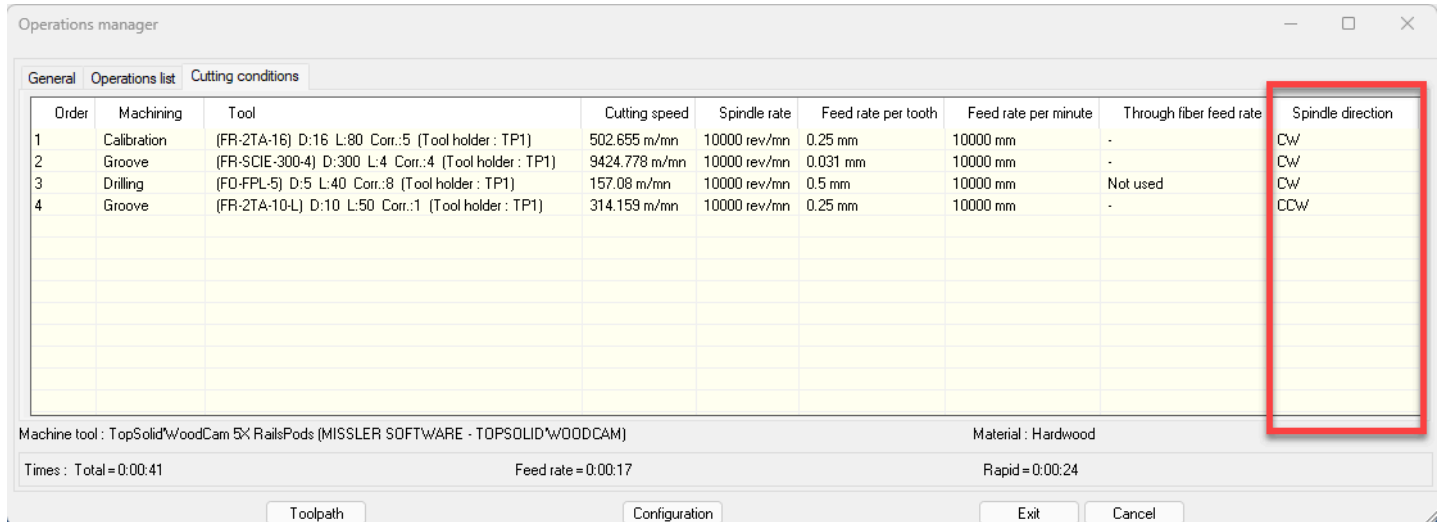
# What's New in TopSolid'WoodCam 6.25

This chapter describes the new features in the version 6.25 of TopSolid'WoodCam.

## Cutting conditions

### Displaying spindle rotation direction

In the **Cutting conditions** tab of the operations manager, the new **Spindle direction** column displays the direction of spindle rotation.



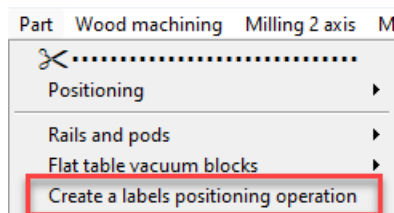
Order	Machining	Tool	Cutting speed	Spindle rate	Feed rate per tooth	Feed rate per minute	Through fiber feed rate	Spindle direction
1	Calibration	(FR-2TA-16) D:16 L:80 Corr.:5 (Tool holder : TP1)	502.655 m/mn	10000 rev/mn	0.25 mm	10000 mm	-	CW
2	Groove	(FR-SCIE-300-4) D:300 L:4 Corr.:4 (Tool holder : TP1)	9424.778 m/mn	10000 rev/mn	0.031 mm	10000 mm	-	CW
3	Drilling	(FO-FPL-5) D:5 L:40 Corr.:8 (Tool holder : TP1)	157.08 m/mn	10000 rev/mn	0.5 mm	10000 mm	Not used	CW
4	Groove	(FR-2TA-10-L) D:10 L:50 Corr.:1 (Tool holder : TP1)	314.159 m/mn	10000 rev/mn	0.25 mm	10000 mm	-	CCW

Machine tool : TopSolid'WoodCam 5X RailsPods (MISSLER SOFTWARE - TOPSOLID'WOODCAM)      Material : Hardwood  
 Times : Total = 0:00:41      Feed rate = 0:00:17      Rapid = 0:00:24

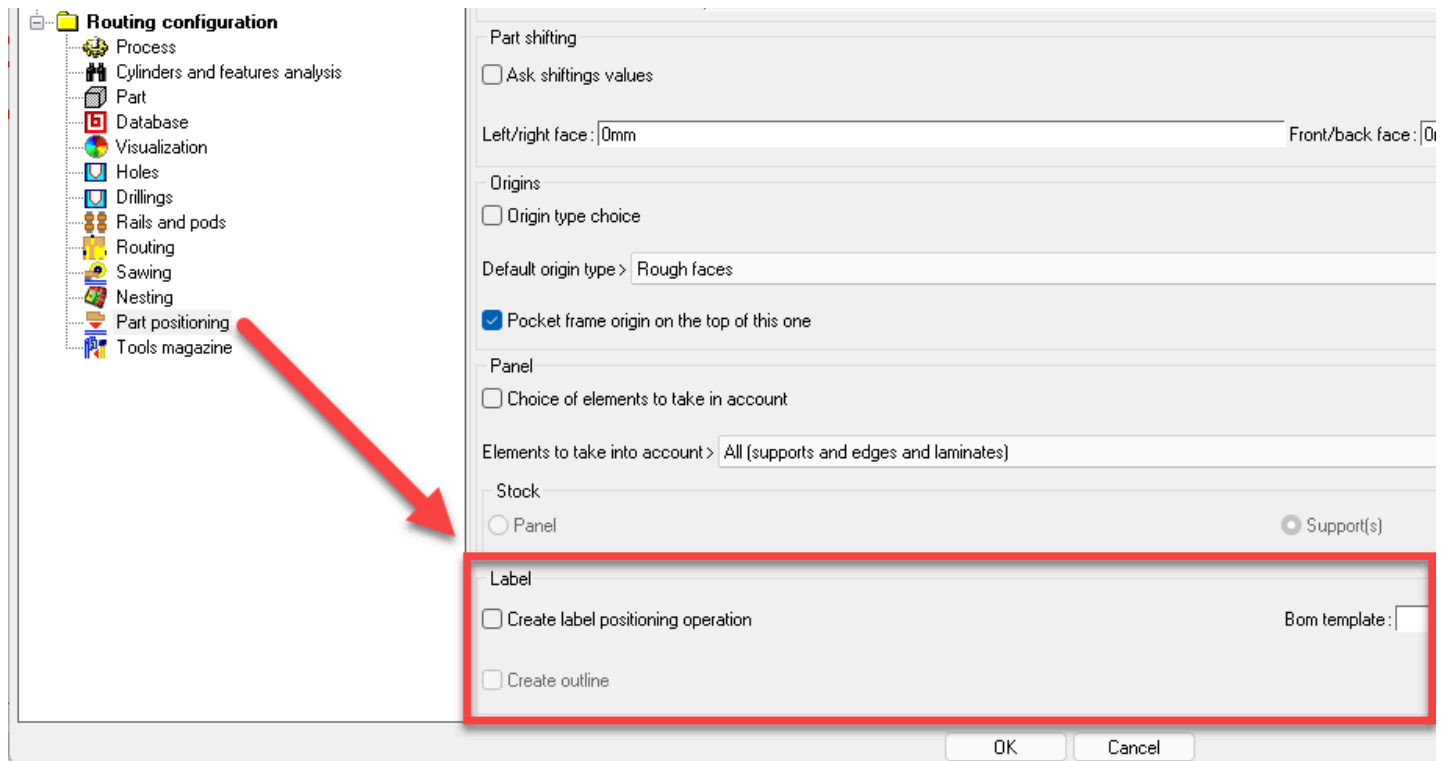
## Label

### Label positioning operation for a single part

You can create a label positioning operation manually from the **Part > Create a label positioning operation** menu if a part is positioned on the machine.



It is also possible to create the label positioning operation automatically when positioning a part by configuring the options in the **Label** section in the part positioning properties. The BOM template is used to define the information contained in the label that corresponds to the columns in the operation dialog box.



By default, the label is positioned at center and on top of the part's enclosing box. This position can be modified in the X, Y, Z columns of the operation dialog box. To apply the modification, validation of this dialog box is required.

The text on the label corresponds to the part designation which becomes visible upon activating the layer (900).

The **Text** column can also be edited from the operation dialog box. To apply the modification, you must validate this dialog box.

You can manually generate the part outline in the label operation, or if the **Create outline** option is activated in the document properties.




The label parameters and outline path are described in the **Operation details** window and are generated in PDB as for nesting.

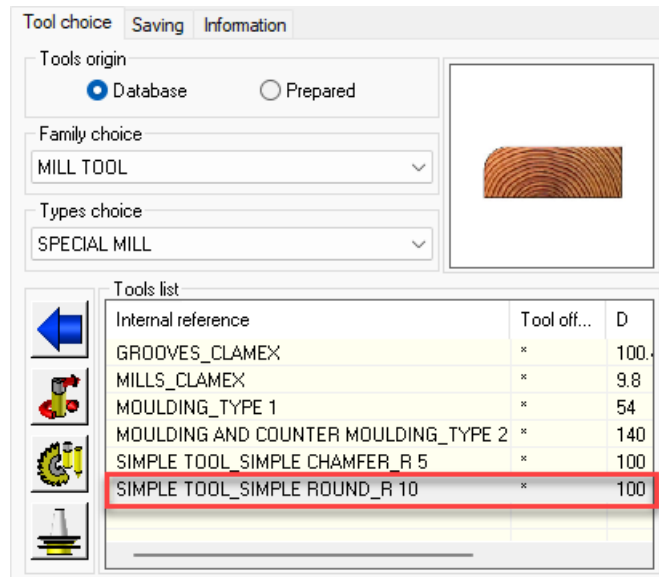
**Important:** To benefit from this new feature, we need to adapt PP in the specifications according to the machine to be driven.

## Tools

### Easy mounting for moulding tools

When creating a new moulding tool via the  icon, the tool is automatically selected from the tool list for quick mounting in the tool magazine.

In addition, the list of special tools is now sorted by ascending creation date rather than ascending diameter, to keep the last tool created at the end of the list.



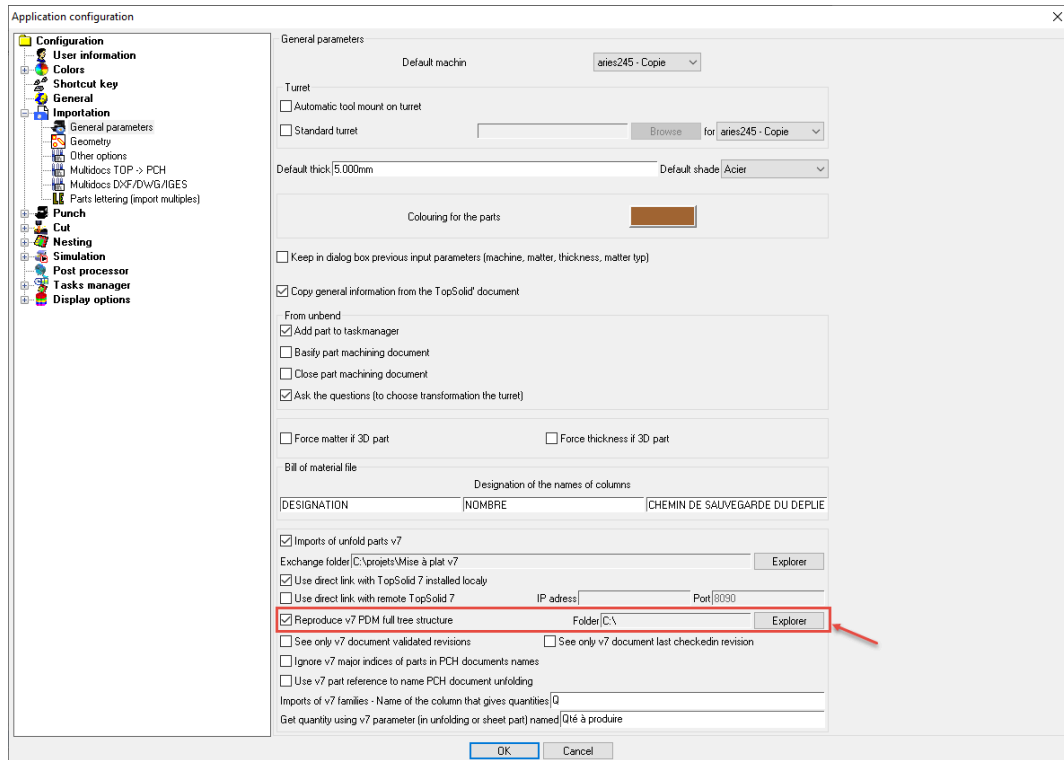
# What's New in TopSolid'Sheetmetal 6.25

This chapter describes the new features in the version 6.25 of TopSolid'Sheetmetal.

## V7 import

### Reproducing the PDM V7 tree structure

When importing parts from TopSolid 7, if the new option **Reproduce v7 PDM full tree structure** available in **Tools > Options > Importation > General parameters** is checked, the destination folder is no longer requested in the import dialog box, and PCH files are stored in the same tree structure as the PDM.

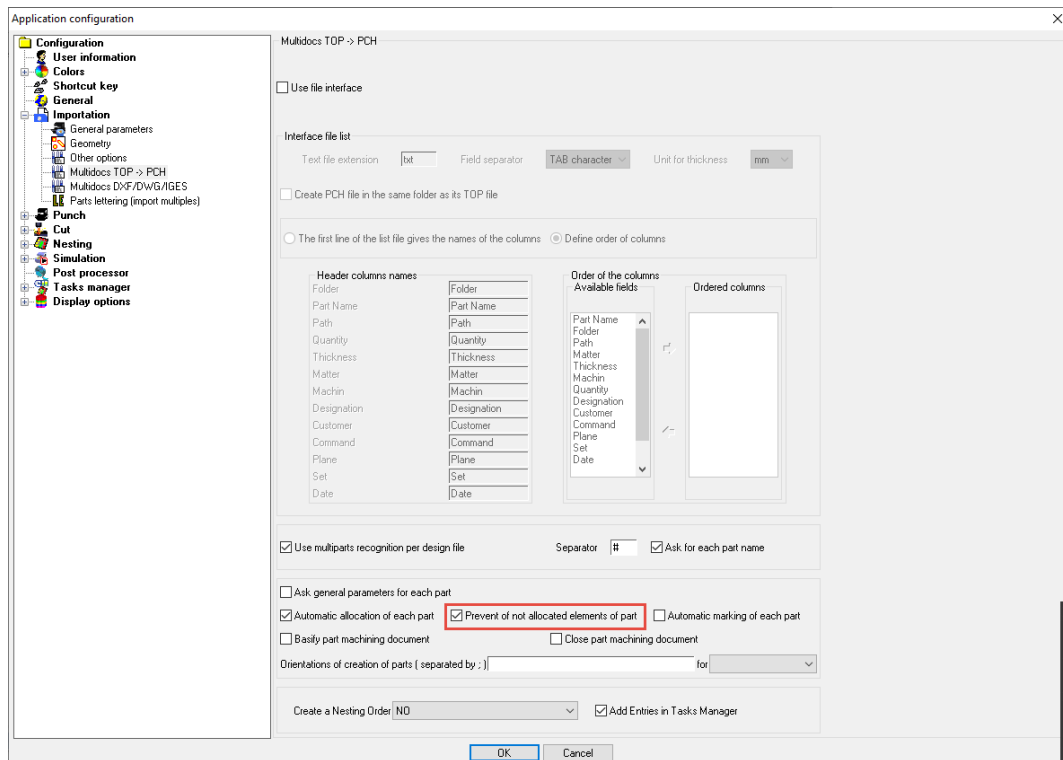




## Multi-document import

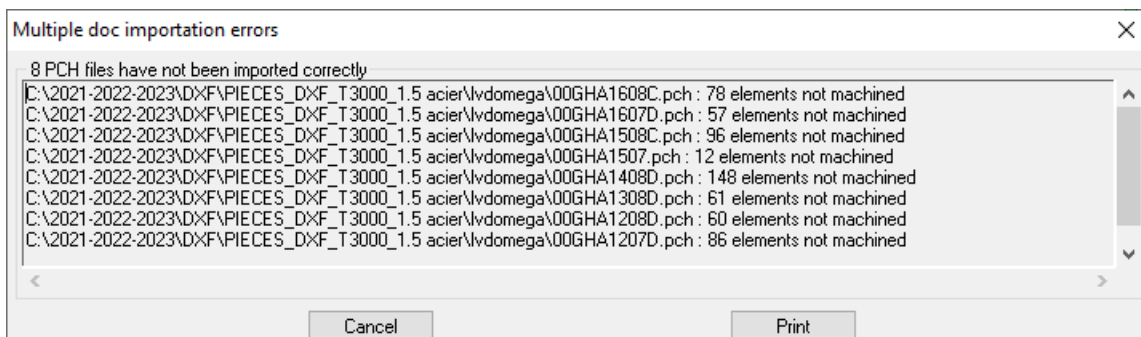
### Verification of unmachined elements

During TOP > PCH multidocs import, the **Prevent of not allocated elements of part** option, available in **Tools > Options > Importation > Multidocs TOP -> PCH** allows you to check on each part if any elements could not be machined by the automatic allocation operation.



**Note:** This option is also available for importing DXF/DWG/IGES files.

During importing, if elements of a part could not be machined, the name(s) of the file(s) concerned, as well as the number of unmachined elements, are displayed in the error report dialog box. This information is also visible in the alpha bar.



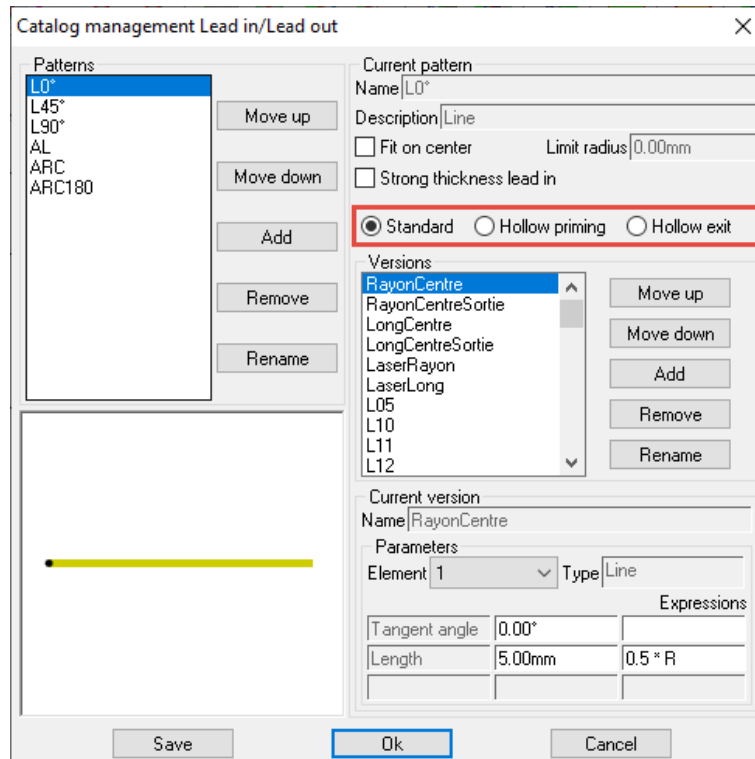
## Cutting

### Hollow lead management

#### Lead-in/out catalog

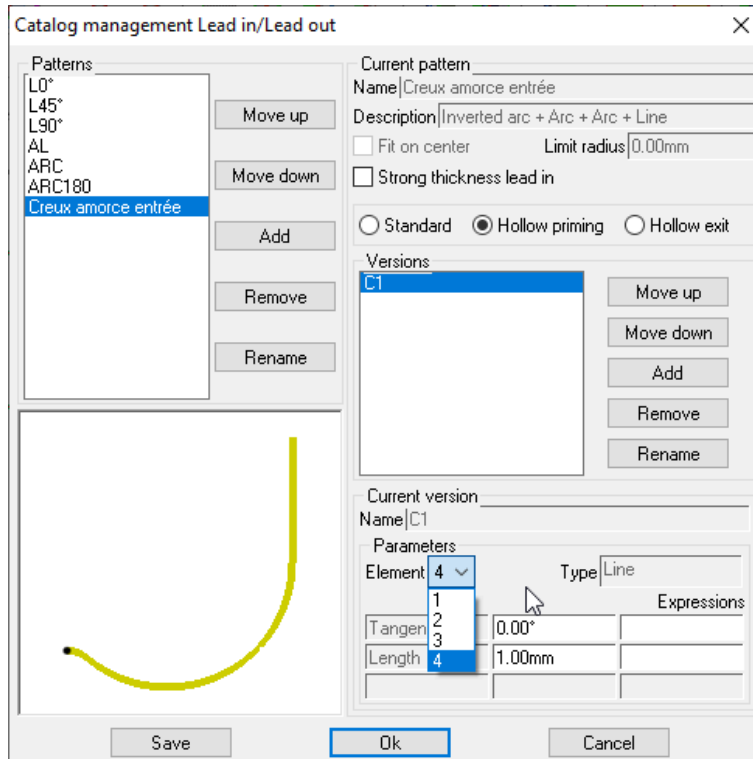
Hollow lead-ins have the advantage of penetrating slightly into the matter, which avoids low roughness in the laser-generated aperture. It is possible to add a lead-out that can cover one part of the lead-in.

The lead-in/out catalog now offers three options for defining lead-ins/lead-outs: **Standard** (classic lead-ins/outs), **Hollow priming** and **Hollow exit**.



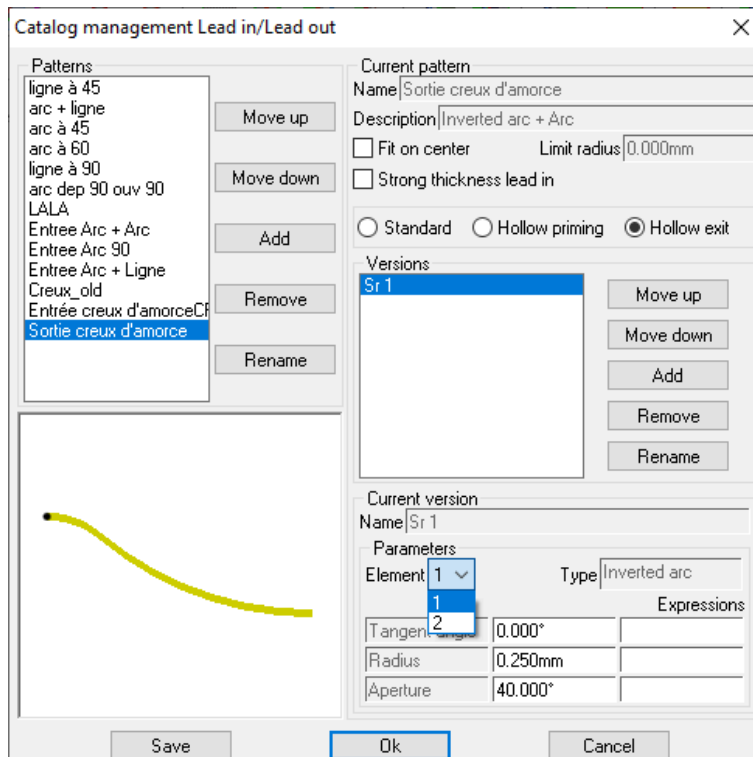
The **Hollow priming** type is composed of four elements:

- Inverted arc
- Arc
- Arc
- Line



The **Hollow exit** type is composed of two elements:

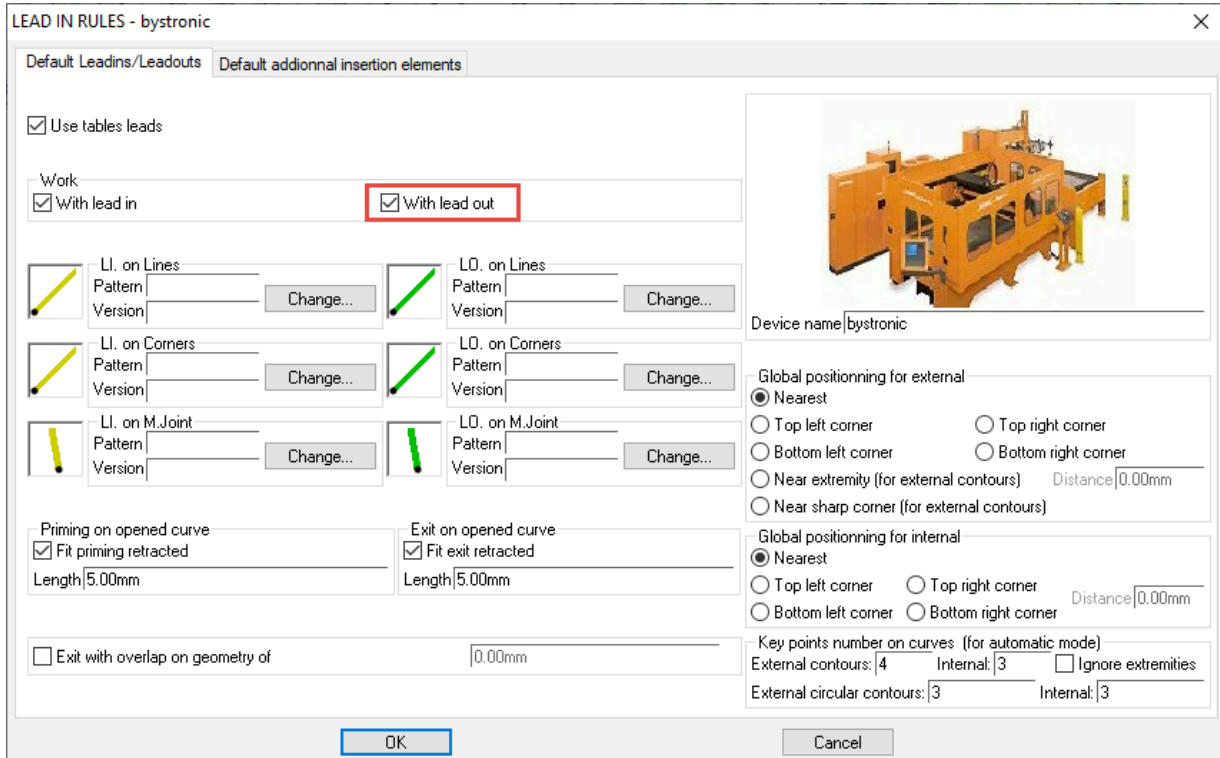
- Inverted arc
- Arc



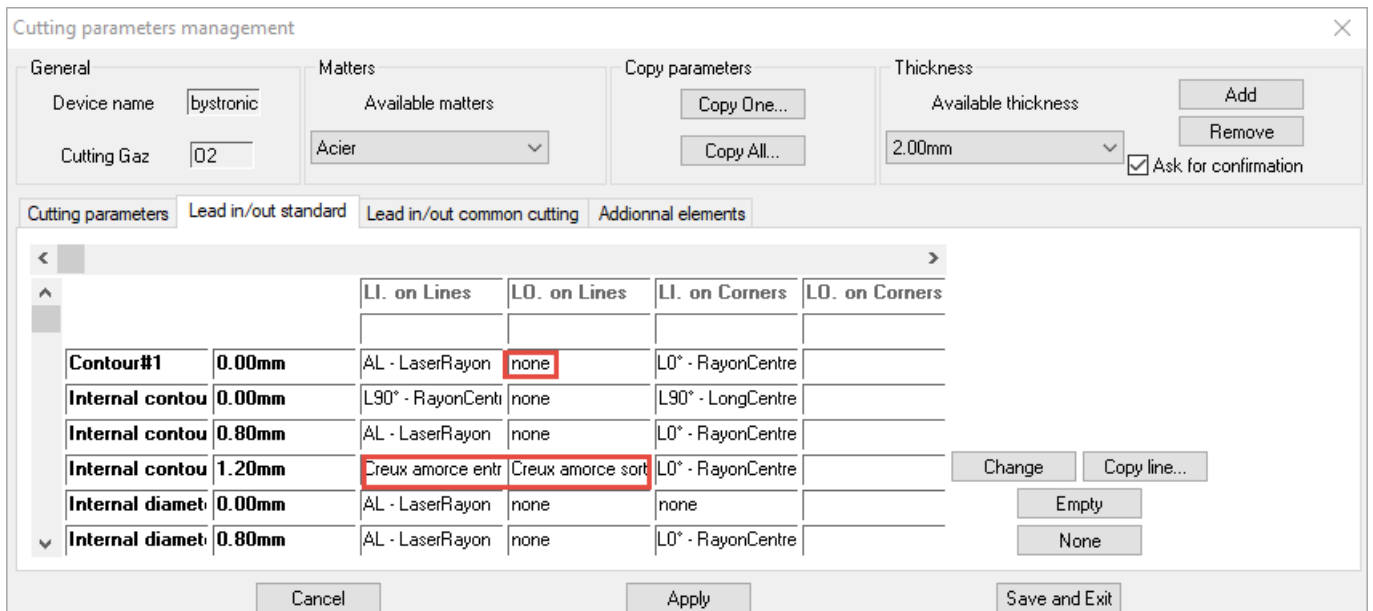
**Note:** In-center priming is not available for hollow leads.

## Priming rules for machine and tables

For the lead-out to cover one portion of the lead-in, you must check the **With lead out** option.



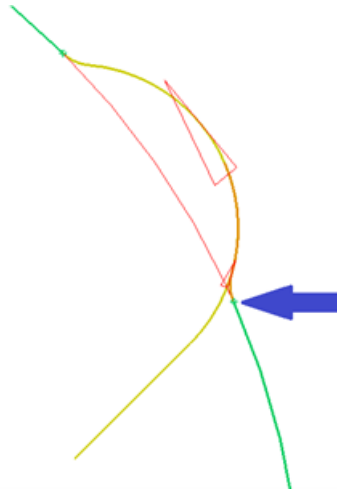
The same applies to tables, where you have to define the geometries on which you want a hollow lead to be present. If you do not wish any hollow lead-in or lead-out on external contours, you must enter "none" in the tables.



## Cutting allocation

Hollow lead-in is only placed if the tool path is closed and does not support any microtab.

A special limitation point is automatically computed to trim the path. It is not possible to move or delete this point with classic commands.

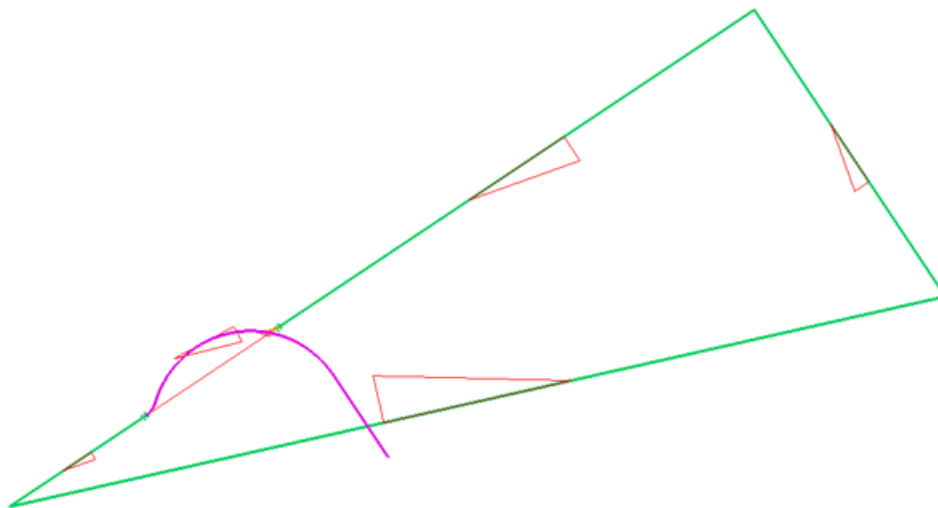


If the lead-out geometry is identical to the first segments of the lead-in geometry, the limitation point is computed to overlap the lead-in, like in the example above. In other cases, the limitation point is computed at the intersection of the lead-out with the support segment.

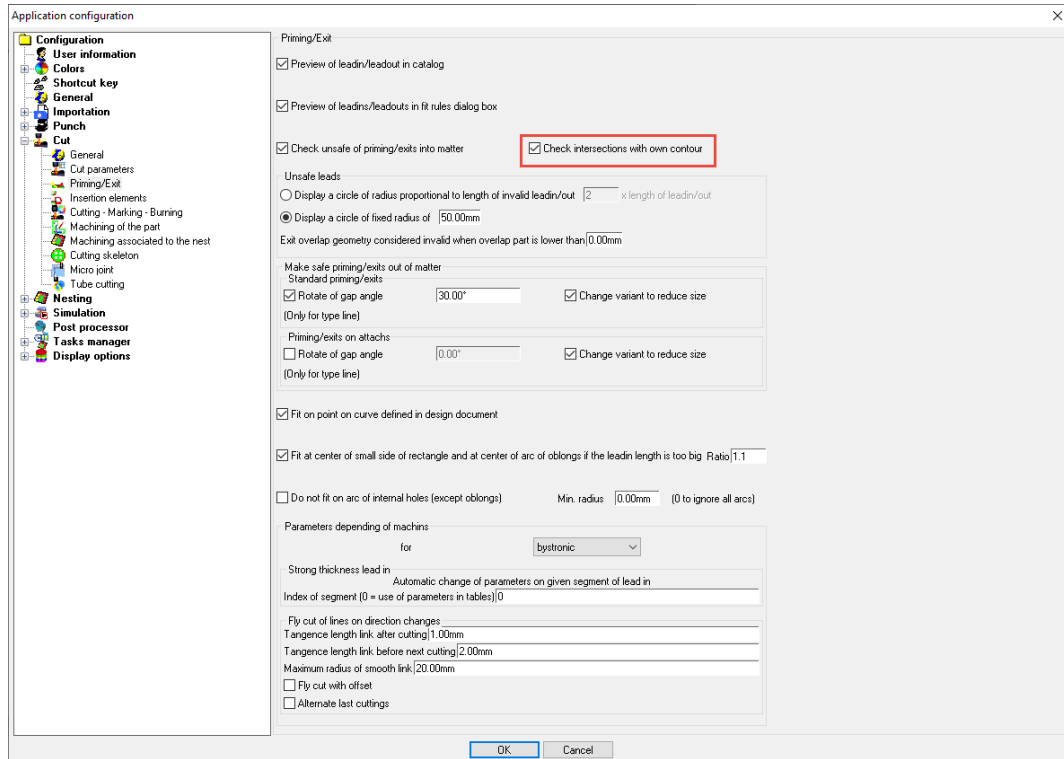
You can use the **Move lead in/out along contour** command to rectify the position of the hollow lead. The hollow lead does not appear during movement; it is created as soon as you click on the lead-in at a valid final position.

## Detection of collisions with matter

Hollow lead-outs are not detected in collision with the matter. Hollow lead-ins are only detected as invalid if the lead-in's extremity point is outside the contour to be machined.



By definition, hollow lead-ins penetrate the matter, which means that the **Check intersections with own contour** option, available in **Tools > Options > Cut > Priming/Exit**, is ineffective for this type of lead-in.



## Lead-in/lead-out replacement

If you replace a standard lead-in with a hollow lead-in, the special limitation point is automatically computed.

Conversely, if you replace a hollow lead-in with a standard lead-in, the special limitation point is automatically removed.

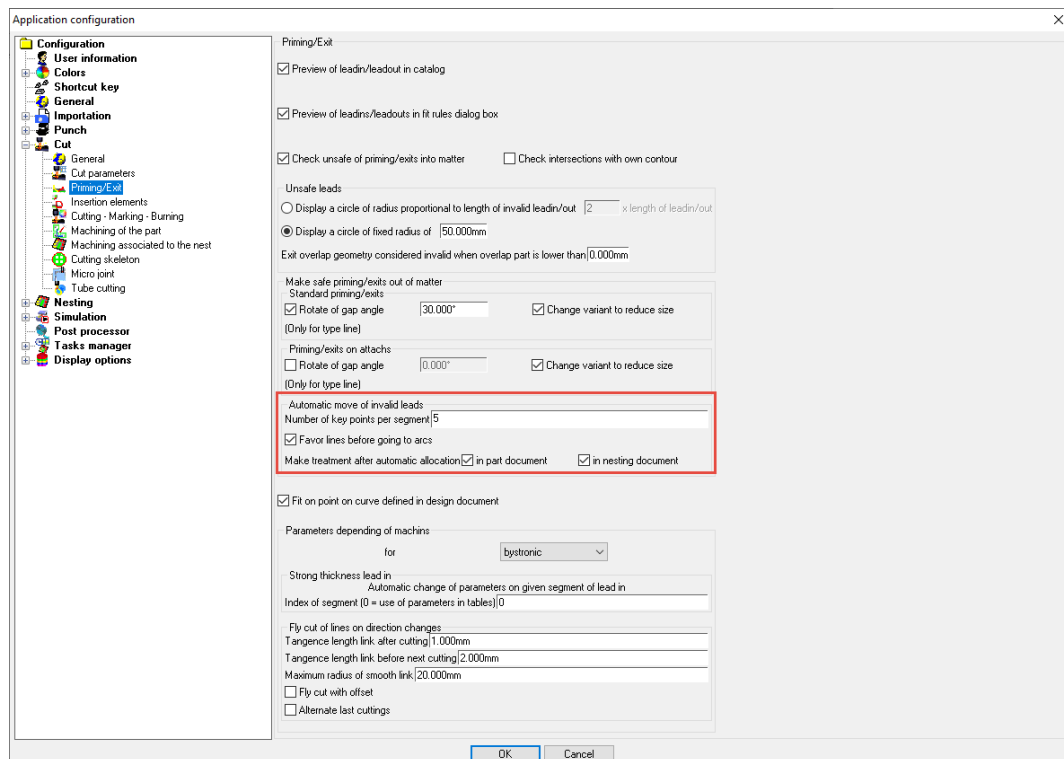
## Automatic movement of invalid lead-ins/lead-outs

New options in **Tools > Options > Cut > Priming/Exit** detect all invalid lead-ins/lead-outs and move them on the contour.

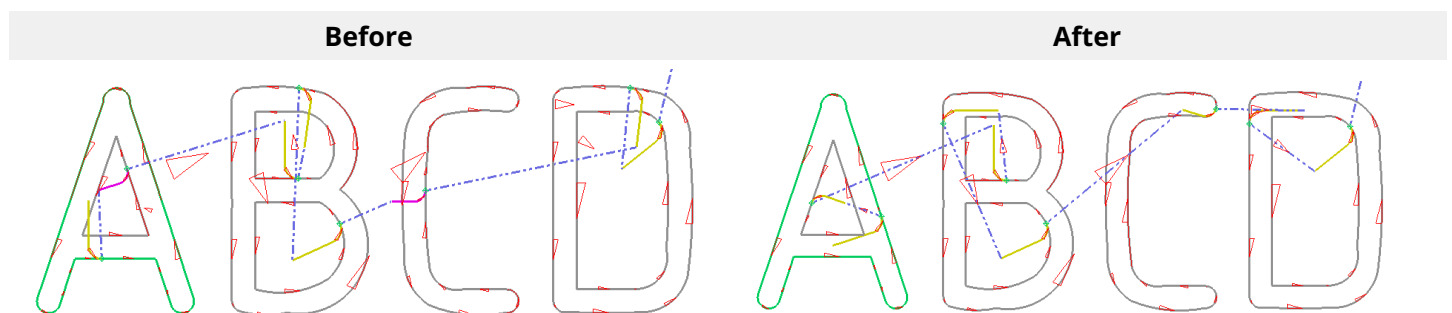
They are moved via distributed key points on each cutting segment. End points of segments are not taken into account. For example, if you indicate a single key point, it will be the middle of the segment.

You can configure the movement so that contour lines are given priority before processing contour arcs. Otherwise, all segment types are processed following the contour.

The treatment can be carried out at the end of automatic machining on a part or nesting document, before the treatment to detect dangerous links.



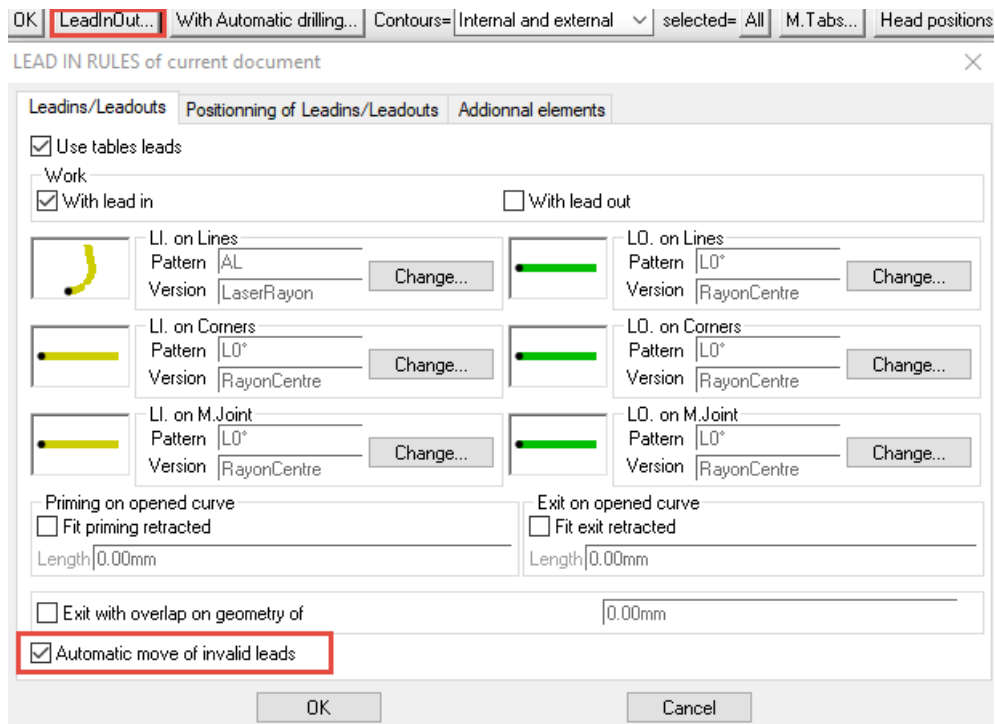
This treatment can be carried out manually in the **Replace unsafe lead in/lead out** command, where you can enter the number of key points per segment and activate the option to give priority to line-type segments, if required. The **Start** button launches the treatment to be applied to the entire document. By default, **TopSolid** positions the lead-in/out so that it does not intersect a contour.



*Example of automatic movement with a safety distance of 0.*

If the safety distance is too great and cannot be respected, **TopSolid** indicates the error in the alpha bar, circles the relevant lead-in(s)/lead-out(s) in brown and suggests moving them manually. Once moved, they are returned to their original color.

Automatic lead-in/lead-out movement treatment can be activated or deactivated via the **LeadInOut** button in the **Automatic cutting and linking** command.

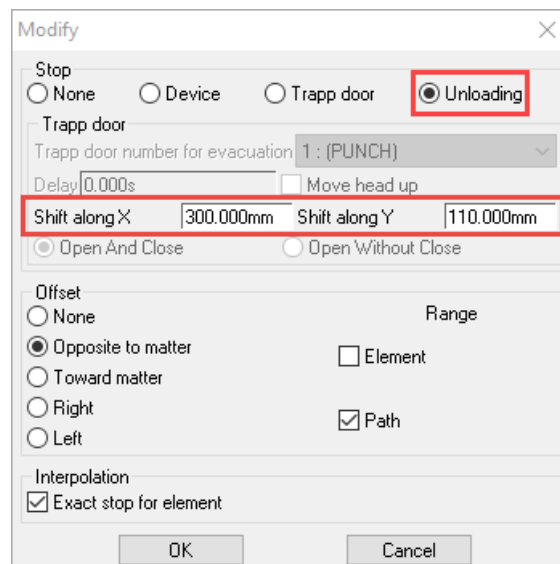


The state of the check box is initialized at document creation with the value defined in **Tools > Options**, and is saved for each document.

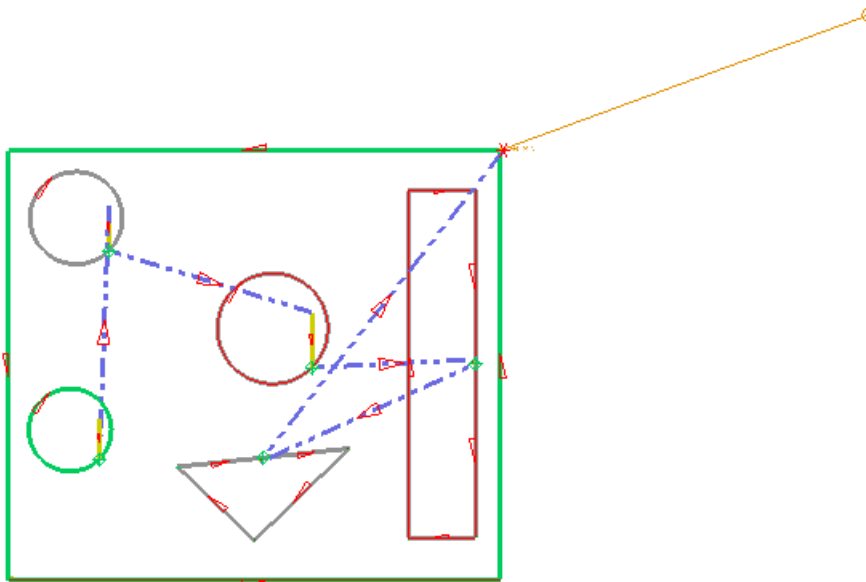


## Additional management for unloader evacuations

On laser machines with unloaders, you can now add an empty movement before unloading after cutting. To do this, use the wrench icon on the cutting allocation, select the **Unloading** option, then enter the desired shift values.

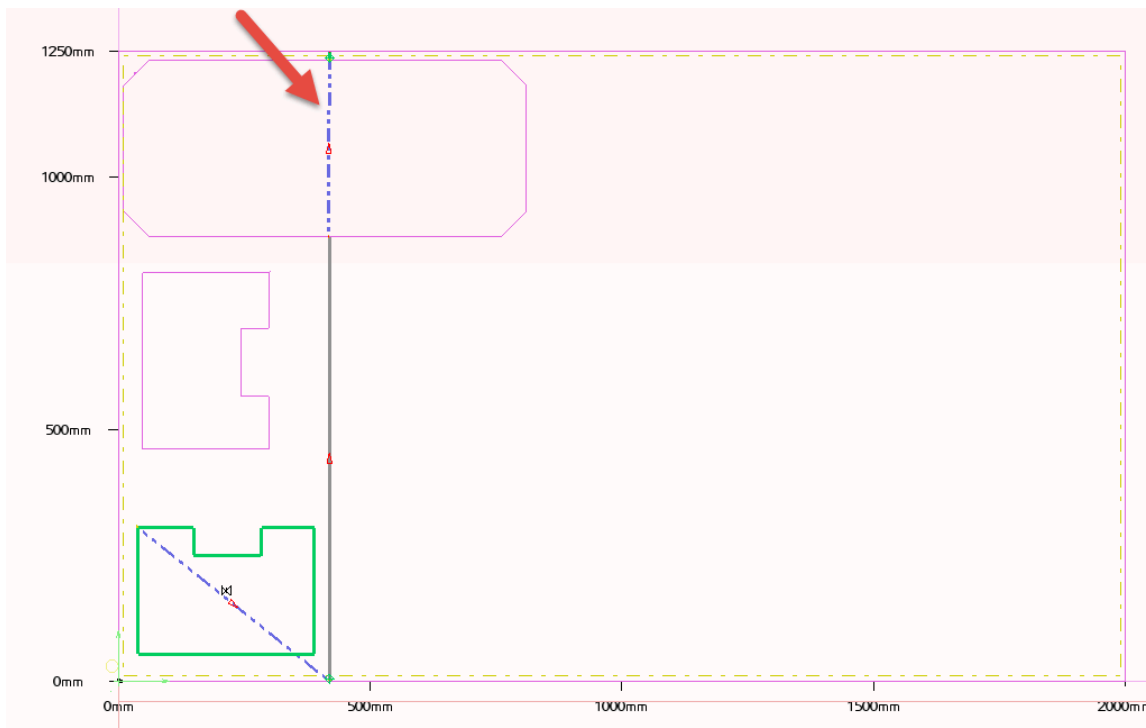


In the graphics area, an orange line shows the movement ending in a small orange circle; it is at this point that the unloader places its origin.



## Holes taken into account in automatic waste cutting

During automatic waste cutting, **TopSolid** now identifies the holes in the sheet and splits the cut into several pieces.



Holes are taken into account on:

- vertical waste cuts
- horizontal waste cuts
- horizontal-vertical waste cuts
- vertical-horizontal waste cuts

In the case of L-shaped waste cutting, if the cutting crosses a hole, **TopSolid** refuses to create the waste and an alert message appears in the alpha bar.

Similarly, in the case of combined horizontal + vertical or vertical + horizontal cuts, it may happen that the end of one of the two cuts (at the intersection level) falls into a hole. **TopSolid** then refuses to create the waste cutting and the alpha bar displays a warning message.

# Punching

## Punching complex areas

In a part or nesting document, the **Punch** menu features the new **Punch area** and **Display external areas not punched** commands.

### Punch area

The **Punch area** command provides a range of options and settings in the question bar of the part document.

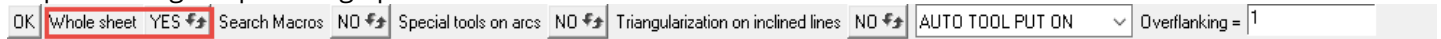


In the nesting document, the options are the same as in the part document, with the additional possibility of :

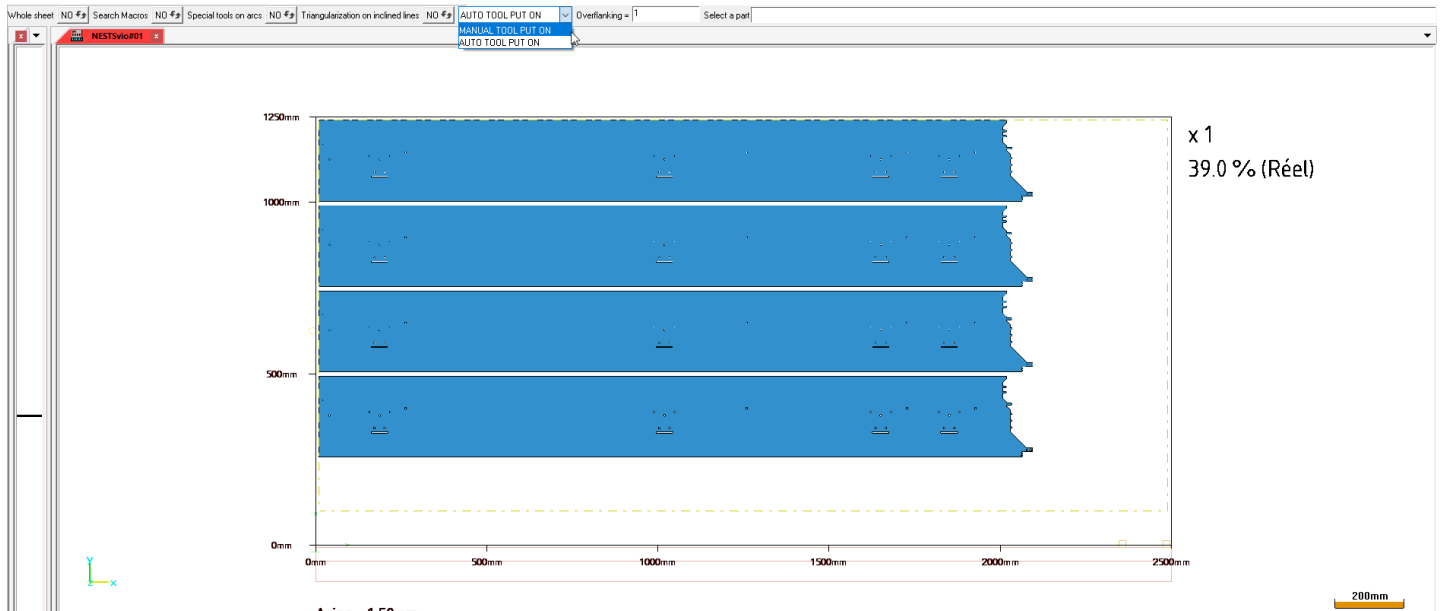
- selecting a part to be punched;



- performing the punching operation on the entire sheet.



A drop-down menu is available for mounting tools.

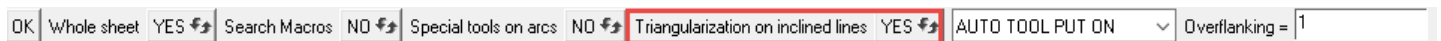


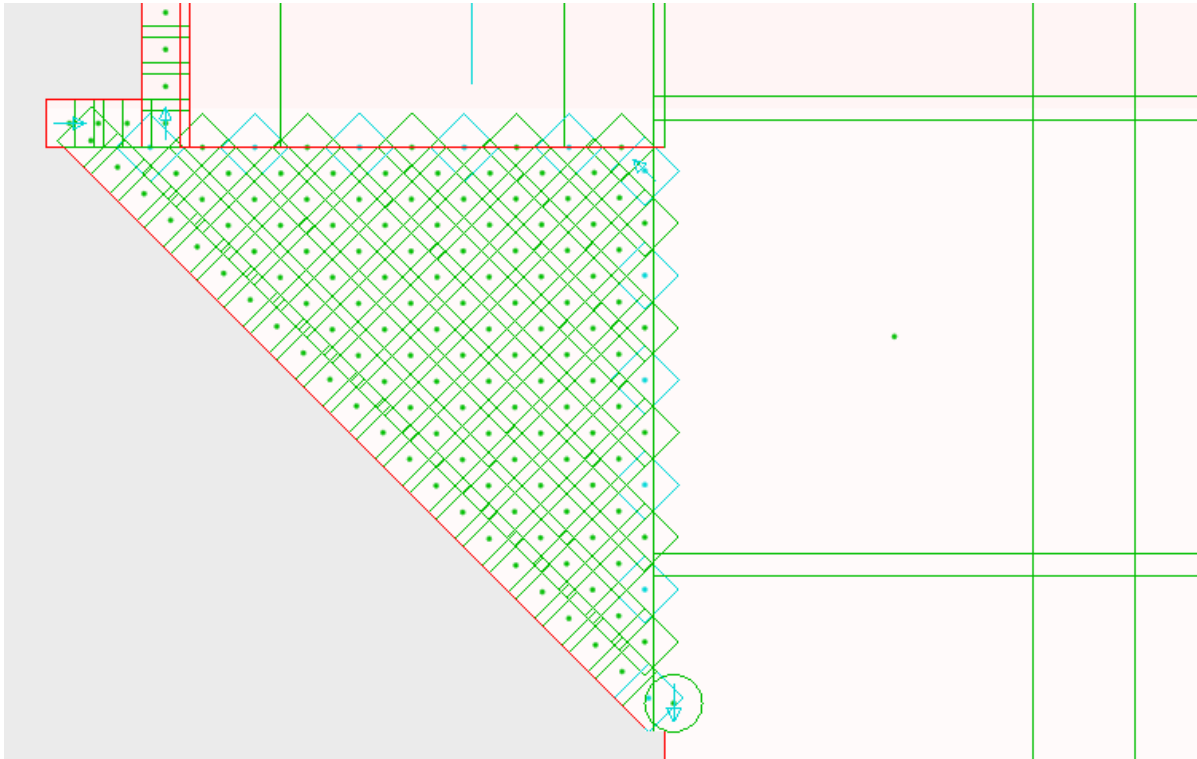
You can set an overflanking value to allow the various areas to overlap.

When activated, the macro search is always performed before the rectangle search.

You can search for special tools on arcs before or after searching for rectangles.

When the **Triangularization on inclined lines** option is activated, a triangular machining is performed on all inclined lines before the rectangles are searched.





*Example of triangularization of inclined lines.*

When the option is deactivated, the search for rectangles may generate small triangular areas which will be machined with small tools.

All machinings created are rectangular or triangular free-form geometries. The segments of the initial external contour that touch these geometries are induced. These free geometries self-destruct as soon as there is no longer a tool assigned to them.

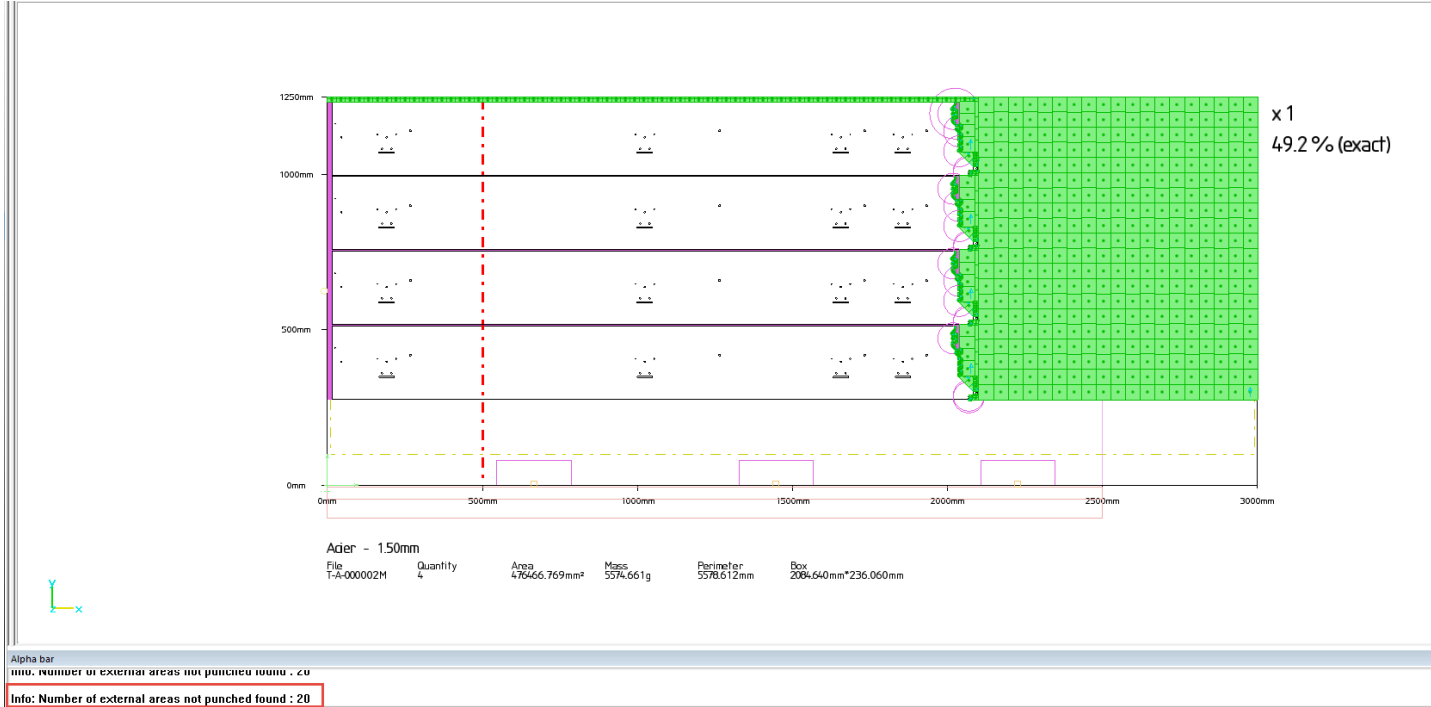
**Note:** The segments of the initial contour for which no rectangle or triangle could be found are automatically machined at the end of processing.

### Display external areas not punched

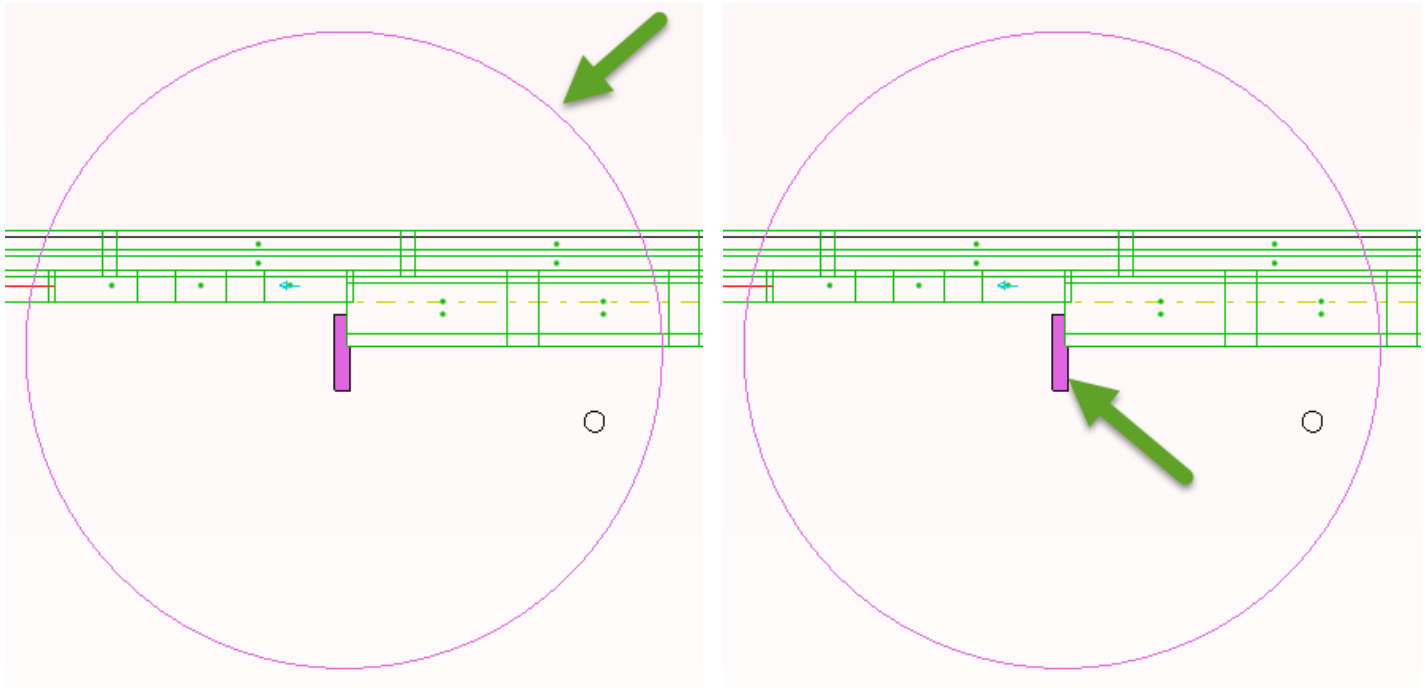
The **Display external areas not punched** command allows you to detect and view unpunched areas around the external contour of the part. These are represented by solid hatched areas in purple.

Before starting the search, the question bar lets you choose a minimum surface size and define the zoom factor on the areas detected.

After clicking on **OK**, **TopSolid** displays the areas that have not been punched, circling them in purple. The number of unpunched areas is indicated in the alpha bar.



For a better view of the first area detected, click on the **Zoom on area** button, or click on the circle around the area or on the area itself.



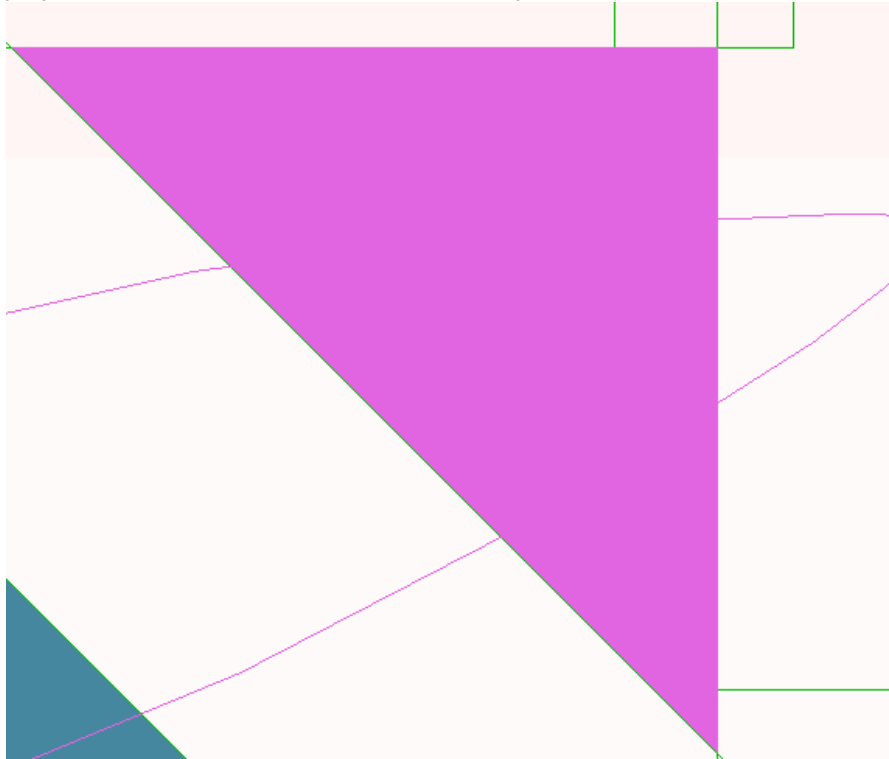
The **Next area** and **Previous area** buttons are used to move from one area to another.

The **Sequences visualization** option allows you to show or hide tool punches without leaving the command.

The **Convert area into free geometry** option converts the border area of the unmachined area into free geometry, which can be used later to manually place tools.



The area currently displayed is converted into a curve, and it is possible to edit the overflanking value to enlarge it.



After clicking on **OK**, the free geometry is created and **TopSolid** moves on to the next unpunched area.

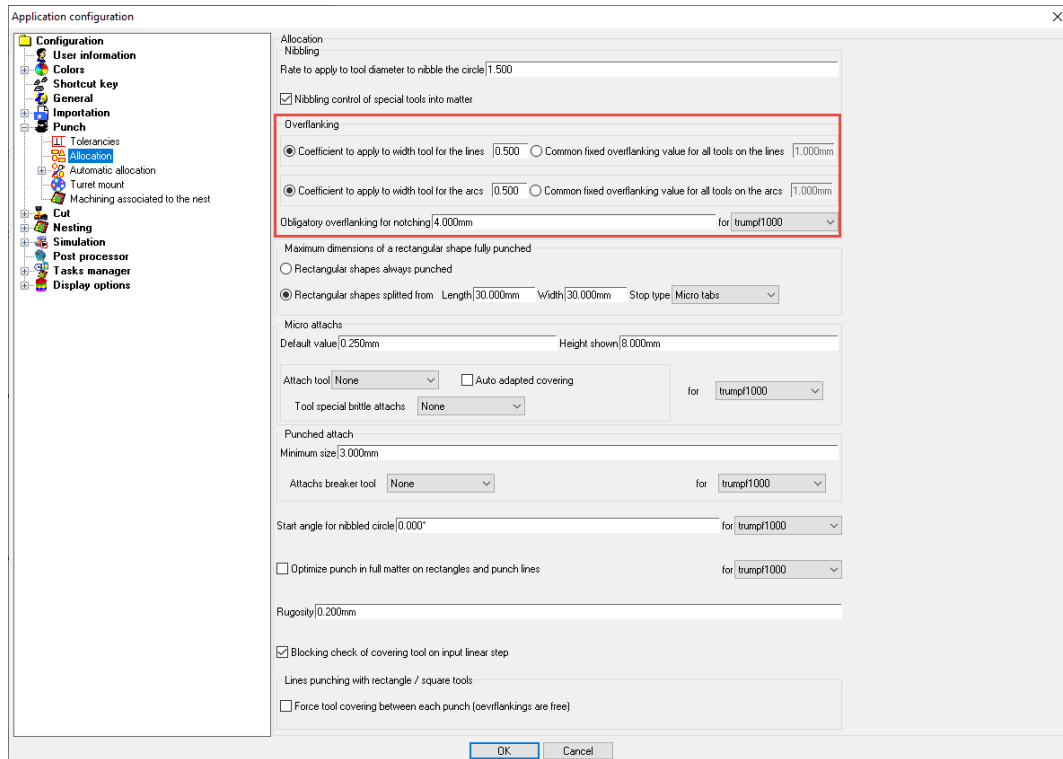
**Note:** Once the unpunched areas have been identified, you do not need to convert them to free geometries. The machining commands **Put a tool on point** and **Put a tool on point along profile** can be used to machine the remaining portions.

## Default overflanking value

In previous versions, the overflanking value for machining lines and arcs was computed with a coefficient proportional to the selected tool width.

You can now define a fixed overflanking value common to all tools in **Tools > Options > Punch > Allocation**.

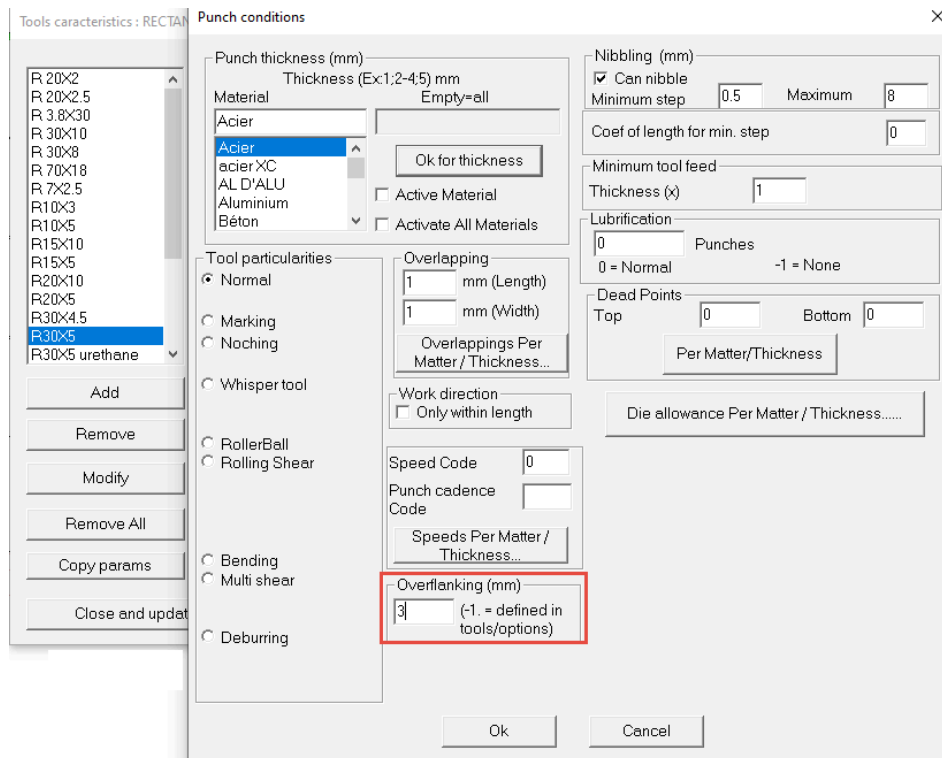
You can enter a different overflanking value for lines and arcs, and choose whether to apply a coefficient or a fixed value.



*In the above example, the overflanking will be 1mm whatever the width of the tool used.*

It is also possible to define an overflanking value per tool.

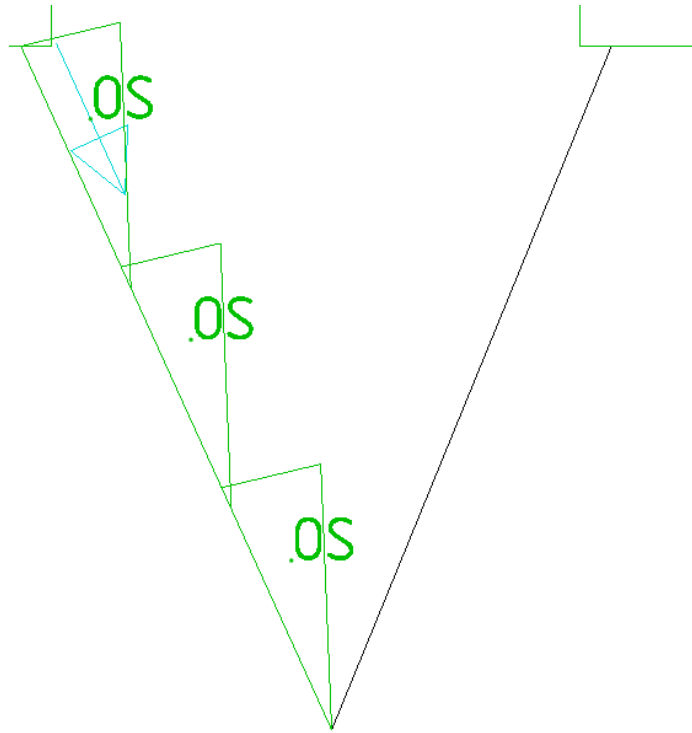
In the case below, the value is different from -1; it will be given priority for overflanking calculation.



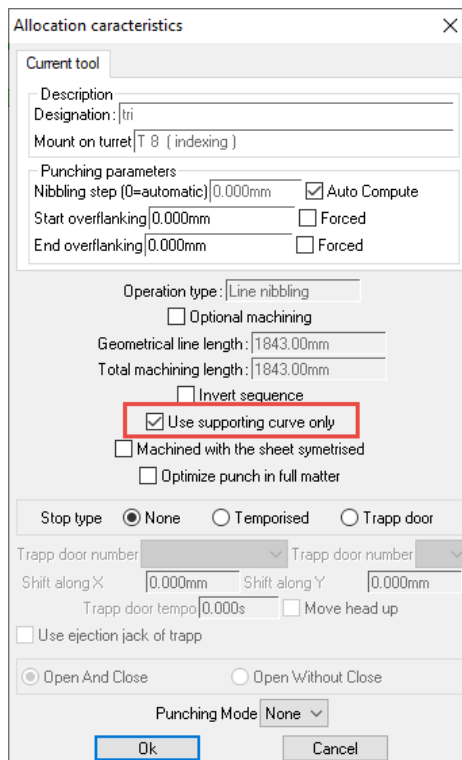
If the default tool overflanking value is -1 (value fixed in **Tools > Options**), the machining overflanking will be 1mm.

## Improved management of special tools

The command for machining lines or arcs with a special tool now manages collision checking with matter and microtabs, and allows you to use the support curve.



Using the wrench, you can modify overflankings while keeping collision management, and return to the **Use supporting curve only** mode.





## Evacuation of the clamp skeleton using trap door with rotation

On TRUMPF-type machines equipped with press strippers, the new **Evacuate skeleton in clamps using trapp door** command allows the remaining clamp skeleton to be evacuated through the trap door.

In some cases, the shift movement performed before rotation of the clamped skeleton portion crossed the clamp's safety zone. This movement is now limited along X to the defined distance between the tool center and the clamp safety zone.



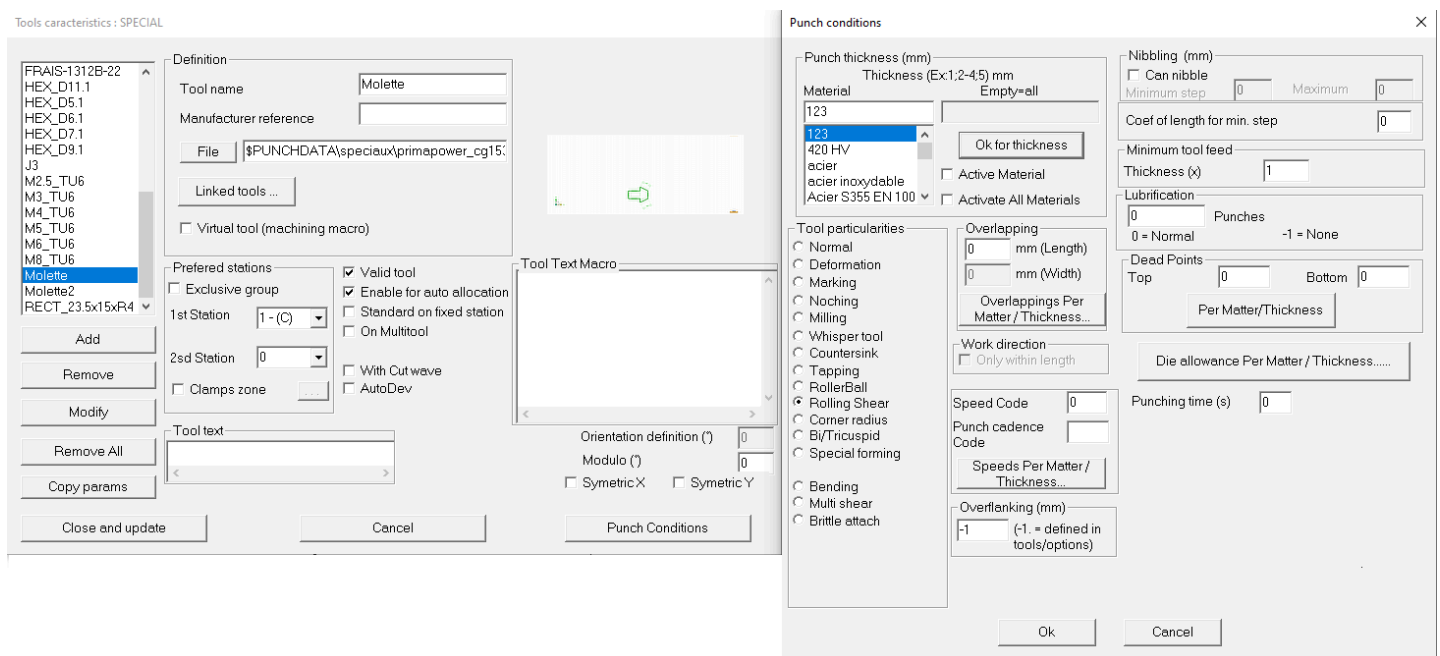
## Improved definition for rolling shear tools

Rolling shear tools can be defined like square or rectangle tools. However, when this type of tool has a very precise work orientation, the rectangle tool is not completely satisfactory, as the rectangle works in modulo 180°. Plus, the rectangle preview is not appropriate.

Rolling shear tools can now be defined from special tools, providing a customized preview that manages a modulo 360° orientation.

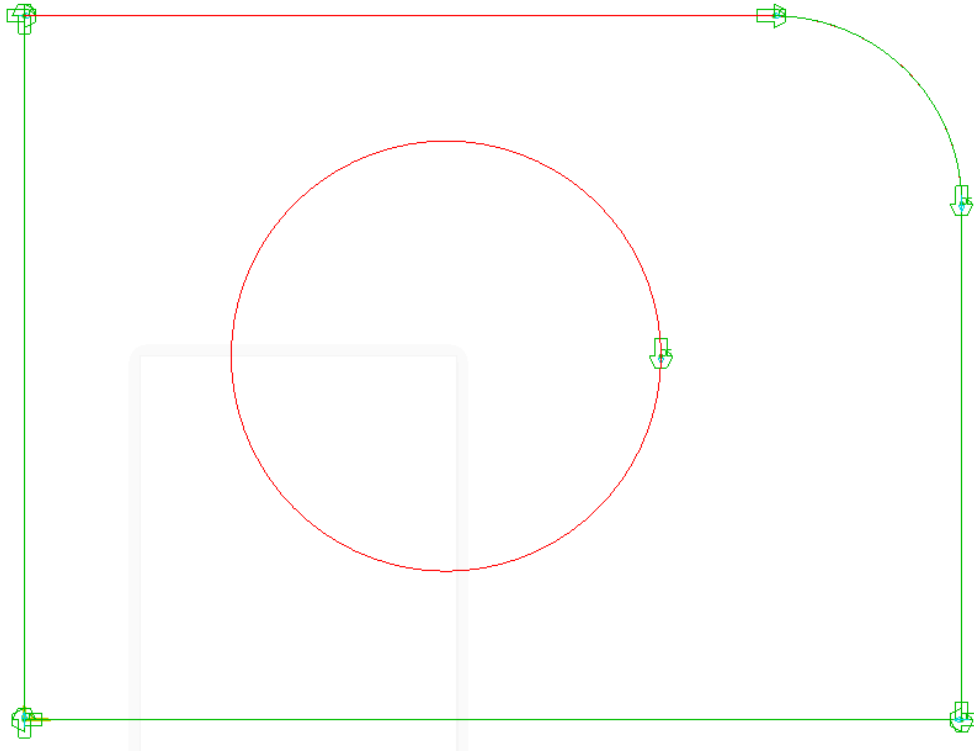
In addition, if the special tool is of the rolling shear type, **TopSolid** no longer proposes to match a segment on the special tool with the geometry to be machined and positions the special tool center directly on the geometry, aligning the tool's X axis with the geometry.

This feature can be configured in the **Line or arc with special tool** command.

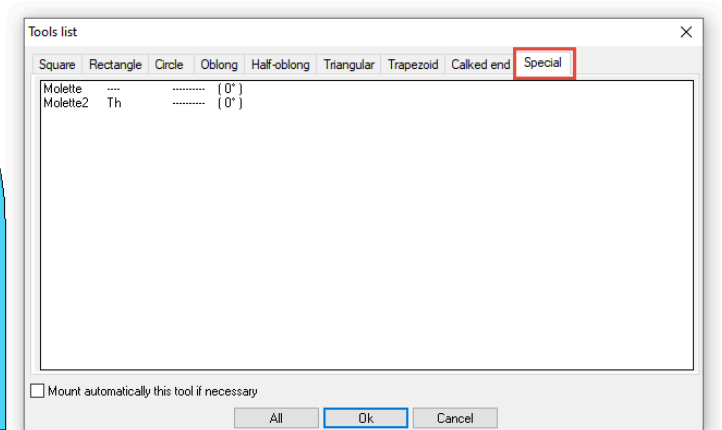
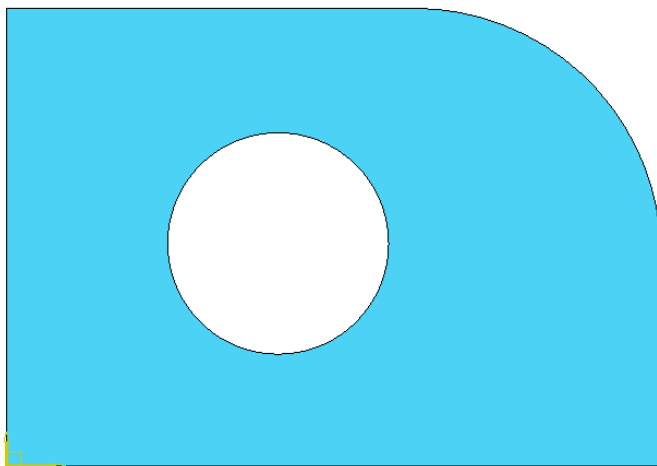


After selecting the rolling shear tool, you can click on the geometry: the tool is immediately placed in the center of the geometry.

**Note:** The geometry can be a complete circle.



The rolling shear tool can be used with the tool machining on profile command. When selecting the tool, the **Special** tab is only displayed if the tool library contains special rolling shear-type tools.

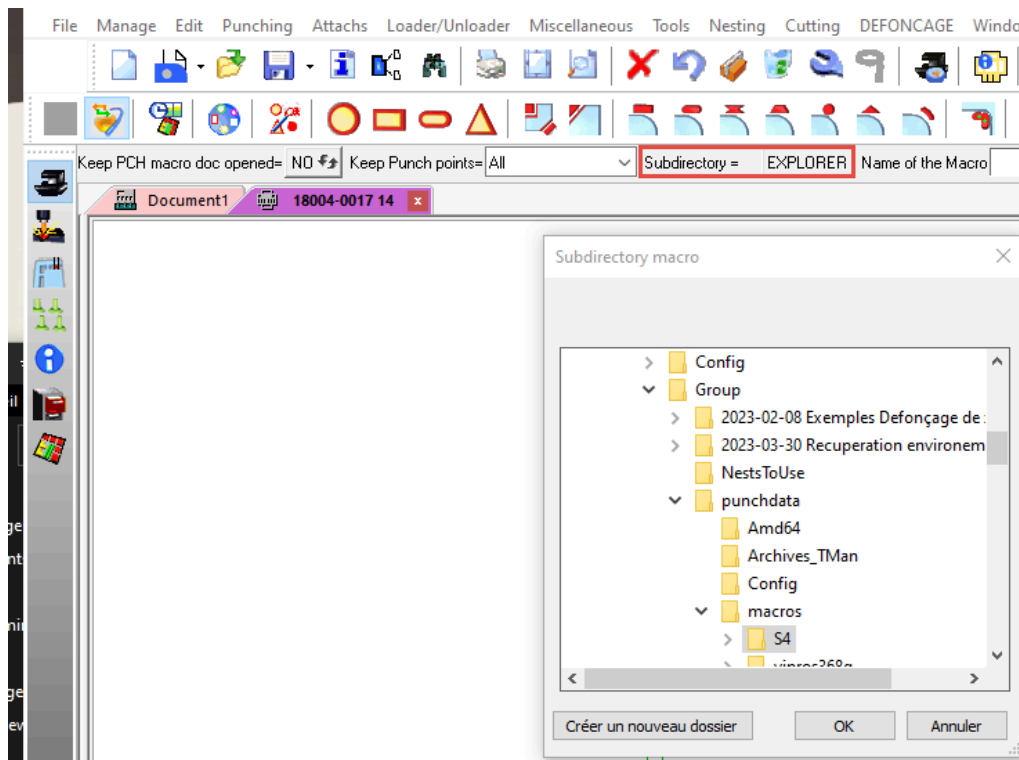


Each independent sequence can be reversed using the wrench.

## Cutting and punching

### Improved management of machining macros

When creating a machining macro, the **Explorer** button allows you to save it in a specific existing subdirectory or that you create.

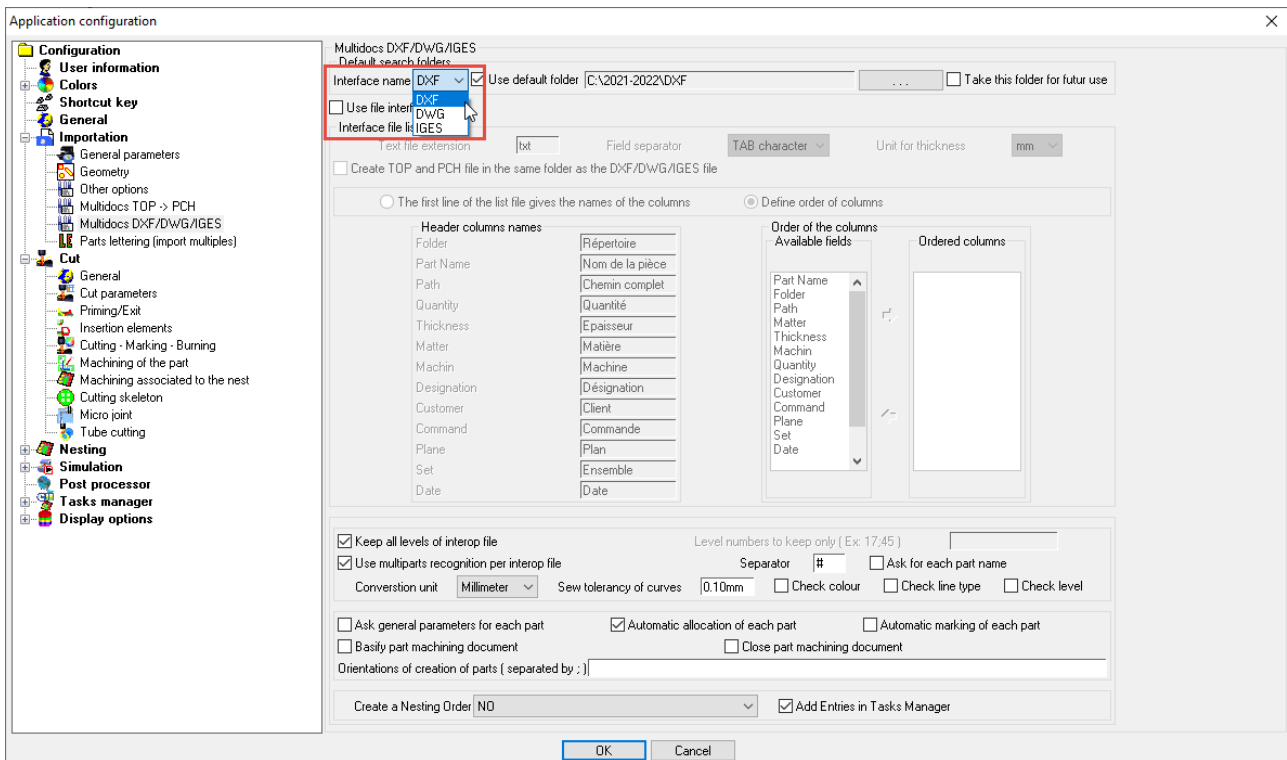


You can also delete several macros at once using multiple selection.

In addition, macros whose tools are not compatible with the matter/thickness of the current part are not loaded into memory.

## Default search folders by import type

In Tools > Options > Importation > Multidocs DXF/DWG/IGES, you can now choose to use a default import folder for each type of import (DXF, DWG or IGES).

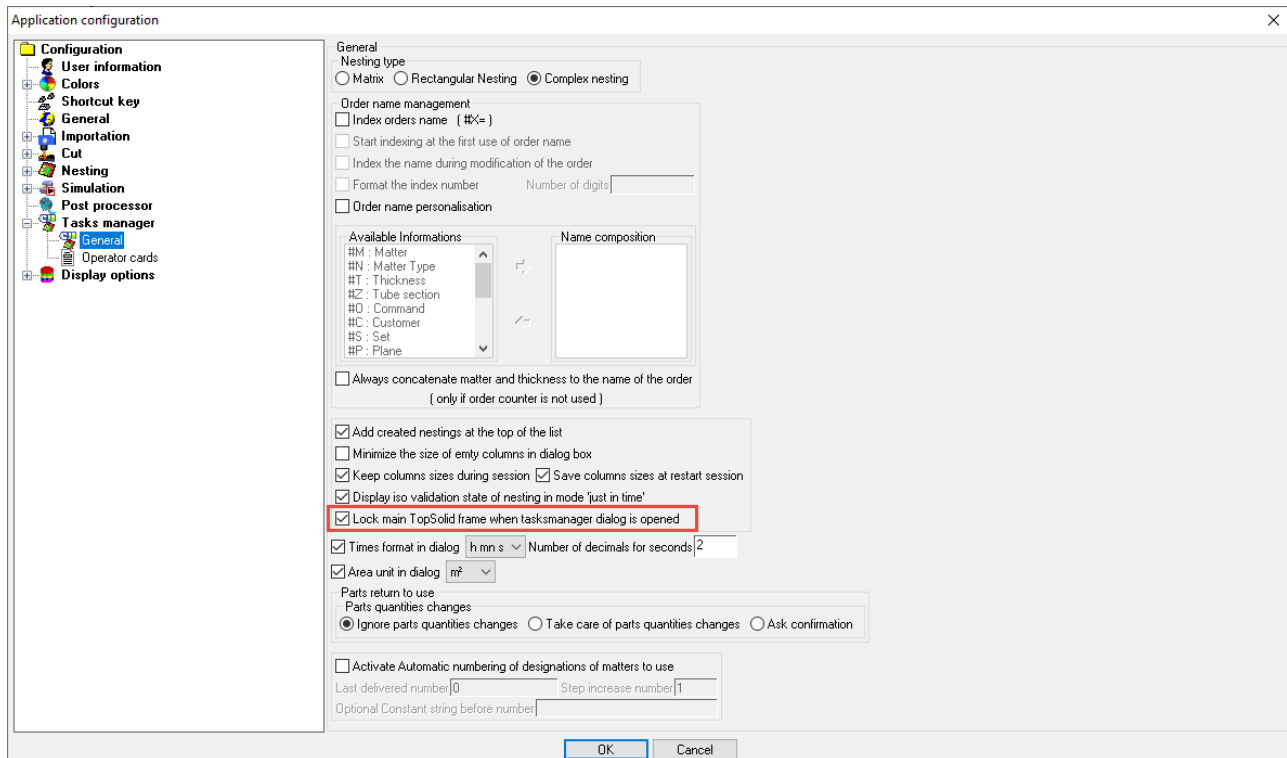


**Note:** If the DSTV interface is available, it is also possible to assign it a default folder.

## Tasks manager

### Locking the main TopSolid window

A new option to lock the main TopSolid window when the tasks manager dialog box is open is available in **Tools > Options > Tasks manager > General**.



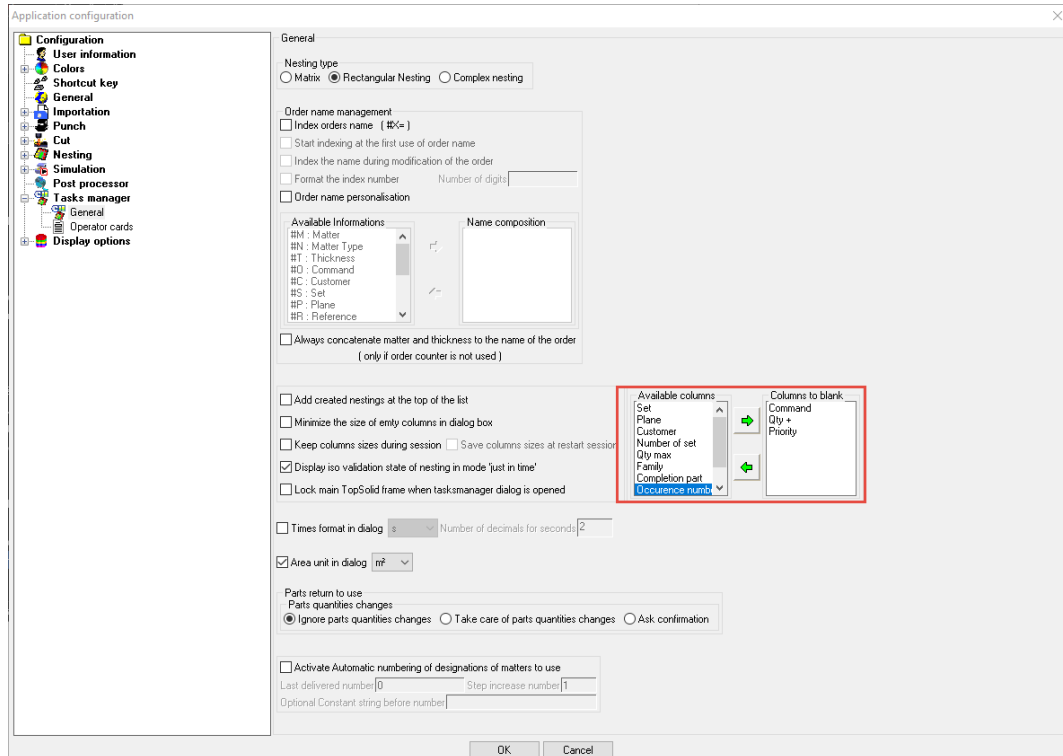
## Optimized tasks manager display

### Dialog box size

The tasks manager dialog box has been extended in height and width to accommodate high resolutions such as 1600 x 1024, 1920 x 1080 and 4K displays.

### Choice of columns to be displayed

In **Tools > Options > Tasks manager > General**, it is possible to lighten the tasks manager display by choosing not to show certain columns in the parts and matters lists.



The columns concerned are as follows:

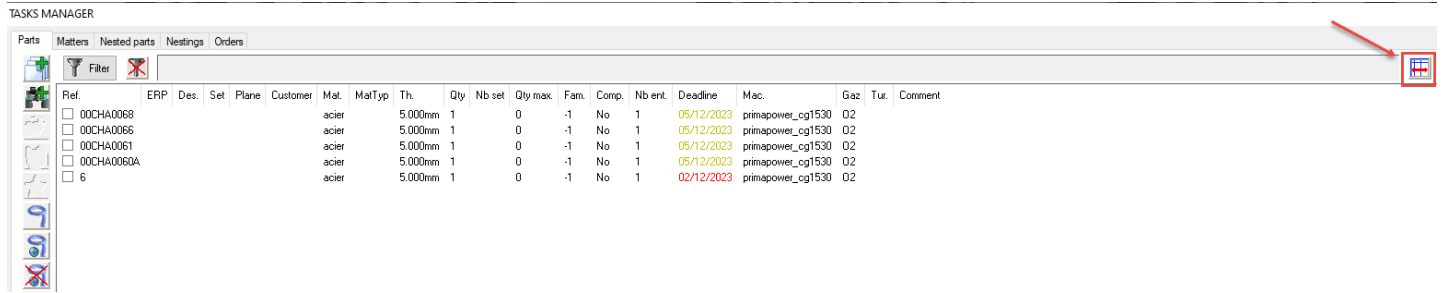
- Set
- Plane
- Command
- Customer
- Number of sets
- Qty +
- Maximum quantity per sheet
- Family
- Completion part
- Nesting priority
- Number of occurrences per PCH file
- Gas
- Turret

When you change one or more columns not to be displayed, the memorized column sizes are reset.

In previous versions, the **Class** and **Tube section** columns were visible even if there was no tube machine, and were reduced to zero width. These columns are no longer displayed in cases where no machine tubes are present, in order to simplify the tasks manager display.

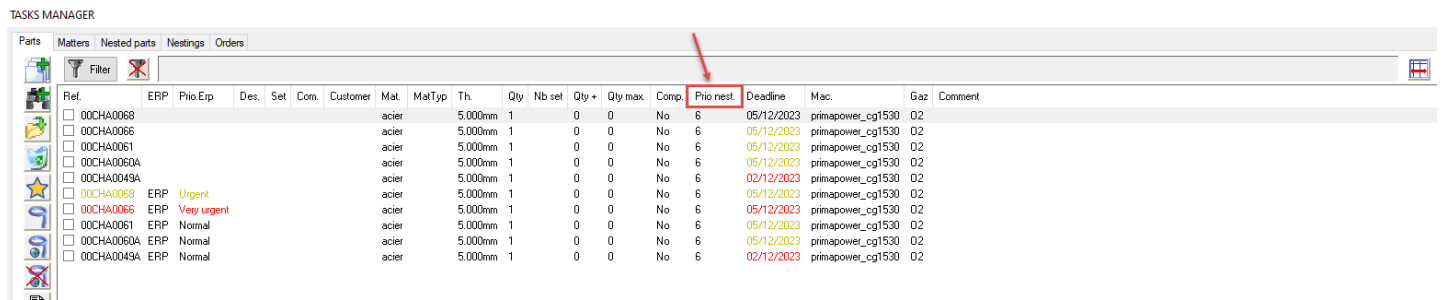
## Column size

A new icon automatically adjusts the size of columns according to their content.



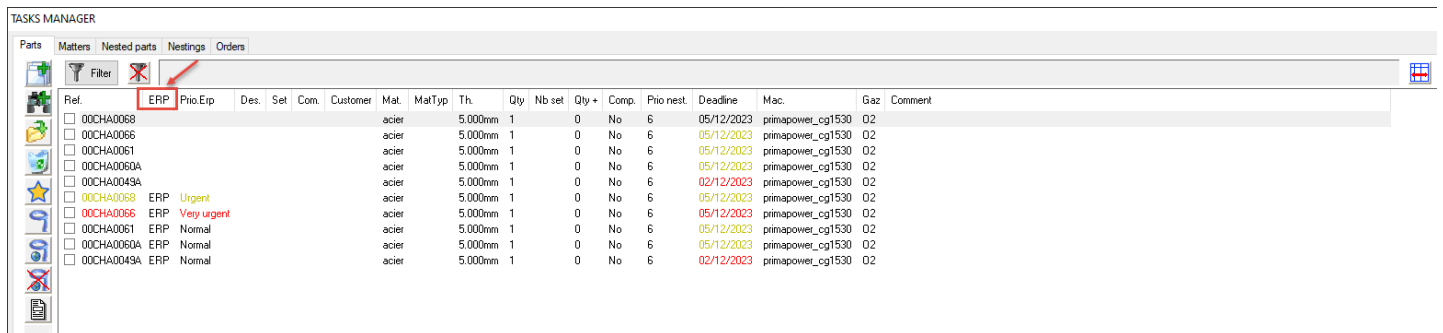
## Nesting priority

The old **Priority** column has been renamed **Prio nest**.



## ERP-related improvements

When the tasks manager is linked to **TopSolid'Erp**, the new **ERP** column appears in the **Parts** and **Matters** tabs of the manager, allowing you to quickly differentiate between parts from the ERP and other parts added manually. This column is always displayed.



In addition, the new **Prio Erp** column indicates the urgency of the part from **TopSolid'Erp**: normal, urgent or very urgent.

**Note:** ERP priority for urgent and very urgent parts overrides nesting priority.

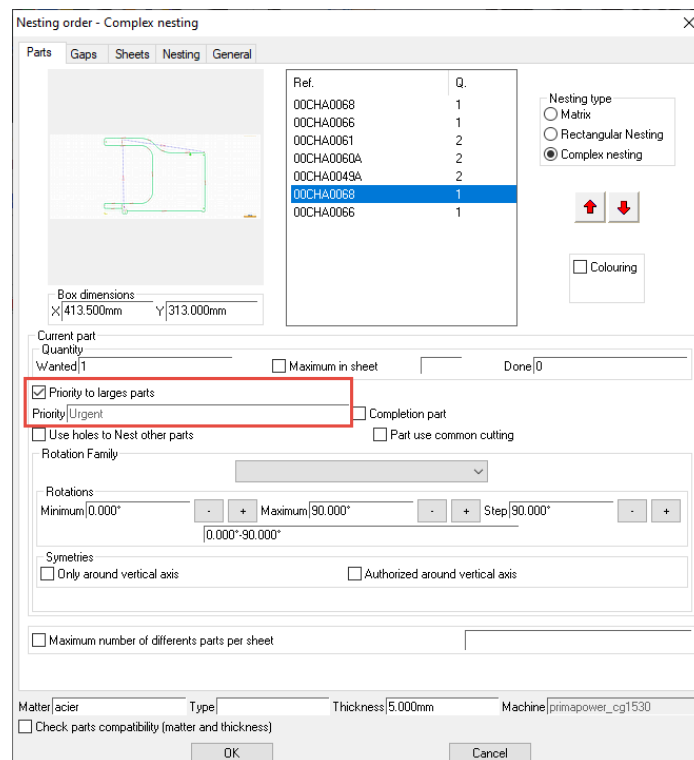
TASKS MANAGER

Parts Matters Nested parts Nestings Orders

Filter

Ref.	ERP	Pro.ERP	Des.	Set	Com.	Customer	Mat.	MatTyp	Th.	Qty	Nb set	Qty +	Comp.	Pho nest.	Deadline	Mac.	Gaz	Comment
00CHA0068							acier		5,000mm	1	0	No	6	05/12/2023	primapower_cg1530		02	
00CHA0066							acier		5,000mm	1	0	No	6	05/12/2023	primapower_cg1530		02	
00CHA0061							acier		5,000mm	1	0	No	6	05/12/2023	primapower_cg1530		02	
00CHA0060A							acier		5,000mm	1	0	No	6	05/12/2023	primapower_cg1530		02	
00CHA0049A							acier		5,000mm	1	0	No	6	02/12/2023	primapower_cg1530		02	
00CHA0068	ERP	Urgent					acier		5,000mm	1	0	No	6	05/12/2023	primapower_cg1530		02	
00CHA0066	ERP	Very urgent					acier		5,000mm	1	0	No	6	05/12/2023	primapower_cg1530		02	
00CHA0061	ERP	Normal					acier		5,000mm	1	0	No	6	05/12/2023	primapower_cg1530		02	
00CHA0060A	ERP	Normal					acier		5,000mm	1	0	No	6	05/12/2023	primapower_cg1530		02	
00CHA0049A	ERP	Normal					acier		5,000mm	1	0	No	6	02/12/2023	primapower_cg1530		02	

ERP priority cannot be modified either in the tasks manager or in the nesting order, since this information is managed by ERP.



TopSolid nests very urgent parts first, then urgent parts and finally others, respecting their nesting priority.

**Important:** ERP priority is only managed with the complex nesting engine. It is inoperative for matrix or rectangular nestings.

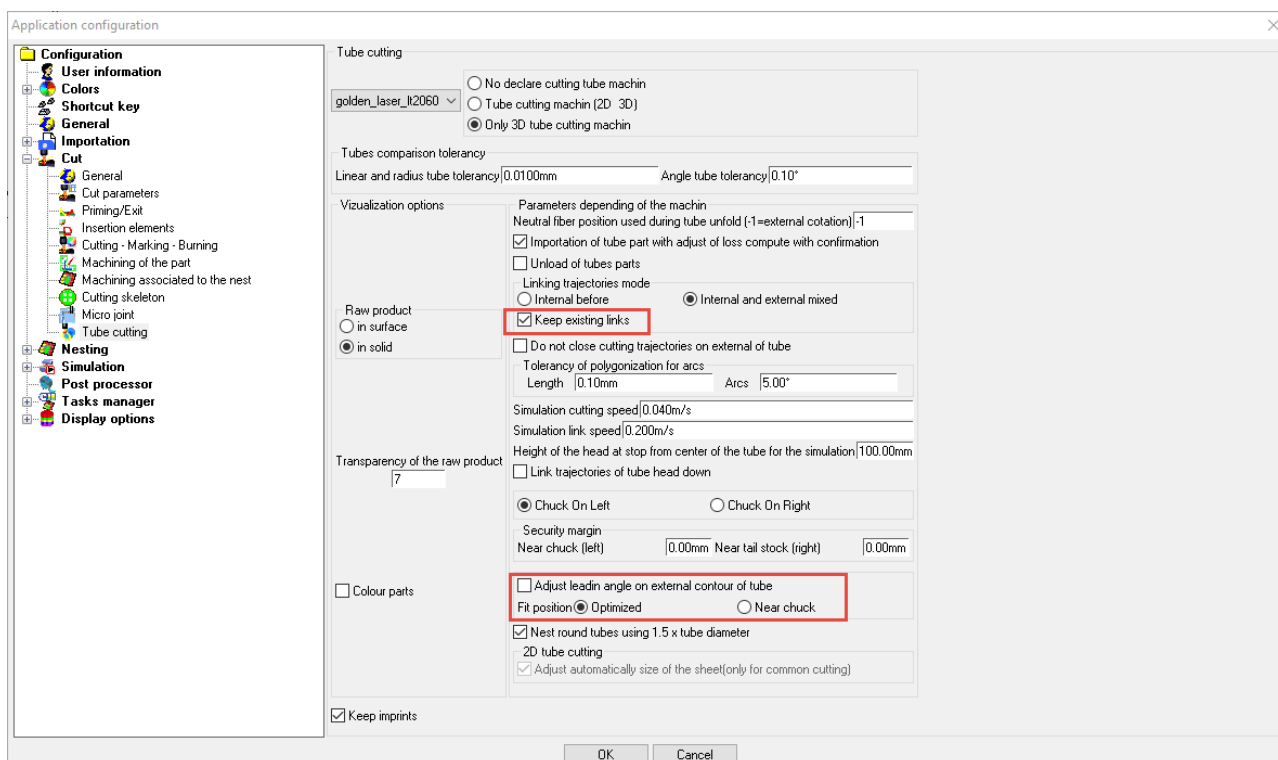


## Tubes

### Automatic tube cutting machining

Two new options are available in **Tools > Options > Cut > Tube cutting** for automatic machining of tube cuttings:

- an option to keep existing links;
- an option for positioning the lead in as close as possible to the chuck.



These options are also available in the question bar of the manual cutting command.

